CENTURION V

Operation Manual

Version 1.3
December , 1990

MILLTRONICS MANUFACTURING COMPANY
7870 Park Drive
Chanhassen, MN 55317
(612) 474-8100

COMPANY BACKGROUND

IIC, Inc., parent company of Milltronics Mfg., was born in the basement of a shopping center in 1973. The first two years showed limited results, but Gary Welch, President, persevered in his design and desire to build his own company.

Gary's roots go back to Racine, Wisconsin, moving to Minneapolis when he was twelve years old. He graduated from the University of Minnesota with a degree in electrical engineering and found employment with Control Data Corporation. It was while working for Control Data that he designed his idea of a small industrial computer.

Tim Rashleger, V.P. Marketing, also hails from Racine, Wisconsin, where he served Tree Machine Tool Company as Sales Manager. Prior employment included Gettys Mfg. and Westamp, where he sold and applied servo systems. He has an Associate Degree in Electronics and served in the USAF as an electronics technician.

Bonar Sabby, V.P. Customer Service, also served in the USAF as an electronics technician after graduating from the University of Wisconsin in Mathematics. Bonar comes from Baraboo, Wisconsin, later moving to Racine to work for Gettys Mfg. and Tree Machine largely as the Field Service Manager.

In 1981 the company was reorganized and Gary, Tim, and Bonar became partners. Most of the IIC business was in manufacturing controls for other machine tool companies.

MILLTRONICS MANUFACTURING COMPANY, a division of IIC, Inc., was formed in 1983 to serve the end-user market with milling machines. The first year of the Milltronics division (1984) is a success story in itself, with over a million dollars of milling machine sales. Milltronics anticipates 1991 sales to exceed $8,000,000.00. Growth in milling machines is also expected to expand even faster with the introduction of the U.S.-manufactured Partner I and the new Centurion 5 CNC.

Today IIC, Inc., now more commonly known as Milltronics Mfg., supplies technically advanced control systems for all forms of motion control. With a base of over 4000 control systems in the field and its new product developments, its growth potential is unmatched in the industry.

Superior products and great employees will continue to keep Milltronics Mfg. a success story. The owners realize their best asset is their employees, and they manage with a unique style that includes incentives, pride in workmanship, and a common goal to succeed.

MILLTRONICS MANUFACTURING is proud of our employees, our products, our reputation, and our prompt, courteous service. We realize our future depends largely on providing the best possible support and product to our customers. We look forward to an opportunity to work with you in developing the best possible results together and invite you to visit our facility and meet the people who make it happen.

MILLTRONICS MANUFACTURING CO.
7870 Park Drive
Chanhassen, MN 55317
Phone (612) 474–8100
FAX (612) 474–7289

Centurion V designed and manufactured in Chanhassen, MN.

TABLE OF CONTENTS

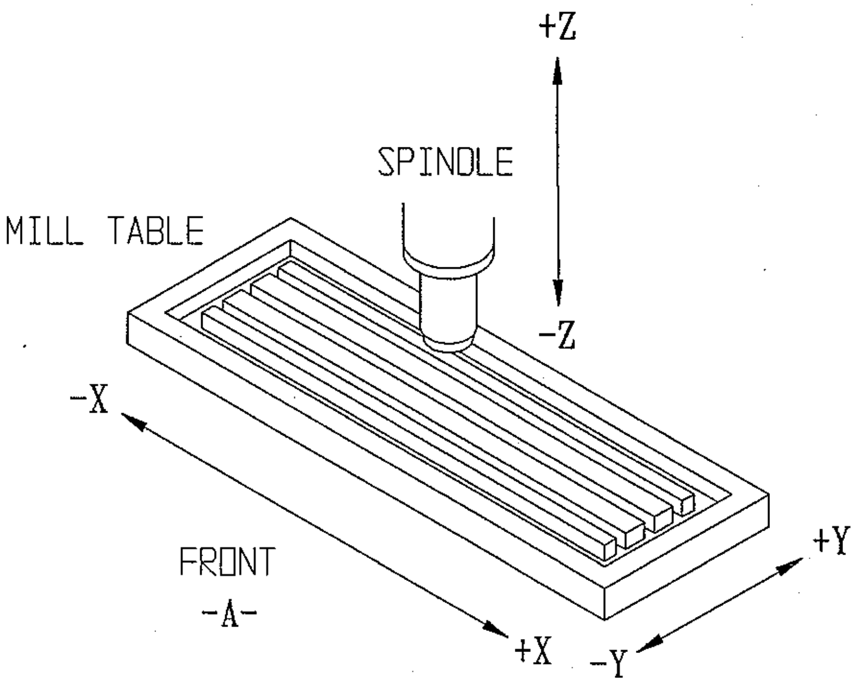
PREFACE

The Centurion V has three controllable axes in its basic configuration: X, Y, and Z. This manual assumes that the tool moves with respect to the workpiece.

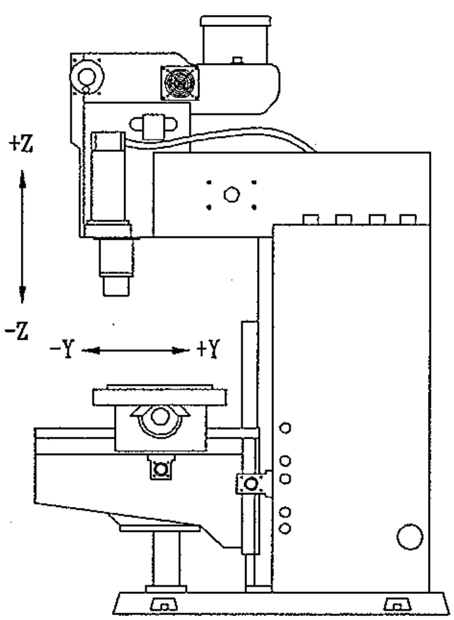
This manual is divided into two sections, M & G code programming and conversational programming. The conversational programming section is designed primarily to explain the various menus, screen entries, and the general flow from one screen to another. It does not go into any detail on how the various functions work. Detailed explanations of each function are covered in the M & G code section and should be referenced there if any questions arise.

AXIS DEFINITIONS

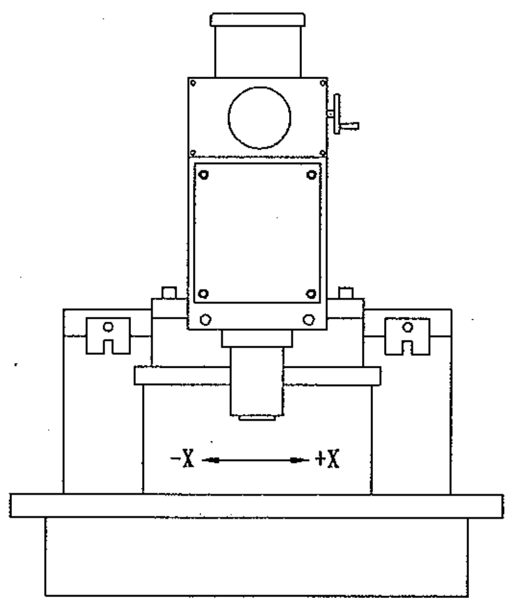
All directions are referenced with respect to the tool. The following illustrates the X, Y and Z directions.



-A-



-B-

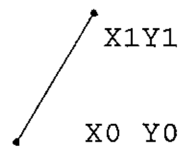


-C-

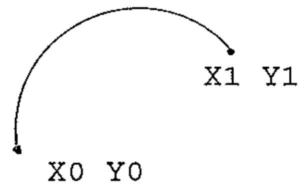
1. INTRODUCTION

A group of commands given to the CNC for operating the machine is called a program. By specifying commands the tool is moved along a straight line or arc, and machine functions such as coolant on/off, tool change or spindle on/off are performed.

The function of moving the tool along straight lines and arcs is called interpolation.

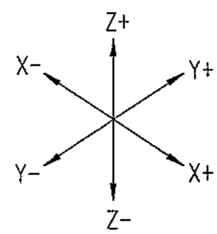


linear interpolation



circular interpolation

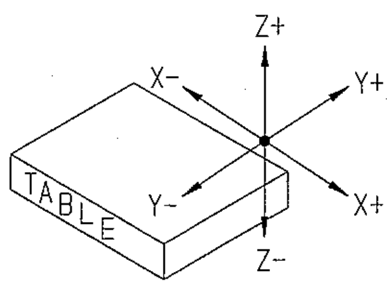
When the commanded position to be reached by the tool is executed, the CNC moves the tool to that position via one of the interpolation modes, circular or linear. The position is given as a coordinate value in a rectangular cartesian coordinate system.



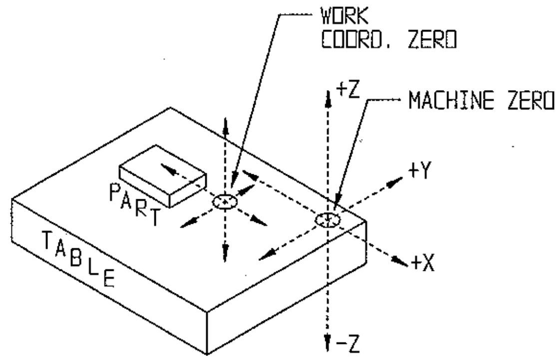
coordinate

The following types of coordinate systems are available:

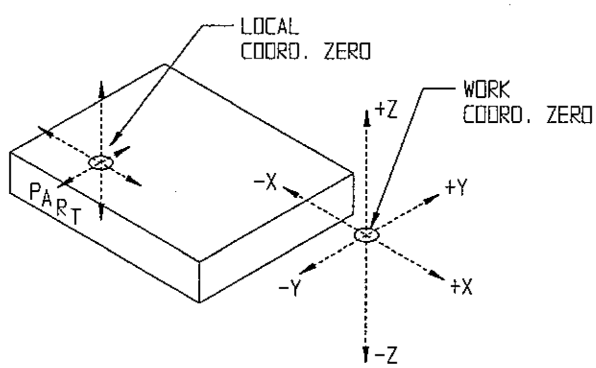
1. Machine system



2. Work coordinate system

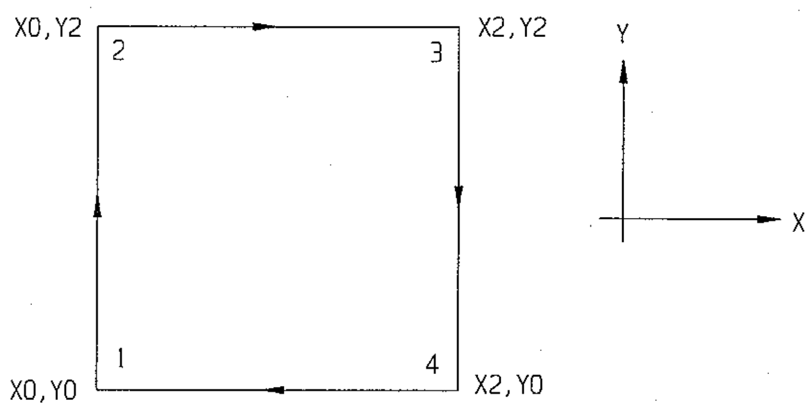


3. Local coordinate system



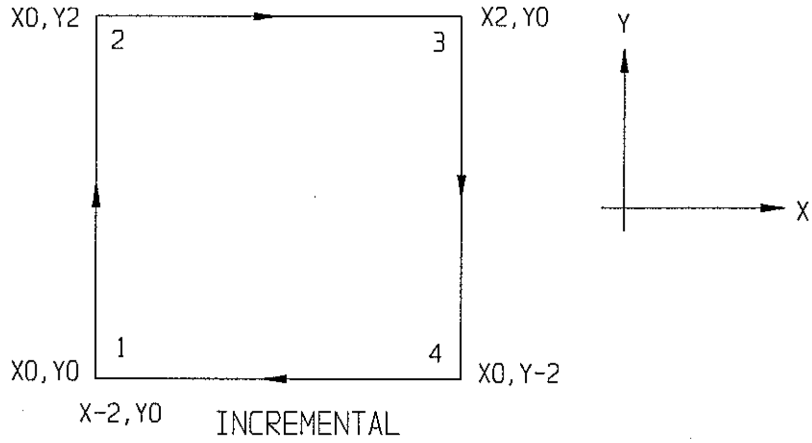
The position to be reached by the tool is commanded with a coordinate value referenced to one of the above coordinate systems. The coordinate value consists of one component for each axis, X, Y and Z.

Coordinate values may be given in either absolute or incremental mode. In absolute mode the tool moves to a point the programmed distance from the zero point of the coordinate system.



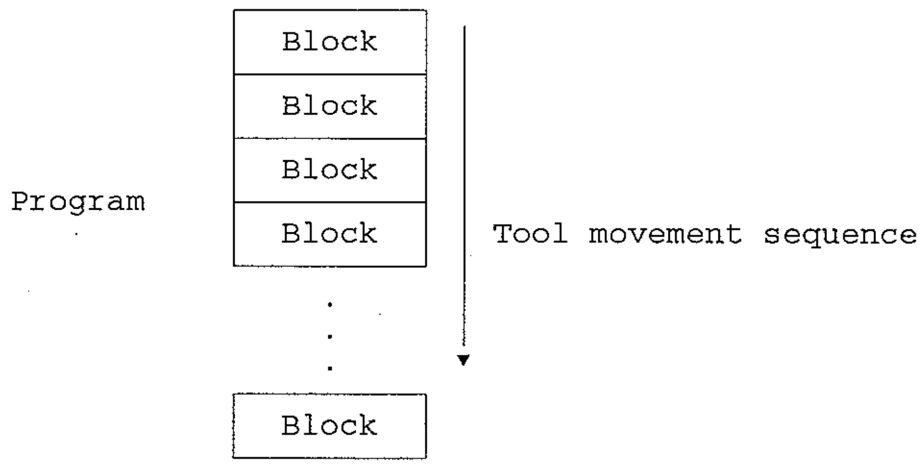
ABSOLUTE

In incremental mode the tool moves to a point the programmed distance from the current tool position.



2. PROGRAM CONFIGURATION

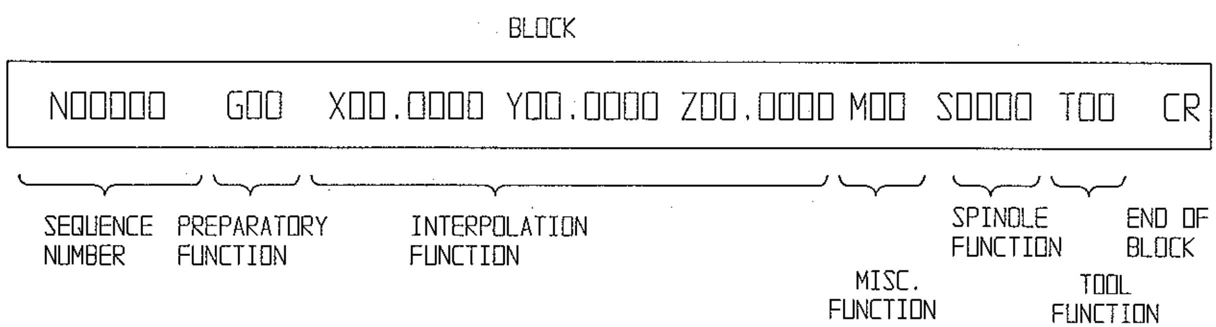
By definition, a program is a group of commands given to the CNC for operating a machine. By specifying commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In a program, specify the commands in the sequence of actual tool movements.



A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machine moves. An optional number for definition of each move is called the block number, and the number for naming each program is called the program number.

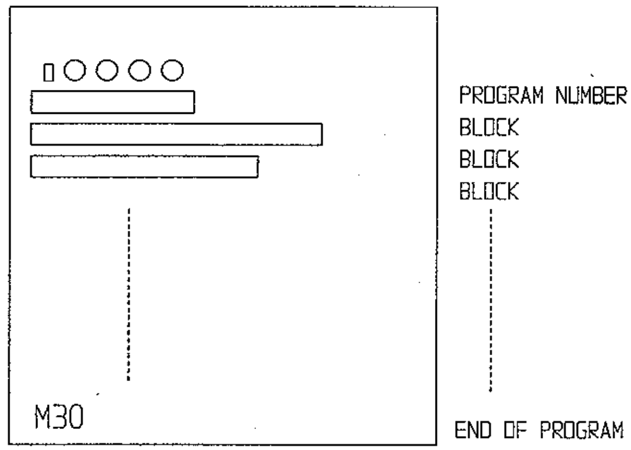
The block and the program have the following configurations:

2.1 Block



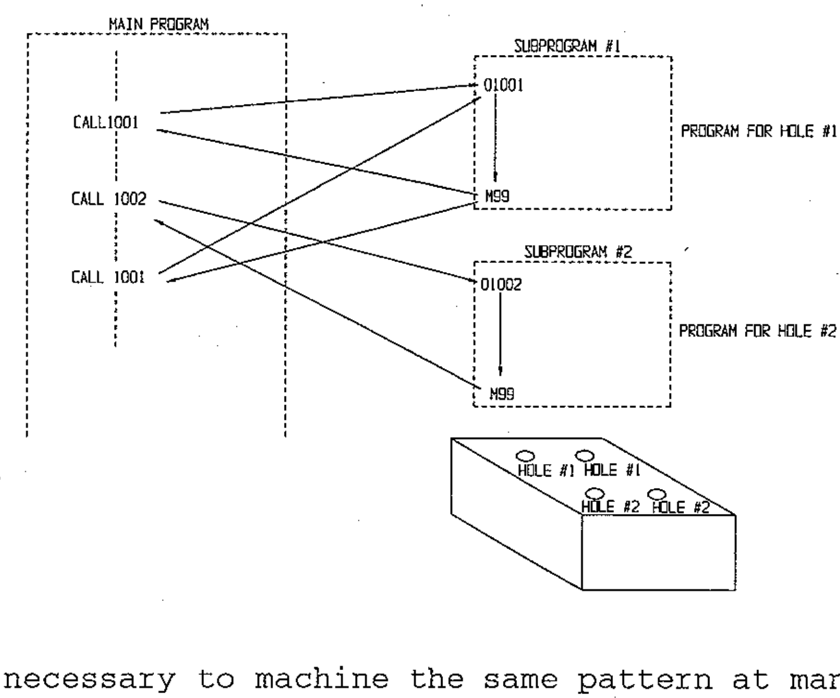
Each block begins with an optional number and ends with a <CR> carriage return.

2.2 Program



Normally a program number is specified at the beginning of a program, and a program end code (M02, M30) is specified at the end of the program. Neither is required, however, and it may be advantageous to omit the program end code from programs which may be used as subprograms. An end program code is assumed when the end of the main program is encountered.

2.3 Main program, subprogram and subroutines



When it is necessary to machine the same pattern at many places on a part, a program for the pattern should be created. This is called a subprogram. When an M98 (subprogram "CALL" command) appears in the main program, the commands of the subprogram are executed before execution of the next block of the main program.

Subprograms can be used to build part libraries of commonly used patterns. Subprograms can reside anywhere in memory.

2.4 Command format ranges

The basic address and command value ranges are listed in Table 1. Note these figures give the maximum numerical limit for the control. These limits will always be greater than or equal to the physical limits of the machine. The machine limits are set via parameters in the machine setup section of the control.

FUNCTIONS	COMMAND LETTER	INCH INPUT	METRIC INPUT
Subprogram # and Program #	O	1 - 9999	1 - 9999
Sequence #	N	1 - 99999	1 - 99999
Preparatory function	G	0 - 99	0 - 99
Dimension * words	XYZUVWQ ABCIJKRP	0 ± 999.9999	0 ± 9999.9999
Dwell	P	.01 - 9999.99	.01 - 9999.99
Feedrates *	F	.1 - 999.9	.1 - 999.9
Spindle speed *	S	1 - 9999	1 - 9999
Tools	T	0 - 99	0 - 99
Misc. function	M	0 - 99	0 - 99
Repeat or loop	L	0 - 9999	0 - 9999

Table 1
Command Format Ranges

* These functions have selectable decimal positions. There may be any number of leading or trailing places as long as the total number of digits fits in the field.

2.5 Command formats for axes: M and G Codes

Axis commands can be programmed in a calculator format. No leading or trailing zeros are necessary. Whole numbers may be programmed without the decimal point. A decimal point may be used with mm, inches or second values. The location of the decimal point is as follows:

Z15.0	Z15 millimeters or Z15 inches
F10.0	10 mm/rev, 10 mm/min., 10 inch/rev or 10 inch/min.
G04 P1	Dwell for one second

These addresses can be used with a decimal point: X, Y, Z, U, V, W, A, B, C, I, J, K, R, F, P, Q, AA, AB

Axis Min/Max Values

	Least increment	Maximum value
Metric	0.001 mm	99999.999 mm
English	0.0001 inch	99999.9999 inch
Degrees	0.001 deg	99999.999 deg

Axis positions are stored in floating point, therefore greater than 8 digit commands will be accepted. These commands will be rounded off to 8 digits at the time of execution.

3. PREPARATORY FUNCTIONS G CODES

The preparatory function code is a two digit number preceded by the letter G. Preparatory functions are used to determine the program operating mode and are divided into two types, one-shot and modal.

One-shot G codes are in effect only during execution of the block in which they are present. **Modal** G codes establish operating modes which remain in effect until replaced by another mode in the same category.

The following is a list of G codes accepted by the Centurion V control system. Each code will have a detailed explanation later in the manual.

G CODES

		Active on Power Up	Modal	One Shot
00	Positioning		X	
01	Linear Interpolation	X	X	
02	Circular/helical interpolation CW		X	
03	Circular/helical interpolation CCW		X	
04	Dwell			X
09	Exact stop			X
10	Set data on		X	
11	Set data off	X	X	
17	XY plane	X	X	
18	ZX plane		X	
19	YZ plane		X	
20	Inch input	X	X	
21	Metric input		X	
22	Safe zone check on		X	
23	Safe zone check off	X	X	
24	Circular pocket clear			X
25	Circular finish inside			X
26	Circular finish outside			X
28-30	Reference point return			X
31	Z to clearance			X
32	Z to tool change			X
34	Rectangular pocket clear			X
35	Rectangular finish inside			X
36	Rectangular finish outside			X
40	Cutter compensation cancel	X	X	
41	Cutter compensation left		X	
42	Cutter compensation right		X	
43	H offset added		X	
44	H offset subtracted		X	
49	Cancel H offset	X	X	
50	Scaling cancel	X	X	
51	Scaling set		X	

52	Local coordinate system set		X	
53	Machine coordinate system			X
54	Work coordinate 1 system	X	X	
55-59	Work coordinate 2-6 system		X	
60	Single direction positioning			X
61	Exact stop mode		X	
63	Tapping mode		X	
64	Cutting	X	X	
65	Non-movement		X	
68	Set rotation		X	
69	Cancel rotation	X	X	
70	Cancel mirror	X	X	
71	Set mirror		X	
72	Bolthole routine		X	
73	Woodpecker		X	
74	Left hand tapping		X	
80	Cancel canned cycle	X	X	
81	Drill		X	
82	Drill/dwell		X	
83	Peck/drill		X	
84	Right hand tapping		X	
85	Bore		X	
86	Bore/spindle stop		X	
89	Bore/dwell		X	
90	Absolute dimension	X	X	
91	Incremental dimension		X	
92	Work coordinate chg. (set fl. zero)		X	
98	Canned cycle initial level return			X
99	Canned cycle R point level return			X

Table 2 G Codes

3.1 Interpolation functions

3.1.1 Positioning (G00) rapid traverse (modal)

G00 specifies positioning in rapid traverse mode. There is no need to program rapid traverse rates because the rates are preset by parameters. Rapid traverse rates can be overridden by the feedrate override switch on the machine operator's panel:

G00 moves the tool at a rapid traverse rate to a position in the work coordinate system for both incremental and absolute commands.

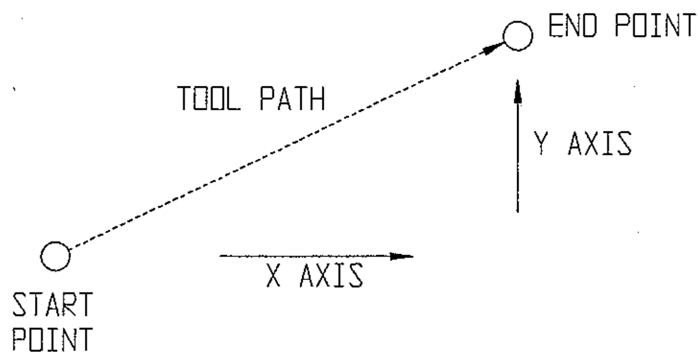
Format: G00 ~-;

where ~- is: a combination of optional axis address (of X, Y, Z, A, B, C) as X-Y-Z-A-...

where ; is: End of block (CR for EIA ASCII code)

This manual uses this notation hereinafter.

The programmed feed remains in the feedrate register and can be activated by cancelling the G00 command with a G01 command. The motions of all axes in G00 mode will be interpolated with all axes reaching the end point simultaneously.



Note 1: *The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, the rapid traverse rate cannot be specified in the address F. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position. "In-position" means that the axis position is within a specified range. (This range is determined by the machine tool builder.)*

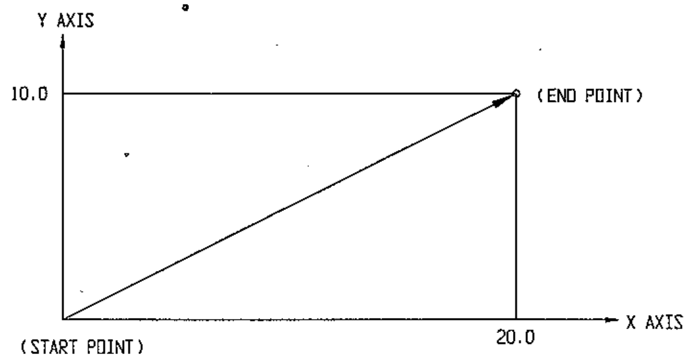
G00 mode automatically accelerates and decelerates in a linear fashion allowing the controlled axis to start and stop smoothly. The rate of accel/decel can be changed by the machine tool builder.

3.1.2 Linear interpolation

G01 ~____F____;

This command actuates the linear interpolation mode. The value of ~ defines the distance the tool will travel. The feedrate is set to a cutting feed by the F code and is modal. An example follows:

(G91) G01 X20 Y10 F20 ;



The feedrate specified by the F code is the vector rate along the path, not the rate of each axis.

3.1.2.1 Polar definition of a line

A polar line is specified by a polar radius/length (R), an angle (AB), and a polar center (AA or I, J, K, or XC, YC, ZC).

Polar definitions are valid in any plane. The 3 o'clock position is always 0 degrees. Positive angles result in CCW rotation of the polar radius, while negative angles result in CW rotation of the polar radius. Polar lines can be used when estimating lengths during trip help.

If the polar radius/length (R) or angle (AB) is not specified, then the previous values will be used. If the polar center is not specified then it is taken to be the current machine position.

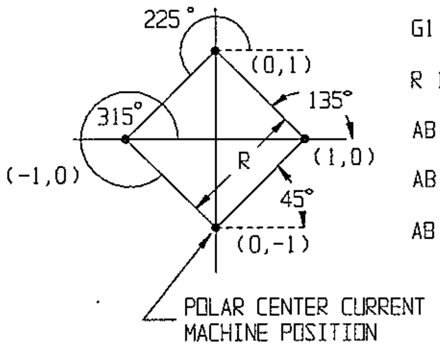
Some examples:

Cutting Feedrate G01, G02, G03 Mode

The feedrate of linear interpolation (G01) and circular interpolation (G02, G03) are commanded with numbers after the F code (FXXX.X).

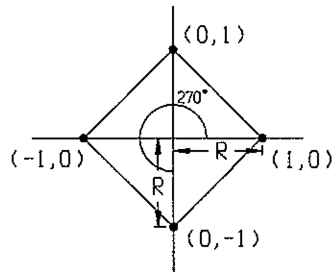
The F command can appear anywhere in a block and specifies the rate of motion in inches or millimeters per minute.

NO POLAR CENTER



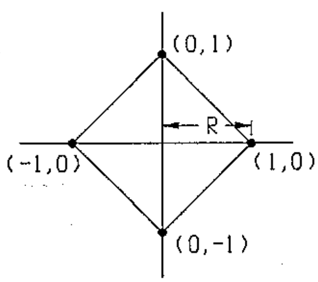

```
G1  G17  X0  Y-1
R 1.414  AB45
AB 135
AB 225
AB 315
```

ANGLE FROM POLAR CENTER



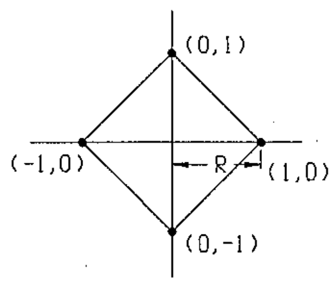

```
G1  G17  X0  Y-1
R1  AA 270  AB 0
AA 0  AB 90
AA 90  AB 180
AA 180  AB 270
```

INCREMENTAL POLAR CENTER




```
G1  G17  X0  Y-1
R1  I0  J1  AB0
I-1  J0  AB90
I0  J-1  AB180
I1  J0  AB270
```

ABSOLUTE POLAR CENTER




```
G1  G17  X0  Y-1
R1  XC0  YC0  AB0
XC0  YC0  AB90
XC0  YC0  AB180
XC0  YC0  AB270
```

Tangential Feedrate Control

The cutting feed is controlled so that speed along the path is always the commanded feedrate.

Feedrate Override

The per minute feed can be overridden using this switch on the machine operator's panel by 0 to 150% (per every 10%). Feedrate override cannot be applied to functions in which override is inhibited (e.g. tapping cycle).

3.1.3 Circular interpolation (G02, G03)

The general command format to move along a circular arc is as follows:

```
G17  G02    X Y  I J  or  XC YC R  or  R  or  AA R   F
G18   or    X Z  I K  or  XC ZC R  or  R  or  AA R   F
G19  G03    Y Z  J K  or  YC ZC R  or  R  or  AA R   F
             or
            AB R

*(1) *(2)   *(3                  *(4 or 5)          *(7)
            or 6)
```

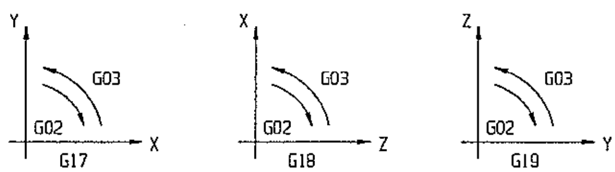
*These numbers are referenced in the chart that follows.

	Data to be given		Command	Meaning
1	Plane selection		G17	Specify arc on XY plane
			G18	Specify arc on ZX plane
			G19	Specify arc on YZ plane
2	Direction of rotation		G02	Clockwise (CW)
			G03	Counterclockwise (CCW)
3	End point position	G90 mode	Two of X, Y, and Z	End point position in work coordinate system
		G91 mode	Two of X, Y, and Z	Distance from start point to end point
4	Distance from start point to center		Two of I, J, and K	The signed distance from start point to center
	Arc radius		R replaces I, J, K	Arc radius
			Two of XC, YC, ZC, and R	The signed absolute coordinates of the arc center

Polar Definition

5	Arc radius	R	Arc Radius
	Start angle	AA	Angle from center to start point CW direction
6	End angle	AB	Angle from center to end point CW direction
7	Feedrate	F	Velocity along arc

The view is from the positive direction of the Z, Y or X axis to the negative direction on XY, ZX or YZ plane in a right hand cartesian coordinate system.



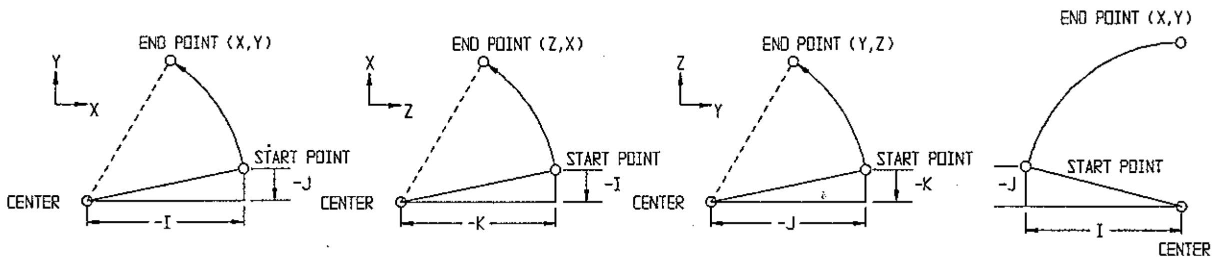
Clockwise and Counterclockwise Directions

Method I

Describing an Arc Using Incremental Center

The end point of an arc is specified by address X, Y or Z, and is expressed as an absolute or incremental value depending on G90 or G91. In incremental the coordinate of the end point is related to the start point of the arc. The arc center is defined by I, J, and K for the X, Y, and Z axes. The numerical value following I, J, or K is the distance from the start point to the arc center in X, Y or Z axes. I, J and K are always incremental values independent of G90 and G91.

The sign of I, J and K depends on the relationship of the center to the start point as shown below:



Programming with Circular Interpolation

Method II

Describing an Arc Using a Radius

When describing an arc using a radius value there are a number of valid formats. The various command formats are as follows:

G17	G02	X____	Y____	R____
G18	or	X____	Y____	R____
G19	G03	X____	Y____	R____

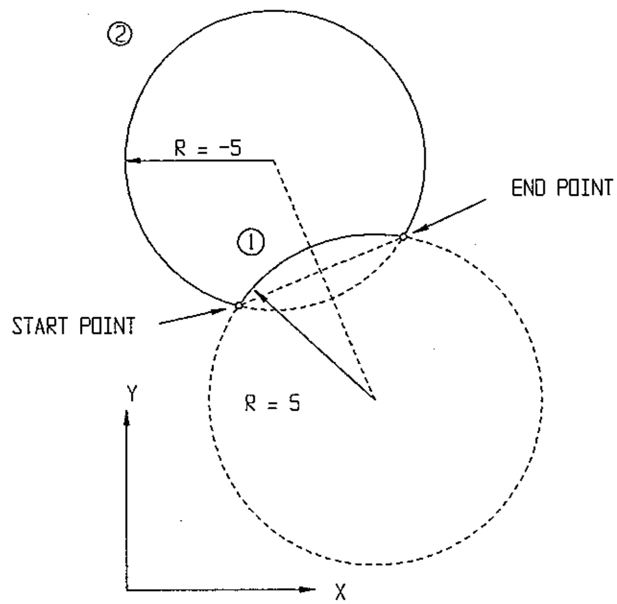
Arc End Points

The radius is always specified as its true value. The end points are incremental or absolute depending on G90 and G91. If a radius is used without a center point there are two types of arcs which can be generated. One is less than 180°, and the other is greater than 180°, as shown in the figure that follows. When the arc exceeds 180° the radius must be specified as a negative value.

Examples:

For arc 1 (less than 180°)
G2 X6 Y2 R5 F30

For arc 2 (greater than 180°)
G2 X6 Y2 R-5 F30



Method III

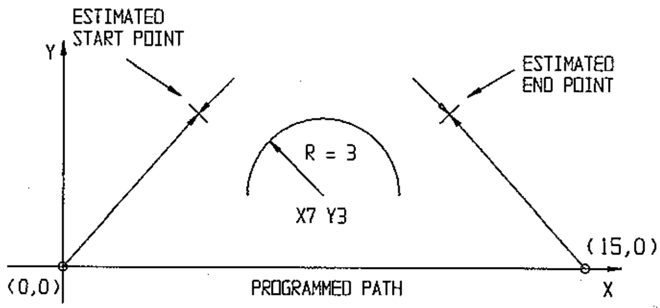
Describing an Arc Using Absolute Center and Trig Help

G17	G02	X___	Y___	XC___	YC___	R___
G18	or	X___	Z___	XC___	ZC___	R___
G19	G03	Y___	Z___	XC___	ZC___	R___
		End Point		Center Point		Radius

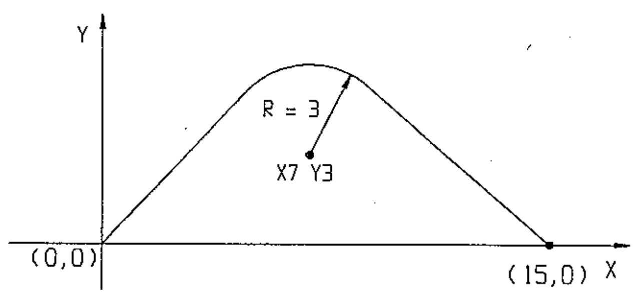
In incremental the center and end points are the distances in X, Y and Z from the start point of the arc. In the incremental mode trig help is not active. However, using this format in the absolute mode will activate the Trig Help function of the control. Trig Help will allow the programmer to estimate both the start and end points of any arc. The control will then calculate the true start and end points based on the moves preceding and trailing the arc. Where there are two possible correct answers, the control will choose the point closest to the estimated point. If the slope of the line entering or leaving the arc is such that no intersection occurs, the line will be made tangent to the arc.

Examples of Trig Help

Program 1

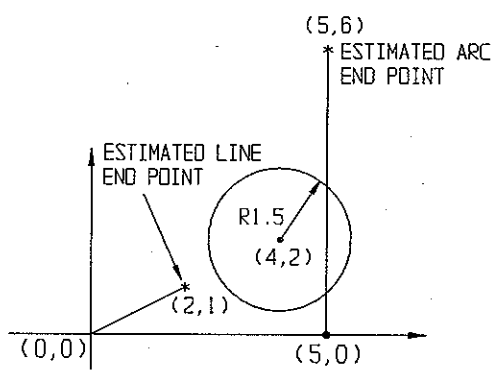



```
    G1  X0  Y0
        X2  Y6   estimated start point
G17 G2  X12  Y6  XC7  YC3  R3

    estimated   absolute
    end point   center point
G1  X15  Y0
```



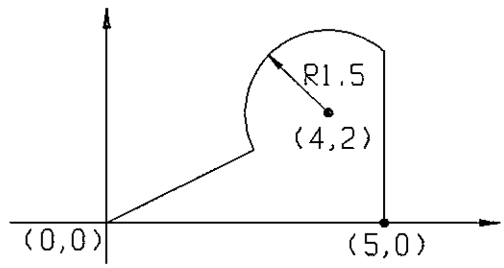
Path generated by Program 1

Program 2



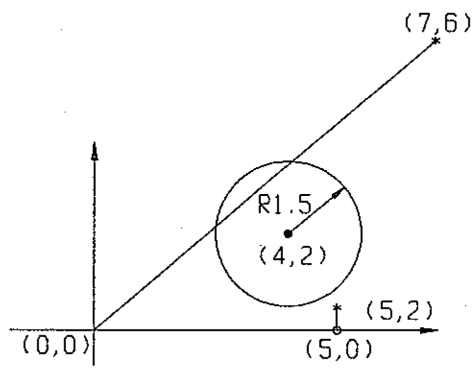
Programmed path

```
     G1 X0 Y0
        X2 Y1  estimated start point
 G17 G2 X5 Y6 XC4 YC2 R1.5

         estimated
         end point
     G1 X5 Y0
```



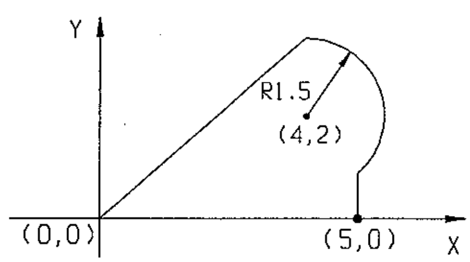
Path generated by Program 2

Program 3



Programmed path

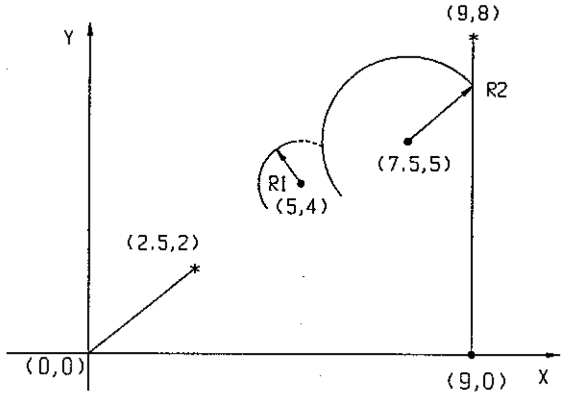
```
     G1 X0 Y0
        X7 Y6  estimated point
 G17 G2 X5 Y.2 XC4 YC2 R1.5

         estimated
           point
     G1 X5 Y0
```



Path generated by Program 3

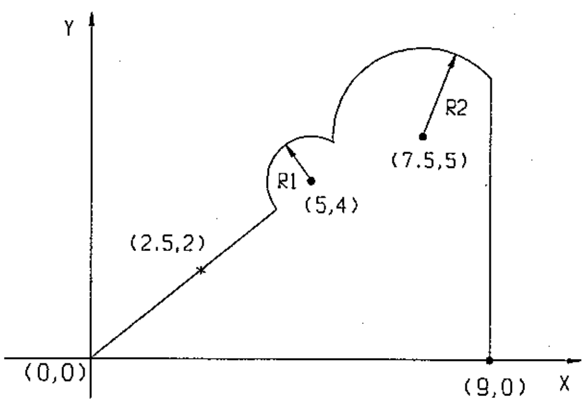
In general, when dealing with lines and arcs, if the line is programmed short of the arc it will be extended to the arc . If the line is programmed past the arc it will be shortened to the arc, and if the line does not intersect the arc it will be made tangent.

Program 4



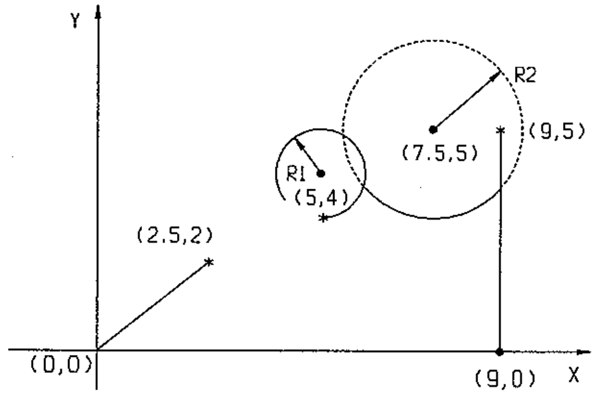
Programmed path

```
     G1 X0   Y0
        X2.5 Y2  estimated point
 G17 G2 X5 Y5 XC5 YC4 R1
         estimated
         end point
 G17 G2 X9 Y8 XC7.5 YC5 R2
         estimated
         end point
     G1 X9 Y0
```



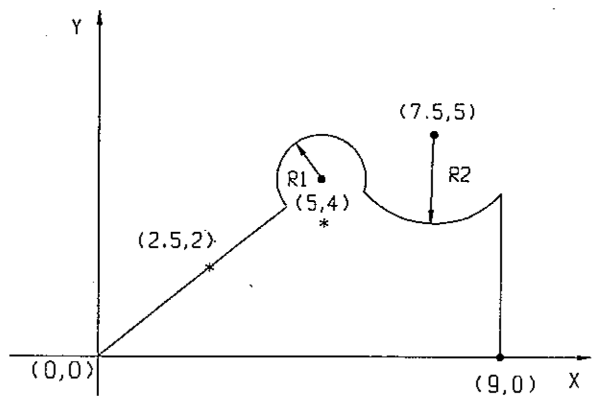
Path generated by Program 4

Program 5



Programmed path

```
    G1  X0  Y0
        X2.5  Y2   estimated point
G17  G2  X5  Y3  XC5  YC4  R1
          estimated
            point
G17  G3  X9  Y5  XC7.5  YC5  R2
          estimated
            point
    G1  X9  Y0
```



Path generated by Program 5

In general when estimating arc-to-arc intersections the end points chosen should be on the arc. The easiest points to pick are one of the quadrant points (0°, 90°, 180°, 270°).

Things To Remember When Estimating Points

** Estimating should only be used with line/circle, circle/circle, and circle/line paths.

** For line/circle and circle/line, the start and end point estimates must lie on the line; i.e. the slopes of the lines entering or leaving the arc must be correct.

** If a line intersects at two points, the estimated point should be closer to the desired point of intersection.

** If the above conditions are met there is no limit on how far the estimated point is away from the correct point.

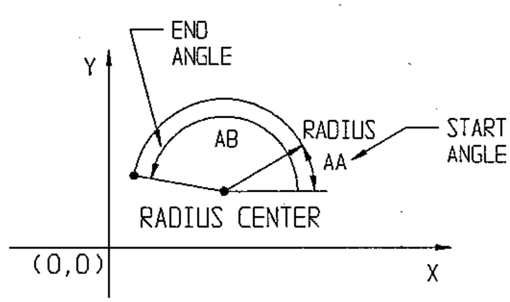
** When estimating the intersection of one arc to another arc, the end points chosen should be on the arc. The easiest point to pick on an arc is at one of the quadrant points (0°, 90°, 180°, or 270°).

Method IV

Describing an Arc Using Polar Definitions

The polar definitions do not change from absolute to incremental. The center of the arc is always considered the pole and all angles are related to it. The basic polar definition is as follows:

G17	G2			
G18	or	AA_____	AB_____	R_____
G19	G3	start angle	end angle	radius



Polar Arc Definitions

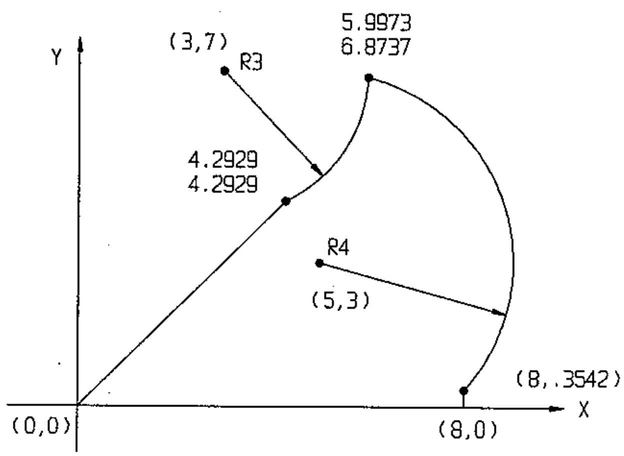
The polar format for arcs can be mixed with the cartesian formats. The following are legal formats.

G17	G2	X_____	Y_____	AA_____	R_____
		end point		start angle	
G17	G2	AB_____	XC_____	YC_____	R_____
		end angle	center point		
G17	G2	I_____	J_____	AB_____	
		center point			

The above formats are written for the XY plane but are valid in any plane or direction. Trig Help is only valid in polar when using an arc with an absolute center point (XC, YC).

Program 6

The following programs will all produce the same part, and which programming method is used is totally optional.



1) Absolute coordinates (Polar No Trig Help)

```
    G90
    G1  X0         Y0
        X4.2929    Y4.2929
G17 G3  AA295.53   AB357.59   R3
    G2  AA75.56    AB318.59   R4
    G1  Y0
        X0
```

2) Absolute coordinates (Polar Trig Help)

```
    G90
    G1  X0    Y0
        AB45  R1
G17 G3  XC3   YC7     AB0   R3
    G2  X8    Y.3542  XC5   YC3   R4
    G1  Y0
        X0
```

Note: When using Trig Help you must have a valid arc center and radius. That is why the G2 and G3 lines have a fixed format.

3) Absolute coordinates (Cartesian No Trig Help)

```
     G90
     G1   X0         Y0
          X4.2929    Y4.2929
 G17 G3   X5.9973    Y6.8737   XC3   YC7   R3
            or
 G17 G3   I-1.2929   J+2.7071  X5.9973  Y6.8737
            or
 G17 G3   X5.9973    Y6.8737   R3
     G2   X8         Y.3542    XC5   YC3   R4
                or
     G2   X8         Y.3542    R4
     G1   Y0
 X0
```

4) Absolute coordinates (Cartesian Trig Help)

```
     G1   X0   Y0
          X1   Y1
 G17 G3   XC3  YC7  X6  Y7  R3
     G2   XC5  YC3  X8  Y.5  R4
     G1   X8   Y0
     X0
```

Note: Most of the dimensions are approximations and the control calculates the exact dimensions.

5) Incremental coordinates

```
     G91
     G1   X0         Y0
          X4.2929    Y4.2929
G17  G3   X-1.7044 Y2.5808   I-1.2929 J2.7071
                or
     G3   X-1.7044 Y2.5808   R3
                or
     G3   X-1.7044 Y2.5808   XC3   YC7   R3
     G2   X2.0027   Y-6.5195 I-.9973   J-3.8737
                or
     G2   X2.0027 Y-6.5195   R4
                or
     G2   X2.0027 Y-6.5195   XC5   YC3    R4
     G1   Y-.3542
          X-8
```

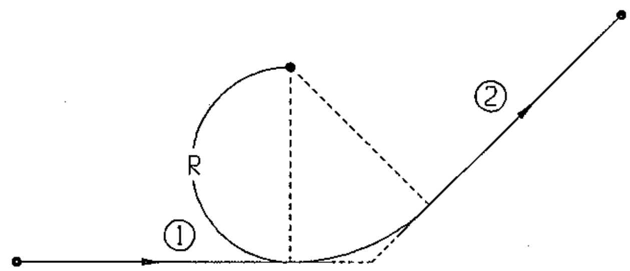
Note: In incremental, Trig Help cannot be used, as each point is related to the current position.

3.1.4 Corner rounding

By adding: ,R___

> to the end of blocks commanding linear or circular interpolation, corner rounding can be automatically inserted.

```
(1)   G91 G01 X0 Y0
(2)   X1,R.25
(3)   X1 Y1
```

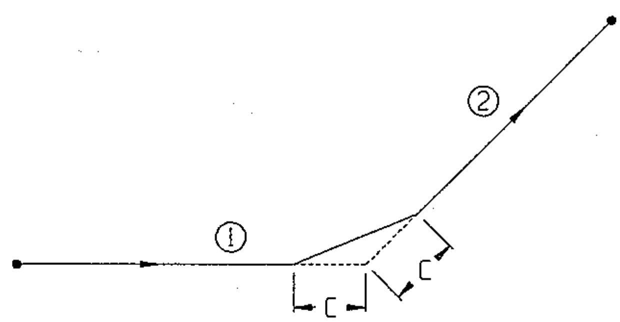


3.1.5 Angle chamfering

By adding: ,C___

> to the end of blocks commanding linear interpolation, angle chamfering is automatically inserted.

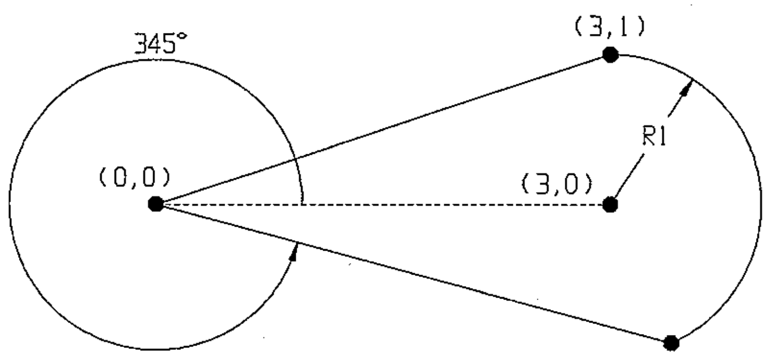
```
(1)   G91 G01 X0 Y0
(2)   X1,C.25
(3)   X1 Y1
```



3.1.6 Back line

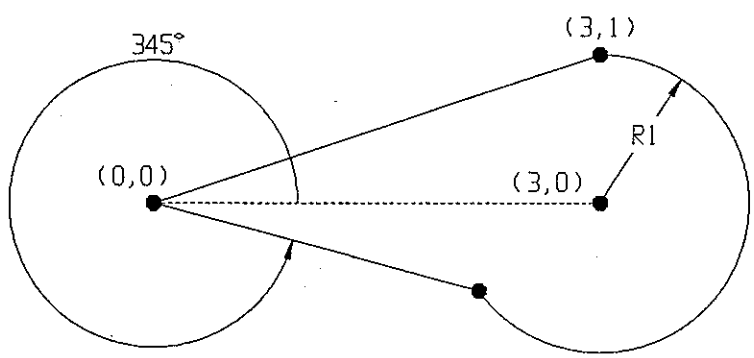
The back line function can be used on any line command. This function reverses the direction of a programmed line. It would normally be used when you know the end point of the line and not its start point. The end point would be programmed and the line would be extended backwards to the start point. When using this function all Trig Help functions are still valid.

```
(1)   X0 Y0
(2)   X3 Y1
(3)   G17 G2 R1 XC3 YC0 AB270
(4)   G01 X0 Y0 BACK C0 W345
```



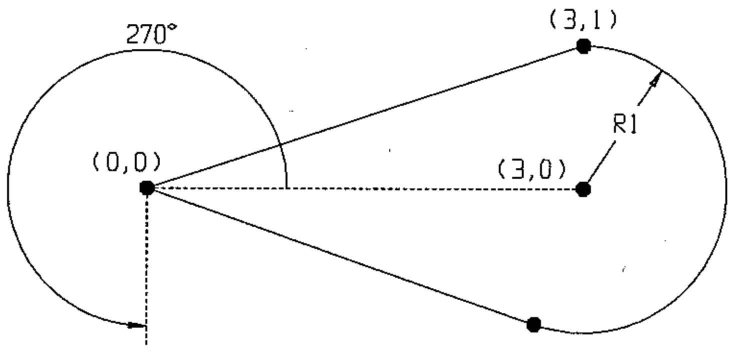
Back	extend line backwards from (0,0)
C0	use the arc intersection farthest from (0,0)
W345	extend the line from (0,0) at an angle of 345°

```
(1)   X0 Y0
(2)   X3 Y1
(3)   G17 G2 R1 XC3 YC0 AB270
(4)   G01 X0 Y0 BACK C2 W345
```



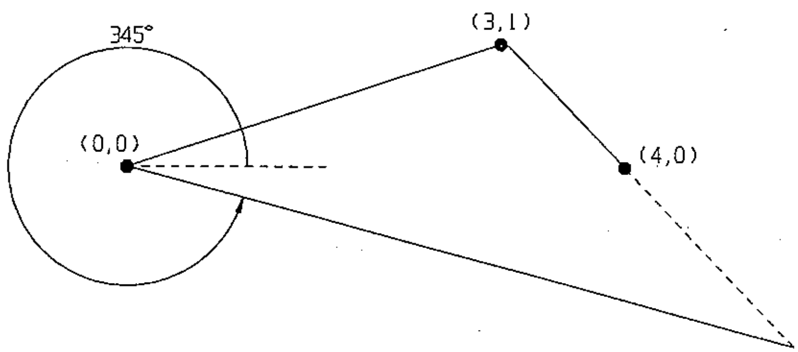
C2 use closest intersection

(1) X0 Y0
(2) X3 Y1
(3) G17 G2 R1 XC3 YC0 X3 Y-1
(4) G01 X0 Y0 BACK C0 W270



W270 This line doesn't intersect with the arc; therefore, the line will be rotated until it is tangent.

(1) X0 Y0
(2) X3 Y1
(3) X4 Y0
(4) X0 Y0 BACK C0 W345



This example used a back line between two lines to program an unknown point.

Other Notes on Circular and Linear Milling

The feedrate in circular and linear is equal to the feedrate specified by the F code. This feedrate is the tangential feedrate along the arc and the vector feed on the linear moves.

Note 1: *I0, J0 and K0 can be omitted.*

Note 2: *If X, Y and Z are all omitted or if the end point is located at the same position as the start point, and the center is commanded by I, J and K, an arc of 360° (a complete circle) is assumed.*
G02I _____; (a complete circle)

When R is used, an arc of 0° is programmed.
G02R _____; (The cutter does not move.)

Note 3: *The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the cutter compensation is applied.*

Note 4: *If I, J, K and R addresses are specified simultaneously, the arc specified by address R takes precedence and the others are ignored.*

Note 5: *If an axis not comprising the specified plane is commanded an error is displayed.*

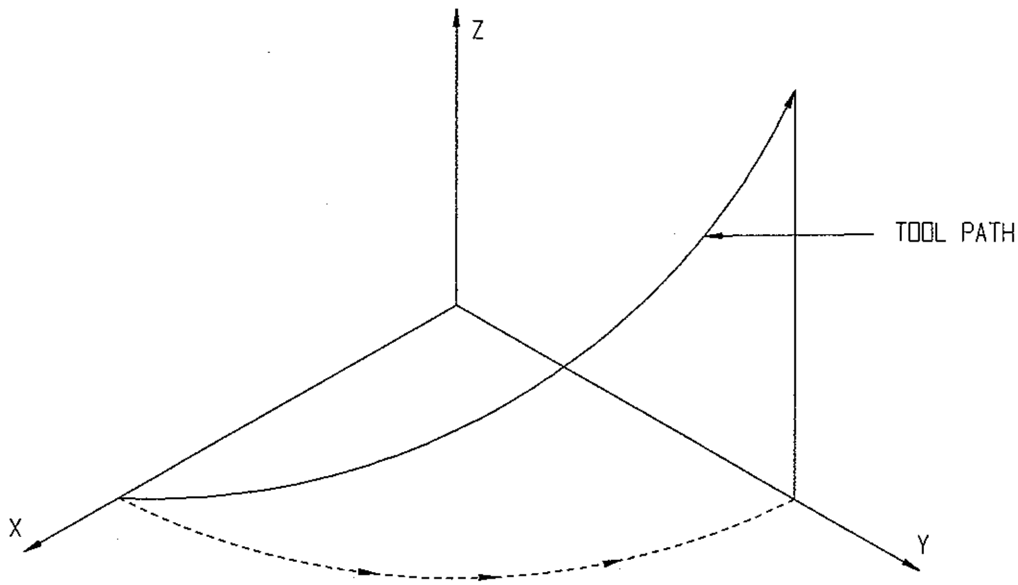
Note 6: *The X Y Z I J K R AA·AB commands are retained by the control. If an interpolation block is left incomplete, the missing axis information will be defaulted to the value previously entered.*

3.1.7 Helical cutting (G02, G03)

Helical interpolation is enabled by specifying another axis which moves synchronously with the circular interpolation. That is, the tool can be moved helically.

```
     G02                       R____
G17   or   X____  Y____         or        Z____  F____;
     G03                    I____  J____

     G02                       R____
G18   or   X____  Z____         or        Y____  F____;
     G03                    I____  K____

     G02                       R____
G19   or   Y____  Z____         or        X____  F____;
     G03                    J____  K____
```

The above formats for helical milling just illustrate the general concept. Any of the previous arc formats can be used to do helical cutting by simply adding the third axis command to the arc command.



An F command specifies a feedrate along a circular arc. Therefore the feedrate of the linear axis is as follows:

$$F \quad X \quad \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.

3.2 Dwell command (G04)

The G4 code must be immediately followed by an FXXX.X instruction. This instruction will then cause the program to stop or dwell for XXX.X seconds.

General Format

G4F2.5 will cause the program to dwell for 2.5 seconds.

G4F25 will cause the program to dwell for 25 seconds.

Note: A P or an X can be used in place of an F following the G4 command.

3.3 Exact stop (G09)

Moves commanded in blocks with G09 decelerate at the end point, and in-position check is performed. This function is used when sharp edges are required for work-piece corners in cutting feed.

3.4 Set data on/off (G10, G11)

This function allows all the CNC's configuration, setup, axis and offset table parameters to be loaded via a program rather than through the front panel. (This function is the only way to change parameters 499 and higher from a program.) The format for loading the parameters is as follows:

```
G10                     Set data On
P*** = value
P*** = value            P*** = parameter number
P*** = value                   to be loaded
P*** = value
G11                     Set data Off
```

When the G11 is performed the CNC will start using the new parameter settings. Refer to Appendix A on parameter assignments.

3.5 XY plane (modal) (G17)

Selects the XY plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G17 command can appear anywhere on the line.

3.6 XZ plane (modal) (G18)

Selects the XZ plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G18 command can appear anywhere on the line.

3.7 YZ plane (modal) (G19)

Selects the YZ plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G19 command can appear anywhere on the line.

3.8 Inch dimensioning mode (modal) (G20)

This function will cause the system to go into the inch mode. In this mode the system will accept dimensions in inches. This function can be initiated in a program or in MDI.

G20 is active after power-up.

G20 cancels G21.

3.9 Metric dimensioning mode (modal) (G21)

This function will cause the system to go into the metric mode. In this mode the system will accept dimensions in millimeters (mm). This function can be initiated in MDI or program mode. In metric the actual machine position may not exactly agree with the program position because of the conversion. Feedrate in the metric mode is in millimeters per minute (mmpm).

Notes on Inch/Metric

The CNC does a conversion from metric to inch, and inch to metric, on all tool offsets. This means that a 1.0 inch offset entered in the inch mode will change to 25.4 mm when the system is switched to metric. The opposite happens for metric entries.

3.10 Safe zone on/off (G22, G23)

This control is equipped with a programmable safe zone. Any area of the machine's travels can be designated as a safe zone. This is an area of the travels the tool cannot enter. If the tool is programmed into this area when the safe zone check is enabled, a safe zone error will be displayed. The safe zone is defined in the machine parameters. (Refer to the section on parameters.) The check is turned on with a G22 and off with a G23. The normal state of this check is off.

3.11 Autoroutines

These G codes select basic patterns which are used over and over in most milling applications. These patterns are circular and rectangular finish cuts, circular and rectangular pocket clearing and bolthole routines. These routines use the CNC's parametrics to input the various cutting and size differences encountered in different applications. The following will explain each G code and give an example of its use in a program.

3.11.1 Circular pocket clear (G24)

The G24 autoroutine is used to clear a circular pocket by starting in the center and spiraling out to the programmed diameter.

Circular Pocket Clear Program

```
N1    G20 G90 (Inch/Absolute)
N2    G00 X0 Y0 (rapids to center of pocket)
N3    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM,
      calls tool #1's offsets)
N4    F25 (X-Y feedrate)
N5    P150=1 (pocket radius)
N6    P153=.015 (X-Y finish stock)
N7    P154=.005 (Z finish stock)
N8    P155=.25 (cut width)
N9    G24 G99 G42 G2 R.1 Z-.5 V-.3 Q.2 F10
      *1  *2  *3  *3  *4  *5   *6   *7  *8

*1    Executes cirle pocket clear autoroutine
*2    Returns Z to clearance plane
*3    CW right or "G41 G3" CCW left
*4    .1 clearance plane
*5    -.5 final Z depth
*6    -.3 first Z depth
*7    .2 Z increment
*8    10 ipm Z feedrate
```

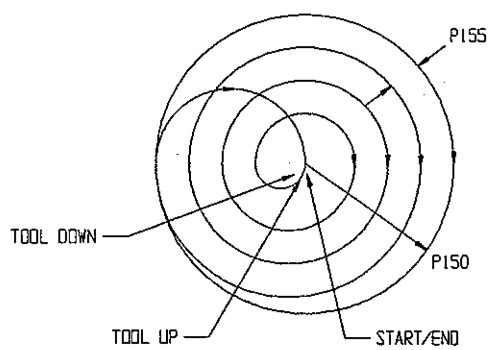


Figure 1.1
CW Circular Pocket Clearing

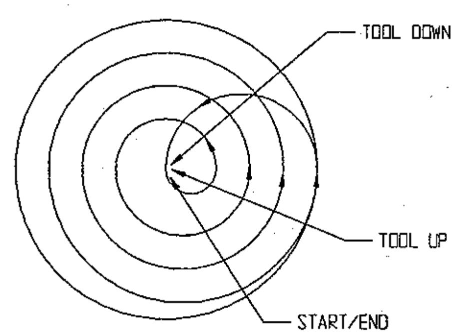


Figure 1.2
CCW Circular Pocket Clearing

Block #	Block Entry Info	Block #	Block Entry Info
N9	G2 G42	N9	G3 G41
Block 9	Selects CW circle, and turns ON <u>right</u> cutter compensation	Block 9	Selects CCW circle and turns ON <u>left</u> cutter compensation

3.11.2 Circular finish inside (G25)

The G2, G3, G41 and G42 codes are used together to determine not only the direction of cut, but whether the cut is to be an outside or inside cut. Parameters are used which determine the dimension of the circle. The radius of the tool will automatically be figured by the control to determine the proper starting point.

If a tool radius is specified, cutter compensation can be used in all autoroutines. The control will automatically decrease or increase the tool path by the radius of the tool.

<u>ALL MILLING AUTOROUTINES MUST BE ACTIVATED WITH THE TOOL AT THE CENTER OF THE ROUTINE.</u>

Figure 2.1 shows the tool path of the following program. Figure 2.2 shows the same program with the change indicated in line N9.

Circular Finish Inside Program

```
N1    G20  G90 (Inch/Absolute)
N2    G00 X0 Y0 (rapid to center of pocket)
N3    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM;
      calls tool #1's offsets)
N4    G99 (return Z to clearance plane)
N5    F20 (X-Y feedrate)
N6    P150=1 (pocket radius)
N7    P153=0 (X-Y finish stock)
N8    P154=0 (Z finish stock)
N9    G25 G42 G2 V-.3 R.1 F5 Z-.5 Q.2
      *1  *2  *2  *3  *4  *5  *6   *7
```

```
*1    Executes circle finish inside
*2    CW right or "G41 G3" CCW left
*3    First Z depth
*4    Clearance plane
*5    Z feedrate
*6    Final Z depth
*7    Z increment
```

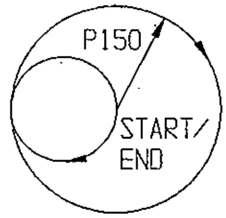


Figure 2.1
Inside CW Finish Circle

N9 G42 G2 selects CW circle and <u>right</u> cutter compensation

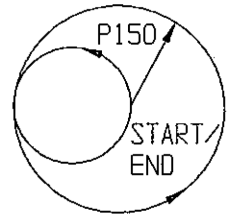


Figure 2.2
Inside CCW Finish Circle

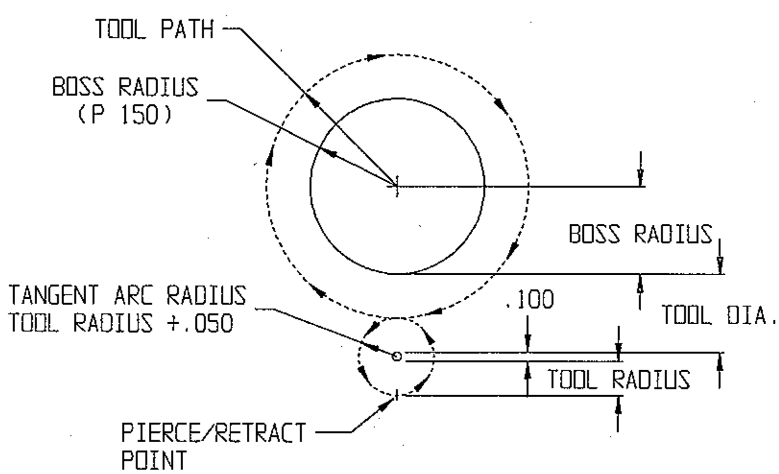
N9 G41 G3 selects CCW direction and <u>left</u> cutter compensation

Note: Parameter P150 is the pocket radius. If no finish stock is desired, parameters P153 and P154 should be set to zero. The F20 programmed in N5 is the XY feedrate and the F5 in N9 affects only the Z axis feed. Once parameters are set to a value they do not change and can be utilized further in the program. When an autoroutine is called, any parameters that are not re-initialized will default to the previous value of the parameter.

3.11.3 Circular finish outside (G26)

The G26 autoroutine is identical in operation to the G25 autoroutine except it cuts the outside of a circular boss rather than the inside. Because the G26 needs to position to the outside of the boss, it will use the following formula to calculate the distance from the center to the feed down point.

$$X_{current\ position} = \text{Circle radius} + .1 + [3 \text{ X Tool radius}]$$



Circular Finish Outside Program

```
N1    G20 G9 (Inch/Absolute)
N2    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM,
      calls tool #1's offsets)
N3    F20 (X-Y feedrate)
N4    P150=1 (Boss radius)
N5    P153=0 (X-Y finish stock)
N6    P154=0 (Z finish stock)
N7    G26 G98 G41 G2 R.1 Z-.5 V-.3 Q.2 F5
      *1  *2  *3  *3  *4  *5   *6  *7  *8
```

```
*1    Executes circle finish outside
*2    Return to initial point
*3    CW left or "G42 G3" CCW right
*4    Clearance plane
*5    Final Z depth
*6    First Z depth
*7    Z increment
*8    Z feedrate
```

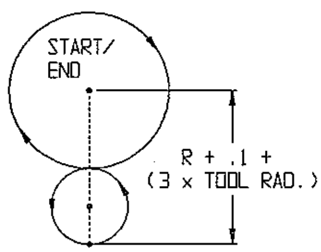


Figure 3.1
Outside CW Finish Circle

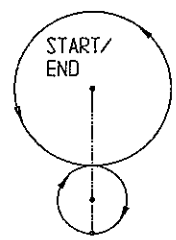


Figure 3.2
Outside CCW Finish Circle

N7 G2, G42 selects CW direction and <u>left</u> cutter compensation

N7 G3, G4 selects CCW direction and <u>right</u> cutter compensation

3.11.4 Reference point return (G28, G29, G30)

These commands allow the machine to be commanded to a fixed point (reference point) by first passing through an intermediate point on the way to the reference point. First a fixed reference point in XYZ is entered via the front panel into the reference point parameters. Once this point is established it will remain unchanged until changed by another front panel command. Each time a G28 or G30 is commanded it will return the machine to the designated reference point. Positioning to the intermediate end reference points are done in rapid traverse. If a G28 or G30 is executed with no axis definitions the machine will position directly to the reference position. If only one or two axes are commanded as intermediate points the remaining axes will stay at their current positions until the intermediate point is reached. Then they will position to the reference point along with the other axes. Once an intermediate point is programmed it will be remembered until the next G28 is executed (i.e. for use in a G29).

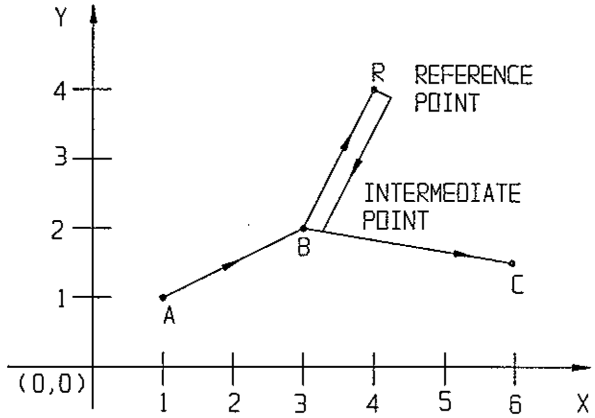
The command format is as follows:

```
        Reference point set at X-2
                               Y 5
                               Z 0


                         Machine positions to:
    X1 Z-2
    G28                  X-2 Y5 Z0
    X1 Y0 Z-2
    G28 X3               X3 Y0 Z-2 then X-2 Y5 Z0
    X-3 Y2 Z-8
    G28 Z-7              X-3 Y2 Z-7 then X-2 Y5 Z0
    G29                  Return from reference point
```

The G29 command is just the converse of a G28. The G29 will return the machine from the reference point to the programmed point via the last intermediate point stored by a G28 command. The command format is as follows:

$$\text{G29}\quad \underbrace{\text{X\_\_\_\_}\quad \text{Y\_\_\_\_}\quad \text{Z\_\_\_\_}}_{\text{programmed point}}$$

Example of G28 and G29



X1 Y1	Point A
G28 X3 Y2	Point B then Point R
G29 X6 Y1.5	Point B then Point C

G30 2nd, 3rd, 4th Reference Point Return

This function works in an identical manner to the G28 reference point return except that a 2nd, 3rd and 4th reference point can be called. The command format is as follows:

```
G30  P2  X____  Y____  Z____
G30  P3  X____  Y____  Z____
G30  P4  X____  Y____  Z____
      ↓
 reference  intermediate
   point        point
```

3.11.5 Z to clearance (G31)

The G31 function will retract Z to the clearance position. This position defaults to the last clearance position but may be changed by editing parameter #140.

3.11.6 Z to tool change (G32)

The G32 function will retract Z to the tool change position. This position defaults to grid position but may be changed by editing the tool change coordinate parameters.

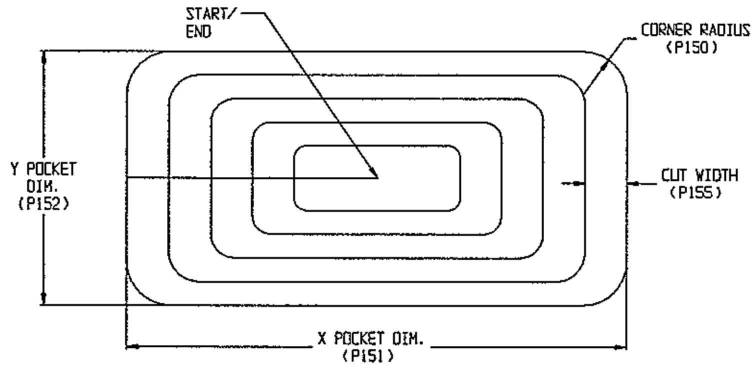
3.11.7 Rectangular pocket clear (G34)

The G34 autoroutine is used to clear a rectangular pocket by starting in the center and working its way out to the finish dimensions. The operation of the autoroutine is identical to the circular routines except that a rectangle with radiused corners is cut. The rectangular routines have the addition of parameters P151 (X pocket dimension) and P152 (Y pocket dimension). Parameters P150, P153, P154 and P155 retain the same meaning as in the circular routines. The X and Y dimensions are the overall pocket dimensions, and if the corner radius is set to 0 the corners will be cut at the tool radius.

Rectangular Pocket Clear Program

```
N1    G20 G90 (Inch/Absolute)
N2    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM,
      calls tool #1's offsets)
N3    G00 X0 Y0 (rapids to pocket center)
N4    F20 (X-Y feedrate)
N5    P150=.75 (corner radius)
N6    P151=4 (X pocket dimension)
N7    P152=2 (Y pocket dimension)
N8    P153=.015 (X-Y finish stock)
N9    P154=.005 (Z finish stock)
N10   P155=.5 (cut width)
N11   G34 G99 G42 G2 R.1 Z-.5 V-.3 Q.2 F5
      *1  *2  *3  *3 *4   *5   *6   *7  *8
```

```
*1    Executes rectangular pocket clear
*2    Returns Z to clearance plane
*3    CW right or "G41 G3" CCW left
*4    Clearance plane
*5    Final Z depth
*6    First Z depth
*7    Z increment
*8    Z feedrate
```



If N11 is G42 G2, the cut direction is CW.
If N11 is G41 G3, the cut direction is CCW.

3.11.8 Rectangular finish inside (G35)

The G35 autoroutine is used to remove the finish stock left by the rectangular clear routine, or to remove some amount of stock in a single pass around the inside of a rectangle. The G35 autoroutine works in an identical manner to the G34 autoroutine. It starts at the center and makes one pass around the rectangle. The circle from the middle of the autoroutine to the outside edge will always be along the longest side of the pocket.

Rectangular Finish Inside Program

```
N1    G20  G90 (Inch/Absolute)
N2    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM,
      calls tool #1's offsets)
N3    G00 X0 Y0 (rapids to center of rectangle)
N4    F20 (X-Y feedrate)
N5    P150=.25 (corner radius)
N6    P153=0 (X-Y finish stock)
N7    P154=0 (Z finish stock)
N8    P151=4 (X pocket dimension)
N9    P152=2 (Y pocket dimension)
N10   G35 G99 G42 G2 R.1 Z-.5 V-.3 Q.2 F5
      *1  *2  *3  *3 *4    *5    *6   *7   *8
```

*1 Executes rectangular finish inside
*2 Returns Z to clearance plane
*3 CW right or "G41 G3" CCW left
*4 Clearance plane
*5 Final Z depth
*6 First Z depth
*7 Z increment
*8 Z feedrate

Block #	Line Entry Info
N10	G2 G42 selects CW direction and right cutter comp

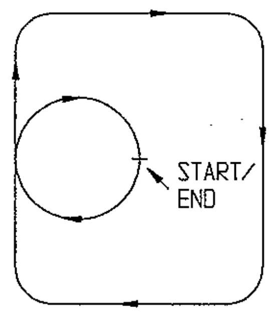


Figure 4.1
Inside CW Finish
Rectangular X≤Y
P151≤P152

Block #	Line Entry Info
N10	G2 G42 selects CW direction and right cutter comp

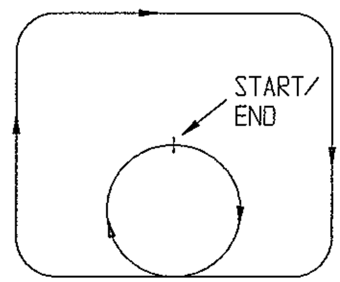


Figure 4.2
Inside CW Finish
Rectangular X>Y
P151>P152

Block #	Line Entry Info
N10	G3 G41 selects CCW direction and left cutter comp

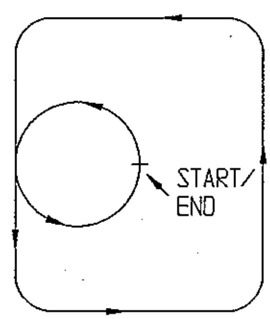


Figure 4.3
Inside CCW Finish
Rectangular w/Circular
Corners and X≤Y
P151≤P152

Block #	Line Entry Info
N10	G3 G41 selects CCW direction and left cutter comp

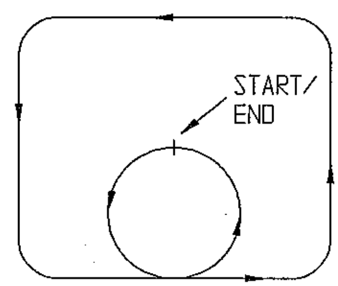
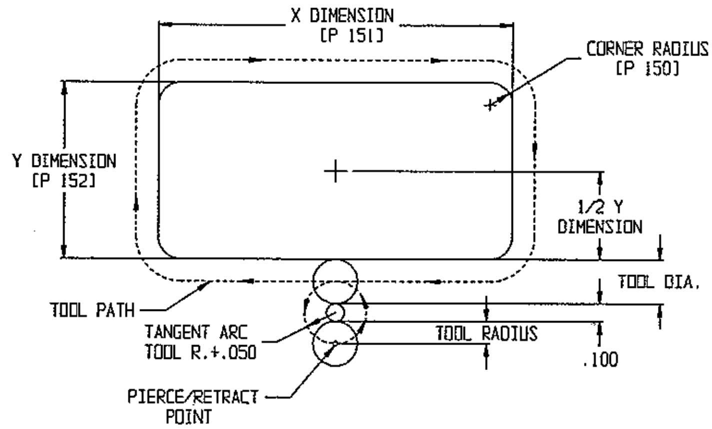


Figure 4.4
Inside CCW Finish
Rectangular w/Circular
Corners and X>Y
P151>P152

3.11.9 Rectangular finish outside (G36)

The G36 autoroutine is used to remove finish stock in a single pass around the outside of a rectangular boss. The G36 autoroutine works in an identical manner to the G34 autoroutine. It starts in the center, makes a rapid move to the outside of the part, and then feeds the tool down. The formula the CNC uses to calculate the distance from the center to the feed down point is as follows:

Y = (3 X tool radius) + .1 + 1/2 Y pocket width



Rectangular Finish Outside Program

```
N1    G20 G90 (Inch/Absolute)
N2    S1000 M3 D1 G43 H1 (spindle CW 1000 RPM,
      calls tool #1's offsets)
N3    G00 X0 Y0 (rapid to X0 Y0)
N4    G98 (returns Z to initial level)
N5    F20 (X-Y feedrate)
N6    P150=.25 (corner radius)
N7    P153=0 (X-Y finish stock)
N8    P154=0 (Z finish stock)
N9    P151=4 (X frame dimension)
N10   P152=2 (Y frame dimension)
N11   G36 G41 G2 V-.25 R.1 Z-.5 Q.25 F10
      *1  *2  *2  *3   *4   *5   *6   *7
```

*1   Executes rectangular finish outside
*2   CW left or "G42 G3" CCW right
*3   First Z depth
*4   Clearance plane
*5   Final Z depth
*6   Z increment
*7   Z feedrate

Block #	Line Entry Info
N11	G3 G42 selects CCW direction and right cutter comp

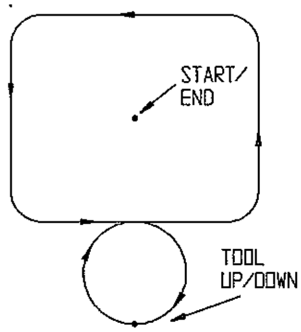


Figure 5.1
Outside CCW Finish

Block #	Line Entry Info
N11	G2 G41 selects CW direction and left cutter comp

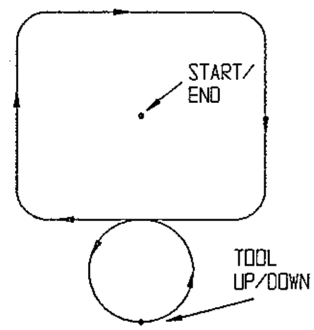


Figure 5.2
Outside CW Finish

3.12 Cutter compensation (G40, G41, G42)

G40 Compensation Off
G41 Left Compensation
G42 Right Compensation

This section will explain how the cutter compensation works and give pointers on how to use it optimally.

Cutter compensation is the displacement of the tool path, perpendicular to the programmed path, by the amount equal to the cutter radius. The programmed path can be figured by the programmer for a zero tool radius.

If the parts program is written for a zero tool radius, i.e., directly off the print, then by entering the actual tool radius into the system and activating cutter compensation the operator can make the control calculate the displaced path.

Throughout the program the control keeps a record of the previous programmed point, the current programmed point, and the next programmed point along the tool path.

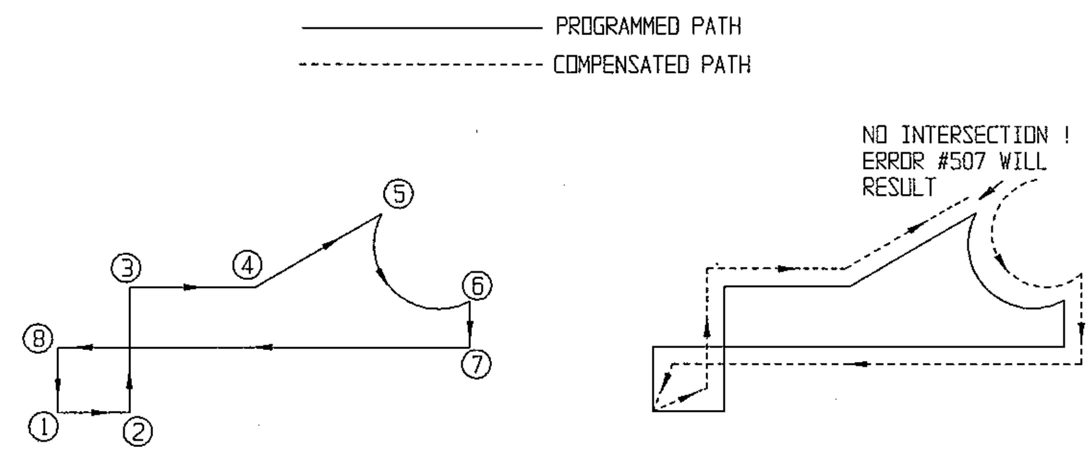
With three points, information on how they are connected, the cutter radius, and whether it is a left or a right compensation, the control can calculate the current compensated point. The control will also employ its Trig Help function discussed earlier to connect lines and arcs during cutter compensation. (See section on Trig Help.)

After each successful calculation of a compensated point the current programmed point becomes the previous programmed point, the next programmed point becomes the current programmed point and a new programmed point is read up to become the next programmed point. This mechanism is repeated over and over again until the end of the program is reached. This sequence should be understood clearly in order to understand many points that will come up later on how the compensation works.

The compensation in this control is truly intersectional. Given these three points, the control calculates the intersection of the compensated path between the previous and the current programmed points <u>and</u> the compensated path between the current and the next programmed points. These paths can be a mixture of straight lines and arcs.

Because of the intersectional nature of the compensation package, there has to be an intersection of all the displaced paths for the system to work. If there is no intersection between two paths the control will give an error.

NOTE: *All cutter compensation examples are shown without Z axis moves.*



(a) Path of a cutter with 0" tool diameter

(b) Path of a cutter with non-zero tool diameter

Figure 6
Explanation of How Displaced Tool Paths
Cannot Have an Intersection

The solution of the above part is to introduce a 00.0001" chamfer or round corner at Point 5 between the non-intersecting surfaces.

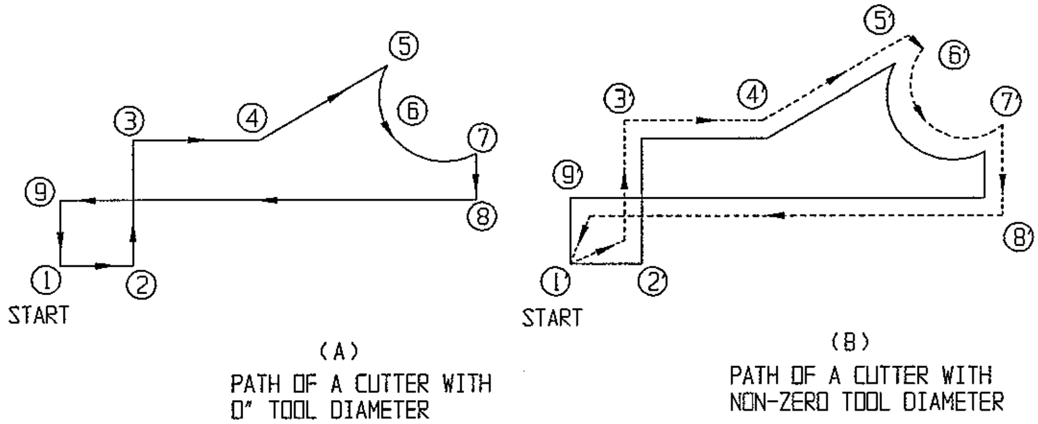
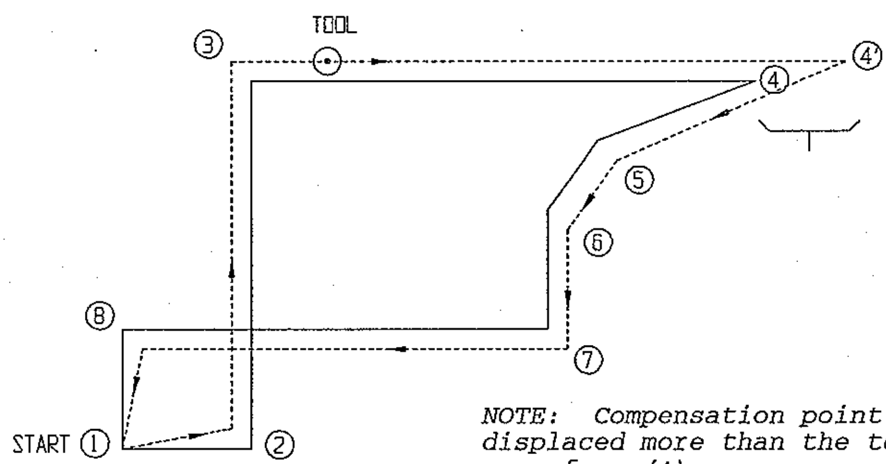


Figure 7
Explanation of How a 00.0001" Chamfer Should Be Introduced to Solve a No-intersection Problem

In some cases the system will find an intersection but it will be unreasonably far away from the part. Again, in such cases a 00.0001" chamfer or round corner should be inserted to solve this problem.



NOTE: Compensation point (4) is displaced more than the tool radius away from (4).

Figure 8
Outside "V" Cutter Compensation

Figure 9 shows how a 00.0001" chamfer or round corner added at point (4) has saved an unnecessary departure.

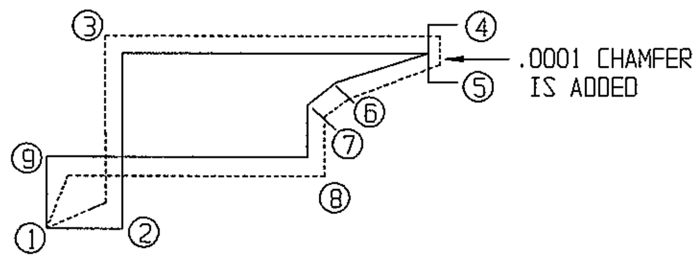


Figure 9
Outside "V" Cutter Compensation Solution

The following case shows how the compensated point for an inside "V" will stay away from the programmed point by more than the tool radius.

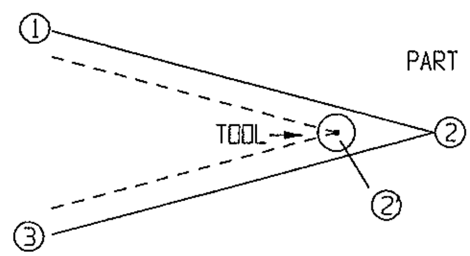


Figure 10
Inside "V" Cutter Compensation

Note: The tool stays away from the programmed point (2) by a distance more than the tool radius. If the compensated point (2) was any closer to (2), the tool would gouge the sides of the part.

3.12.1 Sample part exercise

As the system requires three points to generate a compensated point, care should be taken when the cutter compensation is turned on or off.

The magnitude for the axis movement for leaving or entering the part **has** to be greater than the amount of cutter compensation expected. Looking at Figure 11.1 it will be seen that the machine position at point (1) is identical for both the compensated as well as the uncompensated tool path. This is because the first point cannot be compensated as it has no previous point to make up a three point set for the compensation routine.

Therefore, the compensation should be turned on **before** the tool enters the work.

For ease of programming, the tool should enter and leave the part **perpendicular** to the part surface. This is not a strict requirement, but simplifies understanding how the cutter compensation will behave entering and exiting the workpiece. If in doubt on how the displaced tool path will look it is advisable to rough sketch, by hand, lines parallel to the part surface from start to end. Many times problems will become apparent right away.

How to Determine How the Compensated Path Will Look

Step 1 Sketch actual part and label points in sequence

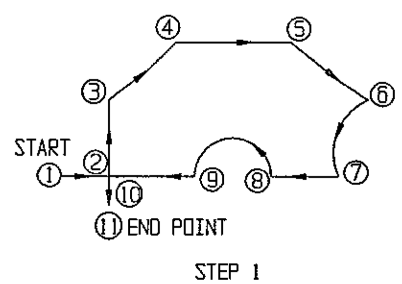


Figure 11.1
Compensation Exercise Step 1

Step 2 Sketch lines displaced tool radius away from part surface from point (1) to point (11)

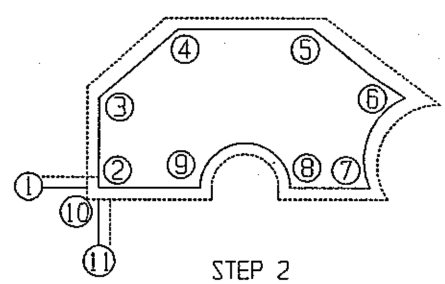


Figure 11.2
Compensation Exercise Step 2

Step 3 Check if all paths in the sequence intersect. If yes, then except for the start and end points, connect the displaced path and label points of intersection. If even one intersection cannot be found, the part will not run if the error is not corrected.

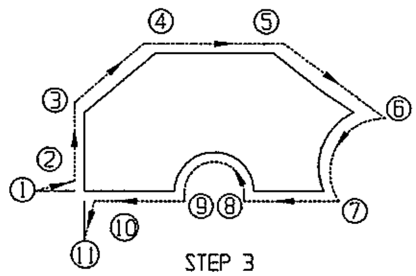


Figure 11.3
Compensation Exercise Step 3

Step 4 Since points (1) and (11) do not have two points on either side, they will be the uncompensated points. Therefore, connect them to their neighboring compensated points to arrive at the actual displaced cutter path.

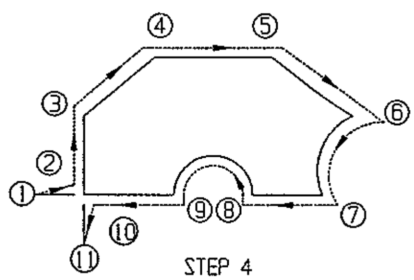


Figure 11.4
Compensation Exercise Step 4

Step 5 The above displaced path is what the system will trace if the part is run. However, a problem has become apparent from the rough sketch.

Note that the lower left hand corner will be left uncut because the tool going from (1) to (2) will leave a little notch of uncut material. A similar case is obvious in the tool path from (10) to (11). There again the corner will be left uncut.

The solution is to rearrange the start and end points so that the corner is properly cut.

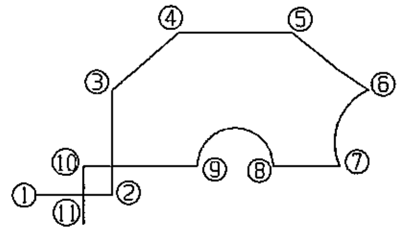


Figure 11.5
Compensation Exercise Step 5

<u>Step 6</u> Note how points (1), (2), (10) and (11) have been moved a little. The result will be as follows:

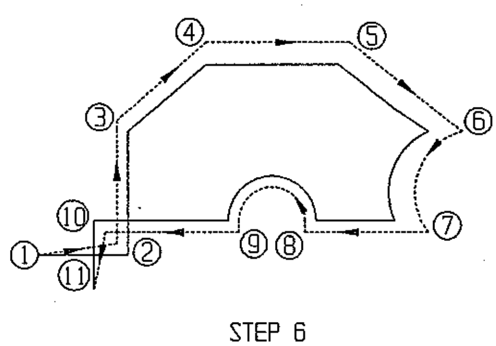


Figure 11.6
Compensation Exercise Step 6

Note: It is now seen that when the tool moved from (2) to (3), and (9) to (10), the corner will be properly cut.

In the above example the cutter compensation was turned on by using the G41 command for turning on left cutter compensation at point (2), and turned off by using the G40 command at point (11).

A few more points are to be noted for the above example:

1. Points (1) and (11), being the start and end points, were chosen so that they lie sufficiently away from the part surface. "Sufficient" is a distance **more** than the total expected compensation.

2. Circle at (9) and (8) points to complications that will arise if the tool radius is increased indefinitely. As the radius is increased (9) and (8) will keep moving closer to one another. For

some value of the radius they will become identical. If the radius is increased further, the tool radius will have become too large to make that circle and the system will give an error telling the operator that an intersection cannot be found at that line.

3. If a compensated path can be successfully sketched by hand, then it will run on the system. However, if the sketch yields a missing intersection, the control will give an error.

4. Until the operator becomes familiar with cutter compensation it is advisable that rough sketches be made for the compensated path before the part is run as a program.

How To Compensate for a Cavity

If the part is a cavity then the start and end points would have to change. Simply changing the G41 to G42 (right cut) will not help. This is because the tool would still come down on point (2) at the beginning. This would be disastrous for the cavity, because in doing so, the tool would cut into the side (9) to (10). The reason for that is as explained earlier; the system uses the previous, current, and the next **programmed** points to calculate its compensation. This procedure does not include line (9) to (10).

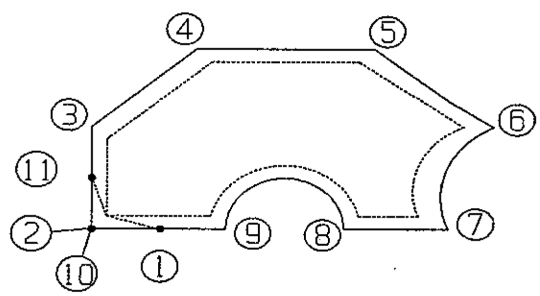


Figure 12.1
Relocation of Points (1), (2), (10) and (11) for a Cavity

In Figure 12.1 the tool should be brought down at points (2) and taken back up at (10).

The purpose of changing line (1) to (2) is to give advance information to the system about line (9) to (10), which happens some 8 blocks later. The length of (1) to (2) can be as small as 00.0001". It is important to note that for the correct advance information the slope of (1) to (2) has to be the **same** as the slope of (9) to (10). In this case the slope is zero.

Similarly, the purpose for changing line (9) to (10) is to give **past** information to the system about line (2) to (3), which happened some 8 blocks **earlier**. Again, the slope of (9) to (10) has to be the same as the slope of (2) to (3).

Regardless of the cutter radius, the tool can now always be brought down on the corner at (2) and brought back up at (10).

Note that the start and end points can either be made to lie inside or outside the cavity.

Programming with Cutter Compensation

When programming with cutter radius compensation, the first and last move the cutter makes should be done off of the part per Figure 12.3. The movement made prior to cutting should be at least a distance of the cutter diameter being used.

Block #	Block Entry Info	
N1001	G0 X-1 Y-1	
N1002	G41 X0 D1	(D offset = tool radius)
N1003	G1 Y3 F10	
N1004	X3.5	
N1005	G3 R.5 XC4 YC3 X4.5 Y3	
N1006	G1 X6.8	
N1007	Y2	
N1008	X4.5 Y0	
N1009	X-1	
N1010	G40 Y-.5	

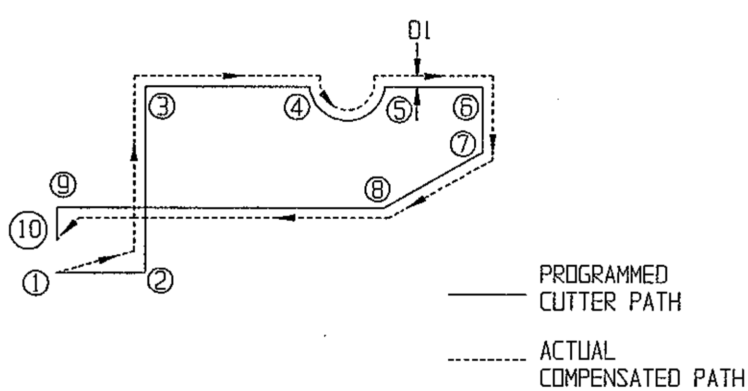


Figure 12.3
Programming with Cutter Compensation

3.12.2 Non-movement (G65)
Starting and Ending Cutter Compensation

The G65 code placed on a line with coordinates will cause these coordinates to be used for cutter compensation points but skipped during machine movement.

G65 X___ Y___ Z___ Machine will not move to the XYZ coordinates

The G65 will allow the programmer to turn cutter compensation on and get the tool to drop or retract at a specific point without doing any extra moves. Generally the no move point would be chosen to be a point on the part that directly precedes the tool down point. On a tool retract the no move point would be a point on the part directly after the tool up point. The no move point does not have to lie on the part but points on the part generally work the best. The following diagrams show various tool start and retract positions given different no move points as indicated by the dotted line ("---") in Figures 13.1 and 13.2. Point 2 is the desired tool start or retract point. Point 1 is the no move point in the cutter compensation on (pierce) case, and Point 3 is the no move point in the cutter compensation off (retract) case. If cutter compensation is turned on or off using the below format, the tool up and down position can be easily predicted.

Starting and Ending Cutter Compensation
G41 Tool Left
D1 = Tool Radius (Previously Set in D1)

PIERCE
1=point on part before pierce point
2=pierce point
3=first cut move

RETRACT
1=last position before retract
2=tool retract position
3=point after retract

```
G41 D1
G65 X0 Y1
X0 Y0
X1 Y0
```

```
G41 PREVIOUSLY ON
X0 Y1
X0 Y0
G65 X1 Y1
G40
```

```
G41 D1
G65 X-1 Y1
X0 Y0
X1 Y0
```

```
G41 PREVIOUSLY ON
X-1 Y1
X0 Y0
G65 X1 Y0
G40
```

```
G41 D1
G65 X-1 Y0
X0 Y0
X1 Y0
```

```
G41 PREVIOUSLY ON
X-1 Y1
X0 Y0
G65 X1 Y0
G40
```

```
G41 D1
G65 X-1 Y0
X0 Y0
X1 Y0
```

```
G41 PREVIOUSLY ON
X0 Y-1
X0 Y0
G65 X1 Y0
G40
```

```
G41 D1
G65 X1 Y1
X0 Y0
X1 Y1
```

```
G41 PREVIOUSLY ON
X1 Y1
X0 Y0
G65 X1 Y0
G40
```

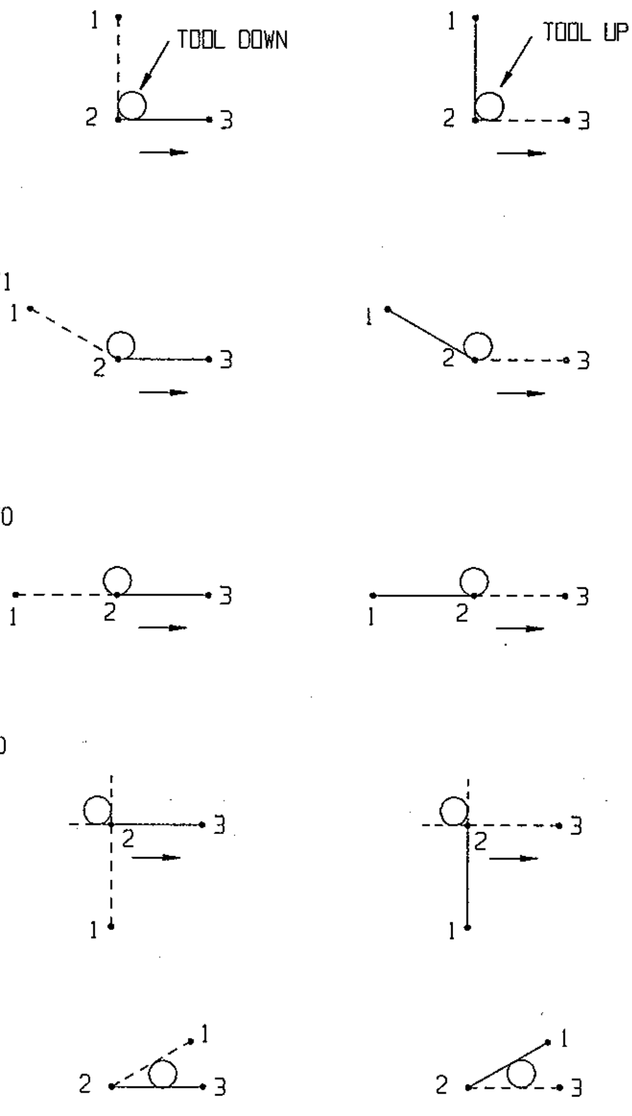


Figure 13.1
Starting and Ending Cutter Compensation (G41)

Starting and Ending Cutter Compensation
G42 Tool Right
D1 = Tool Radius (Previously Set in D1)

PIERCE	RETRACT
1=point on part before pierce point	1=last position before retract
2=pierce point	2=tool retract position
3=first cut move	3=point after retract

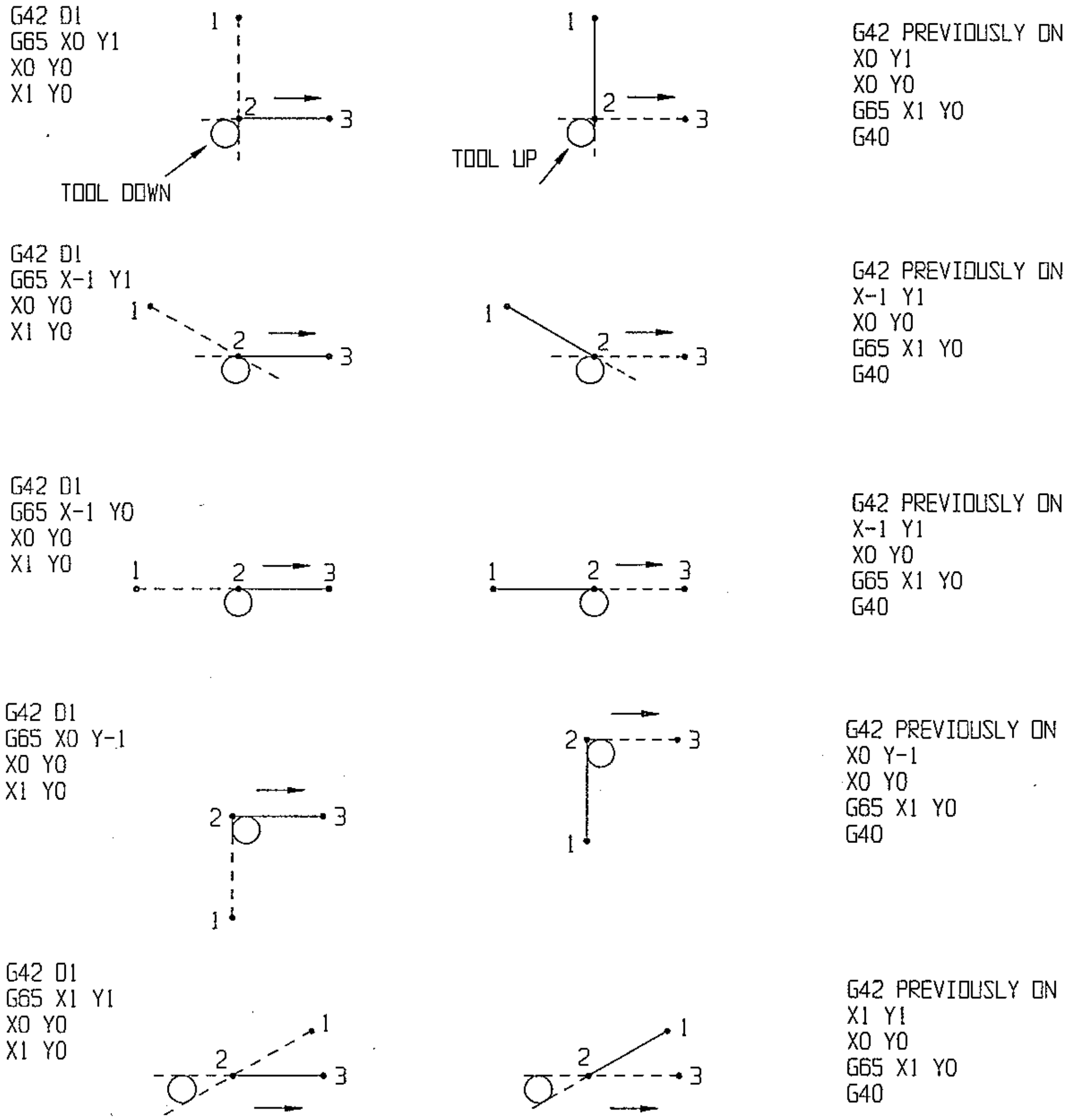
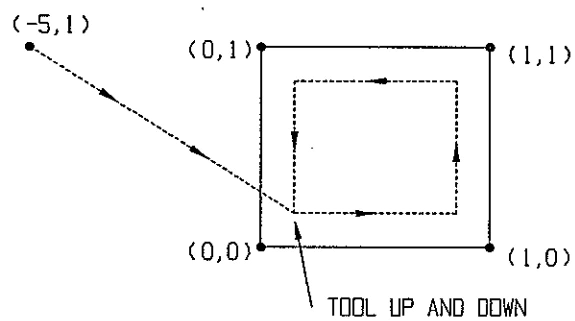


Figure 13.2
Starting and Ending Cutter Compensation (G42)

Enter-Exit Cutter Compensation
Sample Program

```
G0 X-5 Y1       part load/unload point
G41 D1 F10      cutter comp. on offset #1
G65 X0 Y1       no move compensation point
X0 Y0
G1 Z-1          tool down
X1 Y0
X1 Y1
X0 Y1
X0 Y0
G65 X1 Y0       cutter comp. off no move exit point
G40 G0 Z0       tool up
```



Note that the tool enters and exits the part tangent to both walls because of the G65 lines.

<u>Notes on Cutter Compensation</u>

** Cutter Compensation **cannot** be turned **off** at the end of a circular move.

** Turning compensation on can be done both in a block with no axis move, and in a block containing axes moves.

** There is no restriction on how many successive blocks can have no axis information.

** There is no restriction on how lines enter and leave arcs. They can have any angle of intersection as long as an intersection exists.

** Given the correct centers and radii for two intersecting circles, the system automatically checks and corrects the programmed point of intersection, i.e. Trig Help.

** All autoroutines except bolt circle routines use the present axis position as their start point. For this reason it should be made sure that the cutter compensation is turned **off** in a program using these routines so that the axis can position to the programmed center. If the compensation center is used, the whole pocket will be shifted.

** If the programmed point rather than the compensated point is desired, a G40 command should be added to the block containing that point. The G40 could have been a part of the previous block, providing that block had no axis information.

** Similarly, if the point after the desired programmed point is to be compensated, a G41 or G42 should be added to the block containing that next point or to any non-axis command block between the two.

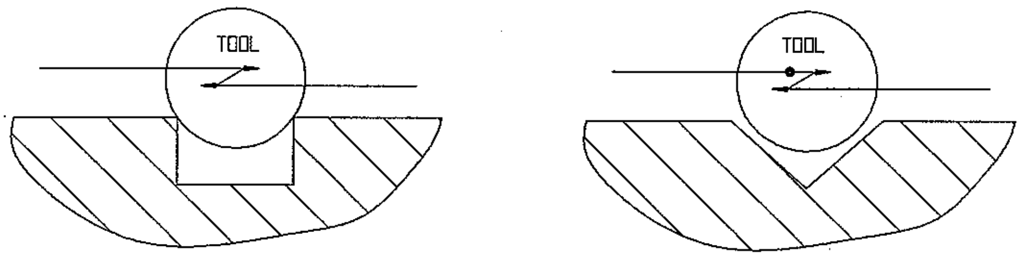
** A program **cannot end** with a circular arc move if cutter compensation is on. If the last move has to be an arc, a 00.0001" linear move should be used for program termination.

** Roughing and finishing passes can be easily made by first entering a tool radius value larger than the actual measured tool radius by the amount of stock to be left on the part for the finish pass.

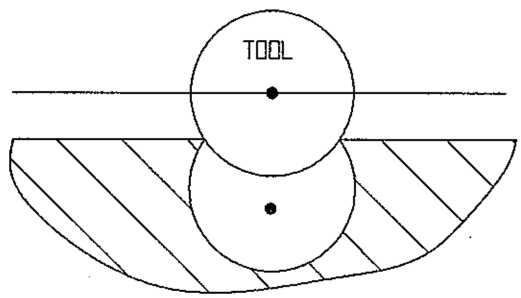
When the program is run, the resultant part will be oversize. Now by entering the actual cutter radius into the system and running the program once more, the finished part size will be obtained.

Thus by entering a **larger** or **smaller** tool radius, the part can be made **under** or **oversize**.

** In general when using cutter compensation, no feature on the part can be smaller than the tool radius. This includes such things as slots, arcs and vees. If a part contains such features they should be replaced by straight lines, cutter compensation should be turned off, or a smaller tool should be used.



In the above cases the tool will back up as it tries to place itself tangent to the walls of the slots or vee.



This case will give a line-to-arc no intersection error.

3.13 Tool length offset (G43, G44, G49)

A tool length offset is activated using a G43 or G44 command.

Command format:

G43 G44	Z ____	H____;	(Z moves to dimension selected referenced to tool length offset selected by H)
or			
G43 G44	H____;		
or			
G43	H10; 1st offset H14; 2nd offset H13; 3rd offset H15; 4th offset		

The direction of the offset is controlled by G43 and G44; the magnitude of the offset is set by the offset value in the H table.

G43 is a + offset (value in H table is added to axis)
G44 is a - offset (value in H table is subtracted from axis)

Once a G43 or G44 offset is activated it will remain in effect until cancelled by a G49 or H00 command. The H offsets can be changed throughout the program without cancelling the previous offset with a G49 or H00. The new H offset will automatically take effect in either the G43 or G44 mode, whichever is active at the time the new H code is executed. However, power-on puts the control in the H00 G49 mode, and no H offsets will be acted upon until one G43 or G44 is executed. The H offsets will always be added to the axis perpendicular to the current plane at the time the H offset is activated.

```
G17 XY plane   H add/sub Z axis
G18 XZ plane   H add/sub Y axis
G19 YZ plane   H add/sub X axis
```

If an H offset is activated in the G17 XY plane and then the plane is switched to G18 XZ plane, the offset will remain in effect and still be added to the Z axis. However if another H offset is activated while still in the G18 XZ plane, it will be added to the Y axis. Both offsets will be in effect, one on the Z axis and one on the Y axis, until they are cancelled by H00's or a G49.

```
H offsets set via front panel commands
          H01 = 1.5
          H02 = -.5
          H03 = -1.25
          H04 = 5

Various program lines and results
          G17 G43 H1
          G90 Z0          Z moves to 1.5
          Z1 H3           Z moves to -.25
          G44 H3
          Z0              Z moves to 1.25
          H4
          Z0              Z moves to -5
          G19 G43 H2
          X0 Z0           Z moves to +5
                          X moves to -.5
          H0
          X0 Z0           Z moves to +5
                          X moves to 0
          G49 X0 Z0       Z moves to 0
                          X moves to 0
```

G90 and G91 modes have no bearing on how the H codes are added/subtracted to the final axis position.

3.14 Cancel scaling (G50)
Set scaling (G51)

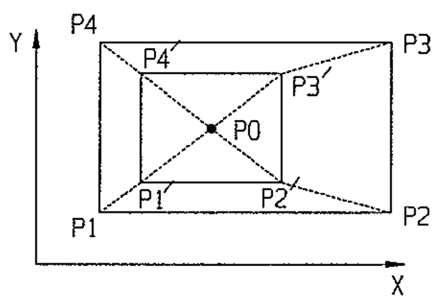
Scaling can be commanded at any time during a program by using the G51 command.

Command format:

G51____ I____ J____ K____ X____ Y____ Z____

I, J, K are the scaling center. If I, J, K are not specified in the G51 line, the scaling center will default to the last center used. The scaling center is set to 0 on power up.

X, Y, Z are the scale factors for each axis. The range of each scale factor is ±999.9999 to ±000.0001. The scale factors, once set, remain in effect until changed or cancelled by a G50. On power up all scale factors are set to 1.



P1 - P4	original program no scaling
P1' - P4'	scaled program
P0	scaling center

Notes on Scaling

1. Once set, scaling remains in effect until cancelled by a G50.

2. If arcs are being scaled, the scale factors for the arc's axes must be equal.

3. Scaling results are rounded down (.00009 = .0000). During scaling some moves may go to zero which could affect cutter compensation.

4. G27, G28, G29, G30 and G92 are not affected by the scale factors.

3.15 Coordinate systems

The machine zero is a fixed point on the machine. The machine zero point is normally decided by the machine tool builder and set by a limit switch and encoder marker pulse on each axis.

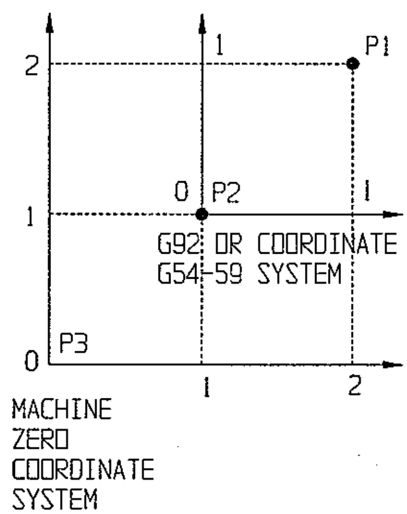
The machine zero point is established when the home command is first executed.

Once the machine zero point is established it is not changed by reset, coordinate system call (G53-G59), coordinate system shift (G92) or local coordinate system setting (G52).

Software limits are set by using the machine zero point.

3.15.1 Machine coordinate system (G53)

A G53 code preceding any XYZ move will cause those dimensions to be relative to the machine zero point.



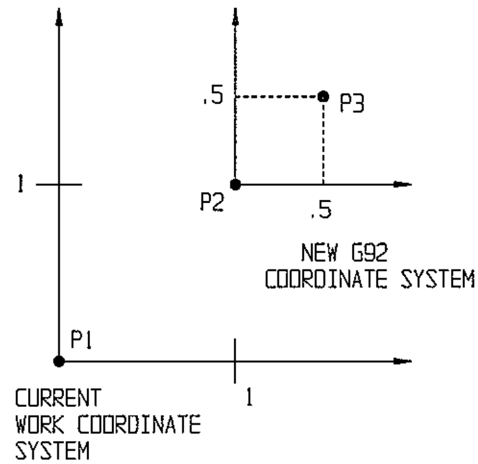

```
X1 Y1          move to P2
G92 X0 Y0
X1 X1          move to P1
X-1 Y-1        move to P3
   or
G53 X0 Y0      move to P3
```

A coordinate system used to align the work part dimensions to the machine's programs is called a work coordinate system. The work coordinate system is set by either of the following methods:

1. using a G92 command
2. using a G53 command
3. using G54 - G59 commands.

3.15.2 Floating zero (G92)

This command establishes the work coordinate system so that the position of the tool becomes the programmed position in the current work coordinate system.



If the machine is positioned at P2, which is a command of X1 Y1, and then a G92 X0 Y0 is commanded, the next time X.5 Y.5 is commanded the machine will position to P3. If the machine is positioned at P1 and G92 X-1 Y-1 is commanded, the next time X.5 Y.5 is commanded the machine will position to P3.

Note:

1. *When using a G92, tool length offsets should be cancelled or accounted for, as they will affect the new floating zero position on the Z axis.*

2. *G92 should not be used when cutter compensation is active; the control should always be in G40 mode.*

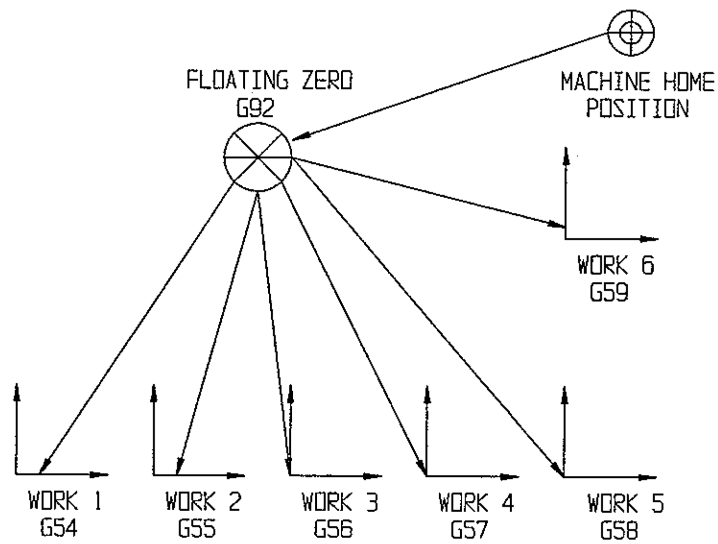
3. *The distance shifted via a G92 in one work coordinate system will be applied to other work coordinate systems when they are activated via G53 - G59 commands. If this is not desirable then a new G92 must be set when changing coordinate systems.*

4. *On power up the G54 coordinate system is active.*

3.15.3 Work coordinate systems (G54 - G59)

These work coordinate systems are set via the keyboard. The dimensions of the coordinate zero point are always relative to the G92 Floating Zero point. To set a work coordinate system, select "PARMS" on the main menu, then "COORD". The work coordinate menu will come up allowing you to type in the offset coordinates for each coordinate system. The "Home Position" offsets are parameters which shift all coordinate systems relative to the

Machine Zero Point. Normally the "Machine Zero Point" and the "Home Position" are the same.



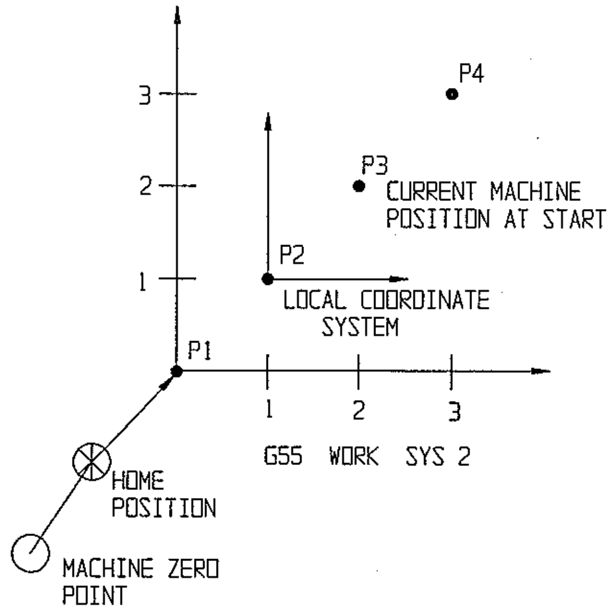
G55 X1 Y1	moves to X1 Y1 in work offset 2
G59 X1 Y1	moves to X1 Y1 in work offset 6
G54	is always the power on coordinate system and is active upon running a program.

3.16 Local coordinate system (G52)

The G52 command is similar to the G92 command in that it uses the current coordinate system zero as its reference point instead of the current machine position (G92).

Notes: *1. A G52 is modal; therefore it will affect all coordinate systems once set.*

2. To cancel a G52, enter G52 X0 Y0.



X2 Y2	moves to P3
G52 X1 Y1	sets zero at P2 dim. rel. P1
X1 Y1	stays at P3
X2 Y2	moves to P4

Using G92

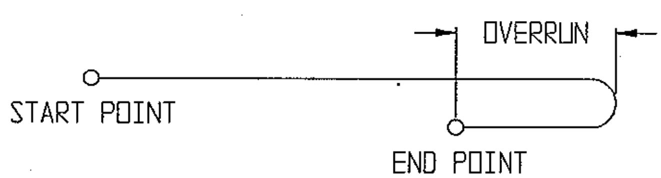
X2 Y2	moves to P3
G92 X1 Y1	sets zero at P2 dim. rel. P3
X1 Y1	stays at P3
X2 Y2	moves to P4

3.17 Exact stop mode (modal) (G61)

When G61 is commanded, deceleration is applied to the end point of the cutting block and in-position is performed per block thereafter. G61 is valid until G63 (tapping mode) or G64 (cutting mode) is commanded.

3.18 Single direction positioning (G60)

For accurate positioning without backlash, positioning from one direction is available.



G60 X___ Y___

G60 is a one-shot G code and is used in place of G00.

Notes: *1. The amount of overrun is preset by the machine tool builder.*

2. During canned cycles Z axis moves will not be affected.

3. Overrun direction is not affected by mirror imaging.

4. If "G00 unidirectional approach" was set by the machine tool builder, the same positioning sequence would happen with each G00 move.

3.19 Tapping mode (modal) (G63)

When G63 is commanded, feedrate override and spindle speed override are ignored (always regarded as 100%), and feedhold becomes invalid. G63 is valid until G61 (exact stop mode) or G64 (cutting mode) is commanded.

3.20 Cutting mode (modal) (G64)

When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode) or G63 (tapping mode) is commanded.

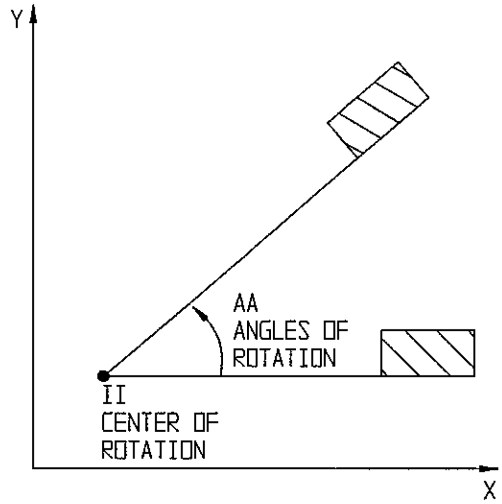
3.21 Coordinate system rotation (G68 - G69)

G68 can be used to rotate a programmed shape about a predefined center point. The plane of rotation is defined by G17, G18, G19; the center of rotation is defined by IJK; and the angle of rotation by AA. The command format is as follows:

G68 AA+_____ I_____ J_____ K_____

AA+ is CCW
AA- is CW

IJK specify the center of rotation in the plane selected by G17 G18 G19. On power up the center of rotation defaults to the current coordinate system zero point. If the IJK's are not present in the G68 block the center of rotation will be the last center specified.



Care needs to be taken when using rotation in conjunction with other functions. Functions such as mirror image, scaling and cutter compensation need to be thought about carefully when used together with rotations. Some of the basic rules are as follows:

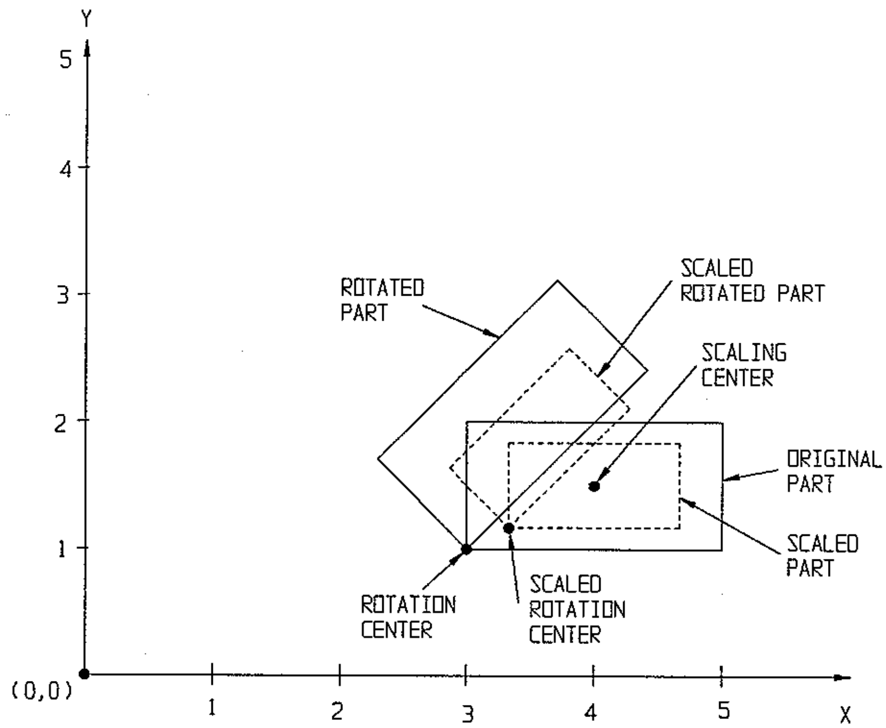
1. Cutter compensation should be off (G40) when rotation is called.

2. If scaling is on before rotation, the rotation center will be scaled; if rotation is called before scaling, the rotation center will not be scaled.

3. The order of on and off is first on last off.

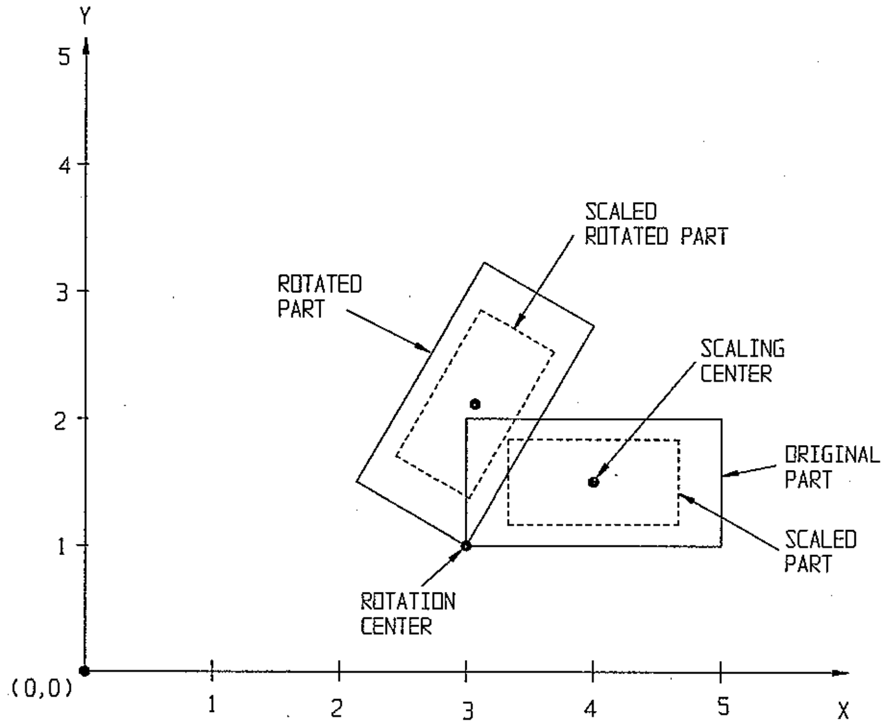
   G51 . . . . scaling on
   G68 . . . . rotation on
   G41 . . . . cutter compensation on
   .
   .
   .
   G40 . . . . cutter compensation off
   G69 . . . . rotation off
   G50 . . . . scaling off

4. If the rotational center is scaled it will remain scaled until replaced by a new center.



Part Scaled then Rotated

```
G51 I4 J1.5 X.9 Y.9
G68 I3 J1 AA45
X3  Y1
X5
Y2
X3
Y1
G69
G50
```



Part Rotated then Scaled

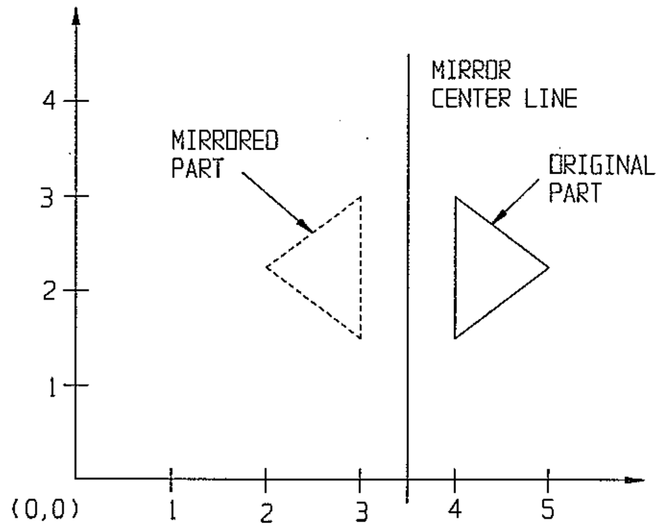
```
G68 I3 J1 AA45.00
G51 I4 J1.5 X.9 Y.9
X3  Y1
X5
Y2
X3
Y1
G50
G69
```

3.22 Cancel mirror image (G70)
Set mirror image (G71)

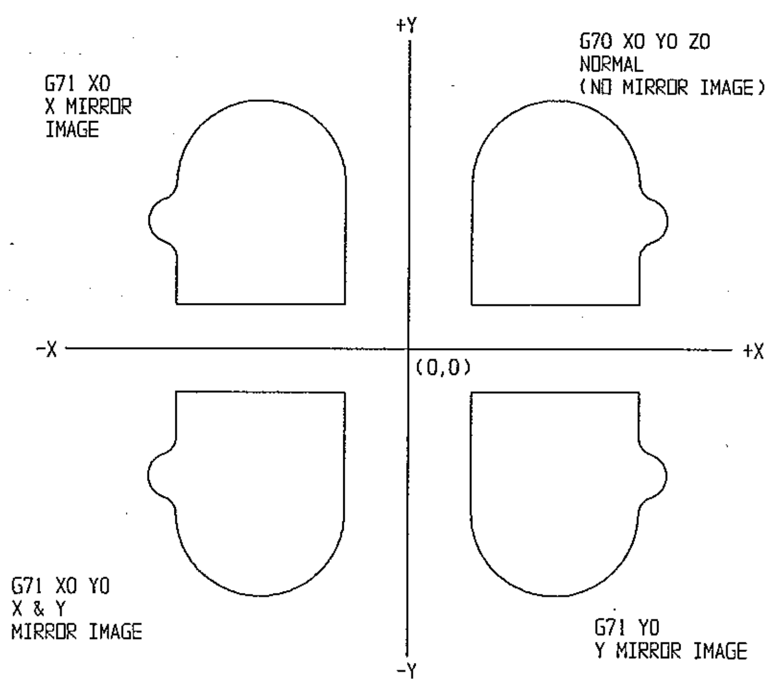
The mirror image commands allow mirroring about any centerline. The mirror image centerline is not affected by either scaling or rotation being on or off.

The command is as follows: G71 X____ Y____ Z____

X,Y,Z are the axes to mirror and the distance from the current coordinate zero to create the mirror centerline. There must be at least one X, Y, or Z after the G71 command.




```
G71 X3.5
X4 Y1.5
X5 Y2.25
X4 Y3
Y1.5
G70
```



MIRROR IMAGE

G70 cancels mirror image.

3.23 Canned cycles

A canned cycle simplifies the program by using a single block with a G code to specify the machining operations usually specified in several blocks.

G code	Drilling -Z	Operation at hole bottom	Retraction +Z	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Dwell→ spindle CW	Feed	Left hand Tapping cycle
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Driling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid	Peck drilling cycle
G84	Feed	Dwell→ spindle CCW	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Table 3
Canned Cycles

Generally a canned cycle consists of a sequence of six operations as shown below.

- Operation 1: Positioning of axes X and Y (or 4th and 5th if enabled)
- Operation 2: Rapid traverse up to point R
- Operation 3: Hole machining
- Operation 4: Operation at the bottom of a hole
- Operation 5: Retraction to point R
- Operation 6: Rapid traverse up to the initial point

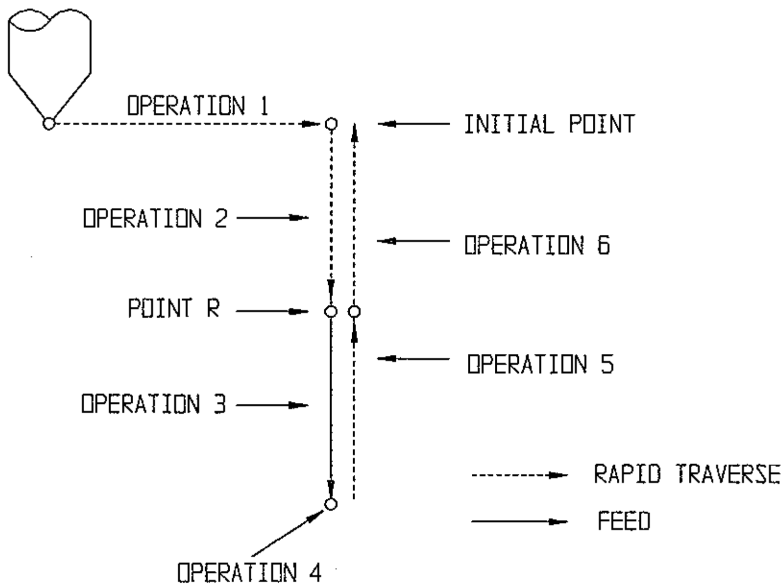


Figure 14.1
Canned Cycle Operation

Positioning is performed on the XY plane and hole machining is performed with the Z axis. Positioning and hole machining must use this plane and axes combination. Canned cycles are indifferent to the plane selection G commands.

Canned cycle operations consist of three basic modes which are specified by particular modal G codes as shown below.

(1)	Data format	G90	Absolute
		G91	Incremental
(2)	Return point level	G98	Initial point level
		G99	R point level
(3)	Drilling mode	G73	
		G80	
		G81	
		.	See Table 3
		.	
		.	
		G89	

Note: *The initial level means the absolute value of the Z axis when the canned cycle is first turned on.*

Figure 14.2 shows how to specify data in G90 or G91 mode.

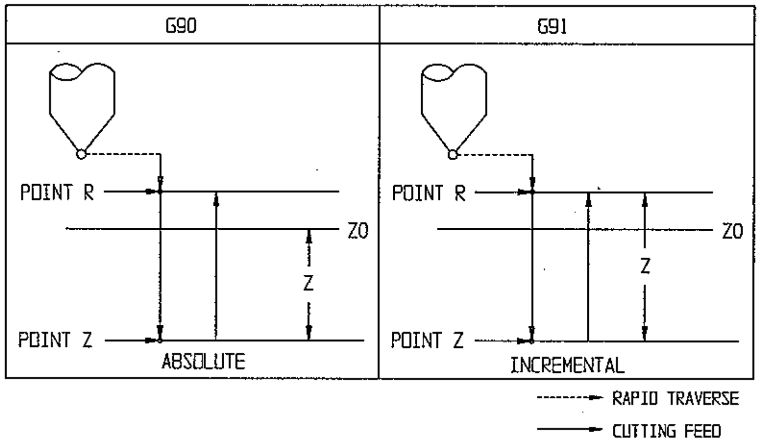


Figure 14.2
Absolute and Incremental Programming

If the tool is to be returned to point R or to the initial level, it is specified by G98 or G99. (See Figure 14.3.) Use G99 for the first hole, and use G98 for the last hole. When the canned cycle is repeated in G98 mode, the tool is returned to the initial level after each hole.

In the G99 mode the initial level does not change and the tool is returned to the R point after each hole.

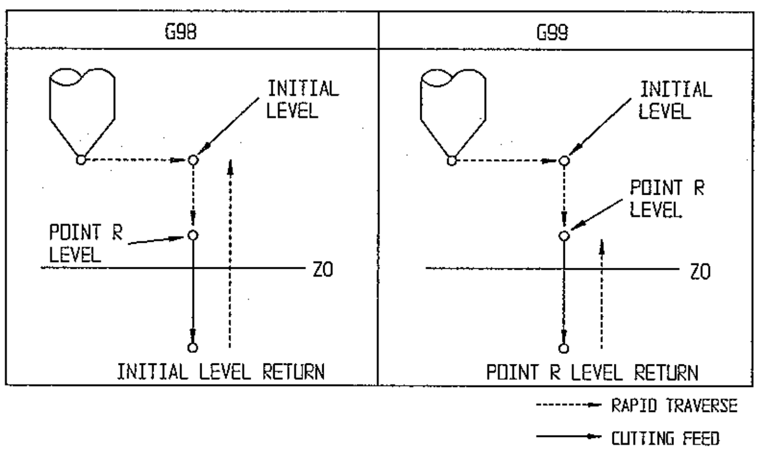


Figure 14.3
Initial Level and Point R Level

The drilling data is specified following G73/G81 to G89. Data is stored in the control as modal values and is retained for future use in other cycles.

The machining data in a canned cycle is specified as shown below:

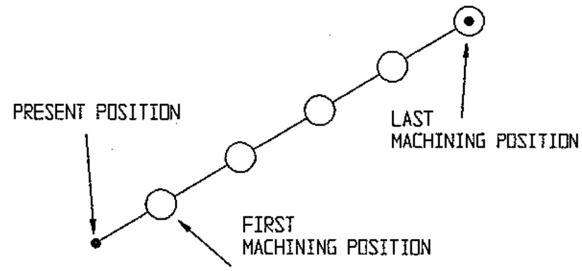
```
G __ __        X__ Y__        Z__ R__ V__ Q__ P__ F__

     |              |                     |     Drilling
     |              └- Hole position      └      data
     |                     data
     └- Drilling mode
```

Drilling mode . . .	G __ __	See Table 3.
Hole position data	X Y	Specifies hole position by an incremental or absolute value. The path and feedrate are the same as G00 positioning.
Drilling data	Z . . .	Specifies the distance from point R to the bottom of the hole with an incremental value, or the position of the hole bottom with an absolute value.
	R . . .	Specifies the distance from the initial level to the point R with an incremental value, or the position of point R with an absolute value.
	V . . .	Specifies the first Z depth in G73 and G83
	Q . . .	Specifies the increment value for G73 and G83.
	P . . .	Specifies the dwell time at the bottom of the hole. The relationship between the time and the specified value is the same as for G04.
	F . . .	Specifies the feedrate.

The drilling mode (G__ __) remains unchanged until another drilling mode is specified or the canned cycle is cancelled with a G80. Once the drilling data has been specified in a canned cycle, it is retained until it is changed or the canned cycle is cancelled. All required drilling data needs to be specified when the canned cycle is started and only data to be changed needs to be specified during the cycle.

When the system is reset during a canned cycle the drilling mode, drilling data, and hole position data are deleted.

Equally spaced holes can be programmed by use of the L address.



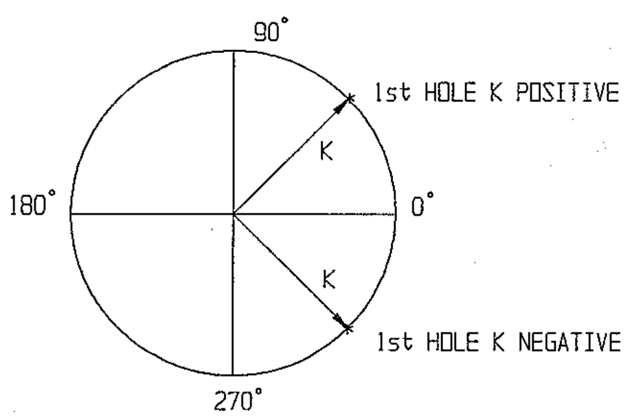
G81 X___ Y___ Z___ R___ L5 F___

X____ Y____ specifies the first and subsequent hole positions in the incremental mode (G91). In the absolute mode (G90), a hole would be repeatedly drilled at the same position. Each hole machining operation is detailed below.

3.23.1 Bolthole routine (G72)

The bolt circle autoroutine can be used with any of the canned cycles. Canned cycles, when used with this autoroutine, differ in that hole positions are not specified. The G72 line indirectly specifies all the hole positions based on the following input: number of holes in 360°, number of holes to be drilled, the radius of the bolt circle, the starting angle of the first hole and the center of the bolt circle. The control will then calculate the position of each hole and rapid to each hole in straight line moves. The angle of the first hole is the angle from the 3 o'clock position. A positive starting angle is counterclockwise from 3 o'clock. A negative starting angle is clockwise from 3 o'clock.

Note: *Holes will be drilled clockwise from the first position.*



Definition of Starting Angle

The following is a format of the G72 command:

```
G72   X___  Y___    R___     Q___      P___      K___
```

position of bolt circle center

radius of bolt circle

holes in 360°

of holes drilled in 360°

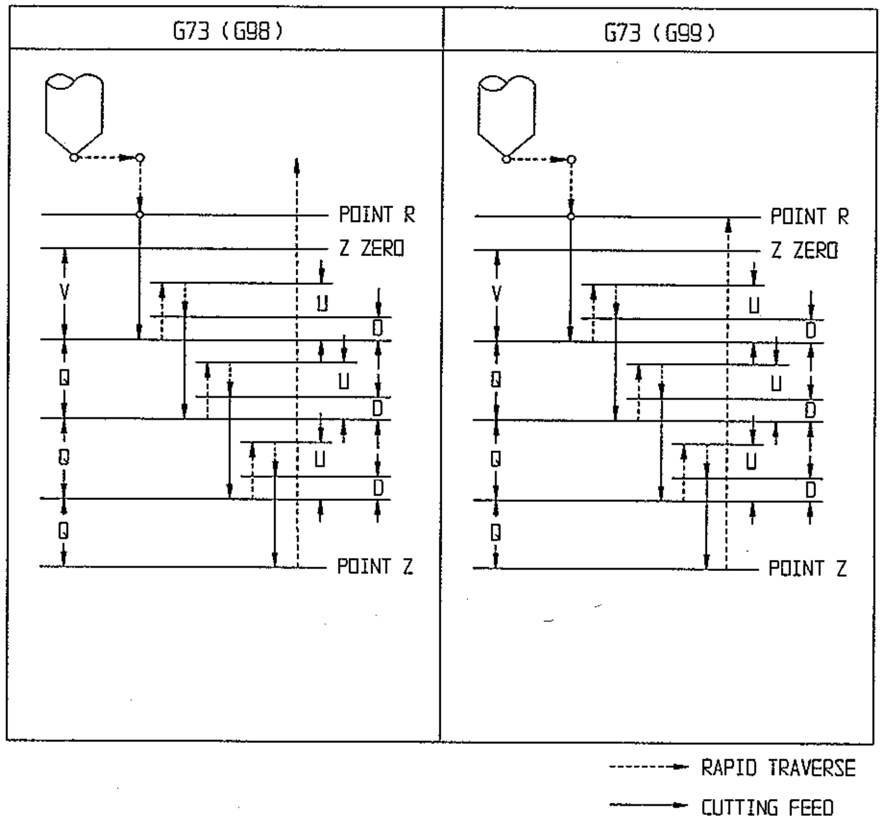
angle of first hole

Program to Drill a 5 Hole 1" Radius Bolt Circle

```
N1    G20 G90 (Inch/Absolute)
N2    S1000 M3 G43 H1 (spindle CW 1000 RPM, activates
      tool #1's length offsets)
N3    P156=1  (Radius of bolt hole)
N4    P157=45 (Angle of first hole)
N5    P158=5  (# of holes in 360°)
N6    P159=5  (# of holes drilled)
N7    G81  G99  Z-1  R.1  F10
                           Z feedrate
                           R plane
                           Drill depth
                           Return to R point
                           Drill
N8    G72  X0  Y0   (Bolthole routine, center of
                    bolthole circle)
```

Note: *If P159 is less than P158, a partial bolt circle will be drilled on a P158 pattern.*

3.23.2 High speed peck drilling cycle (G73)



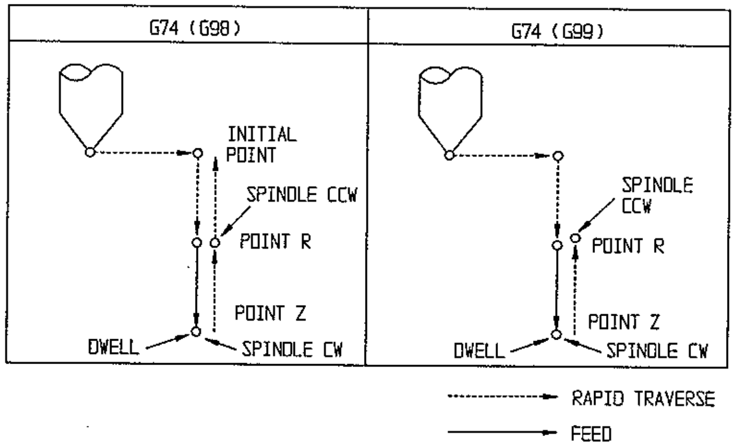
G73 G98/G99 Z___ R___ V___ Q___ U___ D___ F___

The G73 command specifies the high speed peck cycle. This cycle will do the following:

1. Rapids to point R
2. Feeds down to point V
3. Rapids up U value
4. Rapids down to D value
5. Feeds down by Q value or Z point (whichever is less)
6. Repeat steps 3-5 until point Z is reached
7. Rapids to initial point/point R as determined by G98/G99

Note: *The V code is optional. If left out, the first depth would equal R___ - Q___.*

3.23.3 Left hand tapping cycle (G74)



G74 G98/G99 Z___ R___ B___ P___ F___

The G74 command specifies the left hand tapping cycle. At each of the following axis positions, this cycle will do the following:

1. Rapid to point R
2. Feeds to point Z
3. Dwells before reversing (specified by the B code)
4. Reverses spindle (CW)
5. Dwells after reversing (specified by the P code)
6. Feeds to point R
7. Reverses spindle (CCW)
8. Rapids to initial point, if specified by the G98 code

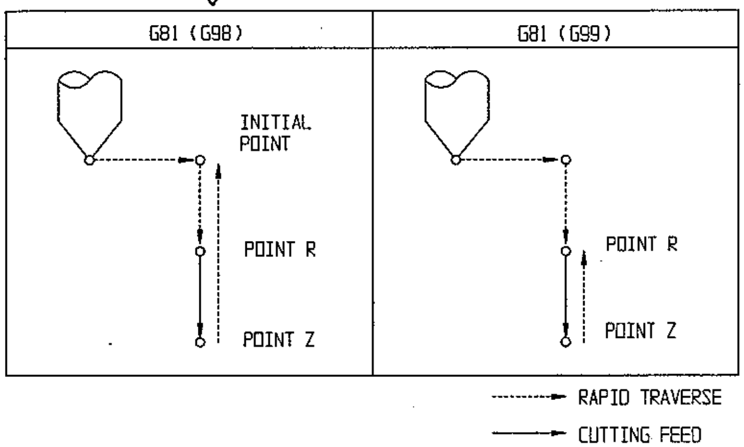
Note: *During tapping the feedrate override and spindle override switches are ignored, and the cycle does not stop until the return operation when feedhold is applied.*

3.23.4 Canned cycle cancel (G80)

The canned cycle (G73, G74, G81 to G89) is cancelled and normal operation is subsequently performed. The points R and Z are also cancelled. (That is, R=0 and Z=0 for the incremental command.) Other drilling data is also cancelled.

3.23.5 Drilling cycle, spot boring cycle (G81)

✓

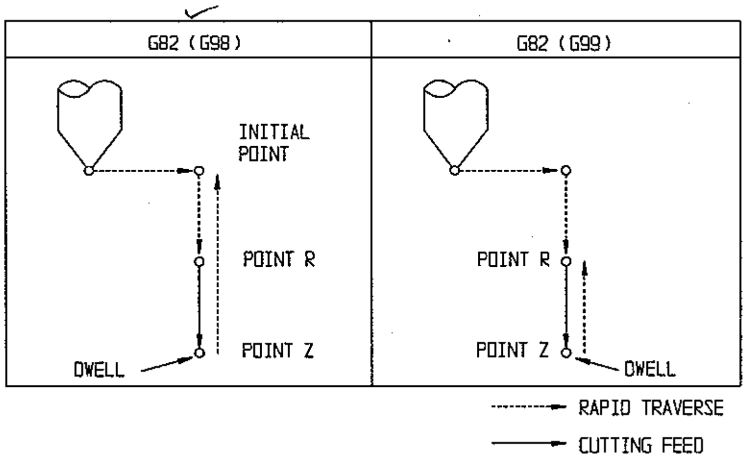


G81 G98/G99 Z___ R___ F___

The G81 command specifies the drilling cycle. This cycle will do the following:

1. Rapids to point R
2. Feeds down to point Z
3. Rapids to initial point/point R as determined by G98/G99

3.23.6 Drilling cycle, counter boring cycle (G82)

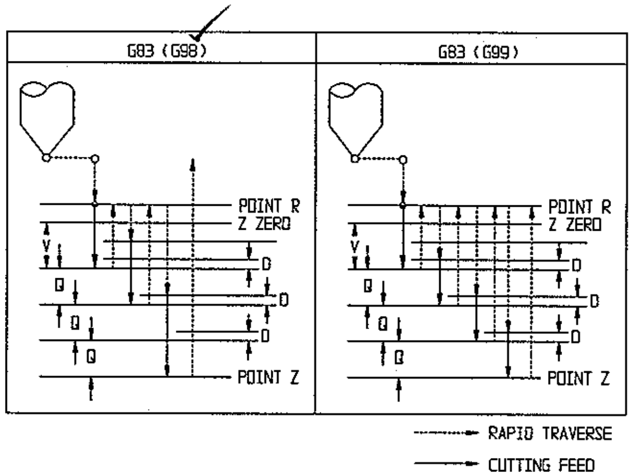


G82 G98/G99 Z___ R___ P___ F___

The G82 command is similar to the G81 command; however, a dwell (specified by the P code) is performed at the bottom of the hole. This cycle will do the following:

1. Rapids to point R
2. Feeds down to point Z
3. Dwells by P___ seconds
4. Rapids to initial point/point R as determined by G98/G99

3.23.7 Peck drilling cycle (G83)



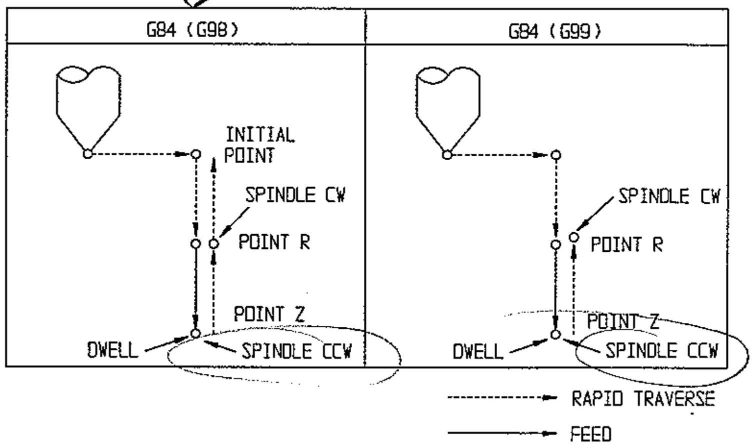
G83 G98/G99 Z___ R___ V___ Q___ D___ F___

The G83 command specifies the peck drill cycle. This cycle will do the following:

1. Rapids to point R
2. Feeds to Point V
3. Rapids up to point R
4. Rapids down to D value
5. Feeds down by Q value or Z point (whichever is less)
6. Repeats steps 3-5 until point Z is reached
7. Rapids to initial point/point R as determined by G98/G99

Note: *The V code is optional; if left out, the first depth would equal R___ - Q___.*

3.23.8 Right hand tapping cycle (G84)



G84 G98/G99 Z___ R___ B___ P___ F___

The G84 command specifies the right hand tapping cycle. At each following axis position this cycle will do this:

1. Rapids to point R
2. Feeds to point Z
3. Dwells before reversing (specified by the B code)
4. Reverses spindle (CCW)
5. Dwells after reversing (specified by the P code)
6. Feeds to point R
7. Reverses spindle (CW)
8. Rapids to initial point, if specified by the G98 code

Note: *During tapping the feedrate override and spindle override switches are ignored and the cycle does not stop until the end of the return operation when the feedhold is applied.*

TAPPING FEEDS AND SPEEDS

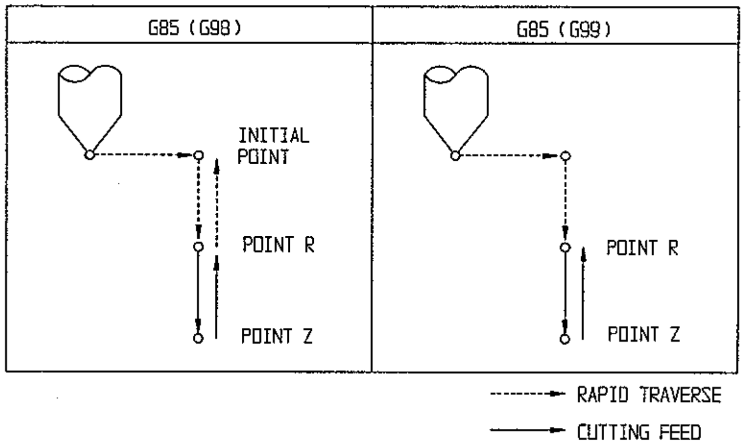
$$\frac{1}{\text{PITCH}} = \text{LEAD}$$

RPM X LEAD = FEEDRATE

Example: 1/4-20 tap, spindle rpm 400
1/20 = .05 (lead)
400 x .05 = 20 (feedrate)

Feedrate may need adjustment for proper operation of tap holder. If tap is pulled too far in the holder, feedrate should be increased. If tap is pushed into the holder, feedrate should be decreased.

3.23.9 Boring cycle (G85)

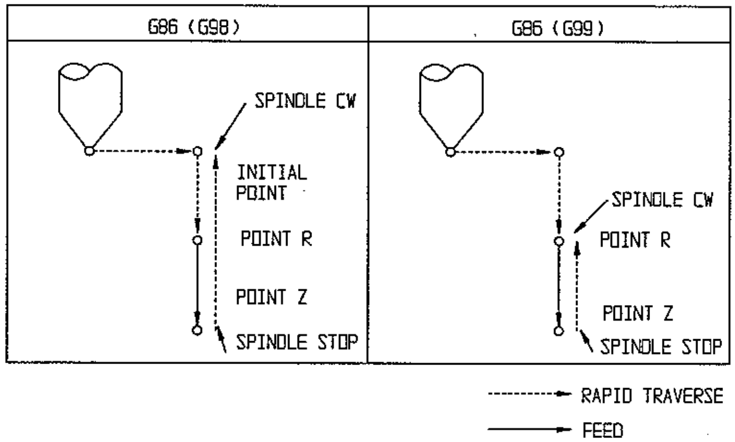


G85 G98/G99 Z___ F___

The G85 command specifies the boring cycle. At each following axis position this cycle will do this:

1. Rapids to point R
2. Feeds to point Z
3. Feeds to point R
4. Rapids to initial point if specified by the G98 code

3.23.10 Boring cycle (G86)

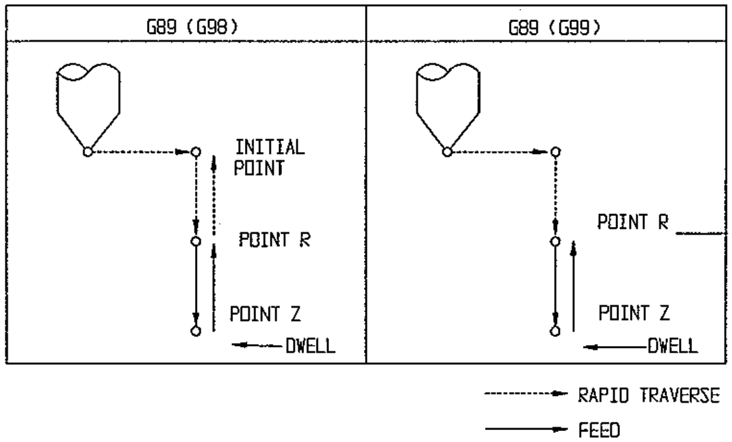


G86 G98/G99 Z___ R___ F___

The G86 command specifies the high speed peck cycle. At each following axis position this cycle will do this:

1. Rapids to point R
2. Feeds down to point Z
3. Spindle stops
4. Rapids to point R
5. Spindle restarts (CW)
6. Rapids to initial point/point R as determined by G98/G99

3.23.11 Boring cycle (G89)



G89 G98/G99 Z___ P___ F___

The G89 command specifies the bore with dwell cycle. At each following axis position this cycle will do this:

1. Rapids to point R
2. Feeds to point Z
3. Dwells at bottom (specified by P code)
4. Feeds to point R
5. Rapids to initial point, if specified by the G98 code

NOTES ON CANNED CYCLE SPECIFICATIONS:

Note 1: *The spindle must be turned on by the miscellaneous function (M code) before a canned cycle is specified.*

```
M3                Spindle CW
.
.
.
G __ __ ......    Correct
.
.
.
M5                Spindle Stop
.
.
.
G __ __ ......    Incorrect (M3 or M4 must be
                  specified before this block.)
```

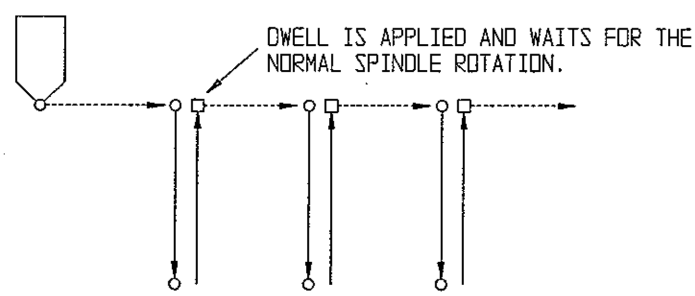
Note 2: *If the block contains X, Y or R data, drilling is performed in canned cycle mode. If the* *block does not contain the X, Y and R data, drilling is not performed. However, when "G4 X___" is specified, drilling is not performed even if X is specified.*

Note 3: *If a following block contains a Z position by itself, drilling will not be performed. The Z axis will, however, rapid to this point. This can be used to manipulate a tool up-and-over obstructions without disabling the canned cycle.*

```
G00 X____
G81X____  Y____  Z____  R____  F____  P____
F____     (Drilling is not performed.  Value F is
          updated.)
M____     (Drilling is not performed.  Only the
          miscellaneous function is executed.)
G4 P____  (Drilling is not performed.  Drilling
          data P is not changed by G04 P__.)
```

Note 4: *Specify drilling data in the block where drilling is performed. Entries (X, Y, Z, R, F or P) are not stored as modal data unless they appear on the same line as the G81.*

Note 5: *When using G74, G84 and G86, if the distance between holes is too small, a Dwell (G04) needs to be inserted between moves to give the spindle time to reverse directions.*




```
G00 M___
G86 X___  Y___  Z___  R___  F___
G04 P___  (Dwell is performed, but drilling is not.)
X__  Y___
G04 P___  (Dwell is performed, but drilling is not.)
X__  Y___
G04 P___  (Dwell is performed, but drilling is not.)
 .    .
 .    .
 .    .
```

This may not have to be considered if spindle up-to-speed is available on the machine tool.

Note 6: *When a miscellaneous function is specified in the same block as a canned cycle command, the M code and MF signals are sent at the first positioning operation (Operation 1, page 3-60). The control waits for the finish signal (FIN) at the end of positioning before starting the next drilling operation.*

Note 7: *When tool length offset (G43, G44, G49) is specified in canned cycle mode, offset is applied when the tool is positioned to point R (Operation 2).*

Note 8: *Operator Precautions*

a) *Single block*
When a canned cycle is performed in the single block mode, the control stops at the end of Operations 1, 2 and 6 as shown in Figure 14.1 on page 3-61. Therefore it must be started three times to drill one hole.

b) *Feedhold*
When a feedhold is applied between Operations 3 to 5 in canned cycle G84, the FEEDHOLD lamp immediately lights, but the control continues to operate up to Operation 6 and stops. If a FEEDHOLD is applied again during Operation 6, it immediately stops.

c) *Override*
The feedrate override and spindle override is assumed to be 100% during the operation of canned cycles G74 and G84.

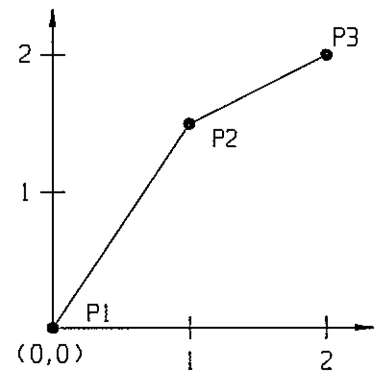
3.24 Absolute/Incremental Mode

3.24.1 Absolute mode (modal) (G90)

This function causes the control to go into its normal Absolute operating mode. In this mode, all dimensions are referenced from a single reference point. This reference can either be the Home Zero" point which is a fixed point on the machine, or an operator defined work coordinate point, which can be anywhere within the physical dimensions of the machine.

X Y Z dimensions relative to the "Home Zero" can only be negative because of the table/saddle limitation. Dimensions relative to work coordinates can be either negative or positive, depending on where the operator sets the zero coordinate.

G90 is active on power up. G90 cancels G91.



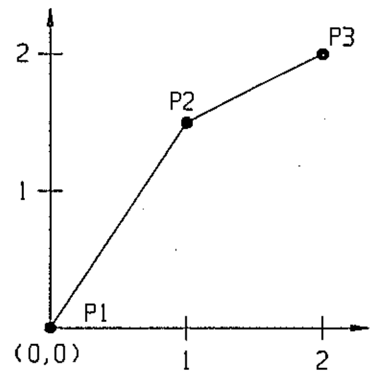
Absolute Positioning

```
G90 X0 Y0       P1
    X1 Y1.5     P2
    X2 Y2       P3
```

3.24.2 Incremental mode (modal) (G91)

This function causes the control to go into the incremental mode. In this mode all dimensions are entered relative to the machine position in the previous block. In the case of MDI, the dimensions are relative to the current machine position. Dimensions in G91 can be either positive or negative. Care should be taken when using G91. Whenever activating tool offsets, R-plane dimensions, or setting "Floating Zeroes" via G92, the control should not be in the G91 mode.

G91 cancels G90.



Incremental Positioning

```
G90 X0 Y0       P1
G91 X1 Y1.5     P2
X1 Y.5          P3
```

3.25 Floating zero (G92)

Refer to Section 5.4 on floating zero.

3.26 Return to initial level or to R level (G98/G99)

These two G codes are only used when the control is in one of the Z axis canned cycles (G73 thru G89) or autoroutines (G24-25, G34-35). A G98 or G99 can be executed anywhere in a program or subroutine. If a canned cycle is active the G98 or G99 will take effect on the next XY move. If a canned cycle is not active the G98 or G99 will take effect on the first XY move after the canned cycle is activated. A G98 will cause a canned cycle to return the Z axis between holes to the same level it was at when the cycle was activated. A G99 will cause a canned cycle to return the Z axis between holes to the current R-level.

```
G98
X1 Y1 Z1
G81 X5  Y-4 Z-1.3 R.2 F10     X5 Y-4 then Z.2 then Z-1.3
X2 Y3                         Z1 then X2 Y3 then Z.2
                               then Z-1.3 at F10
G99 X3 Y-1                    Z.2 then X3 Y-1 then Z-1.3
G80                           cancel cycle
```

4. MISCELLANEOUS FUNCTIONS (M Functions)

The miscellaneous function codes are one or two digit numbers preceded by the letter M. If the code is less than 10, zero entry is optional (M02 or M2). These codes are used to perform a variety of machine and control functions as listed in the table below.

M Codes	Function	Executed Before Move	Executed After Move
M00	Program Stop		X
M01	Optional Stop		X
M02	End of Program		X
M30	End of Program Rewind		X
M03	Spindle on CW	X *	
M04	Spindle on CCW	X *	
M05	Spindle off		X *
M06	Tool Change		X *
M07	Mist Coolant On	X *	
M08	Flood Coolant On	X *	
M09	Coolants Off		X *
M10	Clamps Brake	X *	
M11	Unclamps Brake	X *	
M19	Orient Spindle (ATC option)	X *	
M20-M29	These are pulsed auxiliary outputs.	X *	
M32	Test Wait Channel	X *	
M90	Graphics Off	X *	
M91	Graphics On	X *	
M98	Subroutine Call Statement	X *	
M99	End of Subroutine Statement	X *	

* *These functions are selectable for either before or after move.*

Table 4 M Codes

All M codes except M98 and M99 produce an M strobe and an 8 bit BCD number on the M,S,T buss.

4.1 Program stop (M00)

The execution of the program is halted on the block containing the M00. Program execution will be resumed when CYCLE START is pushed. If M00 is on a line with a move command the move will be executed before the stop.

4.2 Optional stop (M01)

M01 is the same as M00 except it is only executed if the optional stop switch is enabled.

4.3 Block skip (/)

A line of program can be skipped or ignored by the control. Inserting a "/" at the beginning of a line and enabling the BLOCK SKIP will cause the control to skip that line. In the example below with BLOCK SKIP disabled, the machine will move to the first, second, and third points. When BLOCK SKIP is enabled, the machine will move to the first then third points. Block two is skipped.

```
N1   X0 Y0
/N2  X2 Y2
N3   X4 Y0
```

4.4 End of program (M02, M30)

Either of these codes can be used to indicate the end of a program. The only difference is that M02 will leave the spindle and coolants on while the M30 will turn them off. Both codes will return to the beginning of the program and start over when CYCLE START is pushed.

4.5 Spindle on/off (M03, M04, M05)

These codes turn the spindle on CW (M03), CCW (M04) and off (M05). The spindle on commands will be executed before an axis command. M05 will execute after an axis command. The spindle set up parameters determine the exact sequencing each command will use when turning the spindle on and off.

4.6 Tool change (M06)

This command strobes the M function buss and sends out a 150 millisecond (msec) pulse on the M06 I/O output. It then halts program execution until it receives a tool change complete signal. After the tool change complete signal the program will resume running. If the tool change parameters are set to manual tool change, a cycle start is required after the tool change complete signal is received to resume program operation. For safety reasons a manual tool change should never be attempted unless the machine is in an M06 tool change command.

4.7 Coolants on/off (M07, M08, M09)

These codes turn the coolants on (M07 mist, M08 flood) before an axis command is executed. The coolant off command (M09) will be executed after an axis command.

Caution: *The control will accept more than one M code on a line; however, it is recommended that only one M code per line be programmed. When more than one M code per line exists, the order of execution is somewhat undefined and the program may not run as expected. In general the M codes will execute in numerical order "M00 first M99 last" unless they have been defined to execute after the move statements. (See section on machine setup.)*

4.8 Clamp for rotary table (M10)

This energizes the clamp or brake for the optional rotary table.

4.9 Release clamp on rotary table (M11)

This unclamps or releases the brake for rotary table option.

4.10 Orient spindle (M19)

Selecting this function causes the spindle to orient on machines with the automatic tool changer (ATC) option.

4.11 Test wait channel (M32)

These M functions are for general use and will output a 150 msec pulse on their corresponding outputs. When one of these M codes is commanded, a 150 msec pulse will be produced at the I/O output channel and the corresponding input will be poled. Program execution will halt until the appropriate M function complete signal comes back. At this time the next instructions will be executed.

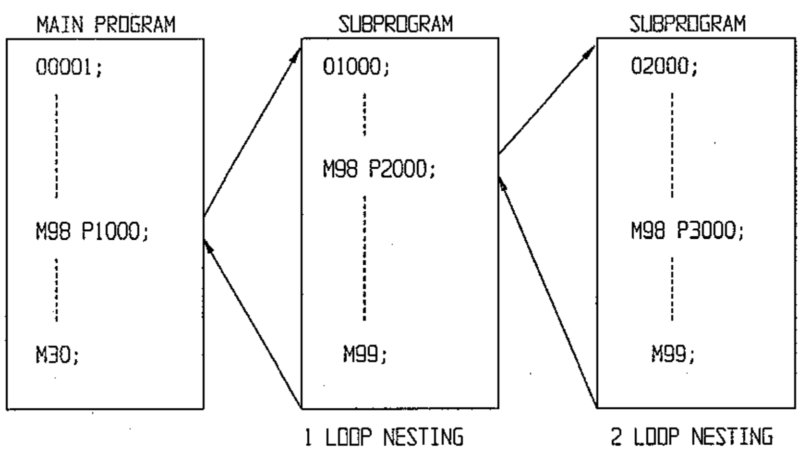
4.12 Graphics off/on (M90, M91)

On a few rare occasions the graphics function on the CNC should be turned off to prevent a graphic memory overflow condition. There are two possible ways to cause a graphic memory overflow. The first is to write a program that loops continuously or has a very large number of loops, 1000 or more. The second is to DNC a very large program and display it as it's running. (Refer to Utilities Section on DNC.) A general rule of thumb when writing a program with loops is to do an M90, Graphics Off, after the first loop. This prevents redundant lines from building up in the graphic memory. After the loop is finished M91, Graphics On, can be executed to display the next section of the program. If the last command executed was a "Graphics Off" and the program is started over, the "Run" program will always restore the CNC to a "Graphics On" state.

4.13 Subprogram call (M98)
Subprogram terminate (M99)

The subprogram call command (M98) can be used to execute any program residing in memory from another program. The called program will be executed until the end, and then control will transfer back to the calling program one block after the M98 command.

When the main program calls a subprogram it is regarded as a one loop subprogram call. Thus a two loop subprogram call can be executed as shown below.



An M98 command when used with an L___ command can call a subprogram repeatedly. An L___ command can specify up to 999 repetitions of a subprogram.

4.13.1 Preparation of subprogram

A subprogram is prepared in the following format:

```
                O XXXX                    subprogram number
                .......................
                .......................
                     .
                     .                    subprogram
                     .
                .......................
                M99
```

At the top of a subprogram a program number identifying the program is specified after "O". Specifying M99 at the end of a subprogram is optional. If the program was called by an M98, an M02, M30 or M00 will return. Subprograms are entered into memory the same as normal programs.

4.13.2 Subprogram execution

A subprogram is executed when called by the main program or another subprogram. A subprogram call has the following format:

```
     M98 PXXXX LXXX
         _____           =  subprogram number
               ____      =  number of times the subprogram
                            is to be repeated
```

Example: M98 P0002 L5

This command is read like this: call subprogram number 0002 five times.

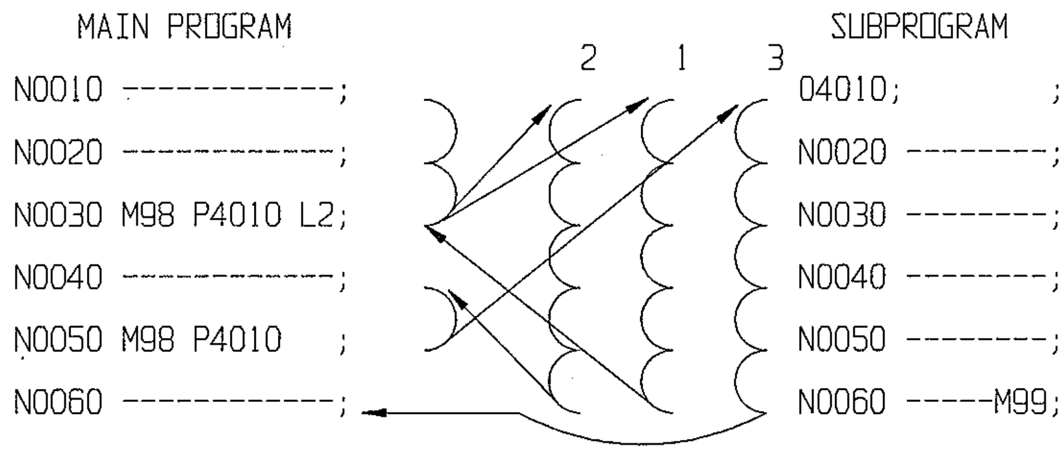
When the loop number is omitted the subprogram is run once.

A subprogram call command and move command can be specified in the same block.

Example: X1 M98 P0200

In this example the subprogram 0200 is called after completing movement in the X axis direction.

The execution sequence of a main program which calls a subprogram is as follows:



When the subprogram is called by another subprogram, it is executed in the same sequence as shown in the above example.

Note 1: M98 and M99 signals are not issued to the machine tool.

Note 2: If the subprogram number specified cannot be found, an error message is displayed.

Note 3: A subprogram call "M98___" cannot be executed from MDI. In this case write a short program to call the subprogram.

OXXXX
M98PXXX
M02 *Then execute it in the Run mode.*

4.14 Text command

The Centurion V has a lettering command which can be used to engrave serial numbers or other descriptions. The text cycles must be loaded by setting the Load Text Cycles parameter, MAIN-PARMS-CTRL. (0 = disable text cycle; 1 = enable text cycle.) The control must be rebooted after setting the parameter. The one-to-one letter size is 1" by 1" and is in a block letter font. All these letters can be scaled or rotated to achieve the desired size and orientation. The depth of cut is contained in P141, the Z plunge feedrate in P145, and the clearance plane in P140. A typical program would be as follows:

```
N1   P140=.1                      Clearance of .1
N2   P141=-.2                     Depth of cut .2 inches
N3   P145=5                       Plunge feedrate of 5 ipm
N4   G1 F10                       XY feedrate of 10 ipm
N5   S1000 M3                     Spindle on CW
N6   X-4 Y.5                      Position of first letter
N7   Text [Milltronics Mfg.]      Desired text
N8   M30                          End program
```

This program will write "Milltronics Mfg." at a depth of -.2" in 1" by 1" block letters starting at a position of X-4 Y.5. The same program written in conversational would be:

```
E1   Program Setup           ABS, ENGLISH
E2   Drill                   Z feedrate 5
                             CW Spindle 1000
                             Clearance .1
                             Z depth -.2
E3   End Drill Cycle
E4   Position X-4, Y.5
E5   Misc
     Type in line            Text [Milltronics Mfg.]
E6   End program
```

5. FRONT PANEL OPERATION

Diagram of Main Screen

```
Runtime:  000:00:00 | MAIN                                  | ACTIVE: 01111

  CURRENT
 X 00.0000
 Y 00.0000
 Z 00.0000

   TIMES AT 100%                 GRAPHICS AREA
 RUN:    000:00:00
 FEED:   000:00:00
 RAPD:   000:00:00
 DISTANCE:
  0000.0000

F1   | F2  | F3  | F4  | F5  | F6    | F7    | F8   | F9   | F10  | ESC
HOME | JOG | HDW | RUN | MDI | DISPL | PARMS | PROG | VERF | UTIL | ESC
```

The above figure represents the primary menu or the number one screen. All operations will start from this menu. The top of the screen will always display the runtime, screen history, and active program #. The screen history line will display the various levels or screens gone through to get to the present screen. Each time the ESC key is pushed the CRT will back down one level until the number one or main screen is reached. Graphic displays are always of the active program.

The remainder of this section will explain each function that can be executed from the front panel. The history line for each function is the sequence of keys to push to get to that function from the main menu. Highlighted keys on the CRT mean they are the currently selected mode or their functions are available for use on this screen. To graphic verify a program it must be the active program.

The remainder of this section will explain each function that can be executed from the front panel. The history line for each

function is the sequence of keys to push to get to that function from the main menu. If a button is blinking, that is the next button to push in a normal sequence of operation. Highlighted keys on the CRT mean they are the currently selected mode or their functions are available for use on this screen.

5.1 Front panel description

The Centurion V front panel has two 16-key keypads and 12 function keys. The keypads are used to enter the alphanumeric data requested by the CNC. The upper keypad is primarily used to enter alpha characters. To enter one of the shifted characters simply hit SHIFT, then the character. After the character has been entered the control automatically returns to the non-shifted character set. The SHIFT key also works in the same manner on the lower or numeric keypad. When data is entered spaces between commands are optional, but an ENTER must be pushed to terminate a line of data or to go on to the next function. The operation of the 12 function keys changes as different menus are displayed on the CRT. The following sections explain in detail each function key meaning.

The lower section of the panel is dedicated to manual machine cycles. On the far left of the panel is the pulse generator which, when turned in the handwheel mode, will cause the selected axis to move. Next to the handwheel are the manual feed controls for the machine's axes. The FEEDRATE override switch will modify the current machine feedrate by the indicated percentage. The FEEDHOLD button will cause axis motion to stop whenever it is illuminated. To restart axis motion, release FEEDHOLD and depress the CYCLE START button. The CYCLE START needs to be depressed anytime a machine command is to be executed. The CYCLE START will blink when it needs to be pushed.

The next section of the panel deals with the spindle and coolant controls. The SPINDLE override switch will modify the current spindle RPM by the selected percentage. If the machine is not equipped with a variable speed spindle option the override switch has no effect on the spindle. The spindle CW, CCW and STOP buttons will override the current control commands giving the operator full manual override capabilities. The active state of the spindle will be represented by which button is lit. The coolant buttons (MIST and FLOOD) work identically to the CW and CCW buttons. When they are illuminated that function is active; however, the coolant will not turn on until the spindle is started. The TOOL RESET button is only active during an M6 command. This button is a safety interlock which prevents the spindle from starting during a manual tool change. The button will start flashing when in a tool change and will need to be pushed after the tool change is completed before program operation can be resumed. The EMERGENCY STOP button, when pushed, will stop all machine actions instantly. Once the EM STOP button is pushed, the RESET button will flash indicating that it must be pushed before any machine functions can be restarted. The control is always in an EM STOP state after Power On. The following diagram shows the layout of a Centurion V front panel.

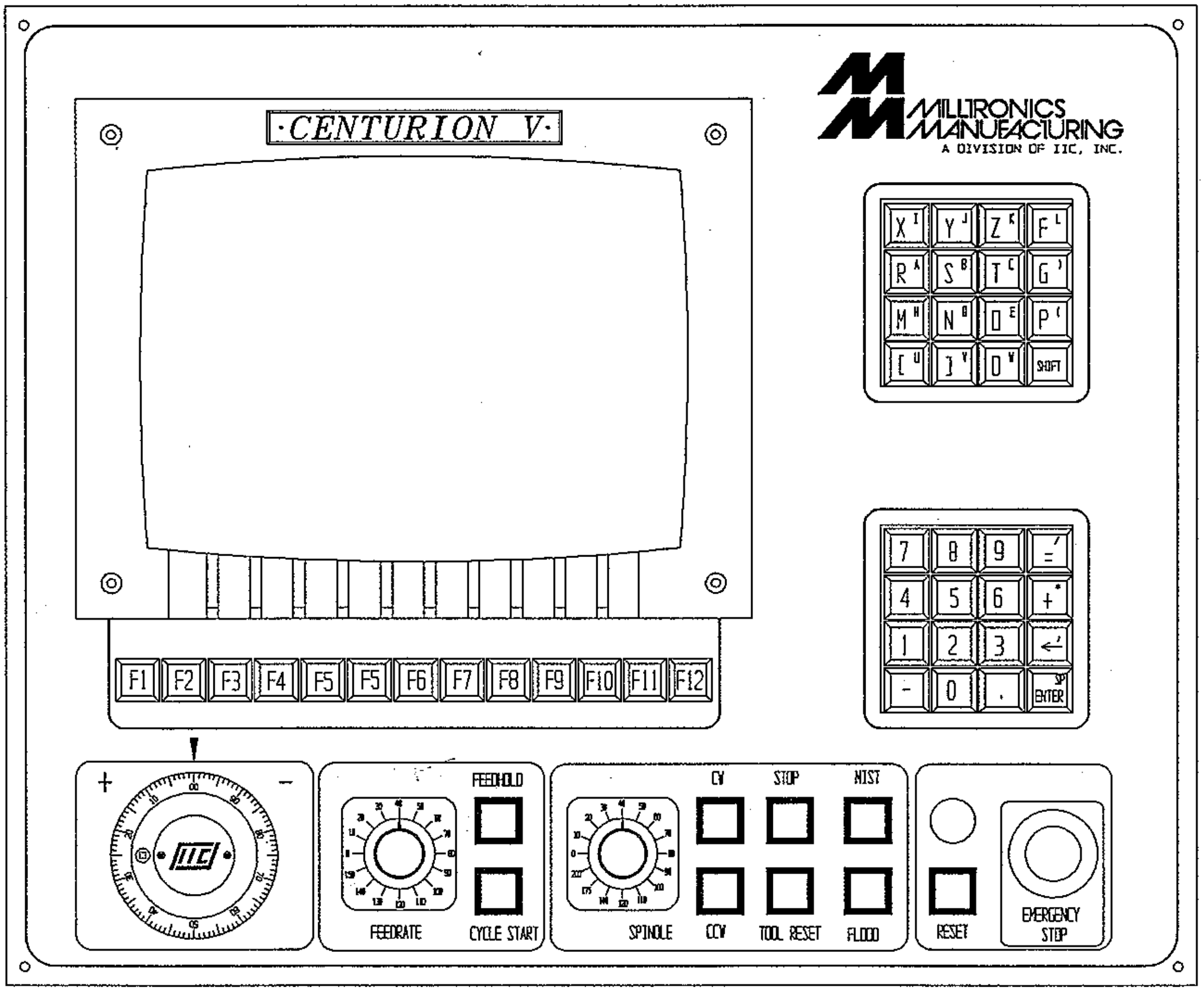


Figure 5
Centurion V Front Panel

5.2 Home sequence (F1 HOME) MAIN-HOME

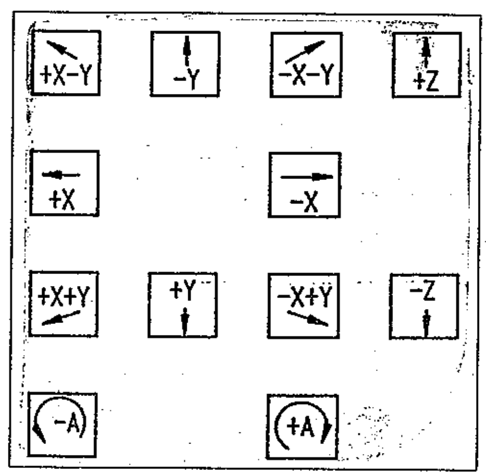
After a power off sequence the control will always have to be homed. Each axis will seek a home limit switch and a marker pulse on the encoder. After this procedure is finished the machine's reference position will be established and will be remembered until another power off. The sequencing of which axis homes first, the direction of home, and whether a limit switch and marker are used or just the marker pulse, can all be set in the axis parameters sections. To initiate a home sequence push ESC until the Main screen is reached; then push F1 "HOME". A message requesting that the Cycle Start button be pushed will appear on the screen and Cycle Start will start flashing. Pushing Cycle Start will start the home sequence and when it is finished the Main screen will return.

5.3 Jog (F2 JOG) MAIN-JOG

Jog is used to move the machine around in a manual mode to pick up zeroes and align parts. Upon pushing JOG the following screen appears:

Runtime: 000:00:00	MAIN-JOG	ACTIVE: O1000

CURRENT
X 00.0000
Y 00.0000
Z 00.0000



F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	ESC
SLOW	RAPID	← →	1	.1	.01	.001	.0001	FL2-X	FL2-Y	ESC

The F keys across the bottom of the screen are used to select the type of Jog desired. F1 selects slow jog which is about 20 ipm at 100% feedrate override. The feed override is active and can be used to speed up or slow down the jog speed. F2 selects rapid jog which is a feedrate of about 100 ipm at 100% override setting. F3 selects between continuous jog or incremental jog. In continuous jog the selected axis will continue to move until the axis key is released. In incremental jog the axis will move the selected increment F4 thru F8 and then stop each time the axis key is pushed and released. F9 and F10 will perform a G92 for X and Y at the current machine position. F11 is the ESC (escape) key and exits Jog, returning to the main menu. The keyboard diagram shows which direction an axis will move when the corresponding key is pushed on the numeric keyboard.

5.4 Handwheel (F3 HDW) MAIN-HDW

The handwheel mode is used to move the machine around using the manual pulse generator. It is used mainly to set tool length offsets, floating zeroes and part alignment. Upon pushing HDW the following screen will appear:

Runtime: 000:00:00	MAIN-HDW	ACTIVE: 01111

CURRENT
X 00.0000
Y 00.0000
Z 00.0000

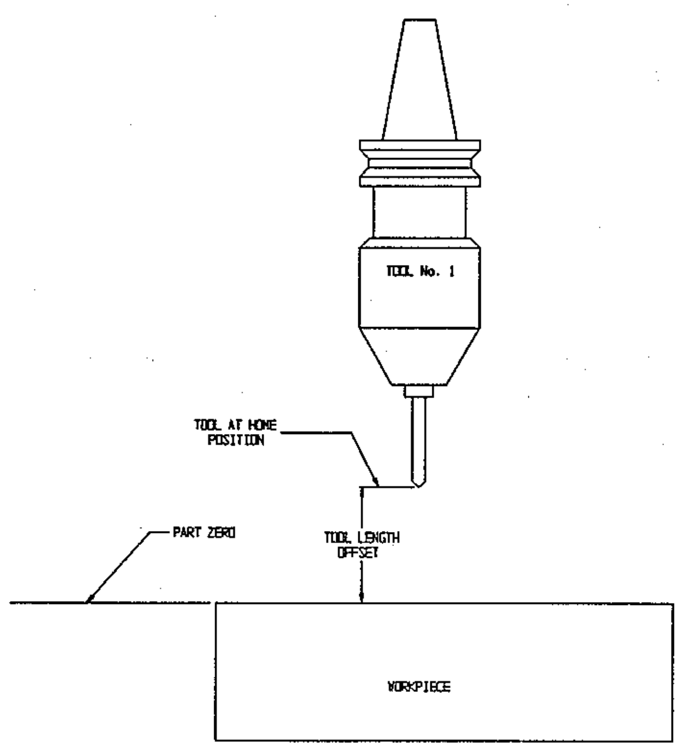
F1 X	F2 Y	F3 Z					F8 ZTOOL		F10 FLZ	ESC ESC

The F keys across the bottom of the screen are used to select which axis will move when the handwheel is turned. The feedrate override switch will determine the distance each axis will move for one click of the pulse generator. A feed override of 10% will cause the axis to move .0001 inch for each handwheel position. The active axis key will be highlighted when it is selected to move.

The F8 key ZTOOL is used to set a Z tool length offset into the tool table or H parameter table. In the handwheel mode a tool could be put into the spindle and moved to its Z zero point. The F8 key can then be pushed indicating that we wish to enter the current Z position as a tool length offset "H parameter". The CNC will prompt an offset number 1 thru 99 to be typed in. When the ENTER key is pushed the current Z position will be used for the H parameter offset.

PROCEDURE FOR SETTING TOOL LENGTH OFFSET

A tool length offset is used to compensate for the difference between Z axis home and part surface (part zero). Setting floating zero in Z axis is not recommended.



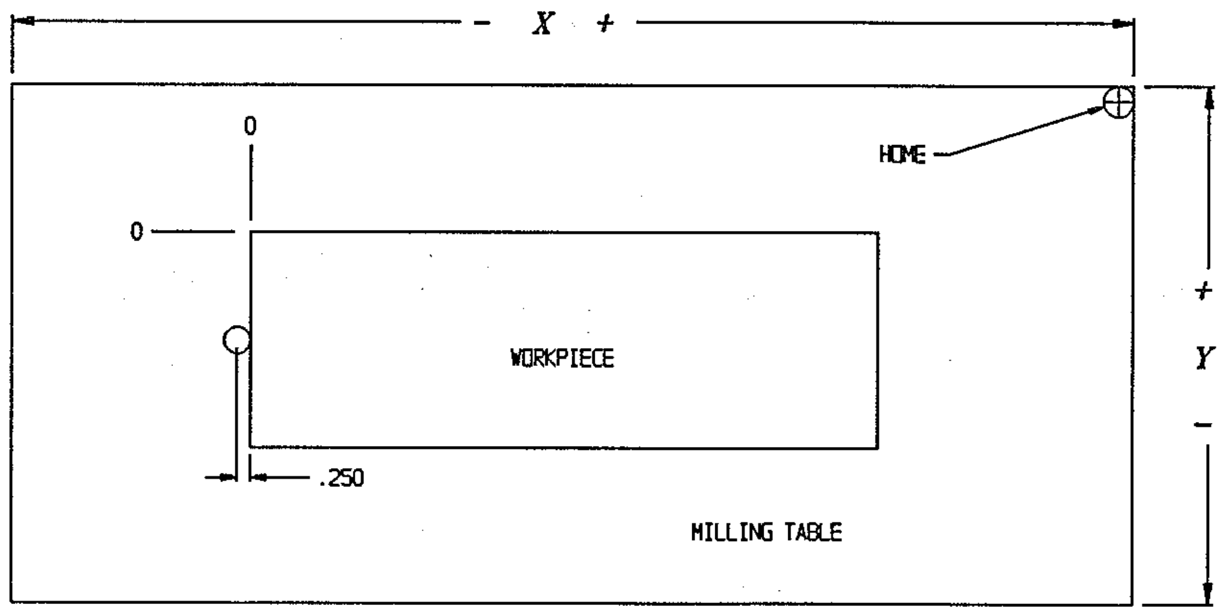
To set tool length offset, place tool #1 in spindle. Use handwheel or jog to touch the tool on the part at the desired Z zero (part zero). Select [Z-TOOL]. The control will prompt you to enter the tool number to be set: enter "1". A tool length offset for tool #1 has been set. Now when tool #1 is programmed to a position, it will position in reference to part zero. Repeat this procedure for each tool.

A tool length offset can also be set by entering a value into a tool offset register, The value can be measured by touching the part with the tool and reading the current position of the Z axis. If a shim is used between the tool and the part, the shim dimension should be added to the offset value. To enter the value select [PARAM] - [TOOL] - [EDIT] and use the arrow keys to move to the correct tool number. Then enter the value as a negative number. For more information on tool offsets, see Section 5.8.9.

The F10 key is used to set a floating zero on the selected axis at the current machine position. So X-FLZ, Y-FLZ, Z-FLZ would be the equivalent of doing a G92 X0 Y0 Z0.

PROCEDURE FOR SETTING A FLOATING ZERO

A floating zero shifts the X and Y axis zero positions to a desired place (edge of the part). Thus a part can be programmed from its part zero. To find and set a floating zero, refer to the example.



Using a 1/2" diameter edge finder in the X or Y axis, handwheel or jog to the edge of the part. Then depress X-FLZ (X axis) or Y-FLZ (Y axis) to set the floating zero for the appropriate axis. Next, to compensate for the edge finder's diameter, move Z up. Then move the X or Y axis back toward the part the distance of the edge finder radius and depress X-FLZ or Y-FLZ again.

Another way to set the floating zero is as follows:

Using a 1/2" diameter edge finder in the X axis, handwheel or jog to the edge of the part. Establish whether the edge finder is positive or negative from the desired zero. Now select [MDI] and type G92 (set floating zero) X -.25 (axis to set and distance from desired zero) [ENTER] [CYCLE START]. A floating zero for the X axis has been set. Current position will read X -.25. Repeat this procedure for the Y axis.

To check a floating zero, select [MDI], type G0 (rapid move) X0 Y0 (X and Y position to 0) [ENTER] [CYCLE START]. Machine will position to the current floating zero.

> *Caution: Machine will move in rapid mode. The tool should be above all parts, vices, etc.*

A floating zero can also be set through the soft keys while in handwheel or jog mode by selecting [FLT-Z]. This will set a floating zero for the axis selected at the current position.

For more information on floating zero, see Section 3.15.2.

5.5 Run F4 RUN (MAIN-RUN)

Run is used to execute the active program. Upon pushing the RUN button the following screen appears:

```
Runtime:  000:00:00 | MAIN-RUN                     | ACTIVE: O1111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                   Comp     :  Cancelled
                                   Tool     :  00
                                   Length   :  00.0000
                                   Radius   :  00.0000
                                   Plane    :  XY (system #1)
                                   Coords   :  Cartesian
                                   Interp   :  Linear (Feed)
                                   Feed     :  000.0 ipm
                                    (00%)   :  000.0 ipm
                                   Units    :  Abs/English
                                   Cycle    :  Cancelled

                                   Dwell    :  0000.00 sec
                                   Spindle  :  0000 rpm
                                    (00%)   :  0000 rpm (OFF)
                                   Coolant  :  Off
BLOCK
F1    |   | F3    | F4    | F5    | F6    | F7   | F8  |  |  | ESC
START |   | BLOCK | OSTOP | BSKIP | DISPL | MENU | DRY |  |  | ESC
```

After the above screen appears, F1 START would be pushed and the following screen would appear:

Runtime: 000:00:00	MAIN-RUN-START	ACTIVE: O1111

```
  CURRENT     NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000

Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 FIRST	F2 BLOCK	F3 TOOL								ESC ESC

The F1 key FIRST is automatically selected when entering this screen from the RUN screen. Therefore if it is desired to run the active program from the beginning, all that is necessary is to push the CYCLE START (Alt F6) button. If F2 BLOCK is pushed, the control will request that the desired block or sequence number be typed in, followed by ENTER. If the Cycle Start button is pushed the active program will start running from the selected block number. If F3 TOOL is pushed, the control will request a tool number. After typing the tool number followed by an ENTER, the CYCLE START (Alt F6) button is pushed, and the active program will start running at the desired tool number and the following screen will appear.

```
Runtime:  000:00:00 | MAIN-RUN                         | ACTIVE: 00000

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                  Comp     :  Cancelled
                                  Tool     :  00
                                  Length   :  00.0000
                                  Radius   :  00.0000
                                  Plane    :  XY (system #1)
                                  Coords   :  Cartesian
                                  Interp   :  Linear (Feed)
                                  Feed     :  000.0 ipm
                                   (00%)   :  000.0 ipm
                                  Units    :  Abs/English
                                  Cycle    :  Cancelled

                                  Dwell    :  0000.00 sec
                                  Spindle  :  0000 rpm
                                   (00%)   :  0000 rpm (OFF)
                                  Coolant  :  Off
BLOCK

      | F2    | F3    | F4    | F5    | F6    |       | F8    | F9    |       | ESC
      | RESUM | BLOCK | OSTOP | BSKIP | DISPL |       | DRY   | HALT  |       | ESC
```

This screen is the basic run screen with two new additions, a block number display and F9-HALT. The block number shows the current line being executed as the program runs. The HALT key is similar to FEEDHOLD in that when pushed the machine will stop. However, unlike FEEDHOLD the HALT also exits the "Run" mode and allows a new program to be started.

5.5.1 F2 RESUM (MAIN-RUN)

Once a program has been "Halted" the "Resume" feature of the control becomes active. The F2 RESUM key will now be displayed on the Run screen. A program can be Resumed as long as one of the following functions is not performed: "Verf", MDI, Home, or EM STOP. If none of the above functions are done the axis can be "Jogged" or Handwheeled away from the work, the spindle turned on or off, or any other function performed, and the "Resume" remains active. As long as the Resume is active the F2 key on the Run screen will show a "Resume" function. If the "Resume" function is selected the active program will be resumed at the halted point. The first thing that will happen when a "Resume" "Cycle Start" is executed is Z will retract to the tool change position, all the way up. Next, X and Y will rapid to the halted point. Once X and Y are in position a "Cycle Start" will be requested. When CYCLE START is pushed the Z axis will rapid to the R plane and then feed to its previous depth. The program will then start running as if nothing had ever happened. FEEDHOLD simply stops axes motion until it is released and CYCLE START is pushed.

Note: *Once the "Run" or "Verify" modes have been entered, a "Halt" must be executed if the mode is to be exited prior to program completion.*

The F3, F4, F5 and F8 keys on the RUN screen set the mode of operation a program will run in. When these keys are in a highlighted state the functions will be active in any currently running program or program to be run.

5.5.2 F3 BLOCK (MAIN-RUN)

When the BLOCK switch is activated the program will stop at the end of each block. Each time the CYCLE START (Alt F6) button is pushed one more block will be run.

5.5.3 F4 OSTOP (MAIN-RUN)

When the optional stop switch (OSTOP) is activated the program will stop at each M01 command. Each time the CYCLE START (Alt F6) button is pushed the program will run to the next M01 or the end of program.

5.5.4 F5 BSKIP (MAIN-RUN)

When the block skip switch (BSKIP) is activated the program will skip all blocks started with a "/".

/M5 When the block skip switch is active the spindle off command will not be executed.

5.5.5 F6 DISPL (MAIN-RUN-DISPL)

The F6 Display key (DISPL) can be accessed from a number of screens. The following screen is shown as though the DISPL was entered from the Main screen. All the display functions and screens are identical, independent of the entry point. Only the return point differs based on the original entry point. When the DISPL key is pushed the following screen will appear:

```
Runtime:  000:00:00 | MAIN-RUN-DISPL | ACTIVE: O1111

 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

                         Comp     :  Cancelled
                         Tool     :  00
                         Length   :  00.0000
                         Radius   :  00.0000
                         Plane    :  XY (system #1)
                         Coords   :  Cartesian
                         Interp   :  Linear (Feed)
                         Feed     :  000.0 ipm
                          (00%)   :  000.0 ipm
                         Units    :  Abs/English
                         Cycle    :  Cancelled

                         Dwell    :  0000.00 sec
                         Spindle  :  0000 rpm
                          (00%)   :  0000 rpm (OFF)
                         Coolant  :  Off
```

F1 NEXT	F2 DIST	F3 GRAPH								ESC ESC

5.5.5.1 F1 NEXT (MAIN-RUN-DISPL)

When the NEXT key is activated the position display on the run screen displays the current position and the next position.

```
 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

5.5.5.2 F2 DIST (MAIN-RUN-DISPL)

When the distance key (DIST) is activated the position display on the RUN screen displays the current position and the distance to go.

```
 CURRENT    DIST
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

5.5.5.3 F3 GRAPH (MAIN-RUN-DISPL-GRAPH)

If the GRAPH key is activated the control switches from displaying text to a graphic display of the active part program. The following screen will appear.

Runtime: 000:00:00	MAIN-VERF-DISPL-GRAPH	ACTIVE: O1111

CURRENT X 00.0000 Y 00.0000 Z 00.0000 TIMES AT 100% RUN: 000:00:00 FEED: 000:00:00 RAPD: 000:00:00 DISTANCE: 0000.0000	GRAPHICS AREA

F1 ROT	F2 PAN	F3 WIND	F4 AUTO	F5 ZOOM-	F6 ZOOM+	F7 LIMIT	F8 ZONE	F9 FRESH	F10 CLEAR	ESC ESC

The graphics on this control are full 3D and will be displayed in the Graphics Area as long as the control remains in the GRAPH mode. When other displays are requested windows will appear in the graphics area showing the requested data. When these functions are finished the windows will disappear and the graphic display will be reinstated. The scale at the top of the screen is to be used as a reference for the part size. As the screen scale is changed the graduations on the ruler will change accordingly. The ruler graduations are in machine units but on an English system the largest graduations equal approximately one inch. The times at 100% and distance displays are normally used during graphic verify but are also valid during normal running. The Run, Feed and Rapid times are intended to be used during a Graphic Verify to estimate the time a part will take to run. All calculations are based on the programmed feedrate and a 100%

feedrate override setting. The distance display gives the total inches the machine has travelled during the program. This display is intended to help in estimating tool wear. The Runtime at the top of the screen is basically a stop watch which starts when a Run Program command is executed and stops at the end of program or when the program is aborted. The two runtimes can be compared at the end of a program to determine how much time the machine sat idle (tool changes, block stops, feedrate less than 100%, etc.) versus actual cutting time.

The next section will explain how to manipulate the part displayed in the graphics area. All the following functions are accessible through the DISPL-GRAPH screen.

5.5.5.3.1 F1 ROT (MAIN-RUN-DISPL-GRAPH-ROT)

When the Display Rotate function (F1 ROT) is selected the following screen is displayed.

Runtime: 000:00:00	MAIN-RUN-DISPL-GRAPH-ROT	ACTIVE: 01111

CURRENT
X 00.0000
Y 00.0000
Z 00.0000

TIMES AT 100%
RUN: 000:00:00
FEED: 000:00:00
RAPD: 000:00:00
DISTANCE:
0000.0000

Orientation

Y
Z X

F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	ESC
XY	XZ	YZ	ISO	X+	X-	Y+	Y-	Z+	Z-	ESC

(F1 -XY, F2 -XZ, F3 -Y2, F4 -ISO)

The F1, F2, F3 and F4 keys give the four standard rotations of a part: XY plane, XZ plane, YZ plane and isometric views. The orientation index in the upper left corner of the screen shows the current part orientation and rotates to show what the new orientation will be. Depressing the F1 thru F4 keys moves the orientation index to its new position, and then pushing the ESC key will cause the part display to rotate to its new position and the previous menus to reappear. The F5 thru F10 keys are used to infinitely rotate any of the selected axes. Again, as the key is pushed the orientation index rotates indicating the orientation of the part display; when ESC is pushed the part will rotate to its new orientation. F5 and F6 rotate X axis ±, F7 and F8 rotate Y axis ±, and F9 and F10 rotate Z axis ±.

5.5.5.3.2 F2 PAN (MAIN-RUN-DISPL-GRAPH-PAN)

The F2 PAN key selects the Pan function (PAN) which allows the operator to pan around a part. The following display will appear.

Runtime: 000:00:00	MAIN-RUN-DISPL-GRAPH-PAN	ACTIVE: 01111

CURRENT
X 00.0000
Y 00.0000
Z 00.0000

TIMES AT 100%
RUN: 000:00:00
FEED: 000:00:00
RAPD: 000:00:00
DISTANCE:
0000.0000

				F5 ENTER		F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

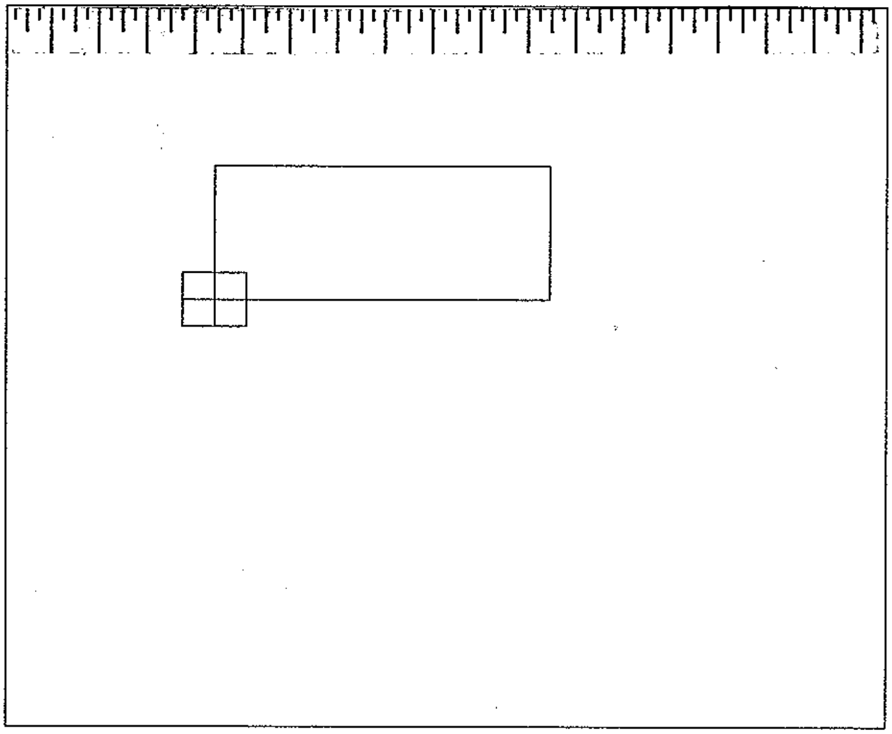
The square cursor which appears on the screen can be moved around using the arrow keys F7 thru F10. To pan simply move the cursor to the point on the display which is desired to be at the center of the screen and push F5 ENTER or the ENTER key on the keyboard. The display will shift to its new position and panning can be started again. The ESC key will terminate the Pan function and drop back to the Graph screen.

5.5.5.3.3 F3 WIND (MAIN-RUN-DISPL-GRAPH-WIND)

The F3 WIND key selects the window function which allows the operator to window in on a particular area of the part. The following display will appear when Window is selected.

Runtime: 000:00:00	MAIN-RUN-DISPL-GRAPH-WIND	ACTIVE: 01111

```
  CURRENT
 X 00.0000
 Y 00.0000
 Z 00.0000

   TIMES AT 100%
 RUN:    000:00:00
 FEED:   000:00:00
 RAPD:   000:00:00
 DISTANCE:
  0000.0000
```



				F5 ENTER		F7 ↑	F8 ↓	F9 ←	F10 →	ESC ESC

The square cursor which appears on the screen can be moved around using the arrow keys F7 thru F10 (the same as Pan). To window, move the cursor to the first corner of the window area and hit F5 ENTER or the ENTER key. Then move the cursor around until the desired area of the part to be viewed is enclosed in the rectangular box being drawn on the screen and hit ENTER. The area enclosed in the box will now be displayed on the entire screen.

5.5.5.3.4 F4 AUTO (MAIN-RUN-DISPL-GRAPH-AUTO)

The F4 key selects the auto zoom function. This function automatically scales and centers any part on the screen. Normally an Auto has to be done after a part is rotated to get it back to the center of the screen.

5.5.5.3.5 F5 ZOOM- (MAIN-RUN-DISPL-GRAPH)

The F5 key selects zoom- which decreases the size of the part currently being displayed on the screen. Generally this function is used to view a larger portion of the part.

5.5.5.3.6 F6 ZOOM+ (MAIN-RUN-DISPL-GRAPH)

The F6 key selects the zoom+ function. This function increases the size of the part being displayed on the screen. Generally this function is used to enlarge a specific area of a part enabling one to see greater detail.

5.5.5.3.7 F7 LIMIT (MAIN-RUN-DISPL-GRAPH)

The F7 key draws a box on the screen which corresponds to the axis limits of the machine. This allows viewing of the part in relation to the machine's overtravels. If the part extends beyond this box it cannot be run on the machine unless some corrective action is taken to reduce the size of the machined areas of the part. The axis overtravel limits are set from the parameter screens. If the tool is programmed outside this box an overtravel error will result.

5.5.5.3.8 F8 ZONE (MAIN-RUN-DISPL-GRAPH)

The F8 key draws a box on the screen which corresponds to an axis safe zone. This is a zone which the tool cannot enter. If the tool is programmed into this box an "attempted to move into safe zone" error will be generated.

5.5.5.3.9 F9 FRESH (MAIN-RUN-DISPL-GRAPH)

The F9 key redraws the currently displayed part on the screen.

5.5.5.3.10 F10 CLEAR (MAIN-RUN-DISPL-GRAPH)

The F10 key clears the current display buffer. After the clear screen command nothing will be displayed until either the program is Verified or Run again. It is generally used to clear MDI moves from the graphic display before running or verifying a program.

5.5.5.4 (F4 DIAG) MAIN-RUN-DISPL-DIAG

The F4 key is mainly used for machine setup or troubleshooting machine functions. The diagnostic screens bring up all the external I/O bits connect-ed to the CNC. The status of each bit is continu-ously displayed on the screen and as they change on the machine the screen will be updated. Function keys F1 thru F5 display the various axes enabled on the CNC. Selecting X, Y, Z, etc. brings up the appropriate axis I/O channels. A "0" displayed on the screen means the I/O channel is not active. A "1" indicates active.

The following screens represent the displayed information for the various axis selections.

Runtime: 000:00:00	MAIN-RUN-DISPL-DIAG	ACTIVE: O1111

```
  X-axis Input          X-axis Output

 Estopped      0      Force Estop     0
 CW Spindle    1      Mist Coolant    0
 CCW Spindle   1      Flood Coolant   0
 Up To Speed   0      Spindle CW      0
 Tool Change   1      Spindle CCW     0
 Lube Fault    0      Spindle Stop    1
 Wait Channel  1      Spindle Allow   0
 X Input 08    0      Tool Change     0
 X Input 09    0      Allow Reset     0
 X Input 10    0      X Output 10     0
 X Input 11    0      X Output 11     0
 Home Switch   1      X Output 12     0

 Marker Pulse  0
```

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 X	F2 Y	F3 Z								ESC ESC

Runtime: 000:00:00	MAIN-RUN-DISPL-DIAG	ACTIVE: O1111

Y-axis Input / Y-axis Output

Y-axis Input		Y-axis Output	
Y Input 01	1	Y Output 01	1
Y Input 02	1	Y Output 02	1
Y Input 03	1	Y Output 03	1
Y Input 04	1	Y Output 04	1
Y Input 05	1	Y Output 05	1
Y Input 06	1	Y Output 06	1
Y Input 07	1	Y Output 07	1
Y Input 08	1	Y Output 08	1
Y Input 09	1	Y Output 09	1
Y Input 10	1	Y Output 10	1
Y Input 11	1	Y Output 11	1
Home Switch	1	Y Output 12	1

Marker Pulse 0

Comp	:	Cancelled
Tool	:	00
Length	:	00.0000
Radius	:	00.0000
Plane	:	XY (system #1)
Coords	:	Cartesian
Interp	:	Linear (Feed)
Feed	:	000.0 ipm
(00%)	:	000.0 ipm
Units	:	Abs/English
Cycle	:	Cancelled
Dwell	:	0000.00 sec
Spindle	:	0000 rpm
(00%)	:	0000 rpm (OFF)
Coolant	:	Off

F1 X	F2 Y	F3 Z								ESC ESC

Runtime: 000:00:00	MAIN-RUN-DISPL-DIAG	ACTIVE: O1111

```
Z-axis Input          Z-axis Output

Z Input    01   1     Z Output   01   1
Z Input    02   1     Z Output   02   1
Z Input    03   1     Z Output   03   1
Z Input    04   1     Z Output   04   1
Z Input    05   1     Z Output   05   1
Z Input    06   1     Z Output   06   1
Z Input    07   1     Z Output   07   1
Z Input    08   1     Z Output   08   1
Z Input    09   1     Z Output   09   1
Z Input    10   1     Z Output   10   1
Z Input    11   1     Z Output   11   1
Home Switch     1     Z Output   12   1

Marker Pulse 0
```

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 X	F2 Y	F3 Z								ESC ESC

5.5.5.5 F5 ERROR (MAIN-RUN-DISPL-ERROR)

The F5 key changes the position display to show each axis following error instead of axis position. The F5 function key will only be active if an access code greater than 0 has been entered into the control. The following error display is intended to help in machine setup or trouble-shooting an axis problem. The following error is displayed in .0001 inch increments.

5.5.6 F7 MENU (MAIN-RUN-MENU)

The F7 key selected from the Run or Verify screen brings up a window containing a listing of all the available programs which can be run. The F7 Menu option is also available from the Program screen (MAIN-PROG-CONV-MENU) as well as the Utilities screen (MAIN-UTIL-FILES-MENU). These programs are currently on the hard disk or in RAM memory. The next screen will be displayed when the program menu is requested.

```
Runtime:  000:00:00   MAIN-RUN-MENU                          ACTIVE: O1111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

          Program Menu Window                 Comp     :  Cancelled
                                              Tool     :  00
  O0001 (Conversational File)    0000657      Length   :  00.0000
  O0002 (Test               )    0000270      Radius   :  00.0000
  O0003 (Sample             )    0000673      Plane    :  XY (system #1)
    .                               .         Coords   :  Cartesian
    .                               .         Interp   :  Linear (Feed)
    .                               .         Feed     :  000.0 ipm
                                               (00%)   :  000.0 ipm
                                              Units    :  Abs/English
                                              Cycle    :  Cancelled

                                              Dwell    :  0000.00 sec
                                              Spindle  :  0000 rpm
                                               (00%)   :  0000 rpm (OFF)
                                              Coolant  :  Off

|    |    |    |    | F5    |    | F7 | F8 | F9   | F10  | ESC |
|    |    |    |    | ENTER |    | ↑  | ↓  | PGUP | PGDN | ESC |
```

To activate one of the programs listed in the window, simply use the arrow and page keys to move the cursor to the desired program and push F5 ENTER or the ENTER on the keyboard. The Menu function can be called from other screens but works the same way from all. When called from the Verify screen the selected program becomes the active program. When called from an edit screen the selected program becomes the current program being edited.

5.5.7 F8 DRY (MAIN-RUN-DRY)

When the dry run switch is active all program feedrates will run at the dry run feedrate.

5.6 Manual data input F5 MDI (MAIN-MDI)

The F5 key on the Main menu selects the MDI (manual data input) function. Through MDI any programmable machine function can be executed one function or one block at a time. When MDI is selected the following screen appears.

```
Runtime:  000:00:00 | MAIN-MDI                          | ACTIVE: O1111

  CURRENT    NEXT                              MESSAGE AREA
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

                                          Comp     :  Cancelled
                                          Tool     :  00
                                          Length   :  00.0000
                                          Radius   :  00.0000
                                          Plane    :  XY (system #1)
                                          Coords   :  Cartesian
                                          Interp   :  Linear (Feed)
                                          Feed     :  000.0 ipm
                                           (00%)   :  000.0 ipm
                                          Units    :  Abs/English
                                          Cycle    :  Cancelled

                                          Dwell    :  0000.00 sec
                                          Spindle  :  0000 rpm
                                           (00%)   :  0000 rpm (OFF)
MDI_________________________________      Coolant  :  Off

|    |    |    |    | F5    |    |    |    | F9   |    | ESC |
|    |    |    |    | DISPL |    |    |    | HALT |    | ESC |
```

As the functions are typed in they will appear on the MDI line at the bottom of the screen. After the data has been typed in the ENTER key must be pushed to end the block. At this point a Cycle Start will execute the MDI line. When MDI is selected any active program will be halted. During MDI two functions remain active: F5 DISPLAY and F9 HALT. The F9 key will terminate any MDI move and F5 will bring up the normal graphic displays discussed earlier. If graphics are turned on during MDI the graphic display will graph all the MDI moves as they are executed.

5.7 Display F6 DISPL (MAIN-DISPL)

See explanation under MAIN-RUN-DISPL. This function can be entered from either screen.

5.8 Parameters F7 PARMS (MAIN-PARMS)

The F7 key from the main screen brings up this parameter screen.

Runtime: 000:00:00	MAIN-PARMS	ACTIVE: 01111

```
 CURRENT     NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	ESC
SETUP	COORD	TOOL	D OFF	H OFF	SAVE	LOAD	PROG	CTRL	USER	ESC

5.8.1 F1 SETUP (MAIN-PARMS-SETUP)

The F1 selection brings up the parameters which make the control unique to a particular machine or application. When F1 SETUP is selected the following screen appears.

```
Runtime:  000:00:00 | MAIN-PARMS-SETUP-LEVEL | ACTIVE: O1111

  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

        Validation Code          Comp     :  Cancelled
                                 Tool     :  00
         Access Level            Length   :  00.0000
                                 Radius   :  00.0000
                                 Plane    :  XY (system #1)
                                 Coords   :  Cartesian
                                 Interp   :  Linear (Feed)
                                 Feed     :  000.0 ipm
                                  (00%)   :  000.0 ipm
                                 Units    :  Abs/English
                                 Cycle    :  Cancelled

                                 Dwell    :  0000.00 sec
                                 Spindle  :  0000 rpm
                                  (00%)   :  0000 rpm (OFF)
                                 Coolant  :  Off

F1    |  |  |  |  |  |  |  |  |  | ESC
LEVEL |  |  |  |  |  |  |  |  |  | ESC
```

The CNC requires a Validation Code and an Access Level number to allow the machine setup parameters to be displayed or changed. The validation code and access levels are supplied by the machine tool builder and should be part of the system parameter setup sheet. Assuming the proper codes have been entered, the following screen will appear.

Runtime: 000:00:00	MAIN-PARMS-SETUP-LEVEL	ACTIVE: 00000

	CURRENT	NEXT
X	00.0000	00.0000
Y	00.0000	00.0000
Z	00.0000	00.0000

MESSAGE AREA

Comp	:	Cancelled
Tool	:	00
Length	:	00.0000
Radius	:	00.0000
Plane	:	XY (system #1)
Coords	:	Cartesian
Interp	:	Linear (Feed)
Feed	:	000.0 ipm
(00%)	:	000.0 ipm
Units	:	Abs/English
Cycle	:	Cancelled
Dwell	:	0000.00 sec
Spindle	:	0000 rpm
(00%)	:	0000 rpm (OFF)
Coolant	:	Off

F1 LEVEL	F2 PREC	F3 MACH	F4 AXIS	F5 MISC						ESC ESC

5.8.2 F2 PREC (MAIN-PARMS-SETUP-PREC)

If the F2 selection for Machine Precision is made, the following screen will be displayed.

Runtime: 000:00:00	MAIN-PARMS-SETUP-PREC	ACTIVE: O1111

```
 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

Decimal Precision

	English		Metric	
	Lead	Trail	Lead	Trail
Cartesian	2	4	3	3
Angular	3	3	3	3
Spindle	4	0	4	0
Feed	3	1	4	0

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 EDIT										ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

The above screen shows some typical settings for leading and trailing zeroes for the different coordinate systems. The number of leading and trailing zeroes is unlimited, but some practical limits do exist. If the numbers get too large they will not fit on the screen in their allotted space, and if they are smaller than the feedback units they will not cause movement. To change a parameter push F1 EDIT. A series of arrow keys will be displayed; simply use them to move the cursor to the desired parameter and type in the new number. Once all the numbers have been edited, pushing the EXIT key will validate the new numbers and return to the previous screen. These parameters are for all numbers entered into the control except axes. The axis parameters are set separately in the "Axis" parameters.

5.8.3 F3 MACH (MAIN-PARMS-SETUP-MACH)

5.8.3.1 F3 POWON (MAIN-PARMS-SETUP-MACH)

Machine parameters are parameters which directly relate to the configuration of the machine tool and will normally be set by the machine tool builder.

The F3 key brings up the power-on defaults as shown in the following screen.

```
Runtime:  000:00:00 | MAIN-PARMS-SETUP-MACH | ACTIVE: 01111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

            Power-On Defaults                 Comp     :  Cancelled
                                              Tool     :  00
 Machine Unit (E/M)      -  English           Length   :  00.0000
 Number of Axes          -  3                 Radius   :  00.0000
 Feed Unit (M/R)         -  Inches/minute     Plane    :  XY (system #1)
 Power-On Feedrate       -  20                Coords   :  Cartesian
 Spindle Unit (R/F)      -  Revolution/min.   Interp   :  Linear (Feed)
 Spindle Axis            -  3                 Feed     :  000.0 ipm
 Tool Change (A/M)       -  Manual             (00%)   :  000.0 ipm
 100% Rapid/Run (Y/N)    -  No                Units    :  Abs/English
 100% Rapid/Dry (Y/N)    -  No                Cycle    :  Cancelled
 Spindle On Dry (Y/N)    -  No
                                              Dwell    :  0000.00 sec
                                              Spindle  :  0000 rpm
                                               (00%)   :  0000 rpm (OFF)
                                              Coolant  :  Off
```

F1 EDIT		F3 POWON	F4 FDOVR	F5 HWOVR	F6 SPOVR					ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

Machine Units	-	Can be either English or Metric and depend on the feedback or screw type
Number of Axes	-	Can be 1 to 7
Feed Unit	-	Can be inch/mm/minute or revolutions/minute
Power On Feedrate	-	Can be any number up to the axis rapid rate
Spindle Unit	-	Always revolutions/minute
Spindle Axis	-	Can be any number 1 to 7 1=X, 2=Y, 3=Z, 4=A ... etc.
Tool Change	-	Either auto or manual. Manual tool changes require a CYCLE START.
100% Rapid/Run	-	**No** means the feedrate override will affect rapid moves in Run mode.
		Yes means the feedrate override will not affect rapid moves in Run mode.
100% Rapid/Dry	-	**No** means the feedrate override will affect rapid moves in Dry Run mode.
		Yes means the feedrate override will not affect rapid moves in Dry Run mode.
Spindle On Dry	-	**No** means the spindle will not come on in Dry Run mode.
	-	**Yes** means the spindle will come on in Dry Run mode.

To change a parameter push F1 EDIT. A series of arrow keys will be displayed; simply use them to move the cursor to the desired parameter and type in the new number.

5.8.3.2 F4 FDOVR (MAIN-PARMS-SETUP-MACH)

The F4 FDOVR key brings up the feedrate override parameter settings. These settings determine which percentage will be used for each of the 16 feedrate override switch positions. The following screen is displayed when F4 FDOVR is selected. To edit the parameters select EDIT and move the cursor to the value to be changed. Type in the desired change and hit EXIT.

Runtime: 000:00:00	MAIN-PARMS-SETUP-MACH	ACTIVE: 01111

```
  CURRENT    NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000
```

Feedrate Override Settings

1-000	5-040	9-080	13-120
2-010	6-050	10-090	14-130
3-020	7-060	11-100	15-140
4-030	8-070	12-110	16-150

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 EDIT		F3 POWON	F4 FDOVR	F5 HWOVR	F6 SPOVR					ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

5.8.3.3 F5 HWOVR (MAIN-PARMS-SETUP-MACH)

The F5 key brings up the handwheel switch settings for the feedrate override switch. These settings determine how far an axis will move for one increment of the handwheel (001=.0001 inches). Editing is performed the same way as the feedrate override parameters. The following screen displays the Handwheel Override Settings.

Runtime: 000:00:00	MAIN-PARMS-SETUP-MACH	ACTIVE: O1111

```
 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

Handwheel Override Settings

1-000	5-015	9-040	13-080
2-001	6-020	10-050	14-090
3-005	7-025	11-060	15-100
4-010	8-030	12-070	16-150

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 EDIT		F3 POWON	F4 FDOVR	F5 HWOVR	F6 SPOVR					ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

5.8.3.4 F6 SPOVR (MAIN-PARMS-SETUP-MACH)

The F6 key brings up the 16 spindle override switch settings. These settings are the percentages a spindle command will be overridden at each switch position. The spindle override parameters are changed the same as the feedrate override parameters. The spindle override screen is displayed below.

Runtime: 000:00:00	MAIN-PARMS-SETUP-MACH	ACTIVE: O1111

```
  CURRENT    NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000
```

Spindle Override Settings

1-000	5-040	9-080	13-120
2-010	6-050	10-090	14-130
3-020	7-060	11-100	15-175
4-030	8-070	12-110	16-200

```
Comp      :  Cancelled
Tool      :  00
Length    :  00.0000
Radius    :  00.0000
Plane     :  XY (system #1)
Coords    :  Cartesian
Interp    :  Linear (Feed)
Feed      :  000.0 ipm
 (00%)    :  000.0 ipm
Units     :  Abs/English
Cycle     :  Cancelled

Dwell     :  0000.00 sec
Spindle   :  0000 rpm
 (00%)    :  0000 rpm (OFF)
Coolant   :  Off
```

F1 EDIT		F3 POWON	F4 FDOVR	F5 HWOVR	F6 SPOVR					ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

5.8.4 F4 AXIS (MAIN-PARMS-SETUP-AXIS)

If the F4 selection, AXIS, is pushed the following screen will be displayed.

```
Runtime:  000:00:00 | MAIN-PARMS-SETUP-AXIS                | ACTIVE: O1111

   CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

    Axis Address Label          ***        Comp     :  Cancelled
    Pulses Per Unit             ***        Tool     :  00
    Home Position               ***        Length   :  00.0000
   >Home Direction              ***        Radius   :  00.0000
    Positive Limit              ***        Plane    :  XY (system #1)
    Negative Limit              ***        Coords   :  Cartesian
   Home Direction                          Interp   :  Linear (Feed)
     X  00.0000                            Feed     :  000.0 ipm
     Y  00.0000                             (00%)   :  000.0 ipm
     Z  00.0000                            Units    :  Abs/English
                                           Cycle    :  Cancelled

                                           Dwell    :  0000.00 sec
                                           Spindle  :  0000 rpm
                                            (00%)   :  0000 rpm (OFF)
                                           Coolant  :  Off
```

F1 EDIT						F7 ↑	F8 ↓	F9 PGUP	F10 PGDN	ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓			ESC EXIT

The PGUP, PGDN and arrow keys move through the tables on the upper display. The lower display changes to display the data associated with the cursor position of the upper display. To edit the values in these tables the cursor is positioned via the arrow keys to the desired parameter in the upper screen. When the EDIT key is pushed the cursor will move to the lower screen. At this point again move the cursor using the arrow keys to the axis or parameter desired and type in the new values using the keypads. After the new values have been entered, push the EXIT key. The new value will be entered and the cursor will go back to the upper screen and the selection process can be started over. The following is a list of all the selectable parameters displayed in this mode and a description of their functions.

Axis Address Label X 88.0000 Y 89.0000 Z 90.0000	ASCII code assigned to each axis
Pulses Per Unit X 10000.0000 Y 10000.0000 Z 10000.0000	The number of pulses the feedback gives per unit of travel. English 1 = 10000 pulses Metric 1cm = 1000 pulses
Home Position X 00.0000 Y 00.0000 Z 00.0000	The dimension assigned to the machine zero or home position.
Home Direction X 00.0000 Y 00.0000 Z 00.0000	Defines the direction of rotation of the motor when a home is commanded. CW = 00.0000 CCW = 01.0000
Positive Limit X 00.0000 Y 00.0000 Z 00.0000	Dimension from machine zero where the positive software limit occurs
Negative Limit X -31.0000 Y -18.0000 Z - 6.5000	Dimension from machine zero where the negative software limit occurs
Maximum Feed X 100.0000 Y 100.0000 Z 100.0000	Sets the maximum G01 feedrate in inches per minute
Dry Run Feed X +75.0000 Y 75.0000 Z 75.0000	Sets the Dry Run feedrate in inches per minute
Rapid Velocity X 200.0000 Y 200.0000 Z 200.0000	Set the maximum G00 feedrate in inches per minute

Rapid Acc/Dec	40.000 The Acc/Dec constant is a number between 1 and 200 that determines the rate at which the axis velocity is stepped up. The smaller the number the longer the Acc/Dec times will be. Acceleration and deceleration in this control are linear ramps from one speed to another.
Home Sequence X 02.0000 Y 02.0000 Z 01.0000	The Home Sequence numbers determine the order the axes will home in: #1 first, #2 next, etc. Axes with the same number home together.
Velocity Toward Home X 60.0000 Y 60.0000 Z 60.0000	Sets the feedrate in IPM at which an axis seeks the home limit switch
Velocity Away From Home X 12.0000 Y 12.0000 Z 12.0000	Sets the velocity in IPM at which an axis feeds off the home limit switch
Velocity Toward Marker X 06.0000 Y 06.0000 Z 06.0000	Sets the velocity in IPM at which an axis searches for the encoder marker pulse
Encoder Multiplier X 02.0000 Y 01.0000 Z 01.0000	Sets an internal multiplier on the number of pulses coming from the encoder. Para. set 1 = 4* multiplication Para. set 2 = 2* multiplication Para. set 4 = 1* multiplication
Slow Jog Velocity	50.0000 Velocity in IPM
Slow Jog Acc/Dec	40.0000 Parameter set the same as Rapid Acc/Dec
Rapid Jog Velocity	100.0000 Velocity in IPM
Rapid Jog Acc/Dec	40.0000 Parameter set the same as Rapid Acc/Dec

In Position X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in feedback units from the destination point where other axes will start their movement. X 00.0001 would set X's distance at 1 encoder count.
G00 Unidirectional X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in inches which an axis will go past the destination tion point in one direction before reversing direction so that the machine will always position from the same direction. Active only in G00 mode. P1——>—P2—>—┐ <————┘ ↑ Uni-distance
G60 Unidirectional X 00.0000 Y 00.0000 Z 00.0000	Same as G00 unidirectional except only active in a G60 block G60 X1 Y2
Backlash X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in inches which the control will compensate for lost motion whenever an axis reversal takes place. Active in all modes.
Excess Error X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in feedback units the machine can lag behind the CNC before the CNC will shut the system down due to an excess following error condition. X 00.0000 = 1 encoder pulse
Rotary=0 Linear=1 X 01.0000 Y 01.0000 Z 00.0000	Sets whether an axis should be treated as circular or linear. In circular the feedrate is interpreted as degrees per minute rather than IPM. A circular axis also will roll over at 360 degrees.
English Leading X 02.0000 Y 02.0000 Z 02.0000	Sets the number of characters to the left of the decimal point for the inch system, for the specified axis only.
English Trailing X 04.0000 Y 04.0000 Z 04.0000	Sets the number of characters to the right of the decimal point for the inch system, for the specified axis only.

Metric Leading X 03.0000 Y 03.0000 Z 03.0000	Same as English Leading except for the metric case.
Metric Trailing X 03.0000 Y 03.0000 Z 03.0000	Same as English Trailing except for the metric case.
Home Switch=0 Marker=1 X 00.0000 Y 00.0000 Z 00.0000	Sets whether an axis will seek a home limit switch and then the marker pulse, or just seek the nearest pulse.
G28 Reference Point X 00.0000 Y 00.0000 Z 00.0000 G30 Reference Point2 X 00.0000 Y 00.0000 Z 00.0000 G30 Reference Point3 X 00.0000 Y 00.0000 Z 00.0000 G30 Reference Point4 X 00.0000 Y 00.0000 Z 00.0000	The G28 thru G30#4 reference point parameters are specified in inches and are relative to the machine zero point. These are the dimensions the control will position to when these functions are used.

5.8.5 F5 MISC (MAIN-PARMS-SETUP-MISC)

The F5 key brings up some miscellaneous setup parameters dealing with the spindle, RS-232 and M codes. When MISC is selected the following screen appears:

Runtime: 000:00:00	MAIN-PARMS-SETUP-MISC	ACTIVE: O1111

```
  CURRENT    NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000
```

Spindle Range 0	1160.0000
Spindle Range 1	2160.0000
Spindle Range 2	5160.0000
Post Mcode #0	+00.0000
Post Mcode #1	+00.0000
Post Mcode #2	+00.0000
Post Mcode #3	+00.0000
Post Mcode #4	+00.0000

Comp	:	Cancelled
Tool	:	00
Length	:	00.0000
Radius	:	00.0000
Plane	:	XY (system #1)
Coords	:	Cartesian
Interp	:	Linear (Feed)
Feed	:	000.0 ipm
(00%)	:	000.0 ipm
Units	:	Abs/English
Cycle	:	Cancelled
Dwell	:	0000.00 sec
Spindle	:	0000 rpm
(00%)	:	0000 rpm (OFF)
Coolant	:	Off

F1 EDIT						F7 ↑	F8 ↓	F9 PGUP	F10 PGDN	ESC ESC

Keys displayed in the Edit Mode:

										ESC EXIT

The Spindle Range 0, 1 and 2 parameters are set to the maximum RPM value for each gear range a machine might have. Range 0 would be the lowest range, Range 2 the highest. There are 10 Post M code slots. Slots #5 thru #9 are on the second page and can be accessed using PGDN or the arrow keys. All M codes will be executed before a move command unless they are entered into the Post M code table. The number of the M code is simply typed in the table if it is to be executed after the completion of an axis move. As an example, if M5 were to be executed after moves, it would be entered in the table as follows:

```
Post M code #1        05.0000
                      ↑
                      M05
```

5.8.6 F2 COORD (MAIN-PARMS-COORD)

The F2 key off the parameter screen brings up the parameters dealing with the various coordinate systems in the control. To edit the work coordinate parameters use the PGUP, PGDN and arrow keys to position the cursor to the correct parameter, and then push the EDIT key and arrow to the desired axis. Type in the new values and push EXIT. The following screen shows the G92, G52 and Work System parameters. The next page of this display contains parameters for:

Tool Change X 00.0000 Y 00.0000 Z 00.0000	Coordinates M6 will move the axes when called. If left blank M6 does not cause any axes movement.
Positive Safe Zone X 00.0000 Y 00.0000 Z 00.0000 Negative Safe Zone X 00.0000 Y 00.0000 Z 00.0000	A position relative to machine zero which, along with the negative safe zone position, describes a cube which the tool cannot enter. If the tool is programmed into this cube an error will be displayed.

Runtime: 000:00:00	MAIN-PARMS-COORD	ACTIVE: O1111

	CURRENT	NEXT
X	00.0000	00.0000
Y	00.0000	00.0000
Z	00.0000	00.0000

Work G92	***
Work G52	***
Work Coord 1	***
Work Coord 2	***
Work Coord 3	***
Work Coord 4	***
Work Coord 5	***
Work Coord 6	***

Work G92
X
Y
Z

Comp	:	Cancelled
Tool	:	00
Length	:	00.0000
Radius	:	00.0000
Plane	:	XY (system #1)
Coords	:	Cartesian
Interp	:	Linear (Feed)
Feed	:	000.0 ipm
(00%)	:	000.0 ipm
Units	:	Abs/English
Cycle	:	Cancelled
Dwell	:	0000.00 sec
Spindle	:	0000 rpm
(00%)	:	0000 rpm (OFF)
Coolant	:	Off

F1 EDIT						F7 ↑	F8 ↓	F9 PGUP	F10 PGDN	ESC ESC

Keys displayed in the Edit Mode:

				F5 MACH		F7 ↑	F8 ↓			ESC EXIT

Operation of the Work Coordinate Systems, G92 and G52, was discussed in an earlier section. These parameters are positions relative to the machine zero and will become the new zero point when they are used. The F5 MACH key in the edit mode enters the current machine position as the work coordinate zero point.

5.8.7 F3 TOOL (MAIN-PARMS-TOOL)

The F3 TOOL key brings up the following screen.

```
Runtime:  000:00:00 | MAIN-PARMS-TOOL                         | ACTIVE: 01111

  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000


  Tool Length (H)          Radius (D)       Comp     :  Cancelled
   T01  01.0000             00.2500         Tool     :  00
   T02  00.0000             00.1000         Length   :  00.0000
   T03 -02.5678             00.0000         Radius   :  00.0000
   T04  00.0000             00.0000         Plane    :  XY (system #1)
    .      .                   .            Coords   :  Cartesian
    .      .                   .            Interp   :  Linear (Feed)
   T15  00.0000              00.0000        Feed     :  000.0 ipm
                                             (00%)   :  000.0 ipm
                                            Units    :  Abs/English
                                            Cycle    :  Cancelled

                                            Dwell    :  0000.00 sec
                                            Spindle  :  0000 rpm
                                             (00%)   :  0000 rpm (OFF)
                                            Coolant  :  Off
```

F1 EDIT						F7 PREV	F8 NEXT			ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

The editing on this screen is the same as on all the other PARMS screens. The F3 tool selection brings up both the H tool length table and the D tool radius table. A value typed into T04 Length is entered into H04 as well. A value typed into T04 Radius is likewise entered into D04. This entry screen is mostly for convenience and is helpful if H and D offsets are always associated with their corresponding T number. This is not mandatory but is a good way to keep tools and offsets straight. There are 100 sets of tool numbers and offsets which can be accessed.

5.8.8 F4 D OFF (MAIN-PARMS-D OFF)

The F4 D OFF key displays the 99 D radius offsets available on the CNC. These offsets are accessed and edited in the same manner as all other "Parameters". Following is the D offset screen.

```
Runtime:   000:00:00 | MAIN-PARMS-D OFF                  | ACTIVE: O1111

  CURRENT    NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000

 D01 00.0000  D02 00.0000  D03 00.0000    Comp     :  Cancelled
 D04 00.0000  D05 00.0000  D06 00.0000    Tool     :  00
 D07 00.0000  D08 00.0000  D09 00.0000    Length   :  00.0000
       .            .            .        Radius   :  00.0000
       .            .            .        Plane    :  XY (system #1)
       .            .            .        Coords   :  Cartesian
       .            .            .        Interp   :  Linear (Feed)
 D40 00.0000  D41 00.0000  D42 00.0000    Feed     :  000.0 ipm
                                           (00%)   :  000.0 ipm
                                          Units    :  Abs/English
                                          Cycle    :  Cancelled

                                          Dwell    :  0000.00 sec
                                          Spindle  :  0000 rpm
                                           (00%)   :  0000 rpm (OFF)
                                          Coolant  :  Off

| F1   |  |  |  |  |  | F7   | F8   |  |  | ESC |
|  EDIT|  |  |  |  |  |  PREV|  NEXT|  |  |  ESC|
```

Keys displayed in the Edit Mode:

```
|  |  |  |  |  |  | F7 | F8 | F9 | F10 | ESC  |
|  |  |  |  |  |  |  ↑ |  ↓ |  ← |   → |  EXIT|
```

5.8.9 F5 H OFF (MAIN-PARMS-H OFF)

The F5 key displays the 99 H tool length offsets available on the control. These offsets are accessed and edited in the same manner as all other "Parameters." The H offset screen follows:

Runtime: 000:00:00	MAIN-PARMS-H OFF	ACTIVE: 01111

```
 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

```
H01 00.0000   H02 00.0000   H03 00.0000
H04 00.0000   H05 00.0000   H06 00.0000
H07 00.0000   H08 00.0000   H09 00.0000
      .             .             .
      .             .             .
      .             .             .
      .             .             .
H40 00.0000   H41 00.0000   H42 00.0000
```

```
Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 EDIT						F7 PREV	F8 NEXT			ESC ESC

Keys displayed in the Edit Mode:

						F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

5.8.10 F6 SAVE (MAIN-PARMS-SAVE)

Store current parameter file on disk.

5.8.11 F7 LOAD (MAIN-PARMS-LOAD)

Load parameter file from floppy drive.

5.8.12 F8 PROG (MAIN-PARMS-PROG)

This set of 125 parameters gives the machine programmer access to all the internal parameters the CNC is using to execute a program. Normally these parameters would be used for display purposes only as an aid to program debugging. However it is possible to read and change these parameters in a parametric program. **Great care** <u>must</u> be taken when doing this because these parameters are used directly by the CNC to produce the next machine movement or function. These parameters are displayed and edited in the same manner as the coordinate parameters. The following is a list and description of these parameters.

P200 thru P207	Contains the previous programmed position relative to the current work offsets of the enabled axis. P200=X P201=Y P202=Z . . . etc.
P208 thru P215	Contains the current programmed position relative to the current work offsets of the enabled axis. P208=X P209=Y P210=Z . . . etc.
P216 thru P223	Contains the previous machine position relative to the machine zero of the enabled axis. P216=X P217=Y P218=Z . . . etc.
P224 thru P231	Contains the current machine position relative to the machine zero of the enabled axis. P224=X P225=Y P226=Z . . . etc.
P232 thru P239	Contains the work coordinate offset relative to the machine zero of the enabled axis. P232=X P233=Y P234=Z . . . etc.
P240 thru P247	Contains the active tool length (H) parameter for the enabled axis. P240=X P241=Y P242=Z . . . etc.
P248	Contains the current arc radius
P249	Contains the current arc I or J or K value
P250	Contains the current arc I or J or K value
P251	Current feedrate

P252	Current dwell time
P253	Current spindle speed
P254	Temporary
P260	Contains active tool number
P261	Contains active D tool radius
P262	Contains active H tool length
P263	Contains active D offset number
P264	Active H offset number
P265	Active canned cycle number
P266	Contains safe zone and overtravel status
P267	Contains ratio of feedback pulses to program units
P270 thru P303	Used as temporary storage
P304	Status if control is in data mode or normal programming
P305	H offset direction or sign
P306	Status of G00, G1 mode
P307	Future
P308	Number of active plane G17, G18, G19
P309	Cutter comp. status G40, G41, G42
P310	Current active canned cycle number
P311	Inch or metric dimensions
P312	Inches/minute or inches/revolution
P313	Revolutions/minute or feet/minute
P314	CW or CCW spindle direction
P315	Inches/minute or mm/minute
P316	Scaling on/off

P317	Rotation on/off
P318	Mirror image on/off
P319	Current work coordinate number
P320 thru P322	Gives the primary, secondary and tertiary axis based on plane selection X=1 Y=2 Z=3 . . . etc. For G17 pri=1 sec=2 ter=3 For G18 pri=1 sec=2 ter=3
P323	Current return plane Z dimension relative machine zero
P24	Tapping mode on/off

5.8.13 F9 CTRL (MAIN-PARMS-CTRL)

This set of 58 parameters is an extension of the Prog parameters. This group deals primarily with parameters used to create the autoroutine and canned cycles. These parameters can be accessed and changed the same as all other parameters in the control. The following is a list and description of these parameters. All coordinates in these parameters are relative to the current work coordinate system.

Orient delay (MS)	Spindle orient delay
P100	Proportional gain
P101	Integral gain
P102	Differential gain
P103	Subscan increment
P104	Detail angle
P105	Probe backlash
P106	Probe radius
P107	Feed 1 - sampling
P108	Feed 2 - searching
P109	Feed 3 - retract
P110	Probe vibration

P111	Wall seek activate
P120 thru P139	Used by 3D pocket
P140	R plane dimension
P141	Final Z depth of canned cycle
P142	Initial level of canned cycle
P143	Z increments of canned cycle
P144	First Z depth of canned cycle
P145	Z axis feedrate for canned cycle
P146	Distance to retract chip brake cycle
P147	R plane distance for each peck cycle
P148, P149	Dwell times for canned cycles
P150	Circular pocket autoroutines radius
P151	X rectangular pocket dimension
P152	Y rectangular pocket dimension
P153	XY finish stock for autoroutines
P154	Z finish stock for autoroutines
P155	Cut width on pocket clearing autoroutines
P156	Radius of bolthole cycles
P157	Bolthole start angle
P158	Number of holes calculated in 360°
P159	Number of holes to be drilled in 360°
P160 thru P171	Scratch
P172 thru P179	Coordinates of the center position for a mirror image command for the enabled axis P172=X P173=Y P174=Z . . . . etc.

P180 thru P187	Coordinates of the scaling center for the enabled axis P180=X P181=Y P182=Z . . . etc.
P188 thru P195	Scale factor for each of the enabled axes P188=X P189=Y P190=Z . . . etc.
P196	I, J, K position of primary axis center of rotation
P197	I, J, K position of secondary axis center of rotation
P198	Angle of rotation
P199	Autoroutines plunge/ramp Z axis 0 = plunge, 1 = ramp

Comm port, baud rate, parity, data bits and stop bits are communications parameters. See the RS-232 section for more information.

Load text cycles	0 = disable text cycles 1 = enable text cycles
P795	RS-232 communications port selection
P796	RS-232 baud rate
P797	RS-232 parity bits
P798	RS-232 data bits
P799	RS-232 stop bits

5.8.14 F10 USER (MAIN-PARMS-USER)

This set of 100 parameters is reserved for the parts programmer to use when writing parametric programs. These parameters are undefined and can be edited, displayed or loaded from this screen. The editing and displaying formats are identical to the parameters discussed in this section.

5.9 Programming F8 PROG (MAIN-PROG)

There are two modes of program file creation/editing available on the Centurion V control: text and conversational. Access to these features is through the PROG softkey.

Pressing the PROG key will change the soft keypad to allow selection of the type of programming wished, or to transfer programs to or from the floppy disk drive.

```
Runtime:  000:00:00 | MAIN-PROG                         | ACTIVE: O0000

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                     Comp     :  Cancelled
                                     Tool     :  00
                                     Length   :  00.0000
                                     Radius   :  00.0000
                                     Plane    :  XY (system #1)
                                     Coords   :  Cartesian
                                     Interp   :  Linear (Feed)
                                     Feed     :  000.0 ipm
                                      (00%)   :  000.0 ipm
                                     Units    :  Abs/English
                                     Cycle    :  Cancelled

                                     Dwell    :  0000.00 sec
                                     Spindle  :  0000 rpm
                                      (00%)   :  0000 rpm (OFF)
                                     Coolant  :  Off

F1   | F2   | F3    |  |  |  |  |  |  |  | ESC
TEXT | CONV | FILES |  |  |  |  |  |  |  | ESC
```

Text and conversational programs are stored in the control in different file formats and have different prefixes to distinguish them. Text programs are prefixed with the letter "O" and are kept in RAM memory in ASCII format. Conversational programs are kept in RAM memory in two formats: ASCII, prefixed with an "O"; and conversational, prefixed with a "P".

Only programs prefixed with an "O" may be run, verified or transferred.

5.9.1 F1 TEXT (MAIN-PROG-TEXT)

Upon entering the text programming mode the upper right hand box (containing the active program number) will switch to show the last text program edited.

```
Runtime:  000:00:00  | MAIN-PROG-TEXT                          | EDITING: O1234

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                              Comp     :  Cancelled
                                              Tool     :  00
                                              Length   :  00.0000
                                              Radius   :  00.0000
                                              Plane    :  XY (system #1)
                                              Coords   :  Cartesian
                                              Interp   :  Linear (Feed)
                                              Feed     :  000.0 ipm
                                               (00%)   :  000.0 ipm
                                              Units    :  Abs/English
                                              Cycle    :  Cancelled

                                              Dwell    :  0000.00 sec
                                              Spindle  :  0000 rpm
                                               (00%)   :  0000 rpm (OFF)
                                              Coolant  :  Off
```

F1 EDIT	F2 NEW	F3 OLD				F7 MENU				ESC ESC

5.9.1.1 F1 EDIT (MAIN-PROG-TEXT-EDIT)

The EDIT key will select the program shown in the upper right corner as the active text edit program and enter the text editing system.

Text Editing Terms

Editor	A program that allows entry and modification of information, then stores it in a file for later retrieval and/or further modification
Text	Text refers to a sequence of characters and/or lines being edited. The individual characters are manipulated using the widely accepted American Standard Code for Information Interchange (ASCII).

Cursor	The cursor is a small line on the screen that marks where changes are being made to the text

Entering and Editing Text

You enter text in much the same way as you enter text on a typewriter, and most of the keys on the keypads behave in the same fashion (pressing ENTER terminates a block, for example). But there are many important differences.

The cursor always indicates where the new text will be entered, and you can move the cursor in a number of ways. You can copy and move text with block commands. You can locate a particular string of text with the FIND command, and optionally replace it with another string using the CHNG command. And, in most cases, you can even undo your last few changes with the REST (restore line) or UNDO commands. These commands, and many more, are described briefly in the following sections.

(MAIN-PROG-TEXT-EDIT)

The first screen you will see when entering the text editor is the edit screen with the first 16 lines of the program displayed. At the bottom of the screen will be the main edit soft keys.

```
Runtime:  000:00:00 | MAIN-PROG-TEXT-EXIT                     | EDITING: 01234

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

 G0 Z0                                    Comp      :  Cancelled
 X1.5                                     Tool      :  00
 Y8                                       Length    :  00.0000
 X-1.5                                    Radius    :  00.0000
 Y-8                                      Plane     :  XY (system #1)
 X1.5                                     Coords    :  Cartesian
 Y0                                       Interp    :  Linear (Feed)
 X0 Y0                                    Feed      :  000.0 ipm
 G98                                       (00%)    :  000.0 ipm
 F20                                      Units     :  Abs/English
 P151=3                                   Cycle     :  Cancelled
 P152=16
 P153=0                                   Dwell     :  0000.00 sec
 P154=.1                                  Spindle   :  0000 rpm
 G81 X0 Y0 G40 G3 R-1 Z-4 F10 Q1           (00%)    :  0000 rpm (OFF)
 G80                                      Coolant   :  Off

F1    | F2    | F3    | F4   | F5  | F6  | F7 | F8 | F9 | F10 | ESC
 BLOCK| CURSR| WORDS | MISC | INS | DEL | ↑  | ↓  | ←  | →   | EXIT
```

Text may now be entered at the current cursor position.

Key Definitions

5.9.1.1.1 F1 BLOCK (MAIN-PROG-TEXT-EDIT-BLOCK)

A block is any arbitrarily defined, contiguous unit of text; a block can be as small as a single character or as large as an entire program. Mark a block by placing a begin-block marker at the first character in the desired block, and an end-block marker just beyond the last character. Once marked, the block can be copied, moved, or deleted.

Although marked blocks are normally highlighted so you can see what you've marked, the block may be hidden (or made visible) with the Hide block command.

F1 BEGIN	F2 END	F3 WORD		F5 HIDE	F6 DEL	F7 COPY	F8 MOVE			ESC ESC

F1 BEGIN Marks the beginning of a block. The marker itself is not visible on the screen, and the block becomes visible only when the end-block marker is set.

F2 END marks the end of a block. Like the begin-block marker, the end-block marker is invisible, and the block itself will not be displayed unless both markers are set.

F3 WORD Marks a single word as a block, combining the functions of the Begin-block and End-block commands. If the cursor is positioned within a word, that word will be marked. If it is not within a word, then the word to the right of the cursor will be marked. And if there is no word to the right of the cursor, then the word to the left will be marked.

F5 HIDE Toggles off and on the visual marking of a block.

F6 DEL Deletes a marked and displayed block. Although the UNDO last deletion command can usually restore portions of an accidentally deleted block, **there is no command** to restore a deleted block in its entirety, so use this command with care.

F7 COPY Creates a copy of a marked and displayed block at the current cursor position. The original block is left unchanged, and the markers are placed around the new copy of the block.

F8 MOVE Moves a marked and displayed block from its current position to the cursor's position. The markers remain around the block at its new position.

5.9.1.1.2 F2 CURSR (MAIN-PROG-TEXT-EDIT-CURSR)

The CURSR menu contains extended cursor movement commands:

F1 BBLOK	F2 EBLOK	F3 TAB	F4 MARK	F5 TOF	F6 EOF	F7 PGUP	F8 PGDN	F9 LEFT	F10 RIGHT	ESC ESC

F1 BBLOK Moves the cursor to the position of the block-begin marker.

F2 EBLOK Moves the cursor to the position of the block-end marker.

F3 TAB Moves the cursor to the beginning of the next word.

F4 MARK Positions the cursor to a marker previously set using MSET.

F5 TOF Moves the cursor to the first character of the program.

F6 EOF Moves the cursor to the last line of the program.

F7 PGUP Moves cursor up 15 lines.

F8 PGDN Moves cursor down 15 lines.

F9 LEFT Moves cursor to beginning of line.

F10 RIGHT Moves cursor to end of line.

5.9.1.1.3 F3 WORDS (MAIN-PROG-TEXT-EDIT-WORDS)

The WORDS softkeys represent reserved words that may be used for programming the control.

F1 IF	F2 THEN	F3 GOTO	F4 CALL	F5 GOSUB	F6 RETRN	F7 WHILE	F8 WEND	F9 SIN	F10 COS	ESC ESC

Pressing a key will cause that word to be printed on the screen. See the section of the manual on programming for the proper usage of these words.

5.9.1.1.4 F4 MISC (MAIN-PROG-TEXT-EDIT-MISC)

This section discusses a number of commands that do not readily fit into any of the other categories.

F1 UNDO	F2 REST	F3 HDW	F4 MSET	F5 MHIDE	F6 UNDEL	F7 CHNG	F8 FIND	F9 FNEXT		ESC ESC

F1 UNDO — Restores whole lines deleted with the Delete line command or the Delete block command. It does not restore single characters or words. To undo your most recent changes to the current line, use the REST command.

F2 REST — Will undo any changes made to a line of text as long as you have not left the line. The line is restored to its previous contents regardless of the changes made.

F3 HDW — Allows the use of the handwheel to move to a position and have that position inserted in the program by pressing the ENTER key.

F1 X	F2 Y	F3 Z	F4 A						F10 FLZ	ESC EXIT

F1 - F4 (X/Y/Z/A) — Choice of axis position to insert

F10 Floating Zero — Sets current axes positions to zero

F4 MSET — Sets one of the ten markers at the current position of the cursor.

F5 MHIDE — Hides or makes all text markers visible. Setting a new text marker automatically turns off Marker display if it was previously turned off.

F6 LNDEL — Deletes the line containing the cursor and moves any lines below it up one line. The cursor moves to column 1 of the next line.

F7 CHNG — This operation works the same as the Find command except that you can replace the found string with any other string of up to 67 characters. After entering the search string, you are asked to enter the replacement string. The last replacement string entered, if any, will be displayed; you may accept it, edit it, or enter a new string. Finally you are prompted for options. The options you used last are displayed at first. You may enter new options (cancelling the old ones), edit the current options, or select them by pressing ENTER. The options available are the same as those for the Find command.

F8 FIND — Lets you search for a string of up to 67 characters. When you enter this command you will be asked for a search string. The last search string entered (if any) will be displayed. You can select it again by pressing ENTER, edit it, or enter a new search string. After the search string is entered you must specify your search options. The options you used last, if any, are displayed. You can enter new options (cancelling the old ones), edit the current options, or select them again by pressing ENTER. The following options are available:

B — Searches backwards from the current cursor position toward the beginning of the program.

G — Searches globally. The entire program is scanned for the search string regardless of the current position of the cursor. The search starts at the beginning of the program if searching forwards; at the end if searching backwards.

L — Limits searches to the currently marked block.

n — Finds the nth occurrence of the string (overridden by the L option).

U — Ignores case; treats all alphabetic characters as if they were uppercase.

W Searches for whole words only; skips matching patterns embedded in other words.

If the text contains a target matching the search string, the target is highlighted and the cursor is positioned just beyond it.

F9 FNEXT Repeats the last search operation. If the last search command called for a Find operation, the same search string and options will be repeated; for a Find-and-replace operation, the replacement string will be reused as well.

5.9.1.1.5 F5 INS (MAIN-PROG-TEXT-EDIT-INS)

If the INS key is on, the editor is in Insert mode and characters will be inserted at the cursor position. If the INS key is off, the editor is in Overwrite mode and characters will overwrite any previous character at the cursor position.

5.9.1.1.6 F6 DEL (MAIN-PROG-TEXT-EDIT-DEL)

Deletes the character under the cursor and moves any characters to the right of the cursor one position to the left. This command does not work across line breaks.

Note: *At any time while in the editor, pressing the ← (backspace) key on the numeric keypad will move the cursor one character to the left and delete the character positioned there. Any characters to the right of the cursor are moved one space to the left.*

F7 ↑ Moves cursor up one line.

F8 ↓ Moves the cursor down one line.

F9 ← Moves the cursor to the left.

F10 → Moves the cursor to the right.

ESC EXIT Upon pressing the EXIT key to leave the editor, the active edit program is checked to determine if it was modified. If it was a prompt will be displayed in the message window asking if the changes should be accepted and stored. Pressing "Y" will accept the changes and alter the program file. Pressing "N" will abort the changes and leave the file unchanged.

5.9.1.2 F2 NEW (MAIN-PROG-TEXT-NEW)

The NEW key will allow entry (in the message box) of a number for a new text program. After the number has been entered, the control will check the text programs currently in RAM memory to see if a text program by that number is already there. If it is found, a warning will be displayed and the operator will be allowed to OK the erasing of the text program prior to entering the text editor.

5.9.1.3 F3 OLD (MAIN-PROG-TEXT-OLD)

The OLD key will allow entry, in the message window, of the number of an existing text program. After the number has been entered, the control will check the text programs currently in RAM memory to see if a program by that number is there. If it is the Edit Window will appear displaying the first 16 lines of the program. If not, an error stating that the program was not found will appear and, after pressing the ESC key, another number may be entered.

5.9.1.4 F7 MENU (MAIN-PROG-TEXT-MENU)

The MENU key will display a list of all text programs currently loaded in RAM memory. By using the F7 - F10 keys the file selection arrows are positioned at the program to edit, and the F5 ENTER key is pressed to make a selection.

```
Runtime:  000:00:00 | MAIN-PROG-TEXT-MENU | EDITING: O1234

 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

O1234 (PART 01-1 FLANGE  ) 002589     Comp     :  Cancelled
O1235 (PART 01-2 BOLTHOLE) 000128     Tool     :  00
O9900 (LOGO              ) 000500     Length   :  00.0000
                                      Radius   :  00.0000
                                      Plane    :  XY (system #1)
                                      Coords   :  Cartesian
                                      Interp   :  Linear (Feed)
                                      Feed     :  000.0 ipm
                                       (00%)   :  000.0 ipm
                                      Units    :  Abs/English
                                      Cycle    :  Cancelled

                                      Dwell    :  0000.00 sec
                                      Spindle  :  0000 rpm
                                       (00%)   :  0000 rpm (OFF)
                                      Coolant  :  Off
```

				F5 ENTER		F7 ↑	F8 ↓	F9 PGUP	F10 PFDN	ESC ESC

The edit window will appear displaying the first 16 lines of the program.

Regardless of which selection mode is used, whenever the edit window is displayed the number of the text program being edited will be shown in the active window.

While programming in the conversational system, three types of softkey configurations will be encountered. They are:

5.9.1.5 STORE/INPUT KEYS

```
Runtime:  000:00:00   MAIN-PROG-CONV                        EDITING: P1234

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

 Program Setup                               Comp     :  Cancelled
                                             Tool     :  00
   Dimensions    [ABSOLUTE]                  Length   :  00.0000
   Units         [ENGLISH ]                  Radius   :  00.0000
                                             Plane    :  XY (system #1)
                                             Coords   :  Cartesian
                                             Interp   :  Linear (Feed)
                                             Feed     :  000.0 ipm
                                              (00%)   :  000.0 ipm
                                             Units    :  Abs/English
                                             Cycle    :  Cancelled

                                             Dwell    :  0000.00 sec
                                             Spindle  :  0000 rpm
                                              (00%)   :  0000 rpm (OFF)
                                             Coolant  :  Off
```

F1 STORE		F3 TOGL		F5 DEL		F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

These softkeys will be available whenever Input is expected. At this time, a screen containing any number of fields will be displayed and the cursor will be positioned in one of the fields.

There are two types of fields which may be on an input screen, data and toggle. Data fields are fields in which datum is entered using the keyboard and may be left blank. Toggle fields are fields that have a limited number of possible input values and whose value may only be changed by pressing the TOGL key.

Definitions of the Store/Input keys are as follows:

F1 STORE	Accepts the entries and adds to the program file. If all required data has not been entered the STORE key will not work and the cursor will position to the field requiring input. Each screen stored is called an event.
F3 TOGL	Pressing this key will result in the next toggle value being displayed in the field. It will have no effect in a data field.
F5 DEL	Used to delete an entry from a data field. It will have no effect in a toggle field.
F7 ↑	Moves cursor to the previous field.
F8 ↓	Moves cursor to the next field.
F9 ←	Moves cursor to the left.
F10 →	Moves cursor to the right.
ESC EXIT	Will abort input and return to the menu keys if creating a new program, or the edit keys if editing an old program.

5.9.1.6 EDIT KEYS

```
Runtime:  000:00:00 | MAIN-PROG-CONV                    | EDITING: P1234

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

            EVENT 0020               Comp     :  Cancelled
  -END OF PART-                      Tool     :  00
                                     Length   :  00.0000
                                     Radius   :  00.0000
                                     Plane    :  XY (system #1)
                                     Coords   :  Cartesian
                                     Interp   :  Linear (Feed)
                                     Feed     :  000.0 ipm
                                      (00%)   :  000.0 ipm
                                     Units    :  Abs/English
                                     Cycle    :  Cancelled

                                     Dwell    :  0000.00 sec
                                     Spindle  :  0000 rpm
                                      (00%)   :  0000 rpm (OFF)
                                     Coolant  :  Off

F1   |     | F3    |   |   | F6  | F7  |   | F9   | F10  | ESC
EDIT |     | EVENT |   |   | INS | DEL |   | PREV | NEXT | EXIT
```

These softkeys will be available whenever the edit menu system has not been entered. At this time it is possible to step through the program, edit events, and insert or delete events.

Definitions of the Edit keys are as follows:

F1 EDIT — Pressing the EDIT key will position the cursor at the first field of the current event. The Store/Input keys will appear and the event may be edited.

F3 EVENT — Allows entry of an event number to search for. If the event number is not found, the -END OF PART- screen will be displayed.

F6 INS — INS is used to insert events in a program. The new event(s) will be inserted before the event that is currently displayed. Inserting will continue until the EXIT softkey is pressed in the menu subsystem.

F7 DEL	Will delete the event currently being displayed.
F9 PREV	Displays the previous event in the program file.
F10 NEXT	Displays the next event in the program file.
ESC EXIT	Exits the conversational system and automatically creates the executable text program with an "O" prefix.

5.9.2 F2 CONV (MAIN-PROG-CONV)

This next section will deal with selecting conversational programs. Upon entering the conversational programming mode, the active window in the upper right hand corner will switch to show the last conversational program edited.

```
Runtime:  000:00:00 | MAIN-PROG-CONV              | EDITING: P1234

  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

                                Comp     :  Cancelled
                                Tool     :  00
                                Length   :  00.0000
                                Radius   :  00.0000
                                Plane    :  XY (system #1)
                                Coords   :  Cartesian
                                Interp   :  Linear (Feed)
                                Feed     :  000.0 ipm
                                 (00%)   :  000.0 ipm
                                Units    :  Abs/English
                                Cycle    :  Cancelled

                                Dwell    :  0000.00 sec
                                Spindle  :  0000 rpm
                                 (00%)   :  0000 rpm (OFF)
                                Coolant  :  Off
```

F1 EDIT	F2 NEW	F3 OLD				F7 MENU				ESC ESC

Four options are available with which to choose the conversational program to edit.

5.9.2.1 F1 EDIT (MAIN-PROG-CONV-EDIT)

Pressing the EDIT key will select the program shown in the active window as the active conversational program and enter the conversational programming system.

5.9.2.2 F2 NEW (MAIN-PROG-CONV-NEW)

Pressing the NEW key will allow entry, in the message box, of a number for a new conversational program. After the number has been entered, the control will check the conversational programs currently in RAM memory to see if a program by that number is already there. If it is found two warnings will be displayed, and the operator will be allowed to "OK" the erasing of the conversational program and its associated text program prior to entering the conversational system. Remember that conversational programs are stored in both formats.

5.9.2.3 F3 OLD (MAIN-PROG-CONV-OLD)

The OLD key will allow entry, in the message window, of the number of an existing conversational program. After the number has been entered, the control will check the conversational programs currently in RAM memory to see if a program by that number is there. If it is the edit window will appear displaying the program setup screen. If not, an error stating that the program was not found will appear and, after pressing the ESC key, another number may be entered.

5.9.2.4 MACHINE OPERATION KEYS (MAIN-PROG-CONV-)

The menu keys are used to move throughout the conversational system and reach the desired input screen. The menu key sequence is listed on the Conversational System Flowchart shown below.

This is what the first level menu keys look like:

F1 POS	F2 MILL	F3 DRILL	F4 BOLT	F5 TCHNG	F6 MISC	F7 CALL	F8 SPEC		F10 EXIT	

CONVERSATIONAL SYSTEM FLOWCHART

```
Menu Subsystem                      Input Screens

                                    Program Setup

POS                                 Position
MILL
  START                             Start Mill Cycle
  GEOM
      LINE                          Line
      ARC                           Arc
   MISC                             Miscellaneous
   END                              End Mill Cycle
   POCK
      SETUP                         Setup Pocket Mill
      CIRC
          CLEAR                     Circular Clear
          F IN                      Circular Finish In
      RECT
          CLEAR                     Rectangular Clear
          F IN                      Rectangular Finish In
   FRAME
      SETUP                         Setup Frame Mill
      CIRC                          Circular Finish Out
      RECT                          Rectangular Finish Out
DRILL
   START
      DRILL                         Enable Drill Cycle
      D/DWL                         Enable Drill/Dwell Cycle
      PECK                          Enable Peck/Drill Cycle
      WPECK                         Enable Woodpecker Cycle
      BORE                          Enable Bore Cycle
      B/DWL                         Enable Bore/Dwell Cycle
      TAP                           Enable Tap Cycle
   POS                              Drill Positioning
   MISC                             Miscellaneous
   SUB                              Call Sub-program
   END                              Disable Cycle
BOLT
   DRILL                            Drill Bolthole
   D/DWL                            Drill/Dwell Bolthole
   PECK                             Peck Drill Bolthole
   WPECK                            Woodpecker Bolthole
   BORE                             Bore Bolthole
   B/DWL                            Bore/Dwell Bolthole
   TAP                              Tap Bolthole
TCHNG                               Tool Call/Change
MISC                                Miscellaneous
CALL                                Call Sub-Program
SPEC
   PARMS                            Set Parameter
   TOOLS                            Set Tool Offset
   SCALE                            Set Scaling
   ROT                              Set Rotation
   MIRR                             Set Mirror Image
   FLZ                              Set Floating Zero
```

F1 thru F8 will either bring up an input screen (e.g. F1-POS) like this:

Runtime: 000:00:00	MAIN-PROG-CONV	EDITING:P1234

```
  CURRENT    NEXT
X 00.0000   00.0000
Y 00.0000   00.0000
Z 00.0000   00.0000
```

```
Position

  Feedrate     [RAPID     ]
  Coordinates  [CARTESIAN]

  X-axis   [          ]
  Y-axis   [          ]
  Z-axis   [          ]
  A-axis   [          ]
```

```
Comp      :  Cancelled
Tool      :  00
Length    :  00.0000
Radius    :  00.0000
Plane     :  XY (system #1)
Coords    :  Cartesian
Interp    :  Linear (Feed)
Feed      :  000.0 ipm
 (00%)    :  000.0 ipm
Units     :  Abs/English
Cycle     :  Cancelled

Dwell     :  0000.00 sec
Spindle   :  0000 rpm
 (00%)    :  0000 rpm (OFF)
Coolant   :  Off
```

F1 STORE		F3 TOGL		F5 DEL		F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

or another menu (e.g. F2 MILL) like this:

F1 START	F2 GEOM	F3 MISC	F4 END	F5 POCK	F6 FRAME	F7 3DPKT	F8		F10 EXIT	F11 BACK

Notice that on all levels except level 1 there is an ESC-BACK key. This key will return you to the previous level menu keys.

Pressing the F10 EXIT key will exit the menu subsystem and display the level 1 EDIT keys.

Whenever an input screen is encountered the Store/Input keys will be displayed:

F1 STORE		F3 TOGL		F5 DEL		F7 ↑	F8 ↓	F9 ←	F10 →	ESC EXIT

5.9.2.5 F7 MENU (MAIN-PROG-CONV-MENU)

The MENU key will display a list of all conversational programs currently loaded in RAM memory.

```
Runtime:  000:00:00 | MAIN-PROG-CONV-MENU                | EDITING: P1234

 CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

P1234 (PART 01-1 FLANGE  )  000090      Comp     :  Cancelled
P1235 (PART 01-2 BOLTHOLE)  000019      Tool     :  00
P9900 (LOGO              )  000030      Length   :  00.0000
                                        Radius   :  00.0000
                                        Plane    :  XY (system #1)
                                        Coords   :  Cartesian
                                        Interp   :  Linear (Feed)
                                        Feed     :  000.0 ipm
                                         (00%)   :  000.0 ipm
                                        Units    :  Abs/English
                                        Cycle    :  Cancelled

                                        Dwell    :  0000.00 sec
                                        Spindle  :  0000 rpm
                                         (00%)   :  0000 rpm (OFF)
                                        Coolant  :  Off

|  |  |  |  | F5    |  | F7 | F8 | F9   | F10  | ESC |
|  |  |  |  | ENTER |  | ↑  | ↓  | PGUP | PGDN | ESC |
```

By using the F7 - F10 keys the file selection arrows are positioned at the program to edit, and the F5 ENTER key is pressed to make the selection. The edit window will appear displaying the program setup screen.

Regardless of which selection mode is used, whenever the edit window is displayed the number of the conversation program being worked with will be shown in the editing window.

5.10 Verify F9 VERF (MAIN-VERF)

The Verify function is used to verify part programs. Most verification is done in the graphics mode, but it doesn't have to be. The distances traveled, and run, feed and rapid times, are valid during verify and can be used to estimate machining times. The program which is verified is the active program. To get coordinate information to compare against a print, put the control in block mode and step through the program. The cursor in the graphic mode will step around the part and the X Y Z display will read out the coordinate values of each point.

Verify is used to verify the active program. Upon pushing the VERF button the following screen appears:

```
Runtime:  000:00:00 | MAIN-VERF                | ACTIVE: O1111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                 Comp     :  Cancelled
                                 Tool     :  00
                                 Length   :  00.0000
                                 Radius   :  00.0000
                                 Plane    :  XY (system #1)
                                 Coords   :  Cartesian
                                 Interp   :  Linear (Feed)
                                 Feed     :  000.0 ipm
                                  (00%)   :  000.0 ipm
                                 Units    :  Abs/English
                                 Cycle    :  Cancelled

                                 Dwell    :  0000.00 sec
                                 Spindle  :  0000 rpm
                                  (00%)   :  0000 rpm (OFF)
                                 Coolant  :  Off
BLOCK
F1    |     | F3    | F4    | F5    | F6    | F7   | F8  |  |  | ESC
 START|     |  BLOCK|  OSTOP|  BSKIP|  DISPL|  MENU|  DRY|  |  |  ESC
```

After the above screen appears, F1 START would be pushed and the following screen would appear:

Runtime: 000:00:00	MAIN-VERF-START	ACTIVE: O1111

```
  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

Comp     :  Cancelled
Tool     :  00
Length   :  00.0000
Radius   :  00.0000
Plane    :  XY (system #1)
Coords   :  Cartesian
Interp   :  Linear (Feed)
Feed     :  000.0 ipm
 (00%)   :  000.0 ipm
Units    :  Abs/English
Cycle    :  Cancelled

Dwell    :  0000.00 sec
Spindle  :  0000 rpm
 (00%)   :  0000 rpm (OFF)
Coolant  :  Off
```

F1 FIRST	F2 BLOCK	F3 TOOL								ESC ESC

The F1 key FIRST is automatically selected when entering this screen from the VERF screen. Therefore if it is desired to verify the active program from the beginning, all that is necessary is to push the CYCLE START (Alt F6) button. If F2 BLOCK is pushed, the control will request that the desired block or sequence number be typed in, followed by ENTER. If the Cycle Start button is pushed the active program will start verifying from the selected block number. If F3 TOOL is pushed, the control will request a tool number. After typing the tool number followed by an ENTER, the CYCLE START (Alt F6) button is pushed, and the active program will start verifying at the desired tool number and the following screen will appear.

```
Runtime:  000:00:00 | MAIN-VERF                                    | ACTIVE: O0000

  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000

                                      Comp     :  Cancelled
                                      Tool     :  00
                                      Length   :  00.0000
                                      Radius   :  00.0000
                                      Plane    :  XY (system #1)
                                      Coords   :  Cartesian
                                      Interp   :  Linear (Feed)
                                      Feed     :  000.0 ipm
                                       (00%)   :  000.0 ipm
                                      Units    :  Abs/English
                                      Cycle    :  Cancelled

                                      Dwell    :  0000.00 sec
                                      Spindle  :  0000 rpm
                                       (00%)   :  0000 rpm (OFF)
                                      Coolant  :  Off
BLOCK

      | F2    | F3    | F4    | F5    | F6    |       | F8    | F9    |       | ESC
      | RESUM | BLOCK | OSTOP | BSKIP | DISPL |       | DRY   | HALT  |       | ESC
```

This screen is the basic verify screen with two new additions, a block number display and F9-HALT. The block number shows the current line being executed as the program verifies. The HALT key is similar to FEEDHOLD in that when pushed the machine will stop. However, unlike FEEDHOLD the HALT also exits the "Run" mode and allows a new program to be started.

5.10.1 F2 RESUM (MAIN-VERF)

Once a program has been "Halted" the "Resume" feature of the control becomes active. The F2 RESUM key will now be displayed on the Run screen. A program can be Re-sumed as long as one of the following functions is not performed: "Verf", MDI, Home, or EM STOP. If none of the above functions are done the axis can be "Jogged" or Handwheeled away from the work, the spindle turned on or off, or any other function performed, and the "Resume" remains active. As long as the Resume is active the F2 key on the Run screen will show a "Resume" function. If the "Resume" function is selected the active program will be resumed at the halted point. The first thing that will happen when a "Resume" "Cycle Start" is executed is Z will retract to the tool change position, all the way up. Next, X and Y will rapid to the halted point. Once X and Y are in position a "Cycle Start" will be requested. When CYCLE START is pushed the Z axis will rapid to the R plane and then feed to its previous depth. The program will then start running as if nothing had ever happened. FEEDHOLD simply stops axes motion until it is released and CYCLE START is pushed.

Note: *Once the "Run" or "Verify" modes have been entered, a "Halt" must be executed if the mode is to be exited prior to program completion.*

The F3, F4, F5 and F8 keys on the RUN screen set the mode of operation a program will run in. When these keys are in a highlighted state the functions will be active in any currently running program or program to be run.

5.10.2 F3 BLOCK (MAIN-VERF)

When the BLOCK switch is activated the program will stop at the end of each block. Each time the CYCLE START (Alt F6) button is pushed one more block will be verified.

5.10.3 F4 OSTOP (MAIN-VERF)

When the optional stop switch (OSTOP) is activated the program will stop at each M01 command. Each time the CYCLE START (Alt F6) button is pushed the program will run to the next M01 or the end of program.

5.10.4 F5 BSKIP (MAIN-VERF)

When the block skip switch (BSKIP) is activated the program will skip all blocks started with a "/".

/M5 When the block skip switch is active the spindle off command will not be executed.

5.10.5 F6 DISPL (MAIN-VERF-DISPL)

The F6 Display key (DISPL) can be accessed from a number of screens. The following screen is shown as though the DISPL was entered from the Main screen. All the display functions and screens are identical, independent of the entry point. Only the return point differs based on the original entry point. When the DISPL key is pushed the following screen will appear:

```
Runtime:   000:00:00 | MAIN-VERF-DISPL                          | ACTIVE: O1111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

                                  Comp     :  Cancelled
                                  Tool     :  00
                                  Length   :  00.0000
                                  Radius   :  00.0000
                                  Plane    :  XY (system #1)
                                  Coords   :  Cartesian
                                  Interp   :  Linear (Feed)
                                  Feed     :  000.0 ipm
                                   (00%)   :  000.0 ipm
                                  Units    :  Abs/English
                                  Cycle    :  Cancelled

                                  Dwell    :  0000.00 sec
                                  Spindle  :  0000 rpm
                                   (00%)   :  0000 rpm (OFF)
                                  Coolant  :  Off
```

F1 NEXT	F2 DIST	F3 GRAPH	F4 DIAG							ESC ESC

5.10.5.1 F1 NEXT (MAIN-VERF-DISPL)

When the NEXT key is activated the position display on the verify screen displays the current position and the next position.

```
  CURRENT    NEXT
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

5.10.5.2 F2 DIST (MAIN-VERF-DISPL)

When the distance key (DIST) is activated the position display on the VERF screen displays the current position and the distance to go.

```
  CURRENT    DIST
X 00.0000  00.0000
Y 00.0000  00.0000
Z 00.0000  00.0000
```

5.10.5.3 F3 GRAPH (MAIN-VERF-DISPL-GRAPH)

If the GRAPH key is activated the control switches from displaying text to a graphic display of the active part program. The following screen will appear.

Runtime: 000:00:00	MAIN-VERF-DISPL-GRAPH	ACTIVE: O1111

CURRENT
X 00.0000
Y 00.0000
Z 00.0000

TIMES AT 100%
RUN: 000:00:00
FEED: 000:00:00
RAPD: 000:00:00
DISTANCE:
0000.0000

GRAPHICS AREA

F1 ROT	F2 PAN	F3 WIND	F4 AUTO	F5 ZOOM-	F6 ZOOM+	F7 LIMIT	F8 ZONE	F9 FRESH	F10 CLEAR	ESC ESC

The graphics on this control are full 3D and will be displayed in the Graphics Area as long as the control remains in the GRAPH mode. When other displays are requested windows will appear in the graphics area showing the requested data. When these functions are finished the windows will disappear and the graphic display will be reinstated. The scale at the top of the screen is to be used as a reference for the part size. As the screen scale is changed the graduations on the ruler will change accordingly. The ruler graduations are in machine units but on an English system the largest graduations equal approximately one inch. The times at 100% and distance displays are normally used during graphic verify but are also valid during normal running. The Run, Feed and Rapid times are intended to be used during a Graphic Verify to estimate the time a part will take to run. All calculations are based on the programmed feedrate and a 100% feedrate override setting. The distance display gives the total inches the machine has travelled during the program.

This display is intended to help in estimating tool wear. The Runtime at the top of the screen is basically a stop watch which starts when a Run Program command is executed and stops at the end of program or when the program is aborted. The two runtimes can be compared at the end of a program to determine how much time the machine sat idle (tool changes, block stops, feedrate less than 100%, etc.) versus actual cutting time.

The graphics functions used in the Verify mode are the same functions used in the RUN mode. For a full explanation of these functions refer to section 5.5.5.3, page 5-14, on Runtime graphics.

5.10.5.4 F4 DIAG (MAIN-VERF-DISPL-DIAG)

The F4 key is mainly used for machine setup or troubleshooting machine functions. The diagnostic screens bring up all the external I/O bits connected to the CNC. The status of each bit is continuously displayed on the screen and as they change on the machine the screen will be updated. Function keys F1 thru F5 diaplay the various axes enabled on the CNC. Selecting X, Y, Z, etc. brings up the appropriate axis I/O channels. A "0" displayed on the screen means the I/O channel is not active. A "1" indicates active.

5.11 Utilities F10 UTIL (MAIN-UTIL)

When F10 UTIL is pressed the following options are available:

		F3 FILES	F4 RS232							ESC ESC

5.11.1 F3 FILES (MAIN-UTIL-FILES)

The files utilities contain basic program manipulation functions. They are:

F1 LOAD	F2 SAVE	F3 NAME	F4 COPY	F5 LIST	F6 DIR	F7 MENU		F9 ERASE		ESC ESC

5.11.1.1 F1 LOAD (MAIN-UTIL-FILES-LOAD)

This function is used to load programs from the floppy disk into the control's program memory. When this function is selected the following screen is displayed:

```
Runtime:  000:00:00 | MAIN-UTIL-FILES-LOAD                | ACTIVE: 01111

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

 ►00001      1243     09/26/90  09:11◄    Comp     :  Cancelled
  00033       164     09/26/90  09:10     Tool     :  00
  09000      3849     09/26/90  09:11     Length   :  00.0000
                                          Radius   :  00.0000
                                          Plane    :  XY (system #1)
                                          Coords   :  Cartesian
                                          Interp   :  Linear (Feed)
                                          Feed     :  000.0 ipm
                                           (00%)   :  000.0 ipm
                                          Units    :  Abs/English
                                          Cycle    :  Cancelled

                                          Dwell    :  0000.00 sec
                                          Spindle  :  0000 rpm
                                           (00%)   :  0000 rpm (OFF)
                                          Coolant  :  Off
```

F1	F2	F3	F4	F5		F7	F8	F9	F10	ESC
START	SET	RESET	ALL	NONE		↑	↓	PGUP	PGDN	ESC

The edit window will display a list of the programs on the floppy drive as well as the length (in bytes) of the programs, their creation date and creation time. The selection cursor (> <) is positioned at the first program.

F1 START — Pressing this key will begin the transfer of the selected programs from floppy disk to program memory.

F2 SET — Selects the file at the cursor position to be loaded from floppy disk.

F3 RESET — Unselects the file at the cursor position.

F4 ALL — Selects all programs on floppy disk to be loaded.

F5 NONE — Unselects all selected programs.

F7 ↑ — Moves selection cursor up one line.

F8 ↓ — Moves selection cursor down one line.

F9 PGUP — Moves selection cursor up 16 lines.

F10 PGDN — Moves selection cursor down 16 lines.

5.11.1.2 F2 SAVE (MAIN-UTIL-FILES-SAVE)

This function is used to save programs from the control's program memory to the floppy disk. When this function is selected the softkeys will change and the type of program to be saved (text, conversational) must be selected.

F1 TEXT	F2 CONV									

After a program type has been chosen, the SAVE function operates the same as the LOAD function with the exception that the transfer direction is changed.

5.11.1.3 F3 NAME (MAIN-UTIL-FILES-NAME)

This function changes the name of a program.

5.11.1.4 F4 COPY (MAIN-UTIL-FILES-COPY)

This function makes a copy of a program under another name.

5.11.1.5 F5 LIST (MAIN-UTIL-FILES-LIST)

This option allows a program in program memory to be selected to list by pressing the ENTER key. The selected program will then be displayed in the edit window and may be looked at, but not edited.

5.11.1.6 F6 DIR (MAIN-UTIL-FILES-DIR)

This key will display a list of programs on a floppy disk.

5.11.1.7 F7 MENU (MAIN-UTIL-FILES-MENU)

This key will display a list of text or conversational programs currently in program memory.

5.11.1.8 F9 ERASE (MAIN-UTIL-FILES-ERASE)

This function is used to erase programs from the control's program memory. The keys available are the same as those in the SAVE option.

5.11.2 F4 RS232 (MAIN-UTIL-RS232)

The RS-232 utilities contain basic communications functions. They are:

		F3 DNC	F4 RUN	F5 SEND	F6 RECEV	F7 PARMS				ESC ESC

5.11.2.1 (F3 DNC) MAIN-UTIL-RS232-DNC

When this key is pressed the following screen appears:

```
Runtime:  000:00:00 | MAIN-UTIL-RS232-DNC | ACTIVE: 01234

  CURRENT    NEXT
 X 00.0000  00.0000
 Y 00.0000  00.0000
 Z 00.0000  00.0000

 Waiting for DNC link . . .        Comp     :  Cancelled
                                   Tool     :  00
 Skipcount : 0                     Length   :  00.0000
 Blocks    : 0                     Radius   :  00.0000
                                   Plane    :  XY (system #1)
                                   Coords   :  Cartesian
                                   Interp   :  Linear (Feed)
                                   Feed     :  000.0 ipm
                                    (00%)   :  000.0 ipm
                                   Units    :  Abs/English
                                   Cycle    :  Cancelled

                                   Dwell    :  0000.00 sec
                                   Spindle  :  0000 rpm
                                    (00%)   :  0000 rpm (OFF)
                                   Coolant  :  Off

F1    | F2    |  |  |  |  |  |  | F9   |  | F11
DISK1 | DISK2 |  |  |  |  |  |  | SKIP |  | ABORT
```

This option should be used when it is necessary to run a large program or a program containing very short moves, and requires no additional calculations (Trig help, cutter compensation).

A file to be DNC'd may only contain coordinate data (X, Y, Z, A), block numbers (N), and an M30 to designate end of program. This is the fastest processing mode.

At this time, the control is waiting for characters to come in the serial port. If communications is established, then the message will change to "DNC link established . . ." and processing will begin. If DNC is not to be run via RS-232 then the following keys may be used.

5.11.2.1.1 F1 DISK1

DISK1 will allow the selection of a program from program memory to be DNC'd.

5.11.2.1.2 F2 DISK2

DISK2 will allow the selection of a program from floppy disk to be DNC'd.

5.11.2.1.3 F9 SKIP

This function key allows entry of a skip count. The skip count is the number of program blocks to be skipped before the program begins execution. This allows execution to start anywhere in the program.

ESC ABORT   Stops execution of the program.

The following messages may be seen on the first line of the edit window when the DNC mode is entered.

-Waiting for DNC link . . .
    No program has been selected from Disk1 or Disk2

-DNC link established . . .
    The program listed in the active program box has been selected to DNC and a CYCLE START will begin execution.

-Executing DNC program . . .
    The program is being executed.

-Skipping DNC program . . .
    The number of blocks entered in skip count is being skipped.

5.11.2.2 F4 RUN (MAIN-UTIL-RS232-RUN)

The RUN option is identical to the DNC option but it does support all the functionality of the control language. This includes all M codes, G codes and transformations (e.g. cutter compensation, scaling, rotation, etc.). This added functionality drastically reduces the throughput and is not well suited to programs with very small moves (less than .0500).

5.11.2.3 F5 SEND (MAIN-UTIL-RS232-SEND)

The SEND option is used to send programs from the control's program memory to an off-line computer. The softkeys for this function are simply:

F1 BEGIN						F7 MENU				ESC ESC

F1 BEGIN — Starts transmission of the active send program.

F2 MENU — Allows selection of a program as the active send program.

Note: *The RS-232 parameters must be set to the same values on both the control and the computer. Always check these parameter settings.*

The RS-232 parameters are found in the control parameters (MAIN-PARMS-CTRL). Normal parameter settings are:

Baud rate	*1200 or 9600*
Parity	*Even*
Data bits	*7*
Stop bits	*2*

5.11.2.4 F6 RECEV (MAIN-UTIL-RS232-RECEV)

The RECEV option is used to receive programs from an off-line computer into the control program memory. Upon pressing the RECEV key a new program number must be entered in the message window. Once a valid program number has been entered, the edit screen will appear and display the program as it is received. Receiving will continue until the ESC key is pressed.

6. PARAMETRIC PROGRAMMING

Parametric programming is similar to macro programming in that equations can be used to specify axis position rather than decimal numbers. The Centurion V does not restrict the use of parametrics to subroutines or macros. They can be used anywhere throughout a program. Parametric expressions cannot be used to specify M or G codes but can be used to specify axes X Y Z A etc., F and S functions. When a parametric expression is used for an axis position it will first be evaluated and then cutter compensation will be applied. All the normal cutter compensation rules will apply to the evaluated point. When using parametric expressions in a program the parameters which are used are the 100 "User Parameters" discussed earlier. P1 is "User Parameter 1." Values generated by equations can be displayed on "User" parameter screens. Other listed system parameters can be used as input data to parametric equations but under normal circumstances these parameters should not be changed.

6.1 Parametric reference

A parameter reference is specified by the letter "P" followed by a valid parameter number. When a parameter reference is used for a coordinate position it must be contained in brackets.

```
Example:        X [P10]
                Y [-P145]
                Z [P2]
```

6.2 Parametric assignment statement

Assigning is the most basic statement in the use of parameters. The assignment character is an equal sign (=).

Assignment statements replace the current value of a variable with a new value specified by an operand type.

```
Example:        P1 = 1.234
```

In this example the value 1.234 is assigned to parameter 1. Therefore both of the following statements would move to the same coordinate position:

```
X1.234 or X[P1]
```

6.3 Parametric operators

6.3.1 Arithmetic operators

The following table shows the available arithmetic operators:

Operator	Operation
+	addition
-	subtraction
*	multiplication
/	division
DIV	integer division
MOD	remainder

The value of A div B is the mathematical quotient of A/B with any fractional portion or remainder dropped. A and B must be integer numbers.

Examples:

3/2 = 1.5	3 DIV 2 = 1
24/5 = 4.8	24 DIV 5 = 4
72/8 = 9.0	72 DIV 8 = 9

MOD (modula) remainder

The MOD operator returns the remainder obtained by dividing its two operands. Both operands must be integer numbers.

3 MOD 2 = 1	remainder = 1
24 MOD 5 = 4	remainder = 4
72 MOD 8 = 0	remainder = 0

6.4 Relational operators

This table shows the available relational operators.

Operator	Operation
EQ or =	equal
NE or <>	not equal
LT or <	less than
GT or >	greater than
LE or <=	less or equal
GE or >=	greater or equal

6.5 Function operators

A function call is specified by the function name (e.g. SIN, ATAN, . . .) followed by the function argument in brackets. When a function is used for a coordinate position it must be contained in brackets.

```
Examples:  X [SIN [45]]
           Y [ATAN[1/2]]
           Z [SQRT[9]]
```

A function returns a value and can be used interchangeably anywhere that a decimal value is accepted. Each function has one argument which can be specified using any of the four operand types.

The functions supported are:

SIN - returns the sine of the argument

SIN [90] = 1

COS - returns the cosine of the argument

COS [180] = 1

TAN - returns the tangent of the argument

TAN [135] = -1

ATAN - returns the arctangent of the argument. The argument must be specified in fractional form (e.g. 1/2, 2/1, 5/6, ...).

ATAN [1/1] = 45

SQRT - Returns the square root of the argument

SQRT [9] = 3

ABS - Returns the absolute value of the argument

ABS [-15] = 15

INT - returns the integer part of the argument

INT [5.5099] = 5

```
ROUND - rounds a decimal value to an integer value.
        Values halfway in-between are rounded up.

        ROUND [2.3] = 2
        ROUND [7.88] = 8
        ROUND [1.5] = 2
        ROUND [-1.5] = 1
```

6.6 Mathematic expressions

Expressions are made up of arithmetic operators and operands. Any combination of the previously described operands may be used to define an expression. See section on Expressions for detailed information.

```
Examples: X[SIN[P123]*COS[P124]]
          Y[2.5+[P2/P3]*SQRT[P4]]
          Z[[P2DIV3]+[P2MOD3]]
```

6.7 Conditional statements

The Centurion V supports two types of conditional statements. These statements are used to transfer control of a program from one point to another based on some condition generated in the program. The first such statement is the **IF THEN** statement.

The IF THEN statement is a way of conditionally executing a block if the results of an expression evaluates to true. The expression must contain one of the relational operators which allows the expression to be reduced to either true or false. If the expression is true, the THEN portion of the IF statement is executed. If the expression is false, the next line of the program is executed.

Example of General Form

```
IF [any         ] (relational) [any         ] then  any action
   [mathematical] (operator  ) [mathematical]         (opt.)
   [expression  ]              [expression  ]

     N20  IF P1 LT P2 THEN GOTO 15
     N21
                      or

     N22  IF P1 LT P2 GOTO 15
     N21
```

The above two statements do the same thing. If the statement is true, N15 is executed; if false N21 is executed.

```
Examples:  IF [P1*P3/COS[P90]] GE [TAN[P6]] THEN X1

           IF [P4/P3] LT [P6] GOTO 25

           IF P1 = P2 THEN P4 = P5 - P6
```

The second type of conditional statement is the **WHILE-WEND** statement. A WHILE statement contains an expression that controls the repeated execution of the blocks contained between the WHILE and WEND statements.

The expression controlling the repetition must contain one of the set of relational operators which allows the expression to be reduced to either true or false. The expression is evaluated before the contained blocks are executed. The contained blocks are executed repeatedly as long as the expression is true. If the expression is false at the beginning, the blocks are not executed at all.

```
Example:  N20  WHILE [[P2*P3]/COS[P6]] LT P2
          N21  P6 = P6 + 1
          N22  Y[P2] Z[P3]
          N23  X[P6]
          N24  X1 Y0 Z0
          N25  WEND
          N26  M30
```

In this example lines N20 thru N25 will be repeated until the WHILE expression becomes false. Then line N26 will be executed instead of N21.

6.8 Transfer statements

These statements transfer control from one section of a program to another. They are unconditional transfers in that when the statement is executed, control always transfers. The GOSUB, RETURN and CALL statements return control to the N+1 block after they are finished, and the GOTO statement transfers control to the specified block without a return.

6.9 GOTO statement

A GOTO statement transfers program execution to the block prefixed by the block label referenced in the GOTO statement.

```
GOTO 30   (The next block executed is block N30.)
```

6.10 CALL statement

A CALL statement transfers control to any program residing in the CNC's memory. Upon completion of the called program or an M99, control is returned to the main program at the block immediately following the CALL statement.

The CALL format is as follows:

```
CALL      XXXX       LXX
         Program   Loop Count
          Number   (Optional)
```

If the L is omitted the called program will be executed once. The call statement is the same as an M98.

6.11 GOSUB and RETURN

A GOSUB transfers program execution to the block number specified in the GOSUB statement. Execution will continue until a block containing a RETURN statement is encountered. The RETURN will transfer control back to the block immediately following the GOSUB statement. To use a GOSUB statement, the called block number must be part of the same program. Generally the subroutines would be at the end of the main program.

The GOSUB format is as follows:

```
GOSUB       XXXX       LXX
           Line #    Loop Count
                     (Optional)
```

If the L is omitted the GOSUB routine will be executed once.

```
N1
N2
N3
N4 GOSUB 100                  Main Program
N5
 .
 .
 .
N90 M30
N100
N101
 .
 .                            Subroutine
 .
N200
N201 RETURN
N202
```

When the GOSUB is executed in N4 the program will jump to N100 and start executing until N201 is reached. At N201 control will transfer to N5 and lines N5 thru N90 will be executed. The M30 will terminate the main program and keep lines 100 thru 202 from being executed.

Parametric Program to Cut One
60° Segment of a Fan Blade

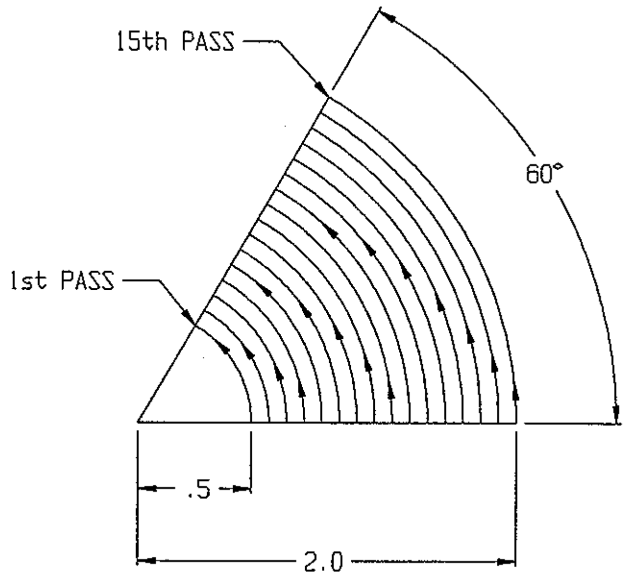


Figure 15
Parametric Program of a Fan Blade

```
N1    P10=15 P11=0 P17=0 P6=60 P8=.5
N2    P9=[.1*P11] P13=P8*SIN[P6]
N3    P14=P8*COS[P6] P16=P8
N4    X[P16] Y[P17] Z0
N5    G3 R[P8] XCO YCO X[P14] Y[P13] Z[P9]
N6    G2 R[P8] XCO YCO X[P16] Y[P17] Z0
N7    G1 X[P16]
N8    P11=P11+1 P8=P8+1
N9    IF P11 LT P10 GOTO 2
N10   Z.1
N11   X0 Y0
N12   M30
```

In the above example the tool makes 15 passes to make a blade 2" long with Z going down 1.4" from one tip to the other. The blade sweeps 60°. This program can be modified and repeated six times to complete the fan.

P6	is the angle it sweeps
P8	is the dynamic radius that increases from .5 by .1 each pass
P9	is the dynamic Z depth that goes from 0 to 1.4"
P10	is the number of times for looping = 15
P11	is the current # of loops, goes from 0-15
P13	X end point at end of CCW arc
P14	Y end point at end of CCW arc
P9	Z end point at end of CCW arc
P16	X end point at end of CW arc
P17	Y end point at end of CW arc

Looking at the program it can be seen that N1 is an initialization line setting the parameters to their starting values. Lines N2 and N3 are the equations needed to describe the fan blade. N4 thru N17 use the values calculated in lines N2 and N3 to move the machine. N8 changes the values in the equations so the next pass can be calculated. N9 tests for the end of the loop and jumps back to N2 if the 15 loops have not been completed. N10 and N11 position the tool back to the start point.

6.12 Computational Functions

1. Tangent Arc [TANA]
2. Tangent Line [TANL]
3. 3 Point Circle Generate [CGEN]

These three functions can be used anywhere throughout a program to solve various intersection problems. These functions receive input data in parameters P90 thru P99, and return the answer in parameters P80 through P89. The answers can then be used in line and circle commands to produce the desired results. The format for these three functions follows.

GENERAL FORMAT FOR TANA, TANL

Input Parameters

P90	= X1	center of arc 1
P91	= Y1	center of arc 1
P92	= R1	radius of arc 1
P93	= X2	center of arc 2
P94	= Y2	center of arc 2
P95	= R2	radius of arc 2
P96	= R3	radius of tangent arc (use in TANA only)

Calculated Output Parameters

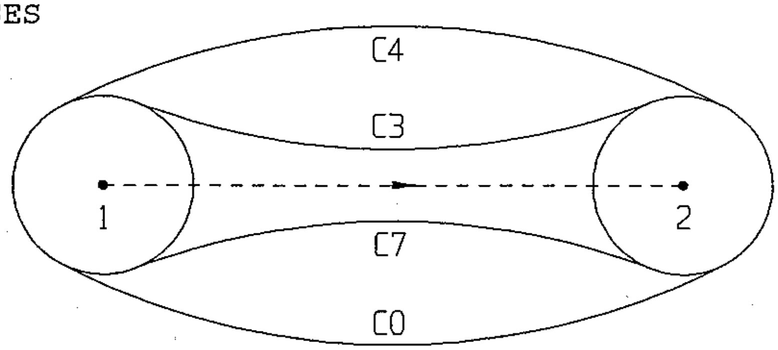
P80 = Xs	X starting point of tangent arc or line
P81 = Ys	Y starting point of tangent arc or line
P82 = Xe	X end point of tangent arc or line
P83 = Ye	Y end point of tangent arc or line
P84 = Xt	XC center of tangent arc (TANA case only)
P85 = Yt	YC center of tangent arc (TANA case only)

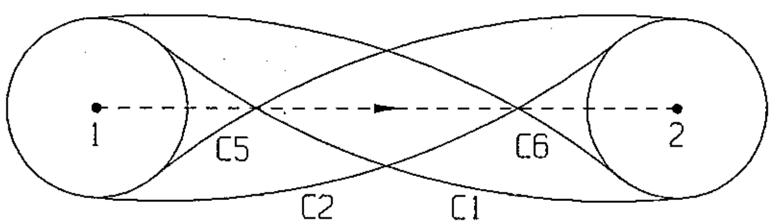
The general form for a tangent arc or line function is

TANA CX or TANL CX

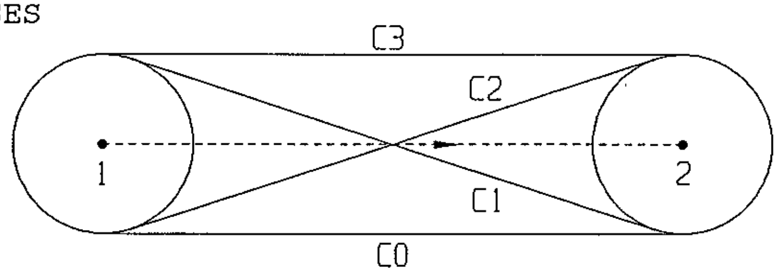
when X is a number 0 through 7 in the tangent arc case, and 0 through 3 in the tangent line case. This number selects one of the eight possible solutions of the TANA or one of the four solutions of the TANL. The values of CX are defined as the tangent point being to the right or left of a line connecting the centers of the arcs when facing in the direction of tool movement. See the following diagrams.

TANA CASES





TANL CASES



The Circle Generate function will calculate the center and radius of an arc through any three non-co-linear points. The general format for the CGEN function is as follows:

Input Parameters

```
P90=X1 P91=Y1  coordinates of first point
P92=X2 P93=Y2  coordinates of second point
P94=X3 P95=Y3  coordinates of third point
```

Output Parameters

```
P80=XC P81=YC  center of calculated circle
P82=R          radius of calculated circle
```

Sample Program using CGEN

```
N1    P90=0 P91=0     first point
N2    P92=1 P93=1     second point
N3    P94=2 P95=0     third point
N4    CGEN
N5    G1 X0 Y0        position to start of arc
N6    G17 G2 R[P82] XC[P80] YC[P81] Xe____Ye____
                                 determined by next line
                                      of program
```

The CGEN function can be used anytime throughout the program to calculate the radius and center of an arc. These calculations can then be used in a normal arc command along with trig help, chamfer, corner round, extend back and any other function available in the Centurion V control.

Sample Program Using TANA or TANL

```
N1    P90=0                 XC of arc 1
N2    P91=0                 YC of arc 1
N3    P92=1.5               radius of arc 1
N4    P93=5                 XC of arc 2
N5    P94=4                 YC of arc 2
N6    P95=2                 radius of arc 2
N7    P96=5                 radius of tangent arc
                            (not used for tangent line)
N8    TANA C3 or TANL C3
N9    G2 R1.5 XC0 YC0 X[P80] Y[P81]

             TANA or TANL calculated end points

N10   G2 R5 XC[P84] YC[P85] X[P82] Y[P83]

             TANA calculated center and end points
   or

N10   G1 X[P82] Y[P83]

      TANL calculated end point

N11   G2 R2 XC5 YC4 Xend Yend

           depends on next line of program
```

7. CONVERSATIONAL INPUT SCREENS

This section contains diagrams of the conversational input screens, a small explanation of each screen, and the M-G code output which each screen will generate.

The setup screen will always appear at the beginning of every program. It initializes certain important functions so no settings remain from a previously run program.

7.1 Program setup

```
Program Setup

  Dimensions   [ABSOLUTE]
  Units        [ENGLISH]
```

CONVERSATIONAL SCREEN

Program Setup

will produce these M-G codes.

GENERATED CODE FOR PROGRAM SETUP

```
G20  G40  G50  G80  G90  H0  D0
```

- D0 — Set tool radius offset to zero
- H0 — Set tool length offset to zero
- G90 — Absolute dimensioning mode
- G80 — Cancel canned cycle
- G50 — Turn off scaling
- G40 — Cancel cutter compensation
- G20 — English (Inch) input

The underlined entries are codes which depend directly on screen input. Those not underlined are added to simplify programming and will remain unchanged. For example, if the dimensions field was toggled from Absolute to Incremental, the G90 would change to a G91.

Not all possible combinations of screen input are shown, so if additional information on any particular screen or field is required the appropriate section of the manual should be referenced.

Although the main history line does not actually change as the conversational menu keys are pressed, a history line will be shown along with each screen to describe the sequence of keys pressed to reach that screen from the main conversational menu:

F1 POS	F2 MILL	F3 DRILL	F4 BOLT	F5 TCHNG	F6 MISC	F7 CALL	F8 SPEC	F9	F10 EXIT	

POSITION KEY

7.2 F1 POS (POS)

The position screen will normally be used to do rapid positioning; however, feed moves may be made by toggling the feedrate field and entering a feedrate.

```
Position

  Feedrate     [RAPID]
  Coordinates  [CARTESIAN]

  X-axis  [1        ]
  Y-axis  [1        ]
  Z-axis  [-2       ]
```

CONVERSATIONAL SCREEN

Cartesian Rapid Positioning

GENERATED CODE FOR POSITION

```
G00  X1  Y1  Z-2
 |        └──────── Axes moves
 └───────────────── Rapid positioning
```

See Section 3.1.1 on page 3-2.

```
Position

  Feedrate     [12       ]
  Coordinates [POLAR]

  Plane  [XY]
  Radius [1.4142    ]
  Angle  [45        ]

  Z-axis [-2        ]
```

CONVERSATIONAL SCREEN

Polar Feed Positioning

GENERATED CODE FOR POSITION

```
G01  F12  G17  R1.4142  AB45  Z-2
                                   Axis move
                                   Polar angle
                                   Polar radius
                                   XY Plane
                                   Feedrate
                                   Linear interpolation
```

See Section 3.1.2.1 on page 3-4.

MILL KEY

7.3 F2 MILL (MILL)

The F2 MILL selection brings up the following softkeys:

F1 START	F2 GEOM	F3 MISC	F4 END	F5 POCK	F6 FRAME	F7 3DPKT	F8 CAD		F10 EXIT	ESC BACK

7.3.1 F1 START (MILL-START)

This screen is used to begin a continuous, single or multi-depth, milling cycle. Milling will start at the first Z depth specified and continue stepping down by the Z increment until the final Z depth has been reached. If cutter compensation is turned on, then entry of a point before the pierce point will be required. See the section on cutter compensation for a full explanation.

```
Tool Pierce - Start Mill Cycle

  Z Pierce Feedrate   [10        ]
  Clearance           [.1        ]
  Final Z depth       [-1        ]
  1st Z depth         [-.25      ]
  Z Increment         [.25       ]

  X Pierce Point      [0         ]
  Y Pierce Point      [0         ]

  Compensation        [OFF]
```

CONVERSATIONAL SCREEN

Tool Pierce
Start Mill Cycle

GENERATED CODE FOR START MILL CYCLE

```
P140 = .1       Set clearance
P141 = -1       Set final Z depth
P143 = .25      Set Z increment
P144 = -.25     Set 1st Z depth
P145 = 10       Set Z feedrate
P160 = P144     Set current Z depth to 1st Z depth
P161 = P140     Set rapid plane to clearance
P162 = 2        Loop control variable (P162) will be 2 when
                final Z depth milling is complete
IF P160 EQ P141 THEN P162=1   If current Z depth is equal to
                              final Z depth then loop control
                              variable will be 1, which means
                              do the mill cycle 1 time at the
                              final Z depth.
IF P160 GT P141 THEN P162=0   If current Z depth is greater
                              than final Z depth, then loop
                              control variable will be 0,
                              which means Z will have to be
                              incremented.
WHILE P162 LT 2               The beginning of the milling
                              loop.  Z will increment down
                              until milling is complete at
                              the final Z depth.
G40  G00  X00  Y00       Rapid to pierce point
 |    |_________________ Rapid positioning
 |______________________ Cancel cutter compensation
Z[P161]                  Rapid Z to rapid plane
G01 F[P145] Z[P160]      Feed Z to current Z depth
```

The mill geometry will now be cut at the current Z depth. The end mill screen will explain what occurs at the end of each loop.

If cutter compensation is turned on, as is the case in the screen below, the resulting output will be identical to the previous with the exception of 2 lines.

```
Tool Pierce - Start Mill Cycle

  Z Pierce Feedrate   [10       ]
  Clearance           [.1       ]
  Final Z Depth       [-1       ]
  1st Z Depth         [-.25     ]
  Z Increment         [.25      ]

  X Pierce Point      [0        ]
  Y Pierce Point      [0        ]

  Compensation        [LEFT]
  X Before Pierce     [1        ]
  Y Before Pierce     [0        ]
```

CONVERSATIONAL SCREEN

Tool Pierce
Start Mill Cycle

GENERATED CODE FOR START MILL CYCLE

```
No Compensation           Compensation
       .                           .
       .                           .
       .                           .
G40  G00  X0  Y0    →    G41  G65  X1  Y0    Point before pierce
       .                  |    |____________ Non-movement
       .                  |_________________ Left compensation
       .                  G00  X0   Y0       Rapid to pierce point
                                   .
                                   .
                                   .
```

See Section 3.12.2 on page 3-43.

7.3.2 F2 GEOM (MILL-GEOM)

The GEOM selection brings up the following softkeys:

F1 LINE	F2 ARC	F3 TANGS	F4 CGEN						F10 EXIT	ESC BACK

7.3.2.1 F1 LINE (MILL-GEOM-LINE)

The line screen is used to do linear interpolation in Feed mode.

```
Mill Geometry - Line

  Feedrate     [20       ]
  Coordinates [CARTESIAN]

  X-axis  [2        ]
  Y_axis  [2        ]
  Z-axis  [         ]




  End Option [---]
  Extend Back [OFF]
```

CONVERSATIONAL SCREENS

Cartesian Linear Interpolation

GENERATED CODE FOR LINE

```
G01  F20  X2 Y2
              Axes move
          Feedrate
     Linear interpolation
```

See Section 3.1.2 on page 3-4.

CONVERSATIONAL SCREENS

Polar Linear Interpolation

```
Mill Geometry - Line

  Feedrate     [20       ]
  Coordinates [POLAR]

  Plane   [YZ]
  Polar Center [ABSOLUTE]
      YC [1        ]
      ZC [-1       ]
  Radius [2        ]
  Angle  [60       ]
  X-axis [         ]

  End Option [---]
  Extend Back [OFF]
```

GENERATED CODE FOR LINE

```
G01  F20  G19  YC1 ZC-1  R2  AB60
```

- AB60 — Polar angle
- R2 — Polar radius
- YC1 ZC-1 — Polar center
- G19 — YZ plane
- F20 — Feedrate
- G01 — Linear interpolation

See Section 3.1.2.1 on page 3-4.

When Extend Back [ON] is selected, these lines will appear:

```
  Extend Back [ON]
  Extend [SHORT]  Angle [110]
```

GENERATED CODE FOR BACK LINE

```
G01 F20 G19 YC1 ZC-1 R2 AB60 BACK C[0] W110
```

- W110 — Angle of extension
- C[0] — Short/long
- BACK — Back line function
- AB60 — Polar angle
- R2 — Polar radius
- YC1 ZC-1 — Polar center
- G19 — YZ plane
- F20 — Feedrate
- G01 — Linear interpolation

See Section 3.1.6 on page 3-17.

Line w/Round Corner

```
Mill Geometry - Line

  Feedrate       [20          ]
   Coordinates  [CARTESIAN]

   X-axis   [2          ]
   Y-axis   [2          ]
   Z-axis   [           ]



  End Option [Round Corner]
      Radius [.15       ]
```

GENERATED CODE FOR ROUND CORNER

```
G01   F20   X2   Y2, R1.5
```

Round corner radius

See Section 3.1.4, page 3-17.

Line w/Chamfer

```
Mill Geometry - Line

  Feedrate       [20          ]
   Coordinates  [CARTESIAN]

   X-axis   [2          ]
   Y-axis   [2          ]
   Z-axis   [           ]



  End Option [Chamfer]
      Length [.15         ]
```

GENERATED CODE FOR CHAMFER

```
G01   F20   X2   Y2, C.15
```

Chamfer length

See Section 3.1.5, page 3-17.

CONVERSATIONAL SCREENS

7.3.2.2 F2 ARC (MILL-GEOM-ARC)

The F2 ARC screen is used to do circular interpolation in Feed mode.

```
Mill Geometry - Arc

  Plane          [XY]
  Feedrate       [20        ]
  Direction      [CW]
  Coordinates    [INC CENTER]

  Arc Center    I[1         ]
                J[-1        ]
  End Point     X[2         ]
                Y[0         ]
                Z[          ]

  End Option [---]
```

CONVERSATIONAL SCREENS

XY Plane
Incremental Center
CW Circular Interpolation

GENERATED CODE FOR ARC

```
G17   G02   F20   I1 J-1   X2 Y0
```

- X2 Y0 — Arc end point
- I1 J-1 — Incremental arc centerpoint
- F20 — Feedrate
- G02 — CW interpolation
- G17 — XY plane

See Section 3.1.3 on page 3-6.

```
Mill Geometry - Arc

  Plane          [ZX]
  Feedrate       [10        ]
  Direction      [CCW]
  Coordinates    [ABS CENTER]
  Arc Radius    R[2         ]
  Arc Center   ZC[0         ]
               XC[2         ]
  End Point     Z[0         ]
                X[4         ]
                Y[          ]

  End Option [---]
```

CONVERSATIONAL SCREENS

ZX Plane
Absolute Center
CCW Circular Interpolation

GENERATED CODE FOR ARC

G18 G03 F10 R2 ZC0 XC2 Z0 X4

- Z0 X4 — End point
- ZC0 XC2 — Absolute center
- R2 — Arc radius
- F10 — Feedrate
- G03 — CCW interpolation
- G18 — ZX plane

See Section 3.1.3 on page 3-7.

```
Mill Geometry - Arc

  Plane           [XY]
  Feedrate        [15         ]
  Direction       [CCW]
  Coordinates     [POLAR]
  Arc Radius     R[3          ]

  Start Angle   AA[45         ]

                 Z[.5         ]

  End Option [Round Corner]
              Radius [.25        ]
```

CONVERSATIONAL SCREENS

XY Plane
Polar
CCW Helical Interpolation
w/Round Corner

GENERATED CODE FOR ARC

```
G17  G03  F15  R3  AA45  AB135  Z.5, R.25
                                      Round corner
                                      Helical axis
                                      End angle
                                      Start angle
                                      Radius
                                      Feedrate
                                      CCW interpolation
                                      XY plane
```

See Section 3.1.3 on page 3-14.

```
Mill Geometry - Arc

  Plane           [XY]
  Feedrate        [           ]
  Direction       [CCW]
  Coordinates     [RADIUS ONLY]
  Arc Radius     R[2          ]

  End Point      X[2          ]
                 Y[0          ]
                 Z[           ]

  End Option [---]
```

CONVERSATIONAL SCREENS

XY Plane
Radius Only
CCW Circular Interpolation

GENERATED CODE FOR ARC

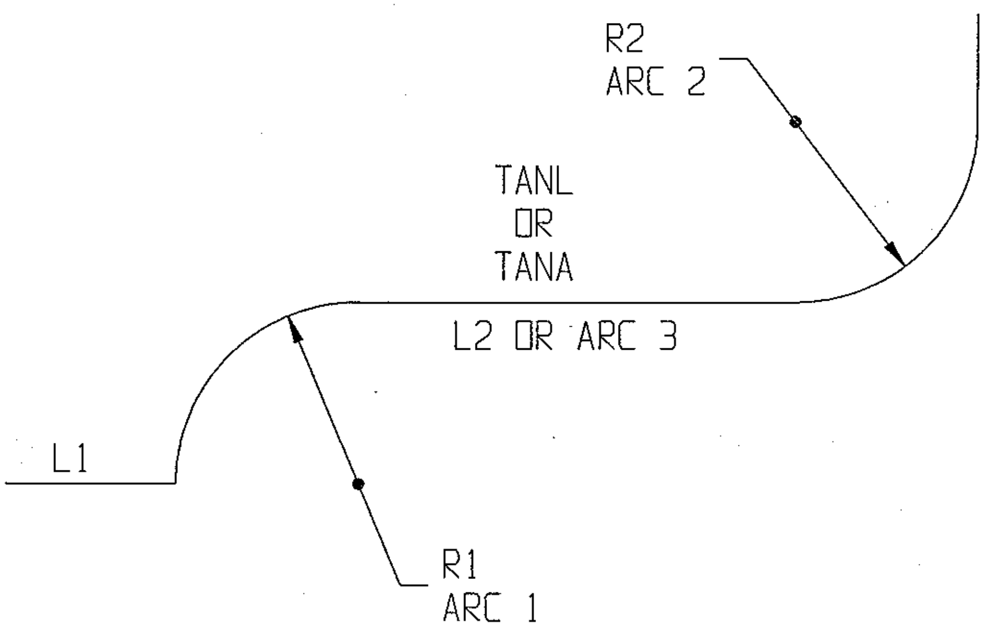
```
G17  G03  R2  X2 Y0
                    End point
                    Radius
                    CCW interpolation
                    XY plane
```

See Section 3.1.3 on page 3-8.

7.3.2.3 F3 TANGS (MILL-GEOM-TANGS)

The TANGS (Tangents) screen is used to compute the intersection points necessary for a tangent arc or tangent line between two arcs. When this function is used the first arc and the tangent line or arc will be entered into the program. The second arc information will only be used for calculation purposes. This was done to enable a series of tangent lines or arcs to be programmed consecutively. Therefore, a TANL and TANA command would normally be followed with an arc command describing the second arc.

To determine the value of the right or left entries on these screens, draw a line connecting the centers of the two arcs in the direction of tool movement. Then determine if the desired points are to the right or left of this line and enter these values.



The general sequence for the above shape would be as follows:

Event 1	Line L1
Event 2	Tangent line or arc function describing arc R1 and line L2 or arc 3
Event 3	Arc R2

7.3.2.3.1 Tangent Line

```
Connect two arcs with tangent line
 or arc in the plane [XY]

Mill first arc in direction   [ CW]
R1   [1.5     ]
XC1  [00.0000]    YC1 [00.0000]

Second arc for computation is:
R2   [2       ]
XC2  [5.0000 ]   YC2  [4.0000   ]

Exit first arc [ LEFT] and enter
 second arc [ LEFT]
Connect with [A LINE]
```

CONVERSATIONAL SCREEN

Tangent Line

GENERATED CODE FOR TANGENT LINE

```
N1    P90=0                  XC of arc 1
N2    P91=0                  YC of arc 1
N3    P92=1.5                radius of arc 1
N4    P93=5                  XC of arc 2
N5    P94=4                  YC of arc 2
N6    P95=2                  radius of arc 2
N7    P96=5                  radius of tangent arc
                             (not used for tangent line)
N8    TANL C3
                             Tangent line type
                             Tangent line function

N9    G17 G02 R[1.5] XC[0] YC[0] X[P80] Y[P81]
                                                End point of
                                                 arc calculated
                                                 by TANL func.
                                                Absolute center
                                                Radius
                                                CW interp.
                                                XY plane

N10   G01 X[P82] Y[P83]
                             End point of line calculated by TANL
                             Linear interpolation
```

See Section 6.12 on page 6-8.

7.3.5.1 F1 SETUP (MILL-POCK-SETUP)

This screen is used to set parameters necessary for the circular and rectangular pocket routines. It must be done prior to any pocket clearing routines.

Mill START and END are not to be used with pocket routines.

<u>ALL MILLING AUTOROUTINES MUST BE ACTIVATED WITH THE TOOL AT THE CENTER OF THE ROUTINE.</u>

```
Pocket Mill Setup

  XY Feedrate          [20        ]
  Z Pierce Feedrate    [10        ]
  Clearance            [.1        ]
  Final Z depth        [-1        ]
  Z Increment          [.25       ]
```

CONVERSATIONAL SCREEN

Pocket Mill Setup

GENERATED CODE FOR POCKET MILL SETUP

```
G99  F20
 |    └──── XY Feedrate
 └───────── Canned cycle rapid plane return
P140 = .1   Set clearance
P141 = -1   Set final Z depth
P143 = .25  Set Z increment
P145 = 10   Set Z feedrate
```

See Section 3.11.1 on page 3-24.

7.3.2.3.2 Tangent Arc

```
Connect two arcs with tangent line or arc
 in the plane [XY]

Mill first arc in direction  [ CW]
R1   [1.5     ]
XC1 [00.0000]  YC1 [00.0000]

Second arc for computation is:
R2   [2.0000 ]
XC2 [5.0000 ]  YC2 [4.0000 ]

Exit first arc [ LEFT] and enter
 second arc [ LEFT]

Connect with [AN ARC] Center to the [LEFT]
 Radius [5.0000] Arc direction [ CW]
```

CONVERSATIONAL SCREEN

Tangent Arc

GENERATED CODE FOR TANGENT ARC

```
N1    P90=0          XC of arc 1
N2    P91=0          YC of arc 1
N3    P92=1.5        radius of arc 1
N4    P93=5          XC of arc 2
N5    P94=4          YC of arc 2
N6    P95=2          radius of arc 2
N7    P96=5          radius of tangent arc
                     (not used for tangent line)
N8    TANA C3
                     Tangent arc type
                     Tangent arc function

N9    G17 G02 R[1.5] XC[0] YC[0] X[P80] Y[P81]
                     End point of
                      arc calculated
                      by TANA func.
                     Absolute center
                     Radius
                     CW interp.
                     XY plane

N10   G17 G02 R[5] XC[P84] YC[P85] X[P82] Y[P83]
                     End pt. tangent
                      arc calculated
                      by TANL func.
                     Center tangent
                      arc
                     Radius of
                      tangent arc
                     CCW interp.
                     XY plane
```

See Section 6.12 on page 6-8.

7.3.2.4 F4 CGEN (MILL-GEOM-CGEN)

To use the Circle Generator function simply fill in any three points on an arc. These three points will be used to compute the center and radius of the specified arc.

```
Circle Generator

  Plane [XY]
  Direction [ CW]

  X1 [          ]  Y1 [          ]
  X2 [          ]  Y2 [          ]
  X3 [          ]  Y3 [          ]

 Use X3 Y3 as end point  [ NO]

 End angle  [          ]
```

CONVERSATIONAL SCREEN

Circle Generator

GENERATED CODE FOR CIRCLE GENERATOR

```
P90=[   ]  P91=[   ]               XY point 1
P92=[   ]  P93=[   ]               XY point 2
P94=[   ]  P95=[   ]               XY point 3
CGEN                               Circle generator function
G17 G2 R[P82] XC[P80] YC[P81] X[ ] Y[ ]
                                       End point determined
                                        by next program line
                                       Center point calc.
                                        by CGEN function
                                       Radius calc. by CGEN
                                       CW interpolation
                                       XY plane
                 or

G17 G2 R[P82} XC[P80] YC[P81] AB[ ]
                                       End angle determined
                                        by screen entry
```

See Section 6.12 on page 6-8, 6-10.

7.3.3 F3 MISC (MILL-MISC)

See explanation under Miscellaneous on page 7-39.

7.3.4 F4 END (MILL-END)

This screen is used to end a previously started mill cycle. An end mill cycle without a start mill cycle, or a start mill cycle without an end mill cycle, will generate a syntax error when the program is run or verified.

COMMON ERROR CODES FOR MILLING CYCLES

Start Mill Cycle	and **NO**	Tool Retract End Mill Cycle	= **ERROR # 602** Missing WEND Statement

This may be caused by a Start Mill Cycle without an End Mill Cycle.

Tool Retract End Mill Cycle	and **NO**	Start Mill Cycle	= **ERROR # 601** Missing WHILE Statement

This may be caused by an End Mill Cycle without a Start Mill Cycle.

```
Tool Retract
End Mill Cycle

  Point On Part
  After Tool Retract  X[0         ]
                      Y[1         ]
```

CONVERSATIONAL SCREENS

Tool Retract
End Mill Cycle

GENERATED CODE FOR END MILL CYCLE

```
G00  Z[P140]
 |      └──── Z to clearance
 └─────────── Rapid positioning

G65  X0 Y1
 |      └──── Point after tool retract
 └─────────── Non-movement

G40
 └─────────── Cancel compensation
```

P161 = P160 + .1 — Set rapid plane to .1 above current Z depth

IF P162 EQ 0 THEN P160 = P160 - ABS[P143]
If loop control variable (P162) =0 then final Z depth has not been reached, so subtract Z increment (P143) from current Z depth (P160).

IF P160 LT P141 THEN P160 = P141
If current Z depth (P160) is now below final Z depth (P141) then set current Z depth to the final Z depth.

IF P160 EQ P141 THEN IF P162=1 THEN P162=2
If current Z depth is at the final Z depth then check loop control variable (P162) to see if milling has been done at the final Z depth (P162=1). If it has, then stop mill cycle (P162=2).

IF P160 EQ P141 THEN IF P162=0 THEN P162=1
If current Z depth is at the final Z depth then check loop control variable

	(P162) to see if milling has not been done at the final Z depth (P162=0). If it hasn't, then tell loop to run one more time at final Z depth (P162=1).
WEND	End of mill cycle. Matches WHILE in start mill cycle.
G01	Linear feed interpolation

See Section 3.12.2 on page 3-43.

7.3.5 F5 POCK (MILL-POCK)

The Pocket Mill selection brings up the following softkeys:

F1 SETUP	F2 CIRC	F3 RECT							F10 EXIT	ESC BACK

7.3.5.2 F2 CIRC (MILL-POCK-CIRC)

The F2 CIRC selection brings up the following softkey selections:

F1 CLEAR	F2 F IN								F10 EXIT	ESC BACK

7.3.5.2.1 F1 CLEAR (MILL-POCK-CIRC-CLEAR)

```
Circular Pocket Clear

  Pocket Radius        [4         ]
  XY Finish Stock      [.01       ]
  Z Finish Stock       [.01       ]
  Cut Width            [-.25      ]

  Cut Direction [CW]
  Compensation  [ON]
```

CONVERSATIONAL SCREEN

CW Circular Pocket Clear

GENERATED CODE FOR CIRCULAR CLEAR

```
P150 = 4        Set pocket radius
P153 = .01      Set XY finish stock
P154 = .01      Set Z finish stock
P155 = .25      Set cut width
G24   G42   G2
 |     |     └── CW circular interpolation
 |     └──────── Right cutter compensation
 └────────────── Circular pocket clear
```

See Section 3.11.1 on page 3-24.

7.3.5.2.2 F2 FIN (MILL-POCK-CIRC-FIN)

```
Circular Finish Inside

  Pocket Radius      [4        ]

  Cut Direction [CW]
  Compensation  [ON]
```

CONVERSATIONAL SCREEN

Inside CW Circular Pocket Finish

GENERATED CODE FOR INSIDE CIRCLE FINISH

```
P150 = 4       Set pocket radius
P153 = 0       Set XY finish stock to zero
P154 = 0       Set Z finish stock to zero
G25  G42  G2
               CW circular interpolation
               Right cutter compensation
               Inside circular pocket finish
```

See Section 3.11.2 on page 3-25.

7.3.5.3 F3 RECT (MILL-POCK-RECT)

The F3 rectangular pocket selection brings up the following softkeys:

F1 CLEAR	F2 F IN								F10 EXIT	ESC BACK

7.3.5.3.1 F1 CLEAR (MILL-POCK-RECT-CLEAR)

```
Rectangular Pocket Clear

  X Pocket Dimension [2         ]
  Y Pocket Dimension [4         ]
  XY Finish Stock    [.01       ]
  Z Finish Stock     [.01       ]
  Cut Width          [.25       ]
  Corner Radius      [.5        ]

  Compensation  [ON]
  Cut Direction [CW]
```

CONVERSATIONAL SCREEN

CW Rectangular Pocket Clear

GENERATED CODE FOR RECTANGULAR POCKET CLEAR

```
P150 = .5      Set corner radius
P151 = 2       Set X pocket dimension
P152 = 4       Set Y pocket dimension
P153 = .01     Set XY finish stock
P154 = .01     Set Z finish stock
P155 = .25     Set cut width
G34  G42  G2
               CW interpolation
               Right cutter compensation
               Rectangular pocket clear
```

See Section 3.11.7 on page 3-30.

7.3.5.3.2 F2 F IN (MILL-POCK-RECT-F IN)

```
Rectangular Finish Inside

   X Pocket Dimension [2         ]
   Y Pocket Dimension [4         ]
   Corner Radius      [5         ]

   Compensation  [ON]
   Cut Direction [CW]
```

CONVERSATIONAL SCREEN

Inside CW Rectangular Pocket Finish

GENERATED CODE FOR INSIDE RECTANGULAR POCKET

```
P150 = .5      Set corner radius
P151 = 2       Set X pocket dimension
P152 = 4       Set Y pocket dimension
P153 = 0       Set XY finish stock to zero
P154 = 0       Set Z finish stock to zero
G35  G42  G2
 |    |    └── CW circular interpolation
 |    └─────── Right cutter compensation
 └──────────── Inside rectangular pocket finish
```

See Section 3.11.8 on page 3-31.

7.3.6 F6 FRAME (MILL-FRAME)

The F6 frame mill selection brings up these softkeys:

F1 SETUP	F2 CIRC	F3 RECT							F10 EXIT	ESC BACK

7.3.6.1 F1 SETUP (MILL-FRAME-SETUP)

This screen is used to set parameters necessary for circular and rectangular frame mill routines. It must be done prior to any frame milling routines.

Mill START and END are not to be used with frame routines.

<u>ALL MILLING AUTOROUTINES MUST BE ACTIVATED WITH THE TOOL AT THE CENTER OF THE ROUTINE.</u>

```
Frame Mill Setup

  XY Feedrate         [20      ]
  Z Pierce Feedrate   [10      ]
  Clearance           [.1      ]
  Final Z Depth       [-1      ]
  Z Increment         [.25     ]
```

CONVERSATIONAL SCREEN

Frame Mill Setup

GENERATED CODE FOR MILL SETUP

```
G99  F20
|    └──── XY feedrate
└───────── Canned cycle rapid plane return
P140 = .1   Set clearance
P141 = -1   Set final Z depth
P143 = .25  Set Z increment
P145 = 10   Set Z feedrate
```

See Sections 3.11.3 and 3.11.8 on pages 3-26 and 3-31.

7.3.6.2 F2 CIRC (MILL-FRAME-CIRC)

```
Circular Finish Outside

  Pocket Radius      [4        ]

  Compensation  [OFF]
  Cut Direction [CCW]
```

CONVERSATIONAL SCREEN

Outside CCW Circular Frame Mill

GENERATED CODE FOR CIRCULAR FRAME

```
P150 = 4       Set circle radius
P153 = 0       Set XY finish stock to zero
P154 = 0       Set Z finish stock to zero
G26  G40  G3
 |    |    └── CCW circular interpolation
 |    └─────── Cancel cutter compensation
 └──────────── Circular frame mill
```

See Section 3.11.3 on page 3-26.

7.3.6.3 F3 RECT (MILL-FRAME-RECT)

```
Rectangular Finish Outside

  X Pocket Dimension [2        ]
  Y Pocket Dimension [4        ]
  Corner Radius      [.05      ]

  Compensation  [OFF]
  Cut Direction [CCW]
```

CONVERSATIONAL SCREEN

Rectangular Finish Outside

GENERATED CODE FOR RECTANGULAR FRAME

```
P150 = .5       Set corner radius
P151 = 2        Set X frame dimension
P152 = 4        Set Y frame dimension
P153 = 0        Set XY finish stock to zero
P154 = 0        Set Z finish stock to zero
G36  G40  G3
           └──  CCW circular interpolation
      └───────  Cancel cutter compensation
 └────────────  Rectangular frame mill
```

See Section 3.11.9 on page 3-32.

7.3.7 F7 3D POCKET (MILL-3DPKT)

The F7 3DPKT selection brings up the following menu:

F1 START	F2 END								F10 EXIT	ESC BACK

7.3.7.1 F1 START 3D Sweep Cycle (MILL-3DPKT-START)

The F1 START key brings up the starting menu.

```
Start 3D sweep cycle

   Clearance              [.1        ]
   Z Pierce Feedrate      [5         ]
   Arc Feedrate           [10        ]
   Start Point           X[0         ]
                         Y[1         ]
                         Z[-.2       ]
   Sweep Start Radius    R[1         ]
   Sweep Start Angle    AA[-.0001    ]
   Sweep End Angle      AB[180       ]
   Pass Width             [.05       ]
   Sweep Plane       [YZ]
   Cutter Comp       [ON]
```

CONVERSATIONAL SCREEN

Start 3D Sweep Cycle

GENERATED CODE FOR 3D SWEEP CYCLE

```
G40
P145 = 5                                Set Z pierce feedrate
F10                                     Set arc feedrate
P127 = 1                                Set start radius
P128 = -.0001                           Set start angle
P129 = 180                              Set end angle
P130=.05                                Set pass width
P140 = .1                               Set clearance
G31                                     Move to clearance
G65 G1 X0 Y1 Z-.2                       (XYZ
P121=P208 P122=P209 P123=P210             start point)
P120 = 9                                Set YZ plane
P167 = 1                                Cutter comp on
M94                                     Turn on sweep cycle
```

Clearance	"P140 = X.XXXX"	Z picks up to this depth after each pass
Z pierce feedrate	"P145 = XXX.X"	Z feeds into the part at this feedrate
Arc feedrate	"FXXX.X"	Arc sweeps (in XZ or YZ) at this feedrate
Start point	"P121=X start position P122=Y start position P123=Z start position"	Arc sweeps start at this point of the XY contour
Sweep start radius	"P127=XX.XXXX"	Radius of arc at the start point of the XY contour

Sweep start angle	"P128=XX.XXXX"	Start angle of arcs: If the start angle $\geq 0°$, a male part is made. If the start angle $< 0°$, a female part is made.
Sweep end angle	"P129=XX.XXXX"	End angle of arcs, always $< 180°$
Pass width	"P130=XX.XXXX"	If sweep is in XZ plane, this is the increment. If sweep is in YZ plane, this is the increment.
Sweep plane	"P120=X"	(=8 is XZ plane; $\neq 8$ is YZ plane)
Cutter comp	"P161=X"	(= 0 is cutter comp off; $\neq 0$ is cutter comp on)

Notes: *If sweep start angle = sweep end angle, then no arc is made.*

If sweep plane is XZ and there is no change in the Y position, then no arc is made.

If sweep plane is YZ and there is no change in the X position, then no arc is made.

After the Start 3D screen is completed, the next step is to program the desired contour using <u>only lines and arcs</u>. These should be entered through the MILL-GEOM-ARC or MILL-GEOM-LINE screen. Once the XY profile of the part has been entered, the above cycle will sweep arcs in the specified plane along the XY programmed lines or arcs until the sweep cycle is ended. The net effect of this cycle is to rotate the programmed XY contour into the XZ or YZ plane. In the case of a bottle mold, only the bottle profile would have to be programmed in the XY plane and then rotated into the YZ or XZ plane.

7.3.7.2 F2 END 3D Sweep Cycle (MILL-3DPKT-END)

This key must be selected to terminate the 3DPKT cycle or an error will occur.

```
Disable 3D Sweep Cycle
```

CONVERSATIONAL SCREEN

Disable 3D Sweep Cycle

GENERATED CODE FOR DISABLE 3D SWEEP CYCLE

M93   Shuts off sweep cycle

DRILL KEY

7.4 F3 DRILL (DRILL)

The F3-DRILL selection brings up the following menu:

F1 START	F2 POS	F3 MISC	F4 CALL	F5 END					F10 EXIT	ESC BACK

7.4.1 F1 DRILL (DRILL-START-DRILL)

All drill cycles must be setup prior to execution and ended after the last hole. This is done with the F1-START and F5-END selections. When F1-START is selected the following softkeys appear:

F1 DRILL	F2 D/DWL	F3 PECK	F4 WPECK	F5 BORE	F6 B/DWL	F7 TAP			F10 EXIT	ESC BACK

After one of the above drill cycles is setup, the sequence of moves specified by the drill cycle will be done after each axis move until the drill cycle is ended.

CONVERSATIONAL SCREENS

F1-DRILL (DRILL-START-DRILL)

```
Enable Drill Cycle

 Z Pierce Feedrate [5          ]
 Spindle On CW RPM [2000       ]

 Clearance         [.1         ]
 Final Z Depth     [-2         ]
```

Drill

F5-BORE (DRILL-START-BORE)

```
Enable Bore Cycle

 Z Pierce Feedrate [5          ]
 Spindle On CW RPM [2000       ]

 Clearance         [.1         ]
 Final Z Depth     [-2         ]
```

Bore

GENERATED CODE FOR DRILL

```
G99  G00  G40
          Cancel compensation
          Rapid positioning
          Canned cycle rapid plane return
P140 = .1     Set clearance
P141 = -2     Set final Z depth
P145 = 5      Set Z feedrate
M03  S2000
          Spindle speed
          Spindle on CW
G81
          Enable drill cycle
```

The bore cycle setup is identical to the drill cycle with the exception of the last line:

```
        .
        .
        G85
            Enable bore cycle
```

See Section 3.23 on page 3-60.

7.4.1.1 F2 D/DWL (DRILL-START-D/DWL)
F6 B/DWL (DRILL-START-B/DWL)

Drill with Dwell

```
Enable Drill Cycle w/Dwell

 Z Pierce Feedrate [5         ]
 Spindle On CW RPM [2000]

 Clearance         [.1        ]
 Final Z Depth     [-2        ]
 Dwell             [.5        ]
```

See Section 3.23.6, page 3-69.

Bore with Dwell

```
Enable Bore Cycle w/Dwell

  Z Pierce Feedrate [5          ]
  Spindle On CW RPM [2000]

  Clearance         [.1         ]
  Final Z Depth     [-2         ]
  Dwell             [.5  ]
```

See Section 3.23.11, page 3-74.

GENERATED CODE FOR BORE

```
G99   G00   G40
 |     |     └─── Cancel compensation
 |     └───────── Rapid positioning
 └─────────────── Canned cycle rapid plane return
P140 = .1         Set clearance
P141 = -2         Set final Z depth
P145 = 5          Set Z feedrate
P148 = .5         Set dwell
M03   S2000
 |     └───────── Spindle speed
 └─────────────── Spindle on CW
G82
 └─────────────── Enable Drill/Dwell cycle
```

The bore/dwell cycle setup is identical to the drill/dwell cycle with the exception of the last line:

```
.
.
G89
 └─────────────── Enable Bore/Dwell cycle
```

7.4.1.2 F3 PECK (DRILL-START-PECK)

```
Enable Drill/Peck Cycle

  Z Pierce Feedrate [5         ]
  Spindle On CW RPM [3000]

  Clearance         [.1        ]
  Final Z Depth     [-1        ]
  First Z Depth     [-.25      ]
  Z Increment       [.25       ]
  Peck Clearance    [.01       ]
```

CONVERSATIONAL SCREENS

Peck Drilling Cycle

GENERATED CODE FOR DRILL/PECK

```
G99  G00  G40
 |    |    └──── Cancel compensation
 |    └───────── Rapid positioning
 └────────────── Canned cycle rapid plane return
P140 = .1        Set clearance
P141 = -1        Set final Z depth
P144 = -.25      Set first Z depth
P143 = .25       Set Z increment
P145 = 5         Set Z feedrate
P147 = .01       Set peck clearance
M03  S3000
 |     └──────── Spindle speed
 └────────────── Spindle on CW
G83              Enable peck drilling cycle
```

See Section 3.23.7 on page 3-70.

7.4.1.3 F4 WPECK (DRILL-START-PECK)

CONVERSATIONAL SCREENS

Woodpecker Drill Cycle

```
Enable Woodpecker Drill Cycle

  Z Pierce Feedrate [10        ]
  Spindle On CW RPM [3000]

  Clearance         [.1        ]
  Final Z Depth     [-1        ]
  First Z Depth     [-.25      ]
  Z Increment       [.25       ]
  Peck Clearance    [.02       ]
  Peckup Increment  [.2        ]
```

GENERATED CODE FOR WOODPECKER DRILL

```
G99  G00  G40
 |    |    └─── Cancel compensation
 |    └──────── Rapid positioning
 └───────────── Return to rapid plane
P140 = .1      Set clearance
P141 = -1      Set final Z depth
P144 = -.25    Set first Z depth
P143 = .25     Set Z increment
P145 = 10      Set Z feedrate
P146 = .2      Set peck up increment
P147 = .01     Set peck clearance
M03  S3000
 |    └──────── Spindle speed
 └───────────── Spindle on CW
G73            Enable woodpecker drill cycle
```

See Section 3.23.2 on page 3-66.

7.4.1.4 F7 TAP (DRILL-START-TAP)

```
Enable Tap Cycle

  Z Pierce Feedrate   [2        ]
  Spindle On CW RPM   [100  ]


  Clearance           [.1       ]
  Final Z Depth       [-2       ]
  Dwell Before Rev.   [.05      ]
  Dwell After Rev.    [.05      ]
```

CONVERSATIONAL SCREEN

Tap Drill Cycle

GENERATED CODE FOR TAP

```
G99  G00  G40
 |    |    └── Cancel compensation
 |    └─────── Rapid positioning
 └──────────── Return to rapid plane
P141 = -2      Set final Z depth
P145 = 2       Set Z feedrate
P148 = .05     Set dwell before reverse
P149 = .05     Set dwell after reverse
M03  S100
 |    └─────── Spindle speed
 └──────────── Spindle on CW
G84            Enable tap cycle
```

See Section 3.23.8 on page 3-71.

7.4.1.4.1 F2 POS (DRILL-POS)
F3 MISC (DRILL-MISC)
F4 CALL (DRILL-CALL)

The F2 POS position key brings up the normal position screens discussed earlier and would be used to enter in the drill positions. F3-MISC brings up the Miscellaneous Function screen and allows those functions to be programmed during drill cycles. The F4-CALL screen allows subroutines to be called during a drill cycle. These subroutines would normally contain the drilling positions. For further explanation see sections on Position (pg.7-2), Miscellaneous (pg. 7-39), and Call (pg. 7-41).

7.4.1.4.2 F5 END

Disable Drill Cycle

CONVERSATIONAL SCREEN

Disable Canned Cycle

This screen does not require any entries but must be stored in the program to terminate the active drill cycle. If this screen is not stored, every move will cause a Z axis drill cycle to be performed.

GENERATED CODE FOR DISABLE DRILL CYCLE

```
G80  G01
 |    └─── Linear feed interpolation
 └──────── Disable canned cycle
```

See Section 3.23 on page 3-63.

7.5 F4 BOLTHOLE (BOLT)

F1 - DRILL (BOLT-DRILL)
F2 - D/DWL (BOLT-D/DWL)
F3 - PECK (BOLT-PECK)
F4 - WPECK (BOLT-WPECK)
F5 - BORE (BOLT-BORE)
F6 - B/DWL (BOLT-B/DWL)
F7 - TAP (BOLT-TAP)

The following screens are displayed upon selecting the bolthole drill cycles. The first part of the screen contains information used to set up the appropriate drill cycle, and the last part contains information used to set up the bolthole cycle. With the exception of a few differences in the drill cycle setups, the M-G codes produced are the same.

CONVERSATIONAL BOLTHOLE DRILL SCREENS

```
Drill Bolthole Cycle

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [          ]
 Clearance         [          ]
 Final Z Depth     [          ]

 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle Of 1st Hole [          ]
 # Of Holes To Be Made [    ]
 # Of Holes In 360 Deg [    ]
```

```
Drill Bolthole Cycle w/Dwell

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [      ]
 Clearance         [          ]
 Final Z Depth     [          ]
 Dwell             [      ]

 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle of 1st Hole [          ]
 # Of Holes To Be Made [    ]
 # Of Holes In 360 Deg [    ]
```

```
Drill/Peck Bolthole Cycle

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [          ]
 Clearance         [          ]
 Final Z Depth     [          ]
 First Z Depth     [          ]
 Z Increment       [          ]
 Peck Clearance    [          ]
 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle of 1st Hole [          ]
 # Of Holes To Be Made [    ]
 # Of Holes In 360 Deg [    ]
```

```
Woodpecker Drill Bolthole Cycle

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [      ]
 Clearance         [          ]
 Final Z Depth     [          ]
 First Z Depth     [          ]
 Z Increment       [          ]
 Peck Clearance    [          ]
 Peckup Increment  [          ]
 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle of 1st Hole [          ]
 # Of Holes To Be Made [     ]
 # Of Holes In 360 Deg [     ]
```

```
Bore Bolthole Cycle

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [          ]
 Clearance         [          ]
 Final Z Depth     [          ]

 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle Of 1st Hole [          ]
 # Of Holes To Be Made [    ]
 # Of Holes In 360 Deg [    ]
```

```
Bore Bolthole Cycle w/Dwell

 Z Pierce Feedrate [          ]
 Spindle On CW RPM [      ]
 Clearance         [          ]
 Final Z Depth     [          ]
 Dwell             [      ]

 Bolthole Center X [          ]
                 Y [          ]
 Bolthole Radius   [          ]
 Angle of 1st Hole [          ]
 # Of Holes To Be Made [    ]
 # Of Holes In 360 Deg [    ]
```

```
Tap Bolthole Cycle

   Z Pierce Feedrate [5         ]
   Spindle On CW RPM [50    ]
   Clearance         [.1        ]
   Final Z Depth     [-1        ]
   Dwell Before Rev. [.25       ]
   Dwell After Rev.  [.5        ]
   Bolthole Center X [1         ]
                   Y [1         ]
   Bolthole Radius   [4         ]
   Angle Of 1st Hole [15        ]
   # Of Holes To Be Made [6    ]
   # Of Holes In 360 Deg [6    ]
```

GENERATED CODE FOR BOLTHOLE

```
G99   G00 G40
 |     |   └─── Cancel compensation
 |     └─────── Rapid positioning
 └───────────── Canned cycle rapid plane return
P140 = .1       Set clearance
P141 = -1       Set final Z depth
P145 = 5        Set Z pierce feedrate
P148 = .25      Set dwell before spindle reverse
P149 = .5       Set dwell after spindle reverse
P156 = 4        Set bolthole radius
P157 = 15       Set angle of 1st hole
P158 = 6        Set number of holes to be made
P159 = 6        Set number of holes in 360 degrees
M3  S50
 |   └───────── Spindle speed
 └───────────── Spindle on CW
G84             Enable tap cycle (Note: This code changes for
                         each different drill cycle.)
G72   X1 Y1
 |      └────── Center of bolthole
 └───────────── Enable bolthole cycle
```

See Section 3.23.1 on page 3-64.

7.6 F5 TOOL CHANGE (TCHG)

When a new tool needs to be put in the machine tool, the Tool Change screen should be used. The two tool change screens are Tool Call and Tool Change. The tool call is used to initiate a new set of tool offsets without physically changing the tool. The tool change puts the machine in a tool change mode and calls for a new tool. When a tool change or tool call is executed, the H and D offsets which are activated will be the ones which are the same as the tool number. For safety reasons, when doing manual tool changes, the machine should <u>always</u> be in a tool change mode. The tool screens are as follows:

```
Tool Change

  Tool [CALL]


  Tool Number [1]

  Spindle Speed [     ]
  Spindle Restart [---]
  Stop for Speed Change [NO]
  Coolant [---]
```

CONVERSATIONAL SCREEN

Tool Call

GENERATED CODE FOR TOOL CALL

```
D1  G43  H1
           └── Call length offset #1
     └──────── Length offset added
└───────────── Call radius offset #1
```

See Section 4.6 on page 4-2.

```
Tool Change

   Tool [CHANGE]
   Tool Change Position X[-10      ]
                        Y[-10      ]

   Tool Number [2]

   Spindle Speed [2000]
   Spindle Restart [CW]
   Stop for Speed Change [YES]
   Coolant [FLOOD]
```

CONVERSATIONAL SCREEN

Tool Change

GENERATED CODE FOR TOOL CHANGE

```
G70  G50  G69  H00
 |    |    |    └──── Call length offset 0 (offset =0)
 |    |    └───────── Cancel rotation
 |    └────────────── Cancel scaling
 └─────────────────── Cancel mirror image
T2                    Strobe tool change carousel
G00  G32
 |    └────────────── Move Z to tool change position
 └─────────────────── Rapid positioning
G40  G80  M05 M09
 |    |    |   └───── Turn off coolant
 |    |    └───────── Turn spindle off
 |    └────────────── Disable canned cycles
 └─────────────────── Cancel cutter compensation
X-10  Y-10            Axes move to tool change position
M06                   Start tool change
S2000  M03  M00
 |      |    └─────── Program stop
 |      └──────────── Spindle on CW
 └─────────────────── Set spindle speed
M08 D02 G43  H02
 |   |   |    └────── Call length offset #2
 |   |   └─────────── Length offset added
 |   └─────────────── Call radius offset #2
 └─────────────────── Turn on flood coolant
```

See Section 4.6 on page 4-2.

7.7 F6 MISCELLANEOUS (MISC)

As a program is being created it may be necessary to add certain miscellaneous functions such as coolant and stop commands. This is done through the MISC screen.

```
Miscellaneous

  Spindle Speed [2000]

  Spindle Command [CW]
  Coolant Command [FLOOD]
  Compensation    [LEFT]
  Stop Command    [---]
  Return Command  [---]
  Cutting Mode    [CONTINUOUS]
  Program Mode    [---]

  Miscellaneous line [              ]
```

CONVERSATIONAL SCREEN

Miscellaneous

GENERATED CODE (DEPENDING ON SELECTION)

```
S2000  M3  M8  G41  G61
                     └── Continuous cutting
                └─────── Left cutter compensation
           └──────────── Flood coolant
       └──────────────── Spindle on CW
 └────────────────────── Spindle speed
```

See Section 4 on page 4-1.

```
Miscellaneous

  Spindle Speed [     ]

  Spindle Command [  ]
  Coolant Command [OFF]
  Compensation    [OFF]
  Stop Command    [PROGRAM]
  Return Command  [XYZ TO ZERO]
  Cutting Mode    [---]
  Program Mode    [ABSOLUTE]

  Miscellaneous line [             ]
```

CONVERSATIONAL SCREEN

Miscellaneous

GENERATED CODE (DEPENDING ON SELECTION)

```
M9  G40  M0  G90  G60 Z0
                          Z Rapid to 0
                          Absolute programming mode
                          Program stop
                          Compensation off
                          Coolant off
```

See Section 4 on page 4-1.

The Miscellaneous line is used to type in any M code that is not part of the standard list.

7.8 F7 CALL

The program call screen is used to transfer program execution to another program for a specified number of loops.

```
Program Call

  Program Number    [9000]

  Number of Loops   [2   ]
```

CONVERSATIONAL SCREEN

Program Number

GENERATED CODE FOR PROGRAM CALL

```
CALL  9000  L2
 |     |     └── Number of loops
 |     └──────── Program number
 └────────────── Function identifier
```

See Section 4.13 on page 4-4.

PARAMETER SETTING KEYS

7.9 F8 SPECIAL (SPEC)

These are screens for setting or adjusting various parameters in the Centurion V control. The parameters control various functions, such as tool offsets, scale factors, rotation angles, mirror image, floating zeroes and the parameters listed in Appendix A. The parameters changed often have their own screens which are shown below. Adjusting a parameter will add the specified value to the current setting. Loading a parameter will set the parameter to the specified value.

7.9.1 F1 PARMS (SPEC-PARMS)

```
Set Parameter
  Set Type [LOAD]
  Parameter Number [97]
  Parameter Value  [1         ]
```

CONVERSATIONAL SCREEN

Set Parameter

GENERATED CODE FOR SET PARAMETER

P97 =1   Set user parameter 97 to 1

```
Set Parameter
  Set Type [ADJUST]
  Parameter Number [97]
  Parameter Value  [.25       ]
```

CONVERSATIONAL SCREEN

Set Parameter

GENERATED CODE FOR ADJUST PARAMETER

P97 = P97 + .25   Add .25 to user parameter 97

7.9.2 F2 TOOLS (SPEC-TOOLS)

```
Set Tool Offset

  Note:  This will only afftect the
         currently active tool.

  Set Type [LOAD]

  Tool Offset  D [.25      ]
               H [.75      ]
```

CONVERSATIONAL SCREEN

Set Tool Offset

GENERATED CODE FOR SET TOOL OFFSET

```
P261 = .25           Set current tool radius
P262 = .75           Set current tool length
```

See Section 3.13 on page 3-48.

```
Set Tool Offset

  Note:  This will only afftect the
         currently active tool.

  Set Type [ADJUST]

  Tool Offset  D [.1       ]
               H [.15      ]
```

CONVERSATIONAL SCREEN

Set Tool Offset

GENERATED CODE FOR ADJUST TOOL OFFSET

```
P261 = P261 + .1     Adjust current tool radius
P262 = P262 + .15    Adjust current tool length
```

7.9.3 F4 SCALE (SPEC-SCALE)

```
Set Scale Factor

  Turn Scaling [ON]

  Scale Factors  X [2        ]
                 Y [2        ]
                 Z [1        ]

  Scaling Origin I [0        ]
                 J [0        ]
                 K [0        ]
```

CONVERSATIONAL SCREEN

Set Scale Factor

GENERATED CODE FOR SCALING

```
G51  I0  J0  K0  X2  Y2  Z1
                               Scale factors
                               Scale center
                               Turn scaling on
```

See Section 3.14 on page 3-50.

```
Set Scale Factor

  Turn Scaling [OFF]
```

CONVERSATIONAL SCREEN

Set Scale Factor

GENERATED CODE FOR SCALING OFF

G50 Turn scaling off

See Section 3.14 on page 3-50.

7.9.4 F5 ROT (SPEC-ROT)

```
Set Rotation Angle

  Turn Rotation [ON]

  Rotation Angle [45]

  Rotation Origin  X [0        ]
                   Y [0        ]
                   Z [0        ]
```

CONVERSATIONAL SCREEN

Set Rotation Angle

GENERATED CODE FOR ROTATION

```
G68  X0  Y0  Z0  AA45
 |     |          └──── Angle of rotation
 |     └─────────────── Rotation origin
 └───────────────────── Turn rotation on
```

See Section 3.21 on page 3-55.

```
Set Scale Factor

  Turn Rotation [OFF]
```

CONVERSATIONAL SCREEN

Set Rotation Angle

GENERATED CODE FOR ROTATION OFF

G69 Turn rotation off

See Section 3.21 on page 3-55.

7.9.5 F6 MIRR (SPEC-MIRR)

```
Set Mirror Image

  Turn Mirror Image [ON]

  Mirror Axis  X [0        ]
               Y [0        ]
               Z [0        ]
```

CONVERSATIONAL SCREEN

Set Mirror Image

GENERATED CODE FOR SET MIRROR IMAGE

```
G71  X0  Y0  Z0
                 Mirror Z axis around Z0
                 Mirror Y axis around Y0
                 Mirror X axis around X0
                 Turn on mirror image
```

See Section 3.22 on page 3-58.

```
Set Mirror Image

  Turn Mirror Image [OFF]
```

CONVERSATIONAL SCREEN

Set Mirror Image

GENERATED CODE FOR MIRROR OFF

G70 Turn off mirror image

See Section 3.22 on page 3-58.

7.9.6 F7 FLZ (SPEC-FLZ)

```
Set Floating Zero

  Axis  X [10        ]
        Y [-5        ]
        Z [0         ]
```

CONVERSATIONAL SCREEN

Set Floating Zero

GENERATED CODE FOR SET FLOATING ZERO

```
G92  X10  Y-5  Z0
 |        └────────── New zero position
 └─────────────────── Set new floating zero
```

See Section 3.15.2 on page 3-52.

8. SAMPLE PROGRAMS

The following sample programs give a variety of programming problems and show possible solutions to these problems using the Centurion V control. The program given for each sample part is by no means the only solution for that sample part.

Each sample part begins with the drawing of the part, then gives the standard EIA (G and M codes), followed by an EIA program explanation, and finally a conversational program of the sample part. The sample part drawings for milling will include the start and end point of the programmed moves, the direction of tool travel, and the tool path indicated by dashed lines running parallel to the edge of the part. The drawings also indicate where the floating zero is set for each part. Feeds and speeds for end mills will assume a 3/8" cutter in aluminum. Other tools will be specified. Before cutting any of these sample parts the operator must set the floating zero and tool offsets.

An asterisk (*) in a conversational program indicates toggled selections made with the "TOGL" key at the bottom of the screen. Where a blank space (underlined value in the samples) is provided in a conversational program, the operator can enter his/her own position information (such as a tool change location) or these entries can be left blank and skipped over. When entering the mill cycle geometry of a conversational program the last selection at the bottom of the "Line" and "Arc" screens is the "End Option." The "End Option" allows the programmer to select the round corner or chamfer feature at that point in the program. If neither selection is desired the"End Option" is skipped over.

When entering conversational programs into the Centurion V control, the "Back" soft key at the base of the screen is used to select a "Previous Menu" to allow entry of a command not found on the current menu. This selection is included in the sample conversational programs to aid the operator in entering these programs, but it is not part of the program. Each event selection in the conversational programs is followed by a label in parentheses. This is the label of the soft key at the base of the screen for making that event selection.

SAMPLE 1

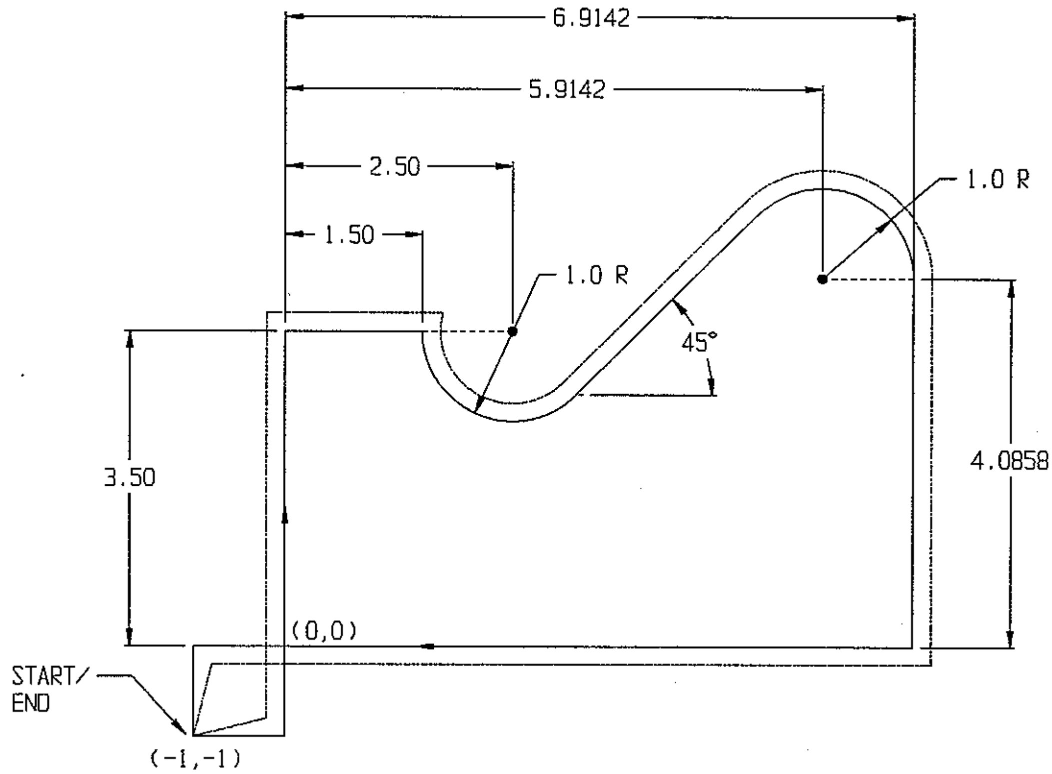


Figure 16.1

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    X-1 Y-1 S3000 M03
N4    G43 H01 Z.1 M08
N5    G01 Z-.375 F5
N6    G41 D01 X0 Y-1 F25
N7    X0 Y3.5
N8    X1.5 Y3.5
N9    G03 R1 AA180 AB-45
N10   G01 AB45 R.5
N11   G02 R1 AB0 XC5.9142 YC4.0858
N12   G01 X6.9142 Y0
N13   X-1 Y0
N14   G40 X-1 Y-1
N15   G00 Z.1 M09
N16   M05
```

Explanation of EIA Program 1

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning

N2 Tool change #1

N3 Positions to X-1 Y-1 and turns spindle on (3000 rpm)

N4 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N5 Feeds Z-.375 at 5 ipm

N6 Selects left cutter compensation, calls tool #1's "D" offset, and feeds to X0 Y-1 at 25 ipm

Note: The cutter compensation will "ramp on" during this move.

N7 Line move to X0 Y3.5

N8 Line move to X1.5 Y3.5

N9 CCW arc 1" radius starting at 180 , ending at -45

N10 Line move using an estimated end point described using polar coordinates; angle 45 radius .5

N11 CW arc 1" radius absolute center of XC5.9142 YC4.0858 and an end angle of 0

N12 Line move to X6.9142 Y0

N13 Line move to X-1 Y0

N14 Turn off cutter compensation during move to X-1 Y-1

Note: Cutter compensation will "ramp off" during move.

N15 Rapids Z to .1 and turns off coolant

N16 Turns off spindle

Conversational Program 1

1. Program setup — Event 0
   - A. Dimensions ABSOLUTE *
   - B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   - A. Tool CHANGE *
   - B. Tool change position X________
     Y________
   - C. Tool number 1
   - D. Spindle speed 3000
   - E. Spindle restart CW *
   - F. Coolant FLOOD *

3. Position (POS) — Event 2
   - A. Feedrate RAPID *
   - B. Coordinates CARTESIAN *
   - C. X axis -1.0
   - D. Y axis -1.0

4. Position (POS) — Event 3
   - A. Feedrate RAPID *
   - B. Coordinates CARTESIAN *
   - C. Z axis .1

5. Mill (MILL) — Event 4
   - A. Geometry (GEOM)
     1. Line (LINE)
        - a) feedrate 5
        - b) coordinates CARTESIAN *
        - c) Z axis -.375
     2. Previous menu (BACK)
   - B. Miscellaneous (MISC) — Event 5
     1. Compensation LEFT *
   - C. Geometry (GEOM) — Event 6
     1. Line (LINE)
        - a) feedrate 25
        - b) coordinates CARTESIAN *
        - c) X axis 0
        - d) Y axis -1
     2. Line (LINE) — Event 7
        - a) coordinates CARTESIAN *
        - b) X axis 0
        - c) Y axis 3.5

3. Line (LINE) — Event 8
   a) coordinates CARTESIAN *
   b) X axis 1.5
   c) Y axis 3.5

4. Arc (ARC) — Event 9
   a) plane XY *
   b) direction CCW *
   c) coordinates POLAR *
   d) arc radius 1
   e) start angle 180
   f) end angle -45

5. Line (LINE) — Event 10
   a) coordinates POLAR *
   b) plane XY *
   c) type CURRENT *
   d) radius .5
   e) end angle 45

6. Arc (ARC) — Event 11
   a) plane XY *
   b) direction CW *
   c) coordinates ABS CENTER *
   d) arc radius 1
   e) arc center XC 5.9142
      YC 4.0858
   f) end point X 6.9142
      Y 4.0858

7. Line (LINE) — Event 12
   a) coordinates CARTESIAN *
   b) X axis 6.9142
   c) Y axis 0

8. Line (LINE) — Event 13
   a) coordinates CARTESIAN *
   b) X axis -1
   c) Y axis 0

9. Previous menu (BACK)

D. Miscellaneous (MISC) — Event 14
   1. Compensation OFF *

E. Geometry (GEOM)
   1. Line (LINE) — Event 15
      a) Coordinates CARTESIAN *
      b) X axis -1
      c) Y axis -1

   2. Previous menu (BACK)

F. Previous menu (BACK)

6. Position (POS) Event 16
   A. Feedrate RAPID *
   B. Coordinates CARTESIAN *
   C. Z axis .1

7. Miscellaneous (MISC) Event 17
   A. Spindle OFF *
   B. Coolant OFF *

8. End program (EXIT) Event 18

SAMPLE 2A

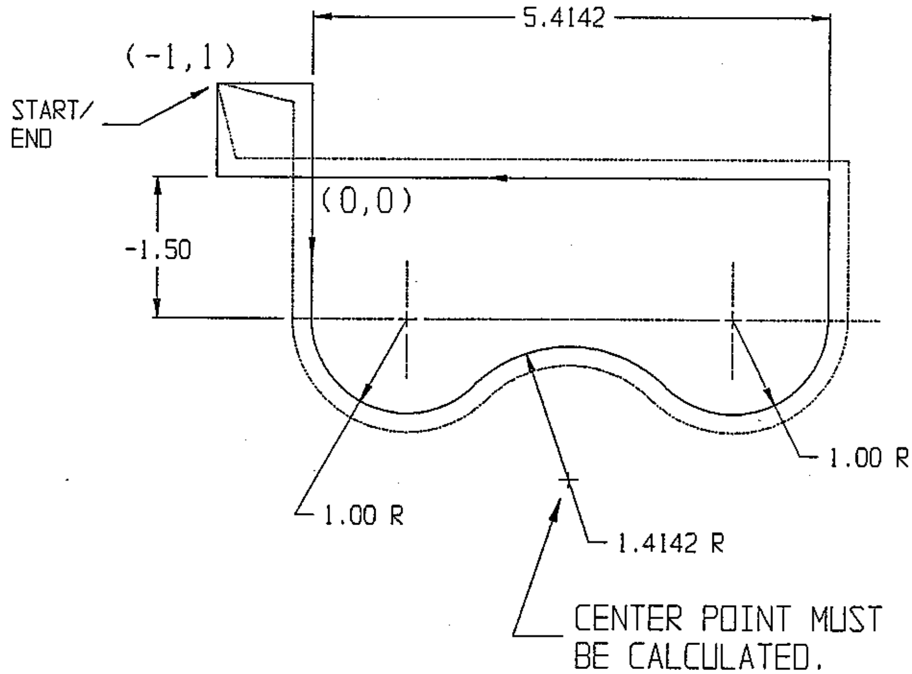


Figure 16.2

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    X-1 Y1 S3000 M03
N4    G43 H01 Z.1 M8
N5    G1 Z-.375 F5
N6    G42 D01 X0 F25
N7    X0 Y-1.5
N8    G03 XC1 YC-1.5 X1 Y-2.5 R1
N9    G02 XC2.7071 YC-3.2071 X2.7071 Y-1.7828 R1.4142
N10   G03 XC4.4142 YC-1.5 X5.4142 Y-1.5 R1
N11   G01 X5.4142 Y0
N12   X-1 Y0
N13   G40 X-1 Y1
N14   G00 Z.1 M09
N15   M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Positions to X-1 Y1; turns spindle on CW (3000 rpm)

N4 Calls tool #1's "H" offset and positions Z to .1; turns on coolant

N5 Feeds Z-.375 at 5 ipm

N6 Selects right cutter compensation, calls tool #1's "D" offset, and moves to X0 at 25 ipm.
Note: Cutter compensation will "ramp on" during this move.

N7 Line move to X0 Y-1.5

N8 CCW arc 1" radius using an XC1 YC-1.5 and an estimated end point of X1 Y-2.5

N9 CW arc 1.4142 radius using an XC2.7071 YC-3.197 and an estimated end point of X2.7071 Y-1.7828
Note: The center point must be calculated prior to programming.

N10 CCW arc 1" radius using an XC4.4142 YC-1.5 and an end point of X5.4142 Y-1.5

N11 Line move to X5.4142 Y0

N12 Line move to X-1 Y0

N13 Turn off cutter compensation, move to X-1 Y1
Note: Cutter compensation will "ramp off" during this move.

N14 Rapid Z axis to .1, turns off coolant

N15 Turns off spindle

Conversational Program 2A

1. Program setup — Event 0
   A. Dimensions ABSOLUTE *
   B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   A. Tool CHANGE *
   B. Tool change position X________
      Y________
   C. Tool number 1
   D. Spindle speed 3000
   E. Spindle restart CW *
   F. Coolant FLOOD *

3. Position (POS) — Event 2
   A. Feedrate RAPID *
   B. Coordinates CARTESIAN *
   C. X axis -1.0
   D. Y axis 1.0

4. Position (POS) — Event 3
   A. Feedrate RAPID *
   B. Coordinates CARTESIAN *
   C. Z axis .10

5. Mill (MILL) — Event 4
   A. Geometry (GEOM)
      1) Line (LINE)
         a) feedrate 5
         b) coordinates CARTESIAN *
         c) Z axis -.375

      2) Previous menu (BACK)

   B. Miscellaneous (MISC) — Event 5
      1) Compensation RIGHT *

   C. Geometry (GEOM) — Event 6
      1) Line (LINE)
         a) feedrate 25
         b) coordinates CARTESIAN *
         c) X axis 0
         d) Y axis 1

      2) Line (LINE) — Event 7
         a) coordinates CARTESIAN *
         b) X axis 0
         c) Y axis -1.5

3) Arc (ARC) Event 8
   a) plane XY *
   b) direction CCW *
   c) coordinates ABS CENTER *
   d) arc radius 1.0
   e) arc center XC 1
      YC -1.5
   f) end point X 1
      Y -2.5

4) Arc (ARC) Event 9
   a) plane XY *
   b) direction CW *
   c) coordinates ABS CENTER *
   d) arc radius 1.4142
   e) arc center XC +2.7071
      YC -3.1970
   f) end point X 2.7071
      Y -1.7828

5) Arc (ARC) Event 10
   a) plane XY *
   b) direction CCW *
   c) coordinates ABS CENTER *
   d) arc radius 1
   e) arc center XC 4.4142
      YC -1.5
   f) end point X 5.4142
      Y -1.5

6) Line (LINE) Event 11
   a) coordinates CARTESIAN *
   b) X axis 5.4142
   c) Y axis 0

7) Line (LINE) Event 12
   a) coordinates CARTESIAN *
   b) X axis -1.0
   c) Y axis 0

8) Previous menu (BACK)

D. Miscellaneous (MISC) Event 13
   1) Compensation OFF *

E. Geometry (GEOM) Event 14
   1) Line (LINE)
      a) coordinates CARTESIAN *
      b) X axis -1.0
      c) Y axis 1.0

   2) Previous menu (BACK)

F. Previous menu (BACK)

6. Position (POS) Event 15
   A. Feedrate <u>RAPID</u> *
   B. Coordinates <u>CARTESIAN</u> *
   C. Z axis <u>.1</u>

7. Miscellaneous (MISC) Event 16
   A. Spindle <u>OFF</u> *
   B. Coolant <u>OFF</u> *

8. End program (EXIT) Event 17

SAMPLE 2B

Same Part as Sample 2A but Programmed Using Tangent Arc Function

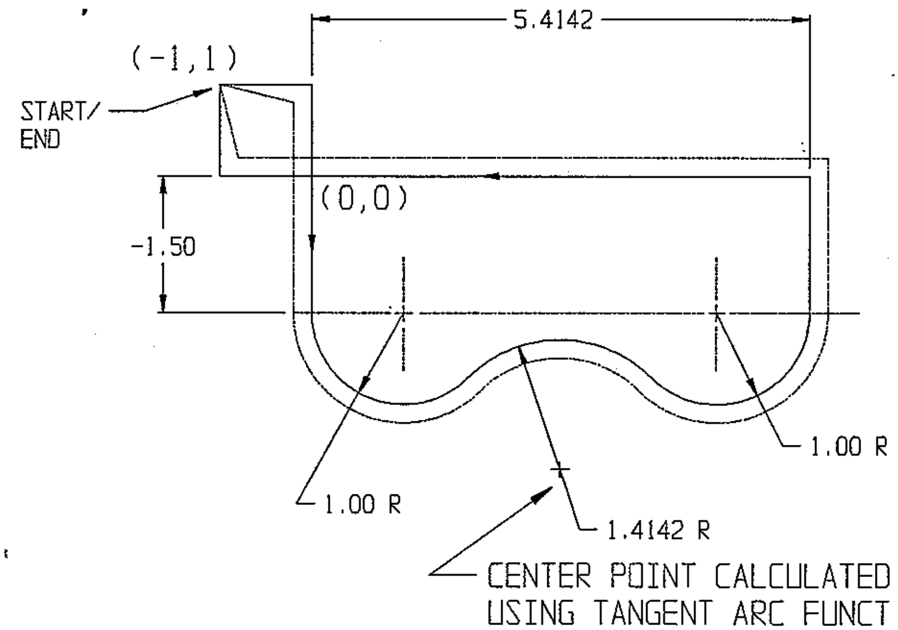


Figure 16.2

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    X-1 Y1 S3000 M03
N4    G43 H01 Z.1 M8
N5    G1 Z-.375 F5
N6    G42 D01 X0 F25
N7    X0 Y-1.5
N8    P90=1
N9    P91=-1.5
N10   P92=1
N11   P93=4.4142
N12   P94=-1.5
N13   P95=1
N14   P96=1.4142
N15   TANA C7 (See TANA explanation for value of C.)
N16   G03 XC[P90] YC[P91] R[P92] X[P80] Y[P81]
N17   G02 XC[P84] YC[P85] R[P96] X[P82] Y[P83]
N18   G03 XC4.4142 YC-1.5 X5.4142 Y-1.5 R1
N19   G01 X5.4142 Y0
N20   X-1 Y0
N21   G40 X-1 Y1
N22   G00 Z.1 M09
N23   M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Positions to X-1 Y1; turns spindle on CW (3000 rpm)

N4 Calls tool #1's "H" offset and positions Z to .1; turns on coolant

N5 Feeds Z-.375 at 5 ipm

N6 Selects right cutter compensation, calls tool #1's "D" offset, and moves to X0 at 25 ipm.
Note: Cutter compensation will "ramp on" during this move.

N7 Line move to X0 Y-1.5

N8, N9 Are the X,Y center point of the first arc

N10 Radius of first arc

N11, N12 Are the X,Y center point of the second arc

N13 Radius of second arc

N14 Radius of tangent arc

N15 Tangent arc function

N16 First arc with end points

N17 Tangent arc with calculated center and end point

N18 CCW arc 1" radius using an XC4.4142 YC-1.5 and an end point of X5.4142 Y-1.5

N19 Line move to X5.4142 Y0

N20 Line move to X-1 Y0

N21 Turn off cutter compensation, move to X-1 Y1
Note: Cutter compensation will "ramp off" during this move.

N22 Rapid Z axis to .1, turns off coolant

N23 Turns off spindle

Conversational Program 2B

1. Program setup — Event 0
   - A. Dimensions <u>ABSOLUTE</u> *
   - B. Units <u>ENGLISH</u> *

2. Tool change (TCHNG) — Event 1
   - A. Tool <u>CHANGE</u> *
   - B. Tool change position X________
     Y________
   - C. Tool number <u>1</u>
   - D. Spindle speed <u>3000</u>
   - E. Spindle restart <u>CW</u> *
   - F. Coolant <u>FLOOD</u> *

3. Position (POS) — Event 2
   - A. Feedrate <u>RAPID</u> *
   - B. Coordinates <u>CARTESIAN</u> *
   - C. X axis <u>-1.0</u>
   - D. Y axis <u>1.0</u>

4. Position (POS) — Event 3
   - A. Feedrate <u>RAPID</u> *
   - B. Coordinates <u>CARTESIAN</u> *
   - C. Z axis <u>.10</u>

5. Mill (MILL) — Event 4
   - A. Geometry (GEOM)
     - 1) Line (LINE)
       - a) feedrate <u>5</u>
       - b) coordinates <u>CARTESIAN</u> *
       - c) Z axis <u>-.375</u>
     - 2) Previous menu (BACK)
   - B. Miscellaneous (MISC) — Event 5
     - 1) Compensation <u>RIGHT</u> *
   - C. Geometry (GEOM) — Event 6
     - 1) Line (LINE)
       - a) feedrate <u>25</u>
       - b) coordinates <u>CARTESIAN</u> *
       - c) X axis <u>0</u>
       - d) Y axis <u>1</u>
     - 2) Line (LINE) — Event 7
       - a) coordinates <u>CARTESIAN</u> *
       - b) X axis <u>0</u>
       - c) Y axis <u>-1.5</u>

3) Tangent arc (TANGS) Event 8

a) plane XY *
b) first arc direction CCW *
c) R1[1] (radius of first arc)
d) XC1[1] YC1[-1.5] (center of first arc)

Second arc information

e) R2[1] (radius of second arc)
f) XC2[4.4142] YC2[-1.5] (center of second arc)
g) Exit 1st arc [RIGHT]* and enter 2nd arc [RIGHT]*
h) Connect with [AN ARC]* Center to the [RIGHT]*
i) Radius [1.4241] Arc direction [CW]*

Note: If a tangent line was desired betwen these two arcs, question h) would be answered with [LINE].

4) Arc (ARC) Event 9

a) plane XY *
b) direction CCW *
c) coordinates ABS CENTER *
d) arc radius 1
e) arc center XC 4.4142
YC -1.5
f) end point X 5.4142
Y -1.5

5) Line (LINE) Event 10

a) coordinates CARTESIAN *
b) X axis 5.4142
c) Y axis 0

6) Line (LINE) Event 11

a) coordinates CARTESIAN *
b) X axis -1.0
c) Y axis 0

7) Previous menu (BACK)

D. Miscellaneous (MISC) Event 12

1) Compensation OFF *

E. Geometry (GEOM) Event 13

1) Line (LINE)

a) coordinates CARTESIAN *
b) X axis -1.0
c) Y axis 1.0

2) Previous menu (BACK)

F. Previous menu (BACK)

6. Position (POS) Event 14

   A. Feedrate RAPID *

   B. Coordinates CARTESIAN *

   C. Z axis .1

7. Miscellaneous (MISC) Event 15

   A. Spindle OFF *

   B. Coolant OFF *

8. End program (EXIT) Event 16

SAMPLE 3A

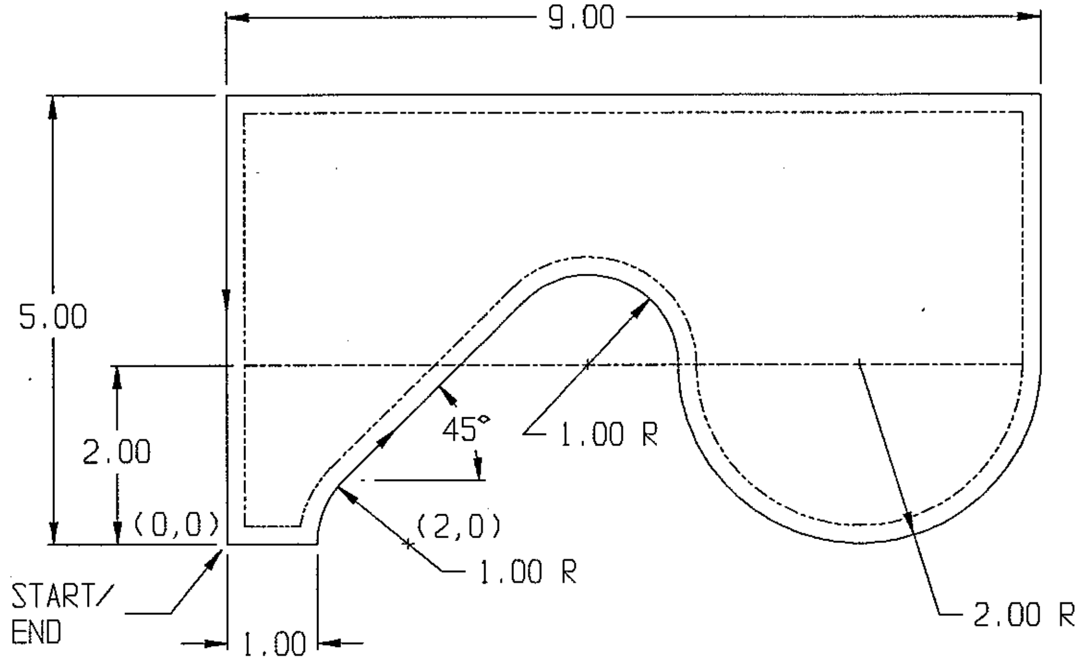


Figure 16.3

EIA PROGRAM

```
N1    G00 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    G41 D01 S3000 M03
N4    G65 X0 Y99
N5    X0 Y0
N6    G43 H01 Z.1 M8
N7    G01 Z-.375 F5
N8    X1 Y0 F25
N9    G02 XC2 YC0 AB135 R1
N10   G01 AB45 R.5
N11   G02 XC4 YC2 X5 Y2 R1
N12   G03 XC7 YC2 X9 Y2 R2
N13   G01 X9 Y5
N14   X0 Y5
N15   X0 Y0
N16   G65 X99 Y0
N17   G40
N18   G00 Z.1 M09
N19   M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Sets a "point before pierce" of X0 Y99
Note: Machine does not move to this position.

N5 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X1 Y0 at 25 ipm

N9 CW arc 1" radius using an XC2 YC0 and an end angle of AB135

N10 Line move using an estimated end point and described polarly, angle 45 radius .5"

N11 CW arc 1" radius using an XC7 YC2 and an end point of X5 Y2

N12 CCW arc 2" radius using an XC7 YX2 and an end point of X9 Y2

N13 Line move to X9 Y5

N14 Line move to X0 Y5

N15 Line move to X0 Y0

N16 Establishes a "point after pierce" of X99 Y0
Note: Machine does not move to this position.

N17 Turns off cutter compensation

N18 Rapids Z to .1, turns off coolant

N19 Turns off spindle

Conversational Program 3A

1. Program setup — Event 0
   A. Dimensions ABSOLUTE *
   B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   A. Tool CHANGE *
   B. Tool change position X________
      Y________
   C. Tool number 1
   D. Spindle speed 3000
   E. Spindle restart CW *
   F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   A. Start (START)
      1. Z pierce feedrate 5
      2. Clearance .1
      3. Final Z depth -.375
      4. First Z depth -.375
      5. Z increment .375
      6. X pierce point 0
      7. Y pierce point 0
      8. Compensation LEFT *
      9. X before pierce 0
      10. Y before pierce 99

   B. Geometry (GEOM) — Event 3
      1. Line (LINE)
         a) feedrate 25
         b) coordinates CARTESIAN *
         c) X axis 1
         d) Y axis 0

      2. Arc (ARC) — Event 4
         a) plane XY *
         b) direction CW *
         c) coordinatess POLAR *
         d) arc radius 1
         e) start angle 180
         f) end angle 135

      3. Line (LINE) — Event 5
         a) coordinates POLAR *
         b) plane XY *
         c) type CURRENT *
         d) radius .5
         e) end angle 45

4. Arc (ARC) — Event 6
   a) plane XY *
   b) direction CW *
   c) coordinates ABS CENTER *
   d) arc radius 1
   e) arc center XC 4
      YC 2
   f) end point X 5
      Y 2

5. Arc (ARC) — Event 7
   a) plane XY *
   b) direction CCW *
   c) coordinates ABS CENTER *
   d) arc radius 2
   e) arc center XC 7
      YC 2
   f) end point X 9
      Y 2

6. Line (LINE) — Event 8
   a) coordinates CARTESIAN *
   b) X axis 9
   c) Y axis 5

7. Line (LINE) — Event 9
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 5

8. Line (LINE) — Event 10
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 0

9. Previous menu (BACK)

C. End mill cycle (END) — Event 11
   1. Point after retract X 99
      Y 0

D. Miscellaneous (MISC) — Event 12
   1. Spindle OFF *
   2. Coolant OFF *

4. End program (EXIT) — Event 13

SAMPLE 3B

Same Part as Sample 3A but Programmed Using Tangent Line Function

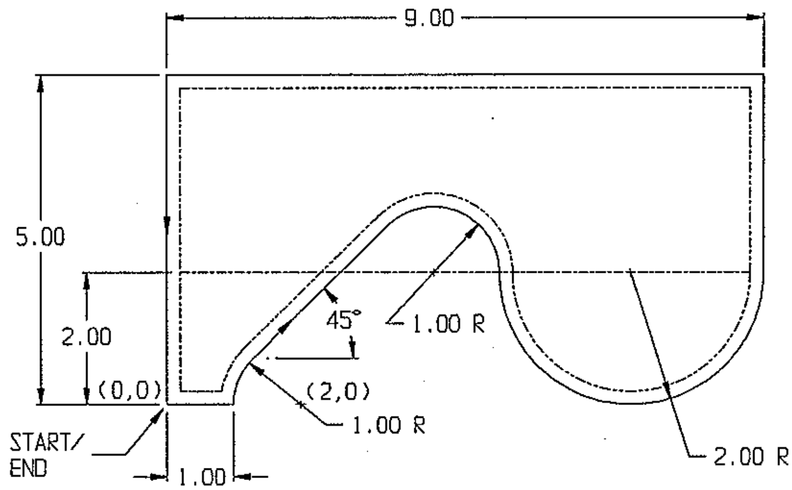


Figure 16.3

EIA PROGRAM

```
N1   G00 G20 G32 G40 G50 G69 G80 G90
N2   T1 M6
N3   G41 D01 S3000 M03
N4   G65 X0 Y99
N5   X0 Y0
N6   G43 H01 Z.1 M8
N7   G01 Z-.375 F5
N8   X1 Y0 F25
N9   P90=2
N10  P91=0
N11  P92=1
N12  P93=4
N13  P94=2
N14  P95=1
N15  TANL C3 (See TANL explanation for values of C.)
N16  G02 XC[P90] YC[P91] R[P92] X[P80] Y[P81]
N17  G01 X[P82] Y[P83]
N18  G02 XC4 YC2 X5 Y2 R1
N19  G03 XC7 YC2 X9 Y2 R2
N20  G01 X9 Y5
N21  X0 Y5
N22  X0 Y0
N23  G65 X99 Y0
N24  G40
N25  G00 Z.1 M09
N26  M05
```

Note: Lines N9 thru N14 could be written as follows:
N9 P90=2 P91=0 P92=1 P93=4 P94=2 P95=1

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Sets a "point before pierce" of X0 Y99
Note: Machine does not move to this position.

N5 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X1 Y0 at 25 ipm

N9, N10 Center of the first arc

N11 Radius of the first arc

N12, N13 The center of the second arc

N14 Radius of the second arc

N15 Tangent line function

N16 First arc with calculated end points

N17 Line tangent to both arcs

N18 CW arc 1" radius using an XC7 YC2 and an end point of X5 Y2

N19 CCW arc 2" radius using an XC7 YC2 and an end point of X9 Y2

N20 Line move to X9 Y5

N21 Line move to X0 Y5

N22 Line move to X0 Y0

N23 Establishes a "point after pierce" of X99 Y0
Note: Machine does not move to this position.

N24 Turns off cutter compensation

N25 Rapids Z to .1, turns off coolant

N19 Turns off spindle

e) arc center XC 4
YC 2
f) end point X 5
Y 2

4. Arc (ARC) — Event 6
a) plane XY *
b) direction CCW *
c) coordinates ABS CENTER *
d) arc radius 2
e) arc center XC 7
YC 2
f) end point X 9
Y 2

5. Line (LINE) — Event 7
a) coordinates CARTESIAN *
b) X axis 9
c) Y axis 5

6. Line (LINE) — Event 8
a) coordinates CARTESIAN *
b) X axis 0
c) Y axis 5

7. Line (LINE) — Event 9
a) coordinates CARTESIAN *
b) X axis 0
c) Y axis 0

8. Previous menu (BACK)

C. End mill cycle (END) — Event 10
1. Point after retract X 99
Y 0

D. Miscellaneous (MISC) — Event 11
1. Spindle OFF *
2. Coolant OFF *

4. End program (EXIT) — Event 12

Conversational Program 3B

1. Program setup — Event 0
   - A. Dimensions ABSOLUTE *
   - B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   - A. Tool CHANGE *
   - B. Tool change position X________
     Y________
   - C. Tool number 1
   - D. Spindle speed 3000
   - E. Spindle restart CW *
   - F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   - A. Start (START)
     1. Z pierce feedrate 5
     2. Clearance .1
     3. Final Z depth -.375
     4. First Z depth -.375
     5. Z increment .375
     6. X pierce point 0
     7. Y pierce point 0
     8. Compensation LEFT *
     9. X before pierce 0
     10. Y before pierce 99

   - B. Geometry (GEOM) — Event 3
     1. Line (LINE)
        - a) feedrate 25
        - b) coordinates CARTESIAN *
        - c) X axis 1
        - d) Y axis 0

     2. Tangent line (TANGS) — Event 4
        - a) plane XY *
        - b) mill first arc direction CW *
        - c) R1[1] radius of first arc
        - d) XC1[2] YC1[0]

        Second arc information
        - e) R2[1] radius of second arc
        - f) XC2[4] YC2[2] center of second arc
        - g) exit 1st arc [LEFT] and enter 2nd arc [LEFT]*
        - h) connect with [A LINE] *

     3. Arc (ARC) — Event 5
        - a) plane XY *
        - b) direction CW *
        - c) coordinates ABS CENTER *
        - d) arc radius 1

SAMPLE 4A

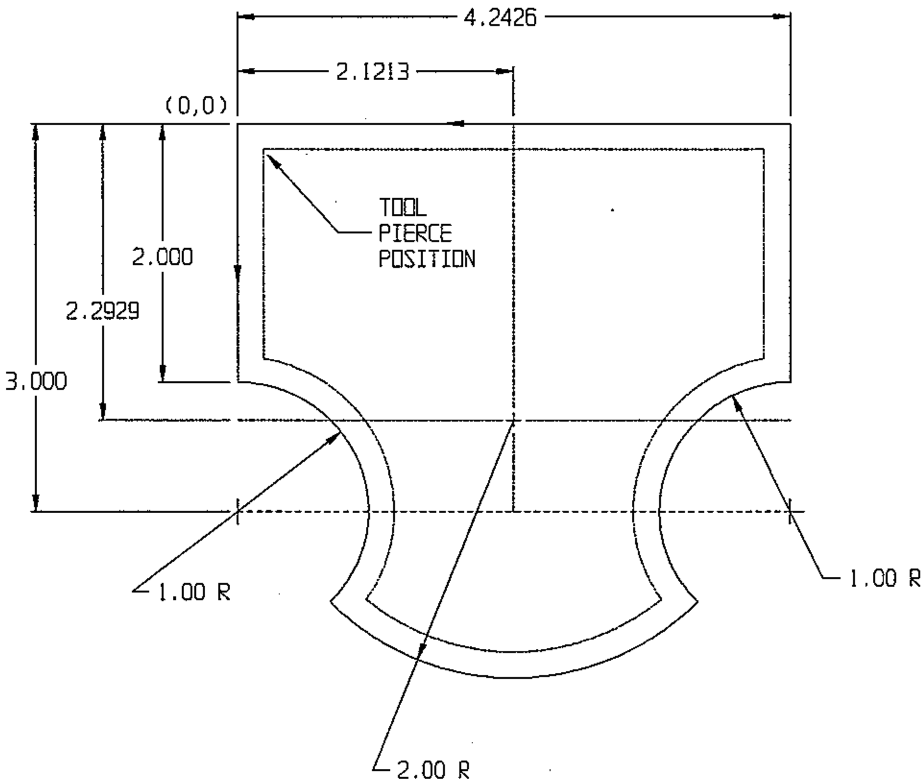


Figure 16.4

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    G41 D01 S3000 M03
N4    G65 X99 Y0
N5    X0 Y0
N6    G43 H01 Z.1 M08
N7    G01 Z-.375 F5
N8    X0 Y-2 F25
N9    G02 XC0 YC-3 X1 Y-3 R1
N10   G03 XC2.1213 YC-2.2929 X2.1213 Y-4.2929 R2
N11   G02 XC4.2426 YC-3 X4.2426 Y-2 R1
N12   G01 X4.2426 Y0
N13   X0 Y0
N14   G65 X0 Y-99
N15   G40
N16   G00 Z.1 M09
N17   M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N5 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X0 Y-2 at 25 ipm

N9 CW arc 1" radius using an XC0 YC-3 and an estimated end point of X1 Y-3

N10 CCW arc 2" radius using an XC2.1213 YC-2.2929 and an estimated end point of X2.1213 Y-4.2929

N11 CW arc 1" radius using an XC4.2426 YC-3 and an end point of X4.2426 Y-2

N12 Line move to X4.2426 Y0

N13 Line move to X0 Y0

N14 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N15 Cancels cutter compensation

N16 Rapids Z to .1, turns coolant off

N17 Turns spindle off

Conversational Program 4A

1. Program setup — Event 0
   A. Dimensions ABSOLUTE *
   B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   A. Tool CHANGE *
   B. Tool change position X________
      Y________
   C. Tool number 1
   D. Spindle speed 3000
   E. Spindle restart CW *
   F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   A. Start (START)
      1. Z pierce feedrate 5
      2. Clearance .1
      3. Final Z depth -.375
      4. First Z depth -.375
      5. Z increment .375
      6. X pierce point 0
      7. Y pierce point 0
      8. Compensation LEFT *
      9. X before pierce 99
      10. Y before pierce 0

   B. Geometry (GEOM) — Event 3
      1. Line (LINE)
         a) feedrate 25
         b) coordinates CARTESIAN *
         c) X axis 0
         d) Y axis -2

      2. Arc (ARC) — Event 4
         a) plane XY *
         b) direction CW *
         c) coordinatess ABS CENTER *
         d) arc radius 1
         e) arc center XC 0
            YC -3
         f) end point X 1
            Y -3

      3. Arc (ARC) — Event 5
         a) plane XY *
         b) direction CCW *
         c) coordinates ABS CENTER *
         d) arc radius 2
         e) arc center XC 2.1213
            YC -2.2929

f) end point X 2.1213
Y -4.2929

4. Arc (ARC) Event 6
   a) plane XY *
   b) direction CW *
   c) coordinates ABS CENTER *
   d) arc radius 1
   e) arc center XC 4.2426
      YC -3
   f) end point X 4.2426
      Y 2

5. Line (LINE) Event 7
   a) coordinates CARTESIAN *
   b) X axis 4.2426
   c) Y axis 0

6. Line (LINE) Event 8
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 0

7. Previous menu (BACK)

C. End mill cycle (END) Event 9
   1. Point after retract X 99
      Y 0

D. Miscellaneous (MISC) Event 10
   1. Spindle OFF *
   2. Coolant OFF *

4. End program (EXIT) Event 11

SAMPLE 4B

Programming Arc Using 3 Point Circle Generate
Points X1, X2, X3 Are the Points Used to Program Each Arc

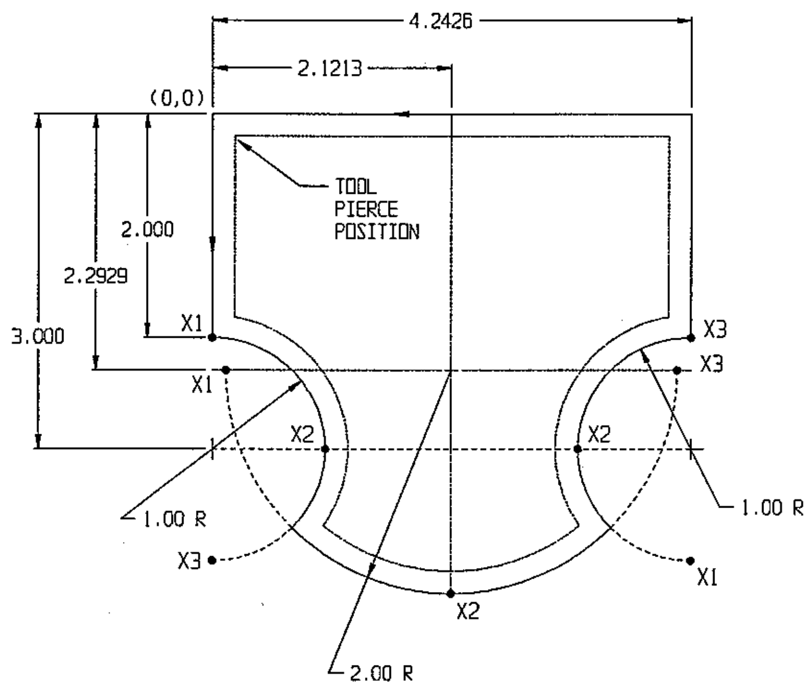


Figure 16.4

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    G41 D01 S3000 M03
N4    G65 X99 Y0
N5    X0 Y0
N6    G43 H01 Z.1 M08
N7    G01 Z-.375 F5
N8    X0 Y-2 F25
N9    P90=0
N10   P91=-2
N11   P92=1
N12   P93=-3
N13   P94=0
N14   P95=-4
N15   CGEN
N16   G02 XC[P80] YC[P81] R[P82] AB300
N17   P90=.1213
N18   P91=-2.2929
N19   P92=2.1213
N20   P93=-4.2929
```

```
N21  P94=4.1213
N22  P95=-2.2929
N23  CGEN
N24  G03 XC[P80] YC[P81] R[P82] AB300
N25  P90=4.2426
N26  P91=-4
N27  P92=3.2426
N28  P93=-3
N29  P94=4.2426
N30  P95=-2
N31  CGEN
N32  G02 XC[P80] YC[P81] R[P82] X[P94] Y[P95]
N33  G01 X4.2426 Y0
N34  X0 Y0
N35  G65 X0 Y-99
N36  G40
N37  G00 Z.1 M09
N38  M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N5 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X0 Y-2 at 25 ipm

N9-N14 are the coordinates of 3 points on the first circle

N15 Calculates circle based on the 3 points

N16 Arc command which moves to the calculated points

N17-N22 Are the coordinates of 3 points on the second circle

N23 Calculates second circle based on the 3 points

N24 Arc command which moves to the calculated points

N25-N30 Are the coordinates of 3 points on the third circle

N31 Calculates third circle based on the 3 points

N32 Arc command which moves to the calculated points

N33 Line move to X4.2426 Y0

N34 Line move to X0 Y0

N35 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N36 Cancels cutter compensation

N37 Rapids Z to .1, turns coolant off

N38 Turns spindle off

Conversational Program 4B

1. Program setup — Event 0
   - A. Dimensions ABSOLUTE *
   - B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   - A. Tool CHANGE *
   - B. Tool change position X________
     Y________
   - C. Tool number 1
   - D. Spindle speed 3000
   - E. Spindle restart CW *
   - F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   - A. Start (START)
     1. Z pierce feedrate 5
     2. Clearance .1
     3. Final Z depth -.375
     4. First Z depth -.375
     5. Z increment .375
     6. X pierce point 0
     7. Y pierce point 0
     8. Compensation LEFT *
     9. X before pierce 99
     10. Y before pierce 0

   - B. Geometry (GEOM) — Event 3
     1. Line (LINE)
        - a) feedrate 25
        - b) coordinates CARTESIAN *
        - c) X axis 0
        - d) Y axis -2

     2. Circle generate (CGEN) — Event 4
        - a) plane XY *
        - b) direction CCW *
        - c) X1[0] Y1[-2]
        - d) X2[1] Y2[-3]
        - e) X3[0] Y3[-4]
        - f) X3, Y3 end point [NO] *
        - g) end angle [300]

     3. Circle generate (CGEN) — Event 5
        - a) plane XY *
        - b) direction CCW *
        - c) X1[.1213] Y1[-2.2929]
        - d) X2[2.1213] Y2[-4.2929]
        - e) X3[4.1213] Y3[-2.2929]
        - f) X3, Y3 end point [NO] *
        - g) end angle [300]

4. Circle generate (CGEN) Event 6
   a) plane XY *
   b) direction CW *
   c) X1 [4.2426] Y1 [-4]
   d) X2 [3.2426] Y2 [-3]
   e) X3 [4.2426] Y3 [-2]
   f) X3, Y3 end point [YES] *

5. Line (LINE) Event 7
   a) coordinates CARTESIAN *
   b) X axis 4.2426
   c) Y axis 0

6. Line (LINE) Event 8
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 0

7. Previous menu (BACK)

C. End mill cycle (END) Event 9
   1. Point after retract X 99
      Y 0

D. Miscellaneous (MISC) Event 10
   1. Spindle OFF *
   2. Coolant OFF *

4. End program (EXIT) Event 11

SAMPLE 5

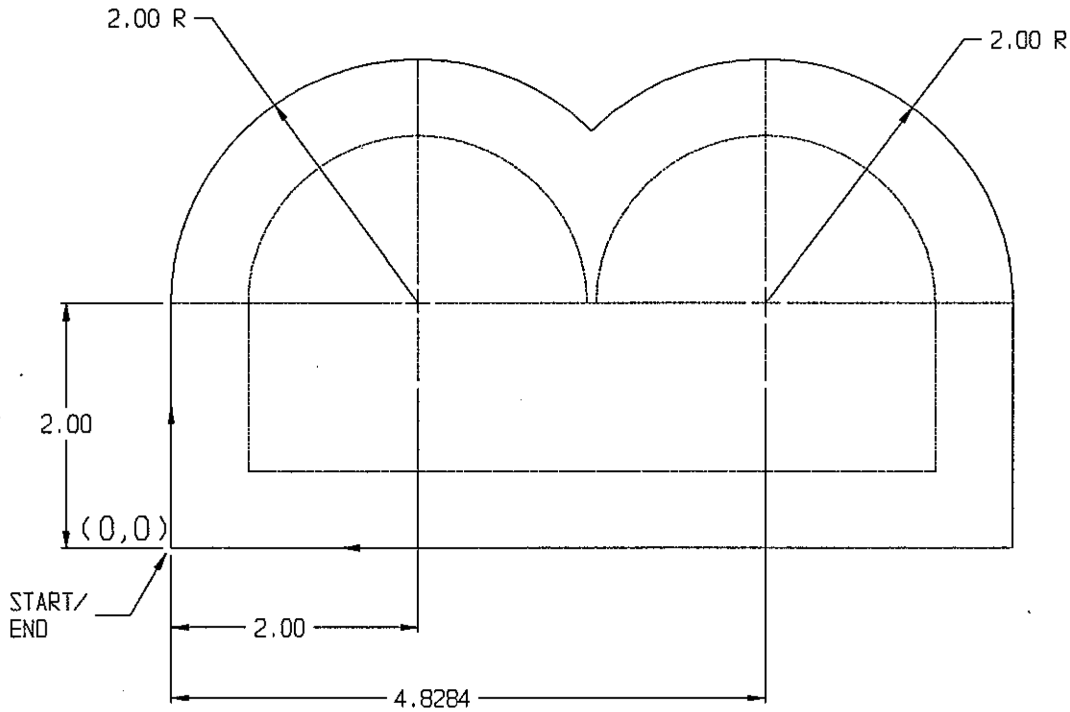


Figure 16.5

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    G42 D01 S3000 M03
N4    G65 X0 Y0
N5    X0 Y0
N6    G43 H01 Z.1 M8
N7    G01 Z-.375 F5
N8    G01 X0 Y2 F25
N9    G02 XC2 YC2 X2 Y4 R2,R.0001
N10   G02 XC4.8284 YC2 X6.8284 Y2 R2
N11   G01 X6.8284 Y0
N12   X0 Y0
N13   G65 X0 Y99
N14   G40
N15   G00 Z.1 M09
N16   M05
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects right cutter compensation, calls tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N5 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X0 Y2 at 25 ipm

N9 CW arc 2" radius using an XC2 YC2, an estimated end point of X2 Y4, and an end option (round corner) of .0001" radius
Note: The end option of a .0001 radius forces an intersection between the arcs.

N10 CW arc 2" radius using an XC4.8284 YC2 and an end point of X6.8284 Y2

N11 Line move to X6.8284 Y0

N12 Line move to X0 Y0

N13 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N14 Turns off cutter compensation

N15 Rapids Z to .1, turns coolant off

N16 Turns spindle off

Conversational Program 5

1. Program setup — Event 0
   - A. Dimensions ABSOLUTE *
   - B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   - A. Tool CHANGE *
   - B. Tool change position X________
     Y________
   - C. Tool number 1
   - D. Spindle speed 3000
   - E. Spindle restart CW *
   - F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   - A. Start (START)
     1. Z pierce feedrate 5
     2. Clearance .1
     3. Final Z depth -.375
     4. First Z depth -.375
     5. Z increment .375
     6. X pierce point 0
     7. Y pierce point 0
     8. Compensation RIGHT *
     9. X before pierce 99
     10. Y before pierce 0

   - B. Geometry (GEOM) — Event 3
     1. Line (LINE)
        - a) feedrate 25
        - b) coordinates CARTESIAN *
        - c) X axis 0
        - d) Y axis 2

     2. Arc (ARC) — Event 4
        - a) plane XY *
        - b) direction CW *
        - c) coordinatess ABS CENTER *
        - d) arc radius 2
        - e) arc center XC 2
          YC 2
        - f) end point X 2
          Y 4
        - 7) end option ROUND CORNER *
          radius .0001

     3. Arc (ARC) — Event 5
        - a) plane XY *
        - b) direction CW *
        - c) coordinates ABS CENTER *
        - d) arc radius 2

e) arc center XC 4.8284
YC 2

f) end point X 6.8284
Y 2

4. Line (LINE) — Event 6
   a) coordinates CARTESIAN *
   b) X axis 6.8284
   c) Y axis 0

5. Line (LINE) — Event 7
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 0

6. Previous menu (BACK)

C. End mill cycle (END) — Event 8
   1. Point after retract X 0
   Y 99

D. Miscellaneous (MISC) — Event 9
   1. Spindle OFF *
   2. Coolant ON *

4. End program (EXIT) — Event 10

SAMPLE 6

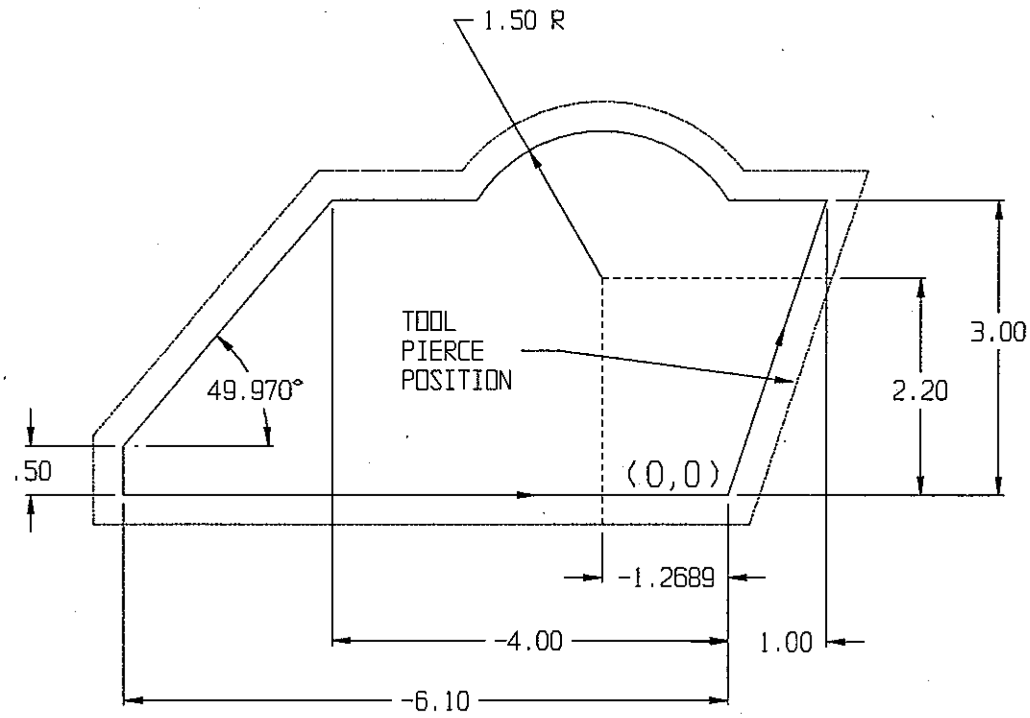


Figure 16.6

EIA PROGRAM

```
N1    G00 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    G42 D01 S3000 M03
N4    G65 X0 Y0
N5    X.5 Y1.5
N6    G43 H01 Z.1 M8
N7    G01 Z-.375 F5
N8    X1 Y3 F25
N9    X.98 Y3
N10   G03 XC-1.2689 YC2.2 X-3.8 Y3 R1.5
N11   G01 X-4 Y3
N12   X-6.1 Y.5
N13   X-6.1 Y0
N14   X0 Y0
N15   X.5 Y1.5
N16   G65 X1 Y3
N17   G40
N18   G00 Z.1
N19   M30
```

Explanation of EIA Program

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning.

N2 Tool change #1

N3 Selects right cutter compensation, activates #1's "D" offset, and turns on spindle CW (3000 rpm)

N4 Establishes a "point before pierce" of X0 Y0
Note: Machine does not move to this position.

N5 Establishes a "pierce point" of X.5 Y1.5; moves to its compensated point as established by the previous block

N6 Calls tool #1's "H" offset, positions Z to .1, and turns coolant on

N7 Feeds Z-.375 at 5 ipm

N8 Line move to X1 Y3 at 25 ipm

N9 Line move to an estimated end point of X.98 Y3

N10 CCW arc 1.5" radius using an XC-1.2689 YC2.2 and an end point of X-3.8 Y3

N11 Line move to X-4 Y3

N12 Line move to X-6.1 Y.5

N13 Line move to X-6.1 Y0

N14 Line move to X0 Y0

N15 Line move to X.5 Y1.5

N16 Establishes a "point after retract" of X1 Y3
Note: Machine does not move to this position.

N17 Turns cutter compensation off

N18 Rapids Z to .1

N19 Ends program, turns spindle and coolant off

Conversational Program 6

1. Program setup — Event 0
   - A. Dimensions ABSOLUTE *
   - B. Units ENGLISH *

2. Tool change (TCHNG) — Event 1
   - A. Tool CHANGE *
   - B. Tool change position X______
     Y______
   - C. Tool number 1
   - D. Spindle speed 3000
   - E. Spindle restart CW *
   - F. Coolant FLOOD *

3. Mill (MILL) — Event 2
   - A. Start (START)
     1. Z pierce feedrate 5
     2. Clearance .1
     3. Final Z depth -.375
     4. First Z depth -.375
     5. Z increment .375
     6. X pierce point .5
     7. Y pierce point 1.5
     8. Compensation RIGHT *
     9. X before pierce 0
     10. Y before pierce 0

   - B. Geometry (GEOM) — Event 3
     1. Line (LINE)
        - a) feedrate 25
        - b) coordinates CARTESIAN *
        - c) X axis 1
        - d) Y axis 3

     2. Line (LINE) — Event 4
        - a) coordinates CARTESIAN *
        - b) X axis .98
        - c) Y axis 3

     3. Arc (ARC) — Event 5
        - a) plane XY *
        - b) direction CCW *
        - c) coordinates ABS CENTER *
        - d) radius 1.5
        - e) arc center XC -1.2689
          YC 2.2
        - f) end point X -3.8
          Y 3

4. Line (LINE) — Event 6
   a) coordinates CARTESIAN *
   b) X axis -4
   c) Y axis 3

5. Line (LINE) — Event 7
   a) coordinates CARTESIAN *
   b) X axis -6.1
   c) Y axis .5

6. Line (LINE) — Event 8
   a) coordinates CARTESIAN *
   b) X axis -6.1
   c) Y axis 0

7. Line (LINE) — Event 9
   a) coordinates CARTESIAN *
   b) X axis 0
   c) Y axis 0

8. Line (LINE) — Event 10
   a) coordinates CARTESIAN *
   b) X axis .5
   c) Y axis 1.5

9. Previous menu (BACK)

C. End mill cycle (END) — Event 11
   1. Point after retract X 1
      Y 3

D. Miscellaneous (MISC) — Event 12
   Miscellaneous line: [M30]

4. End program (EXIT) — Event 13

Note: If an M30 or M02 is used at the end of a program, that program cannot be used as a subroutine since these codes automatically terminate a program.

SAMPLE 7
Sample Program Using Rotary Axis

The "A" axis is programmed in decimal degrees in XXX.XXX format and performs linear interpolation with the X, Y, and Z axes.

The feedrate for the rotary axis is specified in degrees per minute divided by 10, example:

```
G1 A90 F18.0
```

In the above example, A will feed to 90 degrees at a rate of 180 DPM (degrees per minute).

ROTARY AXIS PROGRAMMING EXAMPLE

```
03119                  A axis moves - Cut 4 helical slots
G20  G90
N1 G00 Z.1             Position Z above the work piece
N2 G00 X0 Y0 A0        Position X, Y, and A to the start point
N3 G01 F20 Z-.1        Feed Z to depth
N4 G01 F15 X-5 A90     XA interpolation move
N5 G00 Z.1             Position Z above the work piece
N6 G00 X0              Position X back to the start point

N7 G01 F20 Z-.1        Second slot
N8 G01 F15 X-5 A180
N9 G00 Z.1
N10 G00 X0

N11 G01 F20 Z-.1       Third slot
N12 G01 F15 X-5 A270
N13 G00 Z.1
N14 G00 X0

N15 G01 F20 Z-.1       Fourth slot
N16 G01 F15 X-5 A360
N17 G00 Z.1
N18 G00 X0 Y0
 (End of 03119)
```

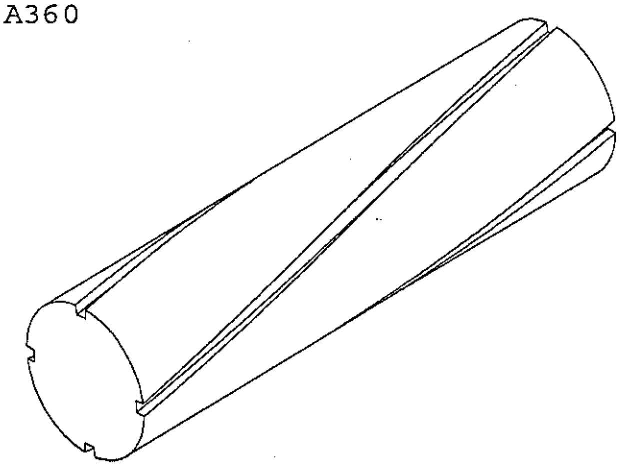


Figure 16.7

APPENDIX A

Control Parameters

<u>Great</u> <u>care</u> must be taken when writing to any parameters other than the User Parameters, P00 - P99.

P140	R plane
P141	Final Z depth
P142	Z initial level
P143	Z increment
P144	First Z depth
P145	Z feedrate
P146	Peck up increment
P147	Peck clearance
P148	Dwell 1
P149	Dwell 2
P150	Pocket radius
P151	X pocket dimension
P152	Y pocket dimension
P153	XY finish stock
P154	Z finish stock
P155	Cut width
P156	Bolthole radius
P157	Bolthole start angle
P158	# holes in 360 degrees
P159	# holes to be made
P160-	Unassigned
P171	"
P172	Mirror position axis 1 (X)
P173	Mirror position axis 2 (Y)
P174	Mirror position axis 3 (Z)
P175	Mirror position axis 4 (A)
P176	Mirror pos. optional axis
P177	Mirror pos. optional axis
P178	Mirror pos. optional axis
P179	Mirror pos. optional axis
P180	Scale position axis 1 (X)
P181	Scale position axis 2 (Y)
P182	Scale position axis 3 (Z)
P183	Scale posiiton axis 4 (A)
P184	Scale position opt. axis
P185	Scale position opt. axis
P186	Scale position opt. axis
P187	Scale position opt.axis
P188	Scale factor axis 1 (X)
P189	Scale factor axis 2 (Y)
P190	Scale factor axis 3 (Z)
P191	Scale factor axis 4 (A)
P192	Scale factor opt. axis
P193	Scale factor opt. axis
P194	Scale factor opt. axis
P195	Scale factor opt. axis

P196	Rotate I position
P197	Rotate J position
P198	Angle of rotation
P199	Unassigned
P200	Previous position (X)
P201	Previous position (Y)
P202	Previous position (Z)
P203	Previous position (A)
P204	Previous pos. opt. axis
P205	Previous pos. opt. axis
P206	Previous pos. opt. axis
P207	Previous pos. opt. axis
P208	Current position (X)
P209	Current position (Y)
P210	Current position (Z)
P211	Current position (A)
P212	Current pos. opt. axis
P213	Current pos. opt. axis
P214	Current pos. opt. axis
P215	Current pos. opt. axis
P216	Previous machine (X)
P217	Previous machine (Y)
P218	Previous machine (Z)
P219	Previous machine (A)
P220	Previous mach. opt. axis
P221	Previous mach. opt. axis
P222	Previous mach. opt. axis
P223	Previous mach. opt. axis
P224	Current machine (X)
P225	Current machine (Y)
P226	Current machine (Z)
P227	Current machine (A)
P228	Current machine opt. axis
P229	Current machine opt. axis
P230	Current machine opt. axis
P231	Current machine opt. axis
P232	Work offset axis 1 (X)
P233	Work offset axis 2 (Y)
P234	Work offset axis 3 (Z)
P235	Work offset axis 4 (A)
P236	Work offset optional axis
P237	Work offset optional axis
P238	Work offset optional axis
P239	Work offset optional axis
P240	Tool offset axis 1 (X)
P241	Tool offset axis 2 (Y)

P242	Tool offset axis 3 (Z)
P243	Tool offset axis 4 (A)
P244	Tool offset optional axis
P245	Tool offset optional axis
P246	Tool offset optional axis
P247	Tool offset optional axis
P248	Arc radius
P149	Arc I value
P250	Arc J value
P251	Feedrate
P252	Dwell
P253	Spindle speed
P254	Temporary X position
P255	Temporary Y position
P256	Temporary Z position
P257	Temporary A position
P258	Temporary axis position
P259	Temporary axis position
P260	Active tool number
P261	Active tool radius
P262	Active tool length
P263	Active radius offset number
P264	Active length offset number
P265	Canned cycle active
P266	Zone status
P267	Unit ratio
P268	Pending tool
P269	Unused
P270	Temporary I position
P271	Temporary J position
P272	Temporary K position
P273-	Unassigned
P299	"
P300	Modal 00
P301	Modal 01
P302	Modal 02
P303	Modal 03
P304	Data mode
P305	H offset direction
P306	Interpolate
P307	Coordinates
P308	Active plane
P309	Cutter compensation
P310	Canned cycle
P311	Dimension
P312	Feed unit
P313	Spindle unit
P314	Spindle direction
P315	Linear unit
P316	Scale
P317	Rotate
P318	Mirror
P319	Work system
P320	Primary

P321	Secondary
P322	Tertiary
P323	Return plane
P324	Tapping
P325-	Unassigned
P399	"
P400	Work G92 axis 1 (X)
P401	Work G92 axis 2 (Y)
P402	Work G92 axis 3 (Z)
P403	Work G92 axis 4 (A)
P404	Work G92 optional axis
P405	Work G92 optional axis
P406	Work G52 axis 1 (X)
P407	Work G52 axis 2 (Y)
P408	Work G52 axis 3 (Z)
P409	Work G52 axis 4 (A)
P410	Work G52 optional axis
P411	Work G52 optional axis
P412	Work coordinate 1 (X)
P413	Work coordinate 1 (Y)
P414	Work coordinate 1 (Z)
P415	Work coordinate 1 (A)
P416	Work coord. 1 opt. axis
P417	Work coord. 1 opt. axis
P418	Work coordinate 2 (X)
P419	Work coordinate 2 (Y)
P420	Work coordinate 2 (Z)
P421	Work coordinate 2 (A)
P422	Work coord. 2 opt. axis
P423	Work coord. 2 opt. axis
P424	Work coordinate 3 (X)
P425	Work coordinate 3 (Y)
P426	Work coordinate 3 (Z)
P427	Work coordinate 3 (A)
P428	Work coord. 3 opt. axis
P429	Work coord. 3 opt. axis
P430	Work coord. 4 (X)
P431	Work coord. 4 (Y)
P432	Work coord. 4 (Z)
P433	Work coord. 4 (A)
P434	Work coord. 4 opt. axis
P435	Work coord. 4 opt. axis
P436	Work coordinate 5 (X)
P437	Work coordinate 5 (Y)
P438	Work coordinate 5 (Z)
P439	Work coordinate 5 (A)
P440	Work coord. 5 opt. axis
P441	Work coord. 5 opt. axis
P442	Work coordinate 6 (X)
P443	Work coordinate 6 (Y)
P444	Work coordinate 6 (Z)
P445	Work coordinate 6 (A)
P446	Work coord. 6 opt. axis
P447	Work coord. 6 opt. axis

P448 Tool change offset (X)
P449 Tool change offset (Y)
P450 Tool change offset (Z)
P451 Tool change offset (A)
P452 Tool chg. offset opt.axis
P453 Tool chg. offset opt.axis
P454 Positive safe zone (X)
P455 Positive safe zone (Y)
P456 Positive safe zone (Z)
P457 Positive safe zone (A)
P458 Positive safe zone opt.
P459 Positive safe zone opt.
P460 Negative safe zone (X)
P461 Negative safe zone (Y)
P462 Negative safe zone (Z)
P463 Negative safe zone (A)
P464 Negative safe zone opt.
P465 Negative safe zone opt.
P466- Unassigned
P499

P500- Tool D table offset
P599 (Tool table radii 0-99)

P500- D table offset
P599 (Offset table radii 0-99)

P600- Tool H table offset
P699 (Radii 0-99)

P600- H Table offset
P699 (Offset table lengths 0-99)

P700 Machine units - ballscrew type: Eng/metric
P701 Number of axes
P702 Feed units-Power on feed (UPM/UPR/Inverse)
P703 Spindle units-Power on spindle units
P704 Power on feedrate
P705- Feedrate override
P720 starting address 0-15
P721- Handwheel override
P736 starting address 0-15
P737 Tool change auto/manual switch
P738 Spindle axis
P739- Spindle RPM/DAC starting
P754 address 0-15
P755- Spindle override starting
P770 address 0-15

P771 English cartesian leading positions
P772 English cartesian trailing positions
P773 Metric cartesian leading positions
P774 Metric cartesian trailing positions
P775 English angular leading positions
P776 English angular trailing positions
P777 Metric angular leading positions
P778 Metric angular trailing positions
P779 English spindle leading positions
P780 English spindle trailing positions
P781 Metric spindle leading positions
P782 Metric spindle trailing positions
P783 English feed leading positions
P784 English feed trailing positions
P785 Metric feed leading positions
P786 Metric feed trailing positions
P787 Run rapid 100
P788 Dry rapid 100
P789 Spindle on dry
P790 Spindle range 0
P791 Spindle range 1
P792 Spindle range 2
P793- Unassigned
P799 "
P800- Post M code table
P809 Table of M codes that occur after move
P810- Unassigned
P899 "
P900 Level 1 password
P901 Level 2 password
P902 Level 3 password
P903- Unassigned
P999 "

P1000- Axis 1 Address
P1045

P1100-
P1145 Axis 2 Address
P1200-
P1245 Axis 3 Address
P1300-
P1345 Axis 4 Address
P1400-
P1445 Axis 5 Address
P1500-
P1545 Axis 6 Address

P1000 Axis address label (X)
P1001 Pulses per unit (X)
P1002 Home position (X)
P1003 Home direction (X)
P1004 Positive limit (X)
P1005 Negative limit (X)
P1006 Maximum feed (X)
P1007 Dry run feed (X)
P1008 Rapid velocity (X)
P1009 Rapid acc/dec (X)
P1010 Home sequence (X)
P1011 Velocity toward home (X)
P1012 Velocity away from home (X)
P1013 Velocity toward marker (X)
P1014 Encoder multiplier (X)
P1015 Slow jog velocity (X)
P1016 Slow jog acc/dec (X)
P1017 Rapid jog velocity (X)
P1018 Rapid jog acc/dec (X)
P1019 In position (X)
P1020 G00 unidirectional (X)
P1021 G60 unidirectional (X)
P1022 Backlash (X)
P1023 Excess error (X)
P1024 Rotary=0 Linear=1 (X)
P1025- Unassigned (X)
P1040 "
P1041 Home switch=0 Marker=1
P1042 G28 reference point (X)
P1043 G30 reference point 2 (X)
P1044 G30 reference point 3 (X)
P1045 G30 reference point 4 (X)

ERROR MESSAGES

001 *Note what just occurred and call for technical support.*

002 *File not found*
File name specified as OLD does not exist. Try MENU.

003 *Path not found*
Check path settings in Centurion V shell.

004 *Too many open files*
Check Config.sys for FILES=20.

005-007 *Note what just occurred and call for technical support.*

008 *Insufficient memory*
The program being loaded is too large to fit into the parts memory. Try erasing some programs from memory.
A program being run or verified is too large to run in the control's memory. Try RS232 run mode.

*009-*013 *Note what just occurred and call for technical support.*

015 *Invalid drive specified*
Drive specified does not exist. Try Set Paths in shell.

016-019 *Note what just occurred and call for technical support.*

020 *Send file aborted*
ESC was pressed while sending the file via RS232.

100 *Disk read error*
An attempt was made to edit a file that has been corrupted in some way, perhaps loss of power while editing, or an error 101 occurred while editing. Try a different file to see if the problem is specific to one particular file. If this is the case, the program must be recreated.

101 *Disk write error - Parts memory is full*
To avoid this error, remove programs from memory as you are done using them (store on a floppy). Also, watch the amount of memory available as you are programming. Deleting some programs from the parts memory will free up space for additional programs.

Procedure to recover from ERROR 101
(Disk write error: Parts memory full)

Type in commands that are shown in CAPITAL letters, followed by the ENT key.

1)	ENT	(B:RAM should be displayed)
2)	CD..	(change directory)
3)	CD PARTS	(change to the parts directory)
4)	DIR	(list PART files)
5)	DEL O####	(#### is your part number)

NOTE: You can recreate your O#### file from your conversational (P####) file by reposting it.

6) Power the machine OFF, then ON again, to RESET the control. Use UTIL-FILES-ERASE to erase programs that are no longer needed.

NOTE: You can check how much parts memory is available by using UTIL-INFO to look at the information page.

102-106 *Note what just occurred and call for technical support.*

150 *Disk is write-protected*
Check the write protect tab on the floppy disk that is being used.

151 *Note what just occurred and call for technical support.*

152 *Drive not ready*
Check to see that there is a disk in the floppy drive.

153-155 *Note what just occurred and call for technical support.*

156 *Disk seek error*
Check the cabling from the control to the floppy drive.

157-161 *Note what just occurred and call for technical support.*

162 *Hardware failure*
Improper format on the floppy disk. Try another disk or try reformatting the same disk.
Verify that floppy cables are properly connected.

163 *Zoom factor is too large*
Zoom+ was pressed too many times in DISPLAY-GRAPH mode.

200-202 *Note what just occurred and call for technical support.*

203 *Heap overflow - Insufficient RAM memory*
Run the DOS command CHKDSK to determine the amount of RAM that is available on the system. If possible, unload unnecessary device drivers that are loaded on the PC before starting FastCAM II.

Steps to take to avoid ERROR 203
(Heap overflow: Insufficient RAM memory)

If text cycles are being loaded and not being used, turn off the Load Text Cycles flag.

1) (F7) PARMS
2) (F9) CTRL
3) Move cursor to Load Text Cycles
4) (F1) EDIT
5) Enter a 0 (zero)
6) (ESC)
7) (ESC)
8) (ESC) to the main menu.
9) Power the machine OFF, then ON, again.

If you are running a large program, try running it through the RS232 RUN mode.

204-207 *Note what just occurred and call for technical support.*

300 *Program already exists*
An attempt was made to use a program name already in use. Try using a different name.

301 *Invalid program number*
Valid program numbers are 1 through 9999.

302 *No programs to select from*
This error may occur anytime a menu is being created for file selection when there are no files.
There may be an unformatted disk in the floppy drive.
Parts memory may be empty.

303 *Problem saving program(s) to disk*
There is no floppy disk in the disk drive.
The floppy disk may not have room to store additional files.
There may be an unformatted disk in the drive.

304 *Problem loading program(s) from disk*
Disk was removed from floppy drive after setting files.

305 *Not formatted for conversational. Try text editor.*

306 Note what just occurred and call for technical support.

307 *Illegal event number*
Event number in conversational program is negative.

308 *Invalid tool number*
Tool number is less than zero or greater than 99.

309 *Can't copy or rename a file to itself*
Try using a different file name.

310 *File not formatted for conversational or parameters*
Problem receiving a file via RS232.
Check to see if the proper file was sent.

311 *Parameter file not valid*
Problem receiving parameter file via RS232.
Check to see if the parameters were sent.

312 *Insufficient parts storage*
Parts memory is full. Try erasing some programs from the parts directory (UTILS-FILES-ERASE).
Program on floppy disk may be too large to fit into the control's memory.

313 *Insufficient storage for compression, unable to post the file*
Erase some programs from the parts memory.

314 *Insufficient storage, post has been aborted*
Erase some programs from the parts memory.

315 *Out of storage space on the floppy*
Floppy disk is full or has too many programs. There is a limit of 224 files that can be stored in the root directory on a floppy disk.
A sub-directory can be created on the floppy disk and the floppy path changed to save files to the sub-directory. This allows full use of the disk space.

316 *Not enough storage to create a new file*
There is not enough parts memory to create a new conversational program.
Erase unnecessary programs to free up memory space.

400 Home required

The machine must be homed before any axis movement can take place on the machine, i.e. MDI, JOG, HDW, etc.

The "home sequence" parameters can be modified so that the machine will not actually home when commanded. Set "home sequence" to X 00.0000 Y00.0000 Z00.0000.

401 X-axis software limit overtravel
402 Y-axis software limit overtravel
403 Z-axis software limit overtravel
404 A-axis software limit overtravel
405 B-axis software limit overtravel
406 C-axis software limit overtravel

These errors are a result of the axis reaching the programmed limits of travel.

When jogging or handwheeling the axis and a limit is reached, the control will allow movement in one allowable direction.

In a program the error may be encountered if the G92 and/or G54 parameters put you out of limit.

Tool length offsets are also a potential cause of Z axis overtravel.

407 X-axis excess error condition
408 Y-axis excess error condition
409 Z-axis excess error condition
410 A-axis excess error condition
411 B-axis excess error condition
412 C-axis excess error condition

These errors are caused by the axis not being able to keep up with the programmed move at the programmed speed.

Does the error occur during rapid moves only? Y__ N__
If so, check buss voltage and rapid feed parameters.

Do any of the drive cards have red LED's lit? Y__ N__
If so, which light on which card? VOL, GF, RMS, or SG

Other causes might be too heavy of a cut, worn tool, low ±15 volts, accel/decel parameters, or drive card failure.

413 Attempted to move into safe zone

414 Note what just occurred and call for technical support.

415 *Can't establish DNC link while program is running or verifying*
The program being run must be halted before the DNC link can be established.

416 *Out of position*

417 *Can't edit parameters while program is running*
The program must be halted before editing parameters.
Is the program in block mode or feedhold?

451 *Lube fault*
The float switch on the autolube pump is indicating that oil needs to be added to the autolube tank.

452 *Tool not found in auto tool sequence*
Check UTIL-TLCHG-SLOTS for that tool number.

453 *Tool pot not up during turret movement*
Check to see if the POT UP switch is functioning as it should be.

454 *Not at tool change position*
Try commanding a G32 before the M6 command.

500 *Last softkey pressed is not supported at this time*

501 *Illegal address " " encountered*
The character within quotes " " is not a valid address, such as X, Y, Z, R, G, etc.
The block where the error occurred is shown in the block display. Check that block for the invalid address.

502 *Undefined canned cycle*

503 *Return without gosub*
Refer to Section 6 about gosub and return.

504 *Coincident points*
The start point and end point are the same on an arc without a center.

505 *Radius too small to span given points*
Start and end points are more than "R" distance apart.

507 *Compensated line/arc does not intersect*

509 *No arc/arc intersection*

517 *Parameter out of range*
Parameter number is less than zero.
For parameter numbers greater than 499 you must use data mode (G10, G11).

518 *Illegal program statement*
Command in program statement is not considered valid.

519 *Feedrate out of range*
The programmed feedrate is beyond the "maximum feedrate" parameter value in the machine setup parameters.
The program feedrate may be negative.

520 *Spindle speed out of range*
The programmed spindle speed is beyond the "spindle range" parameter in the machine setup parameters.
The programmed spindle speed may be negative.

521 *Negative arc radius*
An attempt was made to generate a negative arc radius.

522 *Negative polar radius*
A polar radius must be specified as a positive value.

523 *Illegal tool number*
Valid T numbers are 0 - 99.

524 *Illegal radius number*
Valid D numbers are 0 - 99.

525 *Illegal length number*
Valid H numbers are 0 - 99.

526 *Invalid access code*
The access code does not match that which is loaded in the machine setup parameters.

527 *Invalid access level*
Valid levels are 0 - 4.

529 *Duplicate address encountered*
The same address was found twice on the same block, such as →X0 Y0 X.5.

530 *Collinear line to line in round corner*

531 *Collinear line to arc in round corner*

532 *Collinear arc to line in round corner*

533 *Collinear arc to arc in round corner*

535 *Chamfer length is < 0*
Chamfer length must be a positive number.

536 *Can't chamfer and round the same corner*
Choose either chamfer OR round corner.

537 *Can't chamfer to or from arcs*

538 *Loop counter out of range*
The maximum number of loops for a call is 999.

539 Dwell time out of range
Probably a negative number was specified. The maximum dwell time is 999999999. seconds.

540 *Illegal dwell time " " encountered*
Try G4 F##.####; specify X, P, or F after G4.

541 *No axes moves are allowed on a G31 or G32 block*
G31 and G32 are intended to move Z only.
Relocate X and Y moves to another block.

542 *G30 Illegal return to reference parameter on G30 block*
Should be P2, P3, or P4 for second, third, and fourth reference point.

543 *Illegal G10 statement*

544 *Too many digits in number*
The number of digits used is beyond what the address is expecting, example: G100 should only be two digits.

545 *Illegal K value for number of holes*
K must be a number between 1 and 1000.

546 *Nested calls or gosubs too deep*
Probably a program is calling itself.
Nest limit is 50 for program calls.

547 *Comment not closed*
Always use "( )" (parentheses) in pairs for program comments.

548 *M-Code out of range*
M-code must be a number between 0 and 99.

549 *Unrecognized G-Code*
G-code encountered is not recognized by the control.

550 Bad numeric format
Expecting a numeric value, or a parameter value enclosed within [ ], after an address X, Y, Z, R, etc.

551 *Multiple decimal points*
Multiple decimal points were detected within one numeric value.

552 *Missing "]"*
Always use square brackets in pairs.

553 *Missing "["*
Always use square brackets in pairs.

554 *Tangent function overflow*
Trying to find the tangent of a number close to 90°

555 *Missing "/"*
Arctan "ATAN" syntax is P## = ATAN[#/#].

556-560 *Note what just occurred and call for technical support.*

567 *Unresolved call*
Program being called does not exist (Call ####).

568 *Unresolved goto or gosub*
N#### does not exist in the program (Goto ####).

569 *The tool is too large to cut inside the arc*
"Compensated radius is too small"
Eliminate the arc, or use a smaller tool.

570 *The tool is too large to cut inside the arc*
"1st compensated radius in arc to arc is < 0"
Eliminate the arc, or use a smaller tool.

571 *The tool is too large to cut inside the arc*
"2nd compensated radius in arc to arc is < 0"
Eliminate the arc, or use a smaller tool.

572 *Pocket clear is not in a Start/End mill cycle*
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

573 *Round wall is not in a Start/End mill cycle*
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

574 *Round wall radius will not span 1st Z depth and final Z depth*

575 *Tapered wall is not in a Start/End mill cycle*
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

576 *Z increment is 0*

577 *Input statements must precede axes moves*

578 *Undefined text cycle*
Character specified in a text command is not supported.
Load Text Cycles parameter not set in PARMS-CTRL.

579 *Compensated arc/arc does not intersect*

600 *Can't nest Start/End mill cycles -WHILE WEND loops-*
Do not start a mill cycle within a mill cycle.

601 *Missing WHILE statement*
May be an end mill cycle without a start mill cycle.

602 *Missing WEND statement*
May be a start mill cycle without an end mill cycle.

603 *Program does not exist*
Program being called as a subprogram does not exist. Check to see if the program called is in the memory.

605 *Can't modify dry run status while program is running*
Program must be halted before changing dry run status. Try HALT-DRY-RESUME.

606 *Program is empty*
Text program being run or verified is empty. Try editing and reposting the conversational file.

607 *Can't exit DNC run mode while program is running*
The DNC mode must be halted before exiting.

608 'P' expected in M98 block

800 *Illegal probe block*
Syntax error in guidance file. Check the block format. Blocks must start with an X, Y, or Z which is to be followed by one or two P commands.

801 *Missing end of pick*
Pick boundary is not closed. Input file did not end with a pick boundary definition block.

802 *Reversed scan segment*
Multiple scan segment started that would cause the scan direction to change in the middle of the current scan.

803 *Missing end of scan*
Pick segment terminated before the end of scan was defined. There should be at least two scan definition blocks between each pick boundary.

804 *Reversed pick segment*
Multiple pick segment started that would cause a change in pick direction. Use multiple probe segments if this is the desired intention.

805 *Invalid probe setup*
Input file does not start with a comment containing three asterisks. Also, the following three blocks should be X, Y, Z, or Y, X, Z depending on scan plane.

806 *Scan origin expected*
Multiple pick segment started without defining the start of the scans within that segment.

807 *Probe file not found*
Could not find the selected input file.

808 *Setup not selected*
Tried to probe without selecting both the input file and the output mode from the probe setup screen.

809 *Bad Z limits encountered*
The max Z height is less than the max Z depth value.

810 *Stuck digitizing probe*
Digitizing may have reached max Z height and part contact is still detected. If the probe is not actually touching the part, the probe may require maintenance. If it is touching the part, then the part has to be lowered or the max Z height should be increased.

900 *RS232 overrun error (The system sending data may not have the same baud rate as the CNC.)*
Check RS232 baud rate parameter in PARMS-CTRL.

901 *RS232 parity error (The system sending data may not have the same parity as the CNC.)*
Check RS232 parity parameter in PARMS-CTRL.

902 *RS232 framing error (Remote system and CNC may not have the same line settings or a loose cable.)*
Check line settings in PARMS-CTRL for baud rate, parity, and stop bits.

903 *RS232 break detected (RS232 cable may be loose.)*
Check cabling and connectors for good contact.

New Parameters

PARMS-SETUP-MISC

Minimum Parts Space ______**0 - 255** (Set to about 5% of memory. Usually 005K. The system will warn the operator to free up memory when the specified level is reached.)

G18 is_________________**XZ/ZX** (Change the way G18 works. With XZ the arc is described with 0 degrees at 3 Oclock and CW as viewed from the right of the machine. With ZX, 0 degrees is 12 Oclock and CW is viewed from the left side.

Special Flags_____________**0 or 2** (Normally set to Zero, If you need to disable "Trig Help" because of intersection errors with a CAD/CAM program, set the value to 2.)

Offset round/tapered walls_**Yes or No** (If you don't like the way Round / Tapered walls work, try changing this parameter.)

Screen Blank Time_________**0 - Min**(Set to 000 if no screen blank is desired, Otherwise enter the time in minutes. E.g. 030 means 30 Minutes.Press any key to un-blank screen.)

Use FLZ Instead of G54__**Yes or No** (Recommended setting here is G54. If using G54 instead of FLZ (G92) the part zero is retained in the event of Power loss, Homing the Machine, etc. Whereas with FLZ (G92), the zero would have to be reset.)

PARMS-CTRL

RS232 Buffer Size_________**1 - 255** (Specifies the size of the RS232 Buffer in 256K increments. Must not be set to zero if using RS232)

PARMS-SETUP-POWER

Load Text Cycles__________**Yes or No** (Whether or not to activate TEXT command for lettering. TEXT screen found in PROG-CONV-SPEC)

Don't Load Canned Cycles___**Yes or No** (Normally NO, YES only to save main memory for loading Point To Point Programs that do not use Drill, Mill, and Auto routine cycles.)

Note: After setting the above parameters, recheck them to see that they are all correct. Then power the machine off then back on to let them take effect.

1.29.28 PARTNER 1 AND 6 TOOL CHANGE SETUP PARAMETERS

Parameter	Setting	Meaning
ATC Type is	**P1-6/12**	(Partner 1 or 6, 6 or 12 pocket)
ATC Tool Pocket Count	**6 or 12**	(Number of pockets you have)
M6 (Tool Change) Macro	**A:\TC\P1-6.ATC**	(Path of program in ROM)
Partner 1 ATC is	**Plunger**	(Older plunger style)
	Geneva 1 Step	(Geneva 12 pocket)
	Geneva 2 Step	(Geneva 6 pocket)

M6 (Tool Change) Macro: Type in using following keystrokes (A *shift* . *shift* - TC *shift* - P1 - 6 . ATC *enter*)

Note: After setting the above parameters, recheck them to see that they are all correct. Then power the machine off then back on to let them take effect.