SECTION IX

BRIDGEPORT OPERATING SYSTEM SOFTWARE 5.0 and 6

(BOSS 5 and 6)

The machine must be fitted with the correct ERS card containing the BOSS 5.0 software and also the LSI-11 microcomputer must contain the additional chip for floating point arithmetic.

BOSS 6 is factory fitted only.

9.1 SPECIFICATIONS

The specific features are summarized below. Most features require a close understanding of the use of new preparatory codes and new functions with their parameters:

Multi-Quadrant Circular Interpolation: G75 is a modal command in which the I, J interpolation parameters become the absolute coordinates of the circle center. Any arc up to 360 degrees can be programmed in a single block. Cancel with G74.

Cutter Diameter Compensation: A new tool path, at all times equidistant from the programmed path, will be generated after invoking the G41 or G42 code. The distance that the path is separated from that programmed is the compensation dimension defined as half the difference between the actual tool diameter and the programmed tool diameter. The full diameter difference is the value input to the system either through the MDI Keyboard or embedded in the program.

Polar Coordinate Commands: The radius vector distance (R) and the angle it makes (A) with the polar axis will define any point relative to the pole. The pole is defined in absolute coordinates by its I and J distance from absolute zero. Positioning for Z axis cycles or milling around an arc can thus be generated. Milling an arc with increasing or decreasing Z depth (The Helix) can also be programmed by means of looping.

Cutter Path Transformation: The entire cutter path, and thus a part shape with symmetry, can be rotated through an angle (A) degrees. The initial angle may be preset (G92) and the programmed shape rotated an absolute number of degrees or in conjunction with looping techniques, rotated an incremental amount each time the program is looped. G73 with the (A) angle will turn transformation ON,. G72 turns it OFF.

The cutter path can also be transformed by increasing or decreasing its magnitude (Scaling) relative to its programmed dimensions. G73 followed by X (scale factor) and Y (scale factor) is permitted.

Macro Sub-Routines: The number of macros has been increased from 16 to 36.

Manual Data Input: Permits circular interpolation input in this mode.

Interface: A 20 ma current loop and an RS-232C interface are standard with this system.

Remote Data Input: Permits computer testing at the Factory from a remote source.

9.2 MULTI QUADRANT CIRCULAR INTERPOLATION

G74 (Modal) — The I and J values are the distance from the arc center to the start point of the arc. It is an incremental unsigned value. This is the startup condition.

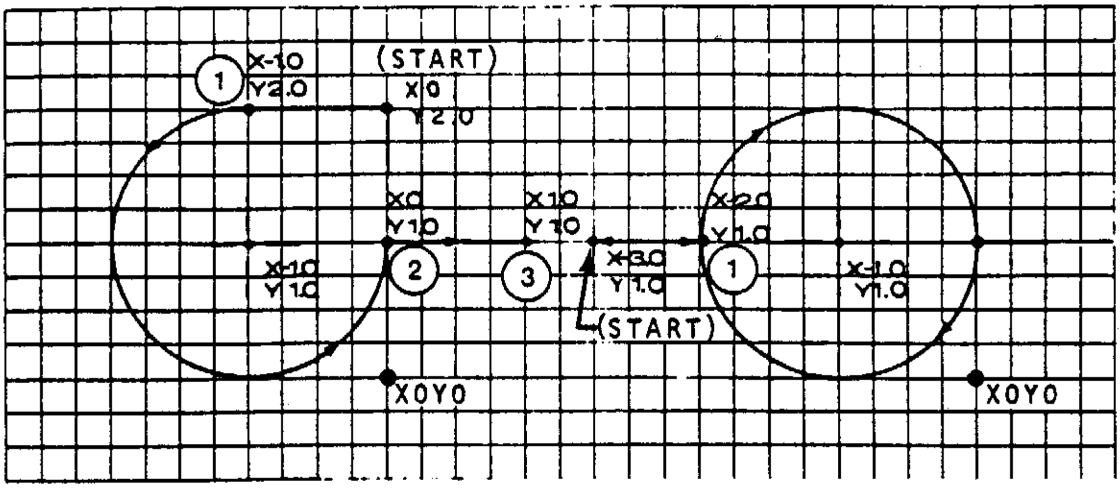
NOTE: (G74) is NOT Multi Quadrant, but is shown here for I and J value comparisons.

G75 (Modal) Abs — The I and J interpolation parameters are the absolute rectangular coordinate dimensions of the arc center from the chosen absolute zero.

*Restricted to G17 (XY plane)

G75 (Modal) Incr — The I and J interpolation parameters are the signed distances from the arc center to the start point of the arc. *Restricted to G17 (X,Y Plane)

G75 permits an arc of up to 360 degrees to be generated in a single block of input data without deceleration taking place at the quadrant cross-over points.

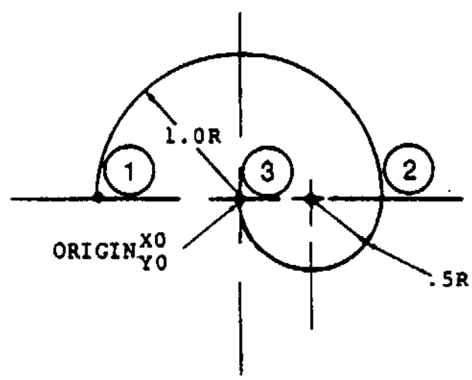


ABSOLUTE	INCREMENTAL	ABSOLUTE	INCREMENTAL
.N10G75G90	.N10G75G91	.N10G75G90	.N10G75G91
(1) N15G1X-1.0Y2.0F100	N15G1X-1.0F100	N15G1X-2.0Y1.0F100	N15G1X1.0F100
(2) N20G3X0Y1.0I-1.0J1.0	N20G3X1.0Y-1.0I0J1.0	N20G2X-2.0Y1.0I-1.0J1.0	N20G2X0Y0I-1.0J0
(3) N25G1X1.0	N25G1X1.0	N25G1X-3.0	N25G1X-1.0
etc.			

Rules in the G75 Mode:

1. All X, Y, I, J values must be entered, even if their value is zero.
2. G02, G03 commands are not modal.
3. G75 must be stated <u>before</u> entering circular.
4. Do not program G75 in the block preceeding G41/G42 (Cutter Compensation).

```
    .N5G75
(1) N10G0G90X-1.Y0
(2) N15G2X1.Y0I0J0F100
(3) N20G2X0Y0I.5J0
```



9.3 CUTTER DIAMETER COMPENSATION

9.3.1. Introduction

Cutter Diameter Compensation has been defined as a displacement normal to the programmed cutter path. By its very title "Compensation" refers to a means whereby the operator allows for a difference between the nominal Cutter Diameter chosen by the programmer and the actual diameter of the end mill. It is then the purpose of the control to generate a new cutter path parallel to the old and perpendicular to the work piece at all times.

Differences in cutter diameters are normally due to tool wear and/or a reground diameter, or due to other deviations found necessary as a normal part of production. The programmer assumes that the chosen cutter diameter, set length and other features of the geometry of the tool will be used. Tools must be to the programmer's specifications. Cutter Diameter Compensation becomes an aid in achieving and maintaining those tolerances on the finished part regardless of changes in the diameter of the tool actually being used. The programmer will allow for cutter diameter variations by special preparatory functions (G40, G41, G42) in the program text.

That portion of the Bridgeport Control containing Cutter Diameter Compensation fulfills the definition by computing new points for the tool ahead of the current tool motion. Such points generate new blocks of data internally within the control. This data is dependent upon the stored program text containing the preparatory functions and the use of stored values of Compensation.

9.3.2. Data Entry

Entry of Cutter Diameter Compensation data to storage with the prescribed format can be made by MDI or in the programmed text (tape input).

By MDI
Entry may be made through the keyboard when the system is in the SETUP mode. (See M-128 - para. 3.6.1).

By Programming
Data with the format Tn//d may be listed at the front of, or may be embedded in a program. These stored values can only be changed utilizing the Editor, provided that the data has first been stored.

Format: Entry of compensation data takes the following form: Tn//d EOB

where n = Tool Number (Values 1 - 24)
d = Desired change to programmed cutter diameter. (Any value from 0.000 to 3.200" (32.00 mm) positive or negative.)

Values to be stored with the format Tn//d are values which change the cutter path from that of one specific tool diameter to one of another specific tool diameter. If the actual tool is larger than the programmed diameter, changes are positive and negative if the tool is smaller.

When cutter diameter compensation value is changed in the middle of a program by whatever means, the new value will not be utilized by the system until the tool in question is called by the part program.

The format for calling the value is Tn without //.
Tn can be part of a line:

For example: N125 G0X-8.0Y5.0T3M6 or N125T3

9.3.3. Programming Commands

Cutter Diameter Compensation will operate properly in any of the following modes of machine operation: -

G01 - Linear Interpolation, Feed Mode
G02 - Circular Interpolation CW, Feed Mode
G03 - Circular Interpolation, CCW, Feed Mode
G17 - XY Plane (only)
G70 - Inch dimension system (also the power ON or initialized state)
G71 - Metric dimension system.
G74 - Circular
G75 - Multi Quadrant Circular Interpolation
G90 - Absolute Coordinates
G91 - Incremental Coordinates

The various programming commands to invoke the action of Cutter Compensation are: - (Figure 9-1)

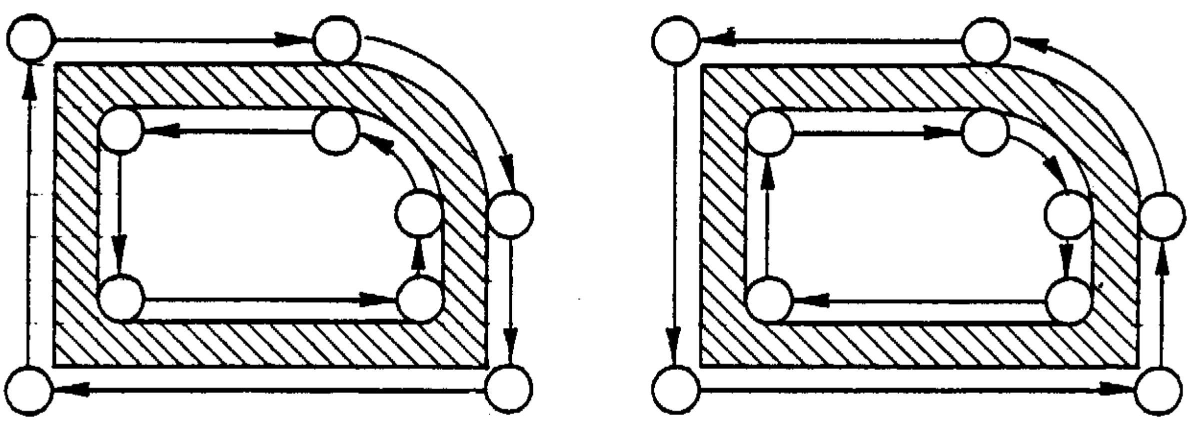
G41 - Turns cutter compensation ON and establishes the cutter to workpiece relationship where the cutter is to the left of the part surface in the direction of the cut.

G42 - Turns cutter compensation ON and establishes the cutter to workpiece relationship where the cutter is to the right of the part surface in the direction of the cut.

G40 - Turns Cutter Compensation OFF.

NOTE: With standard right hand milling cutters:

G41 (Climb Milling)
G42 (Conventional Milling)



(G41) (G42)

Figure 9-1. Programming Commands for Cutter Compensation

To simplify the determination of correct cutter orientation, consider yourself to be the cutter. Look in the direction of the first cut and define which side of the part the cutter is on. Use G41 if the cutter is to the left of the part. Use G42 if the cutter is to the right of the part.

9.3.4 Theory of Operation *

The compensation system is based upon knowledge of where the tool has been and where it is going at all times. To provide the system with sufficient information, it requires three programmed points. These points may be defined as follows:

"Current Point" - The point from which the tool is programmed to move.
"Active Point" - The point to which the tool is programmed.
"Next Point" - The next programmed point.

To determine the compensated cutter path, the system must generate a new point for the center of the tool. It does this by calculating the intersection of:

a path parallel to a line drawn thru the "current point" and the "active point", and
a path parallel to a line drawn thru the "active point" and the "next point";
each displaced by an amount equal to one-half of the compensated value that has been entered for that tool.

To illustrate these points, consider machining the outside surface of the part shown below:

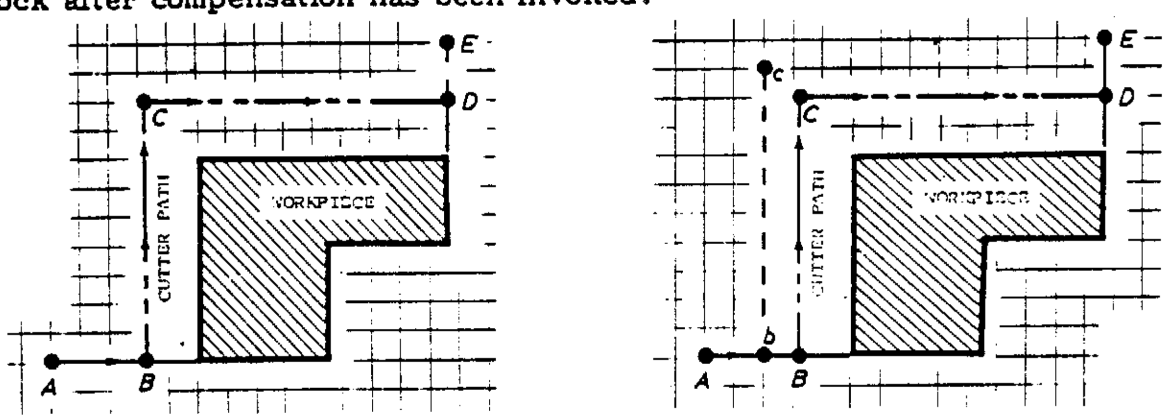
If the cutter was positioned at A and you wished to machine a Path from B to C to D; starting at A, and ending at E, then initially:

Point A would be the "Current Point"
Point B would be the "Active Point"
Point C would be the "Next Point"

To further illustrate: Consider that positive compensation has been invoked at Point A.

The system will generate a path parallel to BC and then compute the intersection point (b) where this path intersects AB.

You will notice that only one axis will be compensated for in the initial motion block after compensation has been invoked.



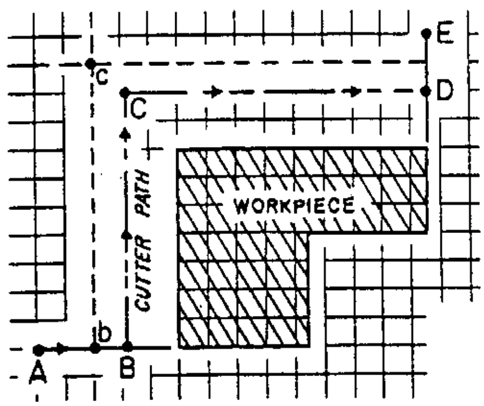
* Cutter Diameter Compensation is restricted to the X-Y plane only.

During continuous path contouring, the points of intersection progressively change as follows: the original "active point" becomes the new "current point," and the original "next point" becomes the new "active point" etc.

Referring to our workpiece again, with the cutter now positioned at b:

Point B has become our "Current Point"
Point C has become our "Active Point"
Point D has become our "Next Point"

The system must now calculate another new intersection point. To do this it will again generate a line parallel to BC and one parallel to CD, both displaced as before. Intersection point c is thus calculated and the cutter will move to this position.

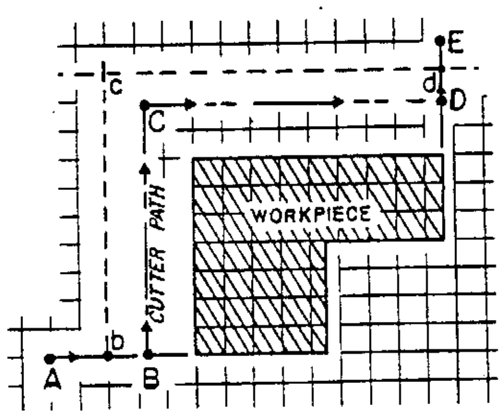


With the cutter now positioned at c:

Point C has become our "Current Point"
Point D has become our "Active Point"
Point E has become our "Next Point" ---Compensation will be turned off off in this motion block to point E.

The system as before, will generate compensated parallel lines and calculate the intersection point d at which time the cutter will move to this point.

The last move (with compensation now turned off) will be directly from d to point E.



9.3.4.1 Limitations

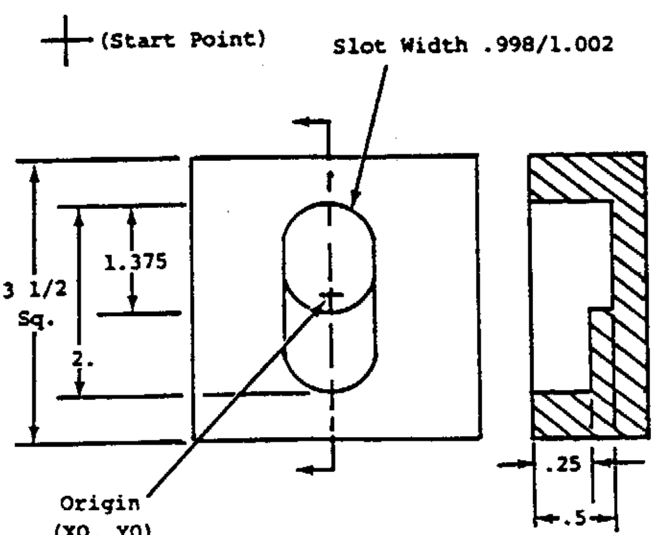
a. NON-COMPENSATED BLOCKS - Only one non-compensated block is permitted. Such a block is one that does not contain linear (G1) X and/or Y data. eg: Z axis motion, Loop Call, Macro Call, etc. If two blocks, one after another, contain such information; the compensation system will fail.

This is more easily understood by referring back to the first paragraph of Sec. 9.3.4 which dealt with the basic theory of Cutter Compensation. To reiterate, the control MUST have three unique points in order to calculate the new compensated cutter path, and they must lie in the "X-Y" plane. If, for example, a "Z" axis motion were programmed while in cutter compensation, the control will buffer (store) this one block momentarily and look ahead one additional block to obtain the third point it requires, however it can buffer ONLY ONE BLOCK.

b. COLLINEAR DATA - Though linear (G1) X and/or Y data will work correctly, the points MUST NOT be collinear (lying on or passing through the same straight line). Therefore, if two blocks, one after another, contain the same XY coordinates even if there is a non - XY block in between (see above), the system will fail.

Again, the system requirement of three unique points clearly points out the problem with Collinear Data; for by definition: "A line consists of an infinite number of points," thus the control is not able to calculate a specific intersection point.

The following example will illustrate one common machining configuration; where Collinear Data is likely to occur, and one method of programming around the problem.



Example 9-1

Programmed: 1/2 Dia. End Mill

Program A

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N15G1X-.25Y.25Z-.5F80
N20Y.125F50
N25G3X.25Y.125I0J.125
N30G1Y.5
N35G3X-.25Y.5I0J.5
N40G1Y.125
N45Z-.25
N50Y-.5
N55G3X.25Y-.5I0J-.5
N60G1Y0
N65G0X-3.Y3.M2
E
```

Program B

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10G41X0Y.25Z.05
N15G1X-.25Z-.5F80
N20Y.125F50
N25G3X.25Y.125I0J.125
N30G1Y.5
N35G3X-.25Y.5I0J.5
N40G1Y.125
N45Z-.25
N50X-.2501Y-.5
N55G3X.25Y-.5I0J-.5
N60G1Y0
N65G0G40X0Y0Z.05
N70G0G90X-3.Y3.M2
E
```

Program "A" in the preceding example, without cutter diameter compensation, executes the data without problem. Program "B", with a cutter compensation invoked (N10), requires a slight modification in Block N50 to avoid collinear data - i.e., one ten thousandth of an inch (.0001) was added to the "X" axis coordinate (.25). This will not be seen at the machine as the output resolution to the drives is .001 (.0005 BOSS 6).

THE FOLLOWING RULES MUST BE FOLLOWED WHEN STARTING COMPENSATION:

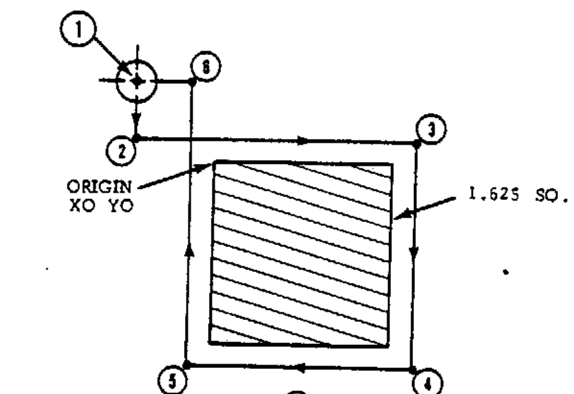
1) When starting Cutter Diameter Compensation G41 or G42 must be programmed in a block containing X and Y data. The system requires the XY coordinates in order to establish a position to start the compensation. The G41 or G42 and X and Y data may be programmed in a Rapid Positioning block, a linear feed block, or a repeat of the present location, and should be programmed before the cutter enters the work.

2) The axis motion for starting into and leaving the work must be as near to perpendicular to the programmed cutter path as possible, and must be greater than the amount of cutter compensation that will be used.

3) When starting cutter diameter compensation G41 or G42 cannot be programmed in a block containing circular interpolation.

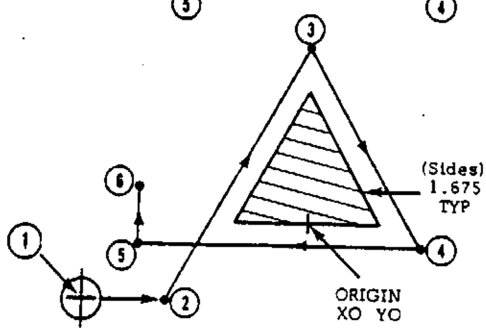
The three examples below illustrate a correct starting procedure:

From the Tool Change location or a convenient start position (1) move to position (2) off the work, turning cutter compensation on. Make next move towards the work perpendicular to cutter path (2) - (3).

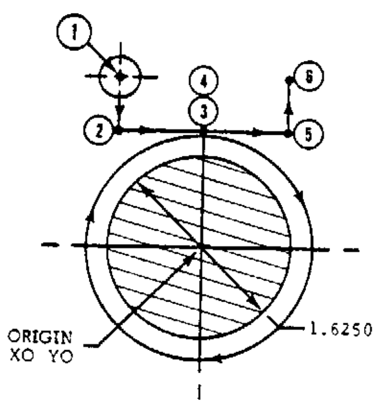


3/8 Diameter End Mill (all examples)

```
%N1G0G90X-1.Y1.T1M6
N5G41X-.687Y.687Z-.1
N10G1Y.187F100
N15X1.812
ETC.
```




```
%N1G0G90X-1.Y1.T1M6
N5G41X-1.75Y-.62Z-.1
N10G1X-1.25Y-.62F100
N15X0Y1.732
ETC.
```




```
%N1G0G90X-1.Y1.T1M6
.N5G75
N10G41X-.75Y1.5Z-.1
N15G1Y1.F100
N20X0
N25G2X0Y1.I0J0
N30G1X.75
N35G0G40Y1.
N40X-1.M2
E
```

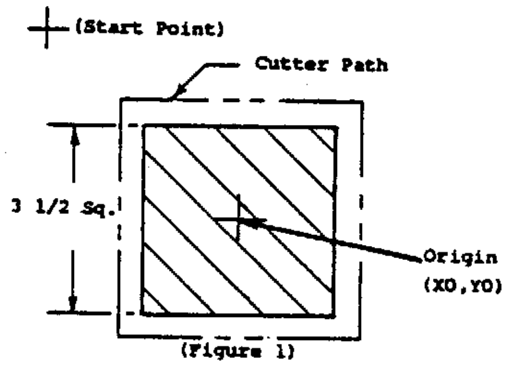
9.3.4.3 Basic Programming Examples

Three simple examples follow illustrating how to program cutter compensation.

1. The first example (Fig. 1) is an external milling cut around a 3-1/2" square part using a 1/2" Dia. End Mill.

Program "A"
Program Cutter Path:
No Cutter Compensation

```
%N1G0G90X-3.Y3.T1M6
N5X-2.26Y2.Z-.1
N10G1X2.F100
N15Y-2.
N20X-2.
N25Y2.26
N30G0X-3.Y3.M2
E
```



(Figure 1)

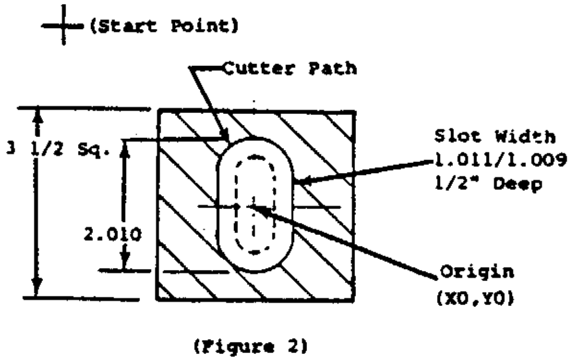
Program "B"
Program Cutter Path:
Cutter comp.-invoked (N5)
revoked (N35)

```
%N1G0G90X-3.Y3.T1M6
N5G41X-2.26Y2.5Z-.1
N10G1Y2.F100
N15X2.
N20Y-2.
N25X-2.
N30Y2.26
N35G0G40X-2.5Z.05
N40X-3.Y3.M2
E
```

2. The next example shows the contour milling of a slot using a 1/2" Dia. End Mill.

Program "A"
Program Cutter Path:
No Cutter Compensation

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N12G1Y-.5Z-.1F50
N13Y.5F100
N15X.255
N20G3X-.255Y.5I0J.5F50
N25G1Y-.5F100
N30G3X.255Y-.5I0J-.5F50
N35G1Y.5F100
N40G0X-3.Y3.M2
E
```

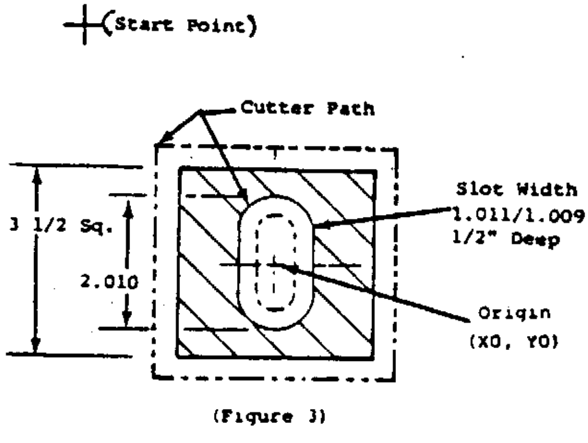


(Figure 2)

Program "B"
Program Cutter Path:
Cutter Comp.-invoked (N10)
revoked (N45)

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N12G1Y-.5Z-.1F50
N15G41X0Y.5F100
N20X.255
N25G3X-.255Y.5I0J.5F50
N30G1Y-.5F100
N35G3X.255Y-.5I0J-.5F50
N40G1Y.5F100
N45G0G40X0Y0Z.05
N50G0X-3.Y3.M2
E
```

3. The last program illustrates both previous examples combined. You will notice in Block N40 it is necessary to turn off compensation after completing the external cut then reinitiating it for the internal slot (N45).



(Figure 3)

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10G41X-2.26Y2.5Z-.1
N15G1Y2.F100
N20X2.
N25Y-2.
N30X-2.
N35Y2.26
N40G0G40X0Y0Z.05
N42G1Y-.5Z-.1F50
N45G41X0Y.5
N50X.255
N55G3X-.255Y.5I0J.5F5
N60G1Y-.5F100
N65G3X.255Y-.5I0J-.5F
N70G1Y.5F100
N75G0G40X0Y0Z.05
N80X-3.Y3.M2
E
```

9.3.5 Guiding Principles for the proper use of Cutter Diameter Compensation

1. Cutter Diameter Compensation is effective only in X and Y.

2. G0 may be programmed in the same block as that containing G41, G42, or G40; however, the system will remove the G0 from any other block within the compensated data.

3. Although a data block containing X and/or Y and Z is permissible, a deviation in the slope of this motion may occur because of the compensation of the X and/or Y move.

4. In compensating for cutter diameter, the system makes no attempt to adjust feedrates. It can be critically important when programming the finished surface to adjust the feedrate in accordance with the difference in path length between work surface and cutter path.

5. The value of Cutter Diameter Compensation for any tool is not scaled when in the Transformation Mode (G73).

6. G99, M00, M01, M25 will not work in Compensation. This code in a compensation block will be stripped out by the control and not sent to the output.

7. It is recommended that the G75 (Multi-Quadrant Circular) mode be used in conjunction with Cutter Diameter Compensation.

8. With Cutter Diameter Compensation in effect, some dwell will occur for blocks that are executed in less than .25 seconds.

9. There may be occasions when a large radius (say 64") is to be tangent to a small radius (say 1") and a tangency is not found. A small adjustment to one of the radii (say .001") will allow the computer to find the tangency point.

9.3.6 Cutter Diameter Compensation Rules

The following MUST be followed, or compensation will NOT work correctly.....

1. When starting compensation:

   G41 or G42 must be programmed in a block containing X and Y data.
   G41 or G42 cannot be programmed in a block containing circular (G2, G3).
   The starting axis motion must be as near perpendicular to the programmed cutter path as possible, and must be greater than the amount of compensation anticipated.

2. Cutter Diameter Compensation cannot be used with Polar Coordinate Commands or the Milling Cycles (G77, G78, G79).

3. The tool to part relationship can be changed from G41 to G42 only after a G40 has been programmed, also the amount of compensation cannot be changed while in G41 or G42.

4. Only one non X and Y block (e.g. "Z" axis motion, loop calls, macro calls, etc. or a combination thereof) will be executed with no ill effect to the compensated X and Y values, provided the block before and the block after contain linear (G1) X and Y data, and the coordinates do not lie in a straight line (co-linear data). See paragraph 9.3.4.1

5. When a radius is programmed for an inside corner, the maximum (+) compensation value which can be used is (2) times the corner radius.

6. When leaving compensation:
   G40 must be programmed in a block containing X and/or Y data.
   The axis motion must be as near to perpendicular to the programmed cutter path as possible.
   G40 cannot be programmed in a block containing circular motion (G2,G3), or the block containing M2 (rewind), M6 (Tool change), or M25 (Retract).

7. The compensation value must be entered (See 9.3.2.) before the call of a tool number.

9.3.7. Special Conditions

The special conditions in this section are also special considerations for the part programmer. The part programmer must exert a close control over the values of Cutter Diameter Compensation he uses or those he allows the operator to use. Proper control can yield many benefits. Figure 9.2 shows a copy of F-127A, a form available from Bridgeport Machines specifically for this purpose.

9.3.7.1 Multiple Compensation Values For One Tool

Input of cutter diameter compensation is by MDI on the special panel provided or by a listing on a separate tape or on the front of the program text to be input to storage through the tape reader or terminal device. The data input could be as follows from Figure 9.2.

```
T3//-.02
T4//-.045
T11//-.121
T12//.26
T13//.04
T14//-.023
```

Inspection of the above values shows the typical use of cutter diameter compensation for T3, T4 and T11. For T12, however, other possibilities become evident:-

a. T12, T13 and T14 use physically the same cutting tool.
b. The cutter path is assumed to be in the form of a macro sub routine containing the appropriate G41/G42 and G40 codes.
c. When the macro is first called it is used with T12 to provide a roughing pass about 1/8" outside the part to be finished.
d. After the first pass, the macro is called again using a different tool number (T13), but the same physical tool, to provide a second roughing pass.
e. The last pass, when the macro is called the third time, is a finish cut where the tool diameter is known to be .023" undersize.

Bridgeport. TEXTRON

Bridgeport Machines Division of Textron Inc.

500 LINDLEY STREET BRIDGEPORT CONN 06606

PART NAME: FLANGE
PART NUMBER: 3673651

TOOL SETTING DATA
MACHINE

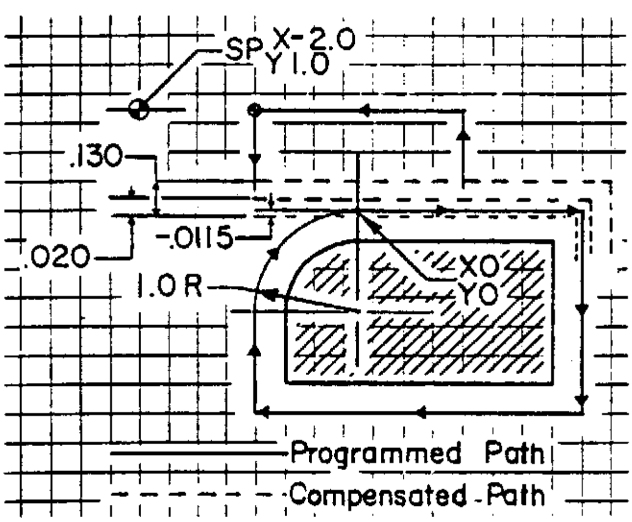
JOB NO 8666
SHEET 1 OF 1

FOR TOOL LENGTH OFFSET OF ANY TOOL TO A SINGLE CLEARANCE PLANE
TLO OF TOOL (LONGEST TOOL LENGTH + ITS TLO) MINUS ACTUAL SET LENGTH OF TOOL

FOR CUTTER DIAMETER COMPENSATION OF ANY TOOL
CUT COMP PROGRAMMED CUTTER DIA. MINUS ACTUAL CUTTER DIA

STATION NO	TOOL NUMBER	DESCRIPTION	SEQ. NO	LENGTH TO BE SET	ACTUAL SET LENGTH	PROGRAM DIAMETER	ACTUAL DIAMETER	STATION NO	TLO	CUT DIA. COMP
T01		4" Dia. Face Mill	5	5.240	3.875			T01	3.2920	
T02		1/2 Dia. Spot Drill	22	5.100	5.113			T02	2.0540	
T03		1/2 Dia. End Mill	35	6.000	5.374	.500	.480	T03	1.7930	-.02
T04		3/8 Dia. End Mill	75	4.750	6.008	.375	.33	T04	1.590	-.045
T05		5/8 Dia. C-Bore (End Mill)	115	5.000	5.127			T05	2.0400	
T06		1 1/4 Dia. Drill	150	6.600	6.632			T06	.5350	
T07		1" Dia. Drill	175	5.812	5.830			T07	1.3170	
T08		3/8 Dia. Spot Drill	435	5.000	4.880			T08	2.2870	
T09		No. 20 Drill (.161)	500	6.600	6.867			T09	.3000	
T10		10-32 Tap	565	3.700	4.122			T10	3.0450	
T11		.750 Dia. End Mill	630	6.875	5.805			T11	1.3620	-.121
T12		1.25 Dia. End Mill	705	6.000	6.0198	.500	.477	T12	1.1472	+.260
T13		" " "	735	"	"	"	"	T13	"	+.040
T14		" " "	760	"	"	"	"	T14	"	-.023
T15								T15		
T16								T16		
T17								T17		
T18		Same physical tool						T18		
T19								T19		
T20								T20		
T21								T21		
T22								T22		
T23								T23		
T24								T24		

Figure 9-2. Tool Setting Data



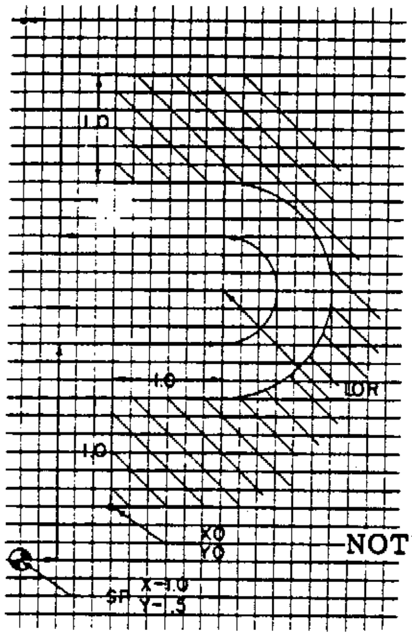

```
%#1
N5G41X-1.Y1.
N10G1Y0F100
N15X2.
N20Y-2.
N25X-1.
N30Y-1.
N35G2X0Y0I0J-1.
N40G1X1.Y.005
N45G40Y1.
$
T12//.26
T13//.04
T14//-.023
N50G0G90X-2.Y1.T12M6
.N55G75
N60X-1.Z-.5
=#1
N65T13
=#1
N70T14
=#1
N75G0X-2.Y1.M2
E
```

NOTE: After programming 2 roughing passes, it may be necessary to program a finish pass using the same dimensions (or finish to spring pass). Coming out of the first pass at coordinates XO YO, the tool must be moved well away from the part, the compensation shut off then reinitiated all in accordance with Rule. 3.

9.3.7.2 Inside Corners

The cutter path for a sharp inside corner can be compensated successfully until the plus compensation becomes too large. This is the condition where the tool chosen is so large that it will not fit into the slot.

The programmer must be aware of the potential problems with large values of positive compensation. Such problems as are demonstrated in example 9-4 might arise with too large a positive value. Also because the tool is describing a new path well away from the workpiece, the possibility of interference with other parts of the workpiece or with the fixture will arise.

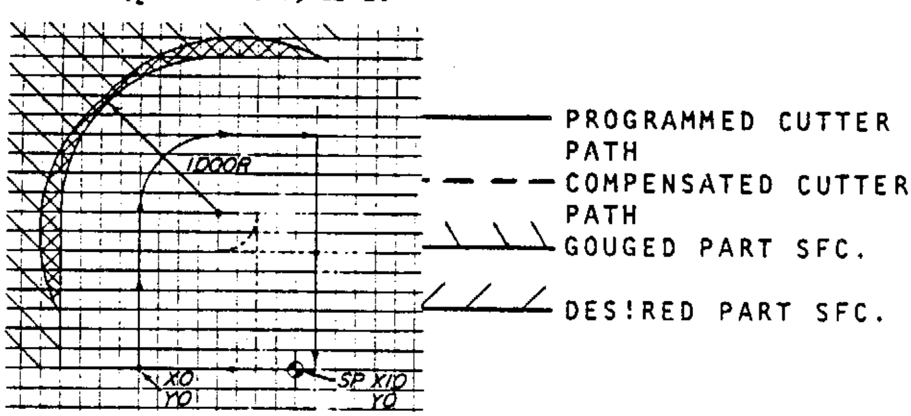


NOTE: PROGRAMMED FOR A 1.0 DIAMETER END MILL

```
%N1G0G90X-1.Y-.5T1M6
.N5G75
N10Z-.5
N15G41X-1.Y-.5
N20G1X-.5F100
N25Y1.5
N30X1.
N35G3X1.Y2.5I1.J2.
N40X-.5
N45Y4.5
N50G40X-1.
N55GOX-1.Y-.5M2
E
```

EXAMPLE 9.3
Inside Corners, Special Conditions

NOTE: Excessive plus compensation will not give the intended corner. The reader may wish to plot this output which shows extensive gouging of the radiused wall. Limiting value of compensation (per Rule 5) is 2."



NOTE: This condition applies to positive compensation in excess of the value of an inside corner radius. The example shows that the computer has logically found the first surface (the radius) and discovered that it must move around that surface backwards to find the next intersection.
Result: Gouging of the workpiece.

```
%N1G0G90X1.Y0T1M6
.N5G75.
N10Z-.5
N15G42X1.Y0
N20G1X0F200
N25Y1.
N30G2X.5Y1.5I.5J1.
N35G1X1.2
N40G40Y0
N45G0X1.Y0M2
E
```

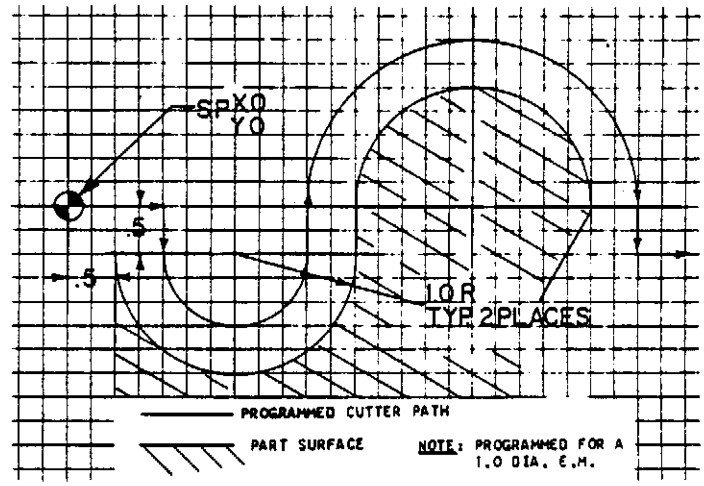
Example 9-4
Inside Corner, Excessive Compensation

9.3.7.3. Feedrate Considerations

In any continuous path contouring operation, the ratio between the feedrate at the tool perimeter (where the cutting takes place) and the tool center (where the feedrate is controlled) can become important. This is one of the normal considerations of the programmer in operations without cutter diameter compensation.

The effect of excessive compensation is illustrated in example 9-2 where it may be assumed that the programmed path has the correct feedrate. When plus compensation is used, the feedrate should be changed to 50% of its programmed value on the inside corner. On the outside radius, the feedrate should be increased by 50%. It is not possible in any system to do this. Minimize the amount of compensation.

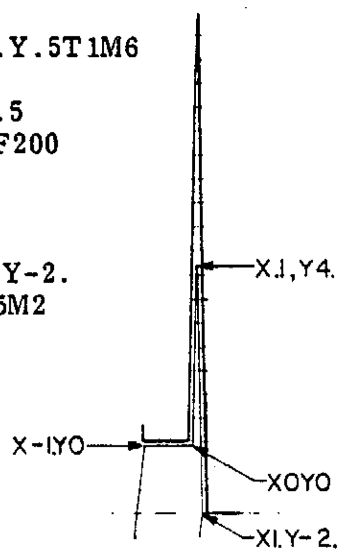
```
%N1G0G90X0Y0T1M6
.N5G75
N10Z-.5
N15G41X0Y0
N20G1X1.F100
N25Y-.5
N30G3X2.Y-.5I1.5J-.5F50
N35G1Y0F100
N40G2X5.Y0I3.5J0F150
N45G1Y0.5F100
N50G40X5.5
N55G0X0Y0M2
E
```



NOTE: The change in chipload resulting from a tool describing the inside of an outside radius at a fixed feedrate has been discussed elsewhere in the programming manual. (Sec. 5.6.1) Large values of compensation will disrupt the efficient cutting at the surface of the part unless the feedrate is reduced in proportion to the amount of compensation for inside corners and increased for outside corners.

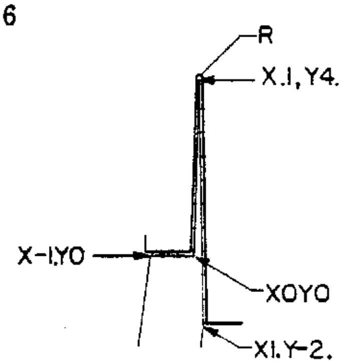
9.3.7.4 Programming with Acute Angles

```
%N10G0G90X-1.Y.5T1M6
N15Z-.5
N20G41X-1.Y.5
N25G1X-1.Y0F200
N30X0Y0F50
N35X.1Y4.
N40Y-2.
N45G40X.6001Y-2.
N50G0X-1.Y.5M2
E
```

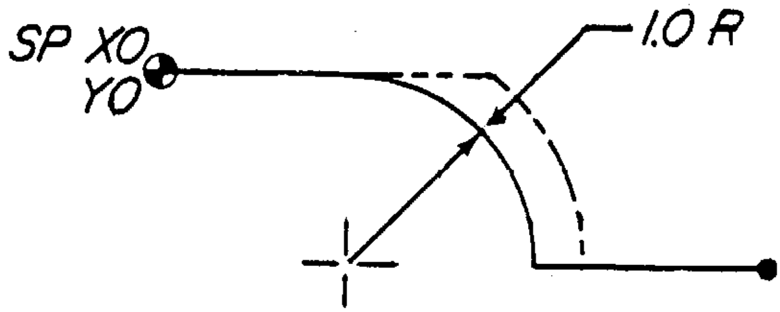


It should be noted that the tool can travel large distances to reach a compensated intersection. A circle can be programmed around the point to reduce this problem. With compensation in effect, the system will generate the radius (R) shown.

```
%N10G0G90X-1.Y.5T1M6
.N12G75
N15Z-.5
N20G41X-1.Y.5
N25G1X-1.Y0F200
N30X0Y0F50
N35X.1Y4.
N40G2X.1Y4.I.1J4.
N45G1Y-2.
N50G40X.6
N55G0X-1.Y.5M2
E
```



9.3.7.5 Starting and Ending, Special Conditions



INPUT TEXT

```
.
.
.N45G75
N50G0G90X0Y0T1M6
N55G41X0Y0
N60G1X1.F100
N65G2X2.Y-1.I1.J-1.
N70G40X3.
N75G0X0Y0M2
E
```

T1//.5000

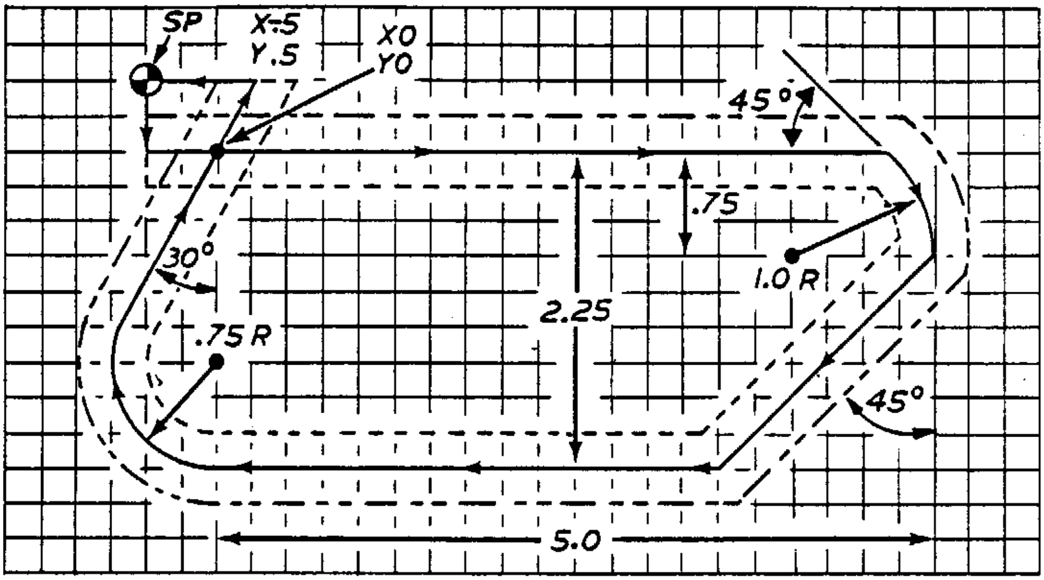
COMPENSATED DATA

```
.N45G75
N50G0G90X000Y000T1M6
N55G41X000Y000
N60G1X17500Y000F100
N65G2G40X22500Y-10000I7500J10000
N75G0X000Y000M2
```

NOTE: Rule 1 states that the starting axis motion must be as near perpendicular as possible to the first fully compensated block. If this condition does not exist, as shown here, compensation works, but not necessarily in the manner intended by the programmer.

9.3.8 Practical Example

The following Example illustrates the principles showing the Input Programmed Text and the simulation of two different outputs. One for T1//.5000 and the other T1//-.5000. Note that these outputs are not available to view externally, but are the outputs of the compensation system as would be seen by a conventional decoding logic and drive system.



Compensated Data - Internally Generated

Program Input

```
%N1G0G75G90X-.5Y.5T1M6
N5Z-.1
N10G41X0.5Y.5
N15G1Y0F300
N20X4.664
N25X4.7071Y-.0429
N30G2X5.Y-.75I4.J-.75
N35G1X3.5Y-2.25
N40X0
N45G2X-.65Y-1.125I0J-1.5
N50G1X.2887Y.5
N55G40X-.5
N60G0M2
E
```

T1//-.5000

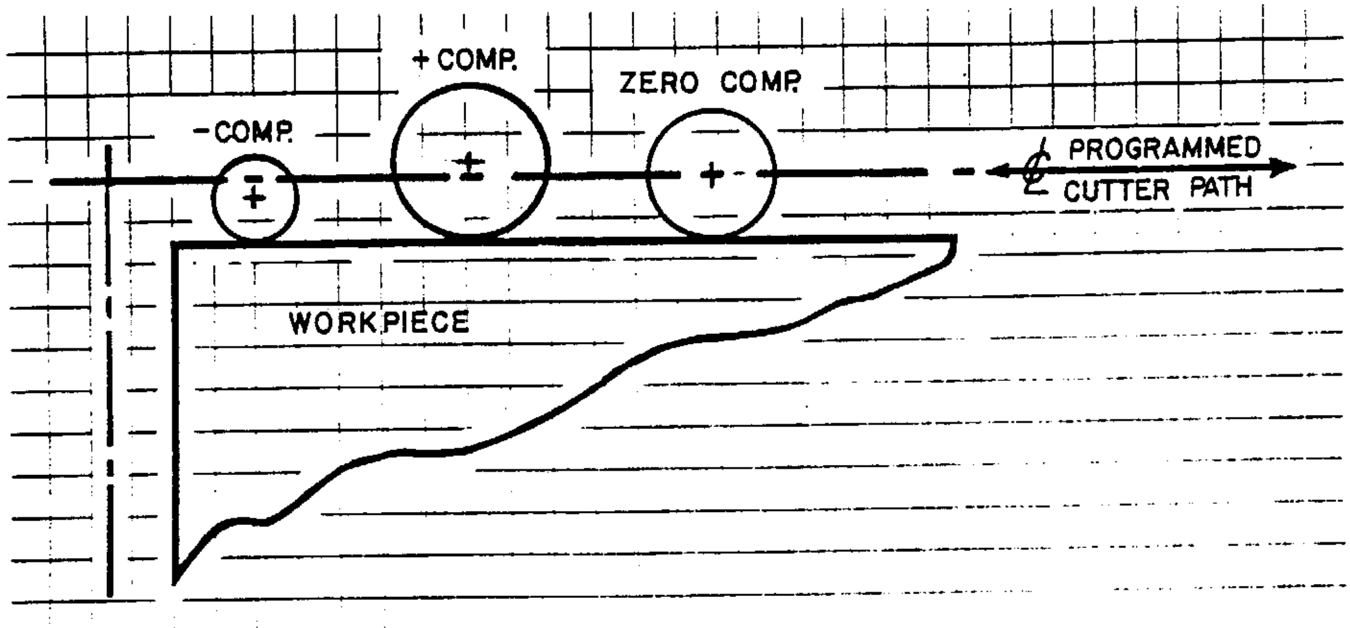
```
%N1G0G75G90X-5000Y5000T1M6
N5Z-1000
N10G42X-5000Y5000
N15G1X-5000Y-2500F300
N20G1X45608Y-2500
N25G1X45305Y-2199
N30G2X47437Y-6528I40000J-7500
N35G1X33964Y-20000
N40G1X000Y-20000
N45G2X-5000Y15000I000J-15000
N45G2X-4333Y-12497I000J-15000
N50G1G40X5774Y5000
N55G1X-5000Y5000
N60G0M2
E
```

T1//.5000

```
%N1G0G75G90X-5000Y5000T1M6
N5Z-1000
N10G41X-5000Y5000
N15G1X-5000Y2500F300
N20G1X47672Y2500
N25G1X48837Y1341
N30G2X52500Y-7500I40000J-7500
N30G2X52453Y-8583I40000J-7500
N35G1X36036Y-25000
N40G1X000Y-25000
N45G2X-10000Y-25000I000J-15000
N45G2X-8663Y-9996I000J-15000
N50G1G40X000Y5000
N55G1X-5000Y5000
N60G0M2
E
```

9.3.9. Effects of Positive, Negative, or Zero Cutter Diameter Compensation

Once Cutter Compensation has been invoked, the tool travels from the current tool location along a path (The programmed center line of the tool) until it encounters the next programmed tool path. At this point, depending on whether positive, zero, or negative compensation has been entered, the system will position the center line of the tool either before, on, or after the programmed path it has encountered.



NOTE: While Cutter Diameter Compensation may be used for part surface programming, with compensation entered as the full diameter of the cutter; it is important to recognize, as several examples in the text point out, possible problem areas with respect to EXCESSIVE Cutter Compensation, (either Positive or Negative). Cutter Compensation is intended to simplify and reduce machining cycle time, by allowing the operator the freedom to utilize resharpened or alternative end mills when the programmed cutter is either not available or reground.

9.4 POLAR COORDINATE SYSTEM

Polar Coordinates present an alternative method of defining point locations whenever they are presented in angular notation, e.g. bolt circles, circumferential slots, cams, etc.

In a polar coordinate system the location of any point (P) may be defined by its radius "R", in relation to a fixed point called the POLE; and by its angle "Θ", in reference to a fixed line Cx called the POLAR AXIS (Figure #1).

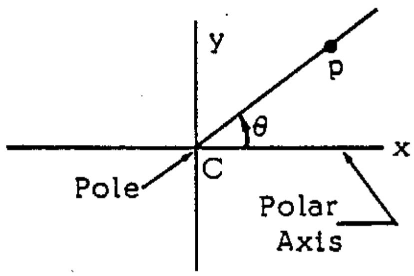


Fig. #1

9.4.1 Polar Coordinate Programming

In order to correlate this concept with CNC programming, consider the part shown below. (Fig. #2) Notice points P1 and P2 are drawn angularly in relation to a circle. (1.125 BCD)

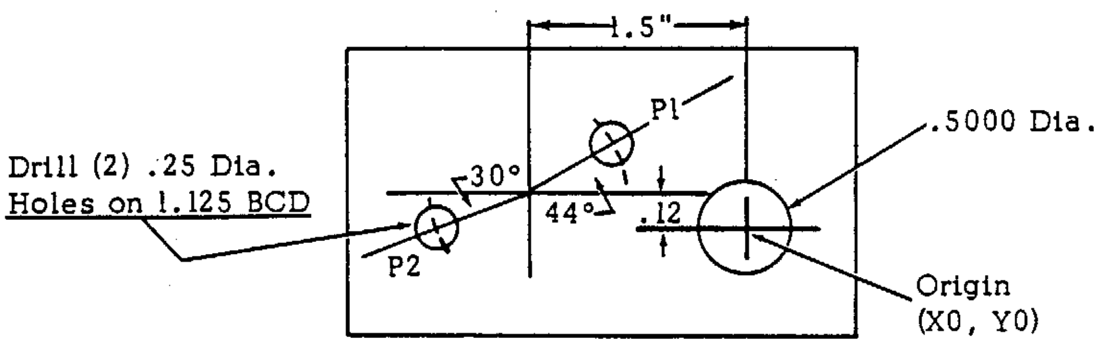


Fig. #2

To simplify determination of the Pole and Polar axis, superimpose rectangular axes thru the circle center. ("C"). Line "Cx" would then be the POLAR AXIS. (Fig #3)

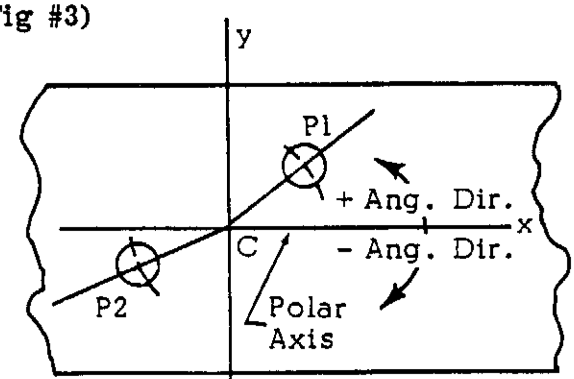


Fig. #3

To program these two points using polar, it is first necessary to define the absolute coordinates of the pole (point "C") and the radius of the points in relation to this pole, i.e.:

.
.
.
N20R.5625I-1.5J.12 (Note: "Z" axis data not permitted in this block)

Where: "R" is the radius of the bolt circle stated (1.125 Dia.)
"I" is the "X" axis absolute coordinate of the pole (Pt. "C")
"J" is the "Y" axis absolute coordinate of the pole (Pt. "C")

Once the Radius and Pole are defined, any point lying on that circle may be located simply by stating the angle this point forms in relation to the polar axis (Cx) within the following limitations:

1. The range of "A" (angle) is +719.999° to -359.999° (resolution .001°).
2. Angles require the use of a decimal point. (e.g. A-30.).
3. The radius "R" and Pole must be defined before using polar angles.
4. Positive angles are measured from the Polar axis, CCW around the circle.
5. Negative angles are measured from the Polar axis, CW around the circle.

Referring again to the part shown in Figure #2:

Point P1 could be programmed:

N25A44. or N25A-316.

Point P2 could be programmed:

N30A210. or N30A-150.

For clarity, assume the two 1/4 dia. holes are to be center drilled as the initial operation of the program for the part shown in Fig. #2.

```
%N1G090X-4.Y-4.T1M6      (Tool change pos. X-4., Y-4.)
.N5R.5625I-1.5J.12
N10A44.Z.05
.N15G81Z.195F100
N20A44.
N25A-150.
N30G0X-4.Y-4.T2M6
.
.
.
.
```

9.4.2 Absolute/Incremental Polar Coordinate Programming

The following example (Fig. #4) and accompanying programs, illustrate Polar Coordinate Programming for a point-to-point drilling operation using Absolute or Incremental angular data.

(Drill .25 Dia. Holes at P1 and P2)

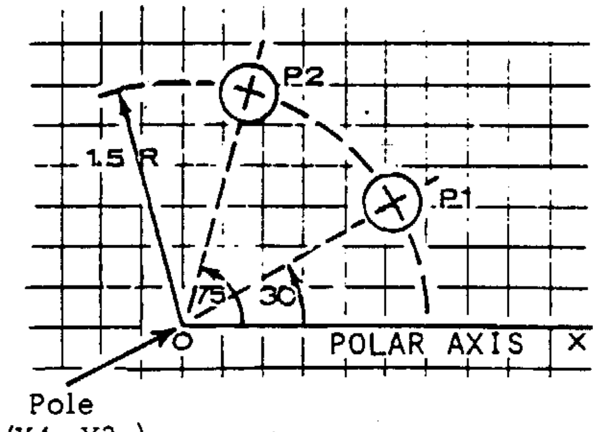


Fig #4

(A) Absolute Mode

```
N10G0G90X0Y0Z.05
N15R1.5I4.J3.
N20G81A30.Z.5F100
N25A75.
N30G0X0Y0
```

(B) Incremental Mode

```
N10G0G90X0Y0Z.05
N15R1.5I4.J3.
N20G81A30.Z.5F100
N25G91A45.
N30G0G90X0Y0
```

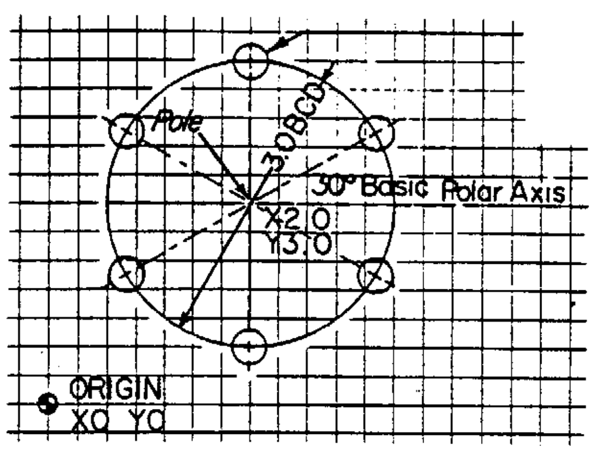
9.4.3 Bolt Circle Programming

(a) Locate Tool on Bolt Circle over first hole before bolt circle block.

(b) Block Format: G(1)A(2)A(3)Z(4)F(5)

Where:

1. The canned Z axis cycle parameter (e.g. - G81, G82, etc.).
2. The polar angle "A" of the last point to be drilled; incrementally (G91) from the first, or absolute (G90) from the polar axis.
3. The signed incremental angle (A) between points.
4. The canned Z axis cycle depth(s).
5. The Feedrate.



3/8" Dia. Drill thru 6 Holes

Program:

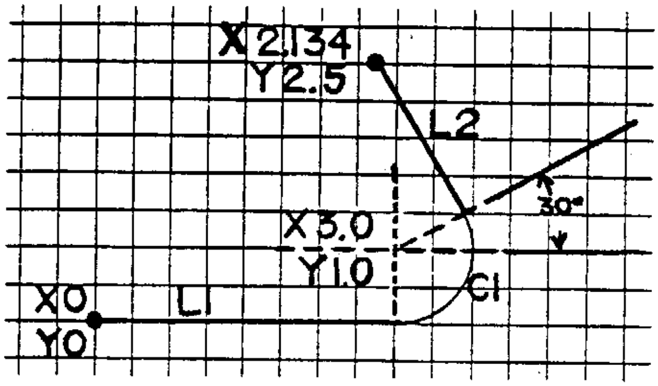
```
%N10G0G90X-5.Y4.T3M6 (Spot Drill)
N15R1.5I2.J2.
N20A30.Z.05
N25G81A330.A60.Z.2475F135
N30G0G90X-5.Y4.T4M6 (Drill)
N35R1.5I2.J2.
N40A30.Z.05
N45G87A330.A60.Z.712Z.4F150
N50G0G90X-5.Y4.M2
E
```

NOTES:

1. Blocks N15 and N35 define the pole and Radius.
2. Blocks N20 and N40 place the tool over the first hole.
3. Blocks N25 and N45 drill the first hole at the existing location, then proceed automatically to the rest. The final hole being at 330 degrees absolute.

9.4.4 Continuous Path Milling

MILL THE PATH L1, C1, L2



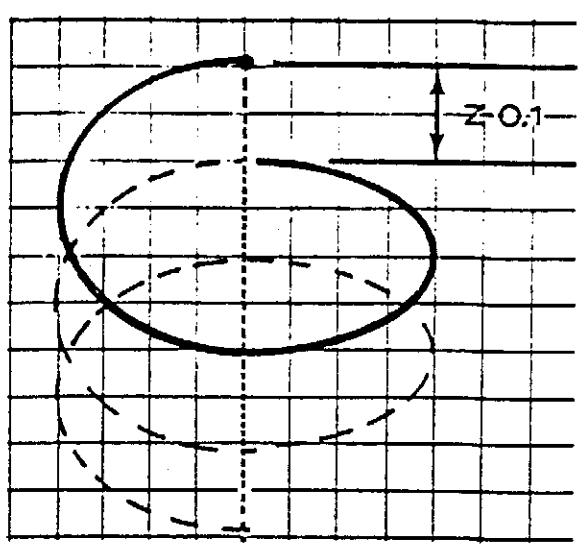

```
N10G0G90X0Y0
N15G1R1.A-90.I3.J1.F100
N20G3A30.
N25G1X2.134Y2.5
ETC.
```

Block N15 defines the pole and is also combined with XY motion data. This is permitted since the motion is other than the Z axis.

9.4.5 Helical Milling

Helical Milling is a special case of polar coordinate programming, which simulates simultaneous three axis continuous path contouring by looping small X,Y,Z incremental moves. This becomes an extremely valuable tool for various difficult machining operations such as: Thread Milling, helical oil grooves, O-ring grooves, undercuts, etc.

The following program segment will illustrate this technique by generating one revolution of a 4." DIA helix with a .1" lead.

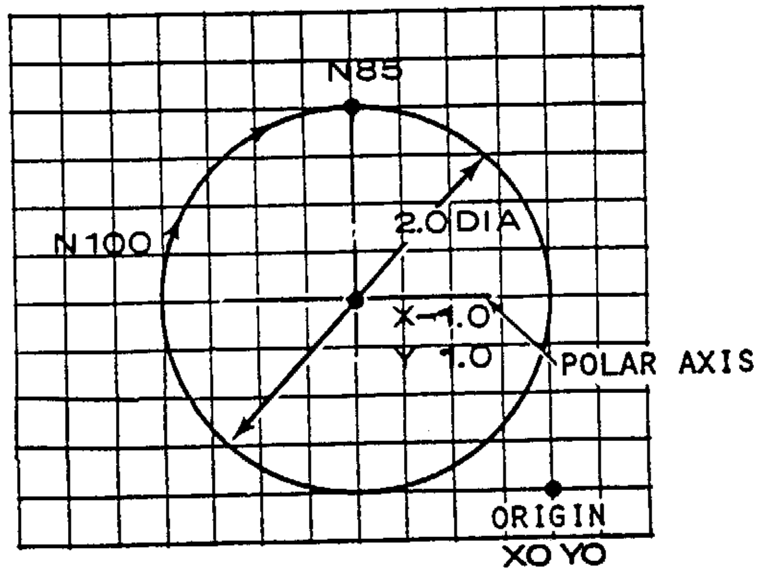


Consider breaking up the 4" dia. circle shown into small segments each having a constant Z move. Repetition of such a program will approximate a circle with Z depth approximately proportional to the angle generated.
Thus:

```
N50G90R2.I0J0
N55G0A90.Z0
=N60/1000
N60G91G1A.36Z-.0001F40
```

9.4.6 A 360° Circle

A partial arc or a full 360 degree circle can be programmed in a manner quite different from the method defined for the rectangular coordinate systems.




```
N80G0G90X-8.Y4.T7M6
N85R1.A90.I-1.J1.
N90Z.05
N95G1Z-.1F120
N100G2A90.
```

9.4.7 Rules for Polar Coordinate Programming

1. The range of A is +719.999 degrees to -359.999 degrees. (Resolution .001 degrees)
2. The I and J dimensions are the absolute coordinates of the Pole.
3. Do not program G75. (The multi quadrant circle mode)
4. The angle A is measured from the polar axis (positive X).
5. No Z axis data is permitted in a block containing I and J data.
6. Polar Coordinate Data cannot be Transformed i.e. Rotated or Scaled (G73).
7. Cannot be used with Cutter Diameter Compensation.
8. The minimum angular increment is A.001 if the value of R is large enough.
9. "A" cannot be used as a macro variable greater than 16. degrees.

NOTE: When redefining a pole location within a program insert the angular start location on the pole definition line, e.g., N55A30.R1.5I2.J2

9.4.8 Cam Track or Cam Profile

Polar coordinate programming can be used most effectively when the cam drawing is presented as a table of Polar Coordinates. Figure 9-3 shows such a drawing with angular data very 10 degrees instead of the finer increments that would normally appear. The adjoining output is all that is necessary to complete the cam track.

```
   %N1G0G90X0Y0T1M6
   N3G92A0 (This block forces "A" register to zero)
   N5R.98A150.I0J0
   N10Z.02
   -N15/120
   N15G91G1A.25Z-10F120
   N20G90G3A0
1. N25G1R.95A10.
2. N30R.93A20.
3. N35R.9A30.
4. N40R.89A40.
5. N45R.88A50.
6. N50R.89A60.
7. N55R.91A70.
8. N60R.94A80.
9. N65R.98A90.
   N70G3A185.
   =N75/90
   N75G1G91A.333Z15
   N80G0G90X0Y0M2
   E
```

	θ°	Radius
1	10	R=.950
2	20	R=.930
3	30	R=.900
4	40	R=.890
5	50	R=.880
6	60	R=.890
7	70	R=.910
8	80	R=.940
9	90	R=.980

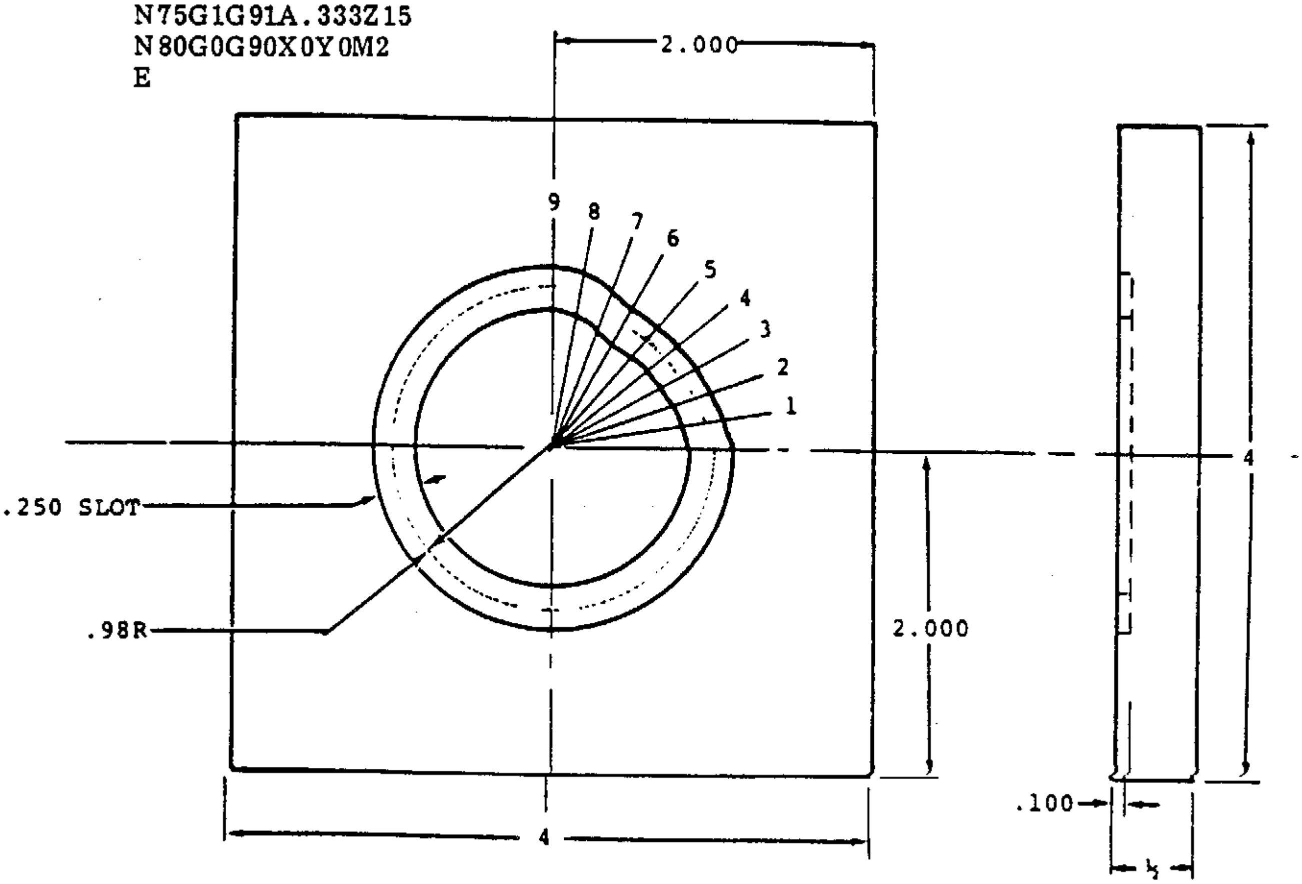
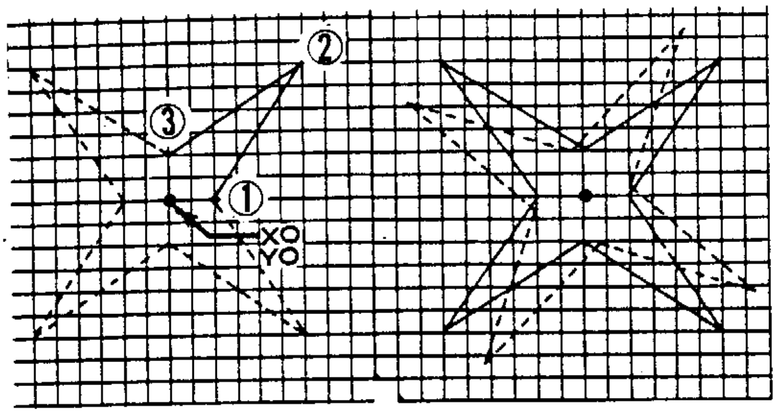


Figure 9-3. Cam Track

9.5 CUTTER PATH TRANSFORMATION

In its strictest definition, transformation can mean changing the shape or form without changing its value. This is indeed the capability when an input programmed shape can be rotated or scaled up or down.

9.5.1 Rotation of a programmed part shape can be done by inserting the appropriate preparatory code (G73) followed by the angle through which the shape is to be rotated (A degrees). The XY coordinate around which the part is rotated is always the origin or X zero Y zero absolute coordinates of the part. The preparatory code (G72) will turn transformation off.




```
① N20X.5Y0
  N25Z-.05
  N30G92A0
  =N60/4
② N40G90G1X1.5Y1.5F120
③ N50X0Y.5
  N60G91G73A90.0
  N70G0G90G72X-4.5Y3.75Z.05
```

```
%N10G0G90X-4.5Y3.75T1M6
N15G92A0
#1
N20X.5Y0
N25Z-.05
=N60/4
N40G90G1X1.5Y1.5F120
N50X0Y.5
N60G91G73A90.0
N65G0G90G72Z.05
$
=#1
N70G73A15.0
=#1
N80G0G90G72X-4.5Y3.75M2
E
```

Features of Rotation:

1. The star shown above can be developed in the manner shown, but the entire shape can form an inner nest with an outer command to rotate through any angle desired.
2. A shape containing Z axis motion (pocket with a sloping bottom) can be rotated and still maintain the Z axis motion at the appropriate places.
3. Though the above example shows linear motion only a part with circular interpolation input can also be rotated. See Figure 9-4 for a Geneva Mechanism Rotor.

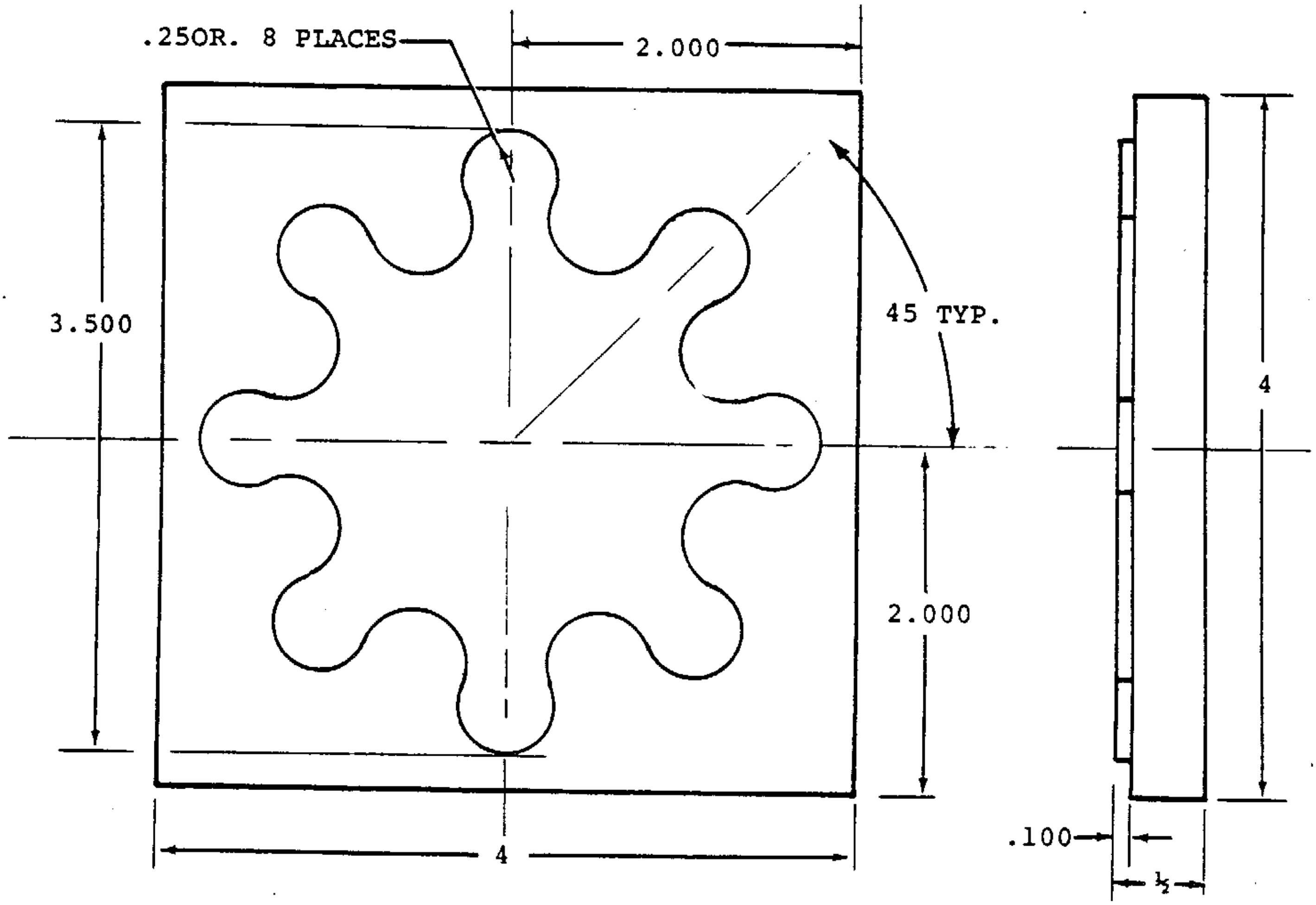
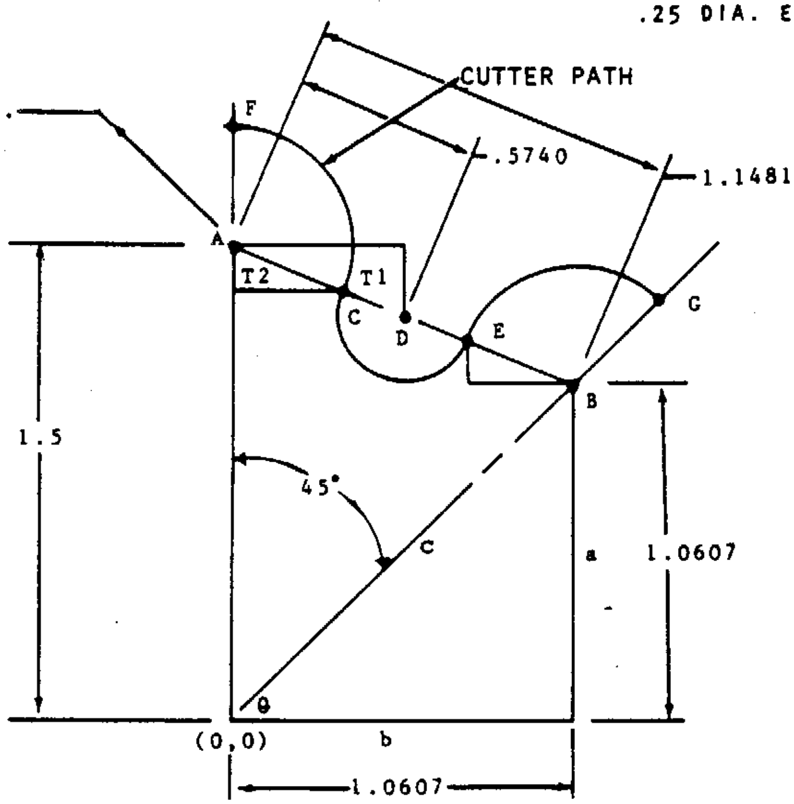
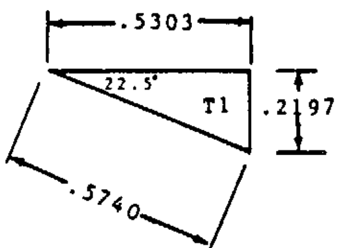


Figure 9-4. Rotor, Geneva Mechanism

PROGRAMMED FOR
.25 DIA. E.M.





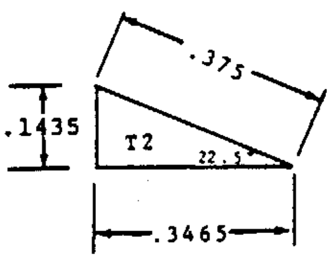
$$\sin\theta = \frac{\text{OPP.}}{\text{HYP.}}$$

$$\cos\theta = \frac{\text{ADJ.}}{\text{HYP.}}$$

$$\text{*DISTANCE A to B} = \sqrt{(A_X - B_X)^2 + (A_Y - B_Y)^2}$$

NOTE	X CO'ORD	Y CO'ORD
A	0.0000	1.5000
B	1.0607	1.0607
C	0.3465	1.3565
D	0.5303	1.2803
E	0.7142	1.2042
F	0.0000	1.8750
G	1.3258	1.3258

$a = \sin 45° \; (1.5) = 1.0607$ (NOTE: b=a)



$$AB = \sqrt{(1.0607)^2 + (1.5-1.0607)^2} = 1.1481$$

$$AD = 1.1481/2 = .5740$$

* The Distance Equation is the general form of Pythagorean Theorem which states $c=\sqrt{a^2+b^2}$ or hypotenuse of a right triangle is equal to the square root of the sum of the sides squared.

The programmed tape image is as follows:

```
%N1G0G75G90X0Y0T1M6
N5X-.1Y1.875Z.05
N10G1Z-.1F60
N15X0F120
N20G92A0
=N40/8
N25G2G90X.3465Y1.3565I0J1.5
N30G3X.7142Y1.2042I.5303J1.2803F50
N35G2X1.3258Y1.3258I1.0607J1.0607F100
N40G73G91A-45.
N45G1G72G90X.1
N50G0G90X0Y0M2
E
```

4. One of the most useful applications for rotation is the case of a part containing a number of slots. See Figure 9-5 in which 5 slots are symmetrically placed while the sixth, or timing slot is placed in a unique position. The programming technique used is to minimize the amount of effort by programming the simplest slot, i.e., the one located on an axis of motion. The following have been generated for 5 and 6 slots:

5 Slots

```
%N1G0G75G90X0Y0T1M6
N3G92A0
=N45/5
N5G0G90X0Y1.25Z.05
N10G1X0Y1.75Z-.1F100
N15X-.025Y1.25
N20G3X.025Y1.25I0J1.25F20
N25G1X.025Y1.75F100
N30G3X-.025Y1.75I0J1.75F20
N35G1X-.025Y1.25F100
N40G3X0Y1.27I0J1.25F20
N45G73G91A60.
N50G0G72G90X0Y0M2
E
```

6 Slots

```
%N1G0G75G90X0Y0T1M6
N3G92A0
#1
N5G0G90X0Y1.25Z.05
N10G1X0Y1.75Z-.1F100
N15X-.025Y1.25
N20G3X.025Y1.25I0J1.25F20
N25G1X.025Y1.75F100
N30G3X-.025Y1.75I0J1.75F20
N35G1X-.025Y1.25F100
N40G3X0Y1.27I0J1.25F20
$
=N45/5
N42(See 6-2, Rule b)
=#1
N45G73G91A60.
N47G73A5.
=#1
N50G0G72G90X0Y0M2
E
```

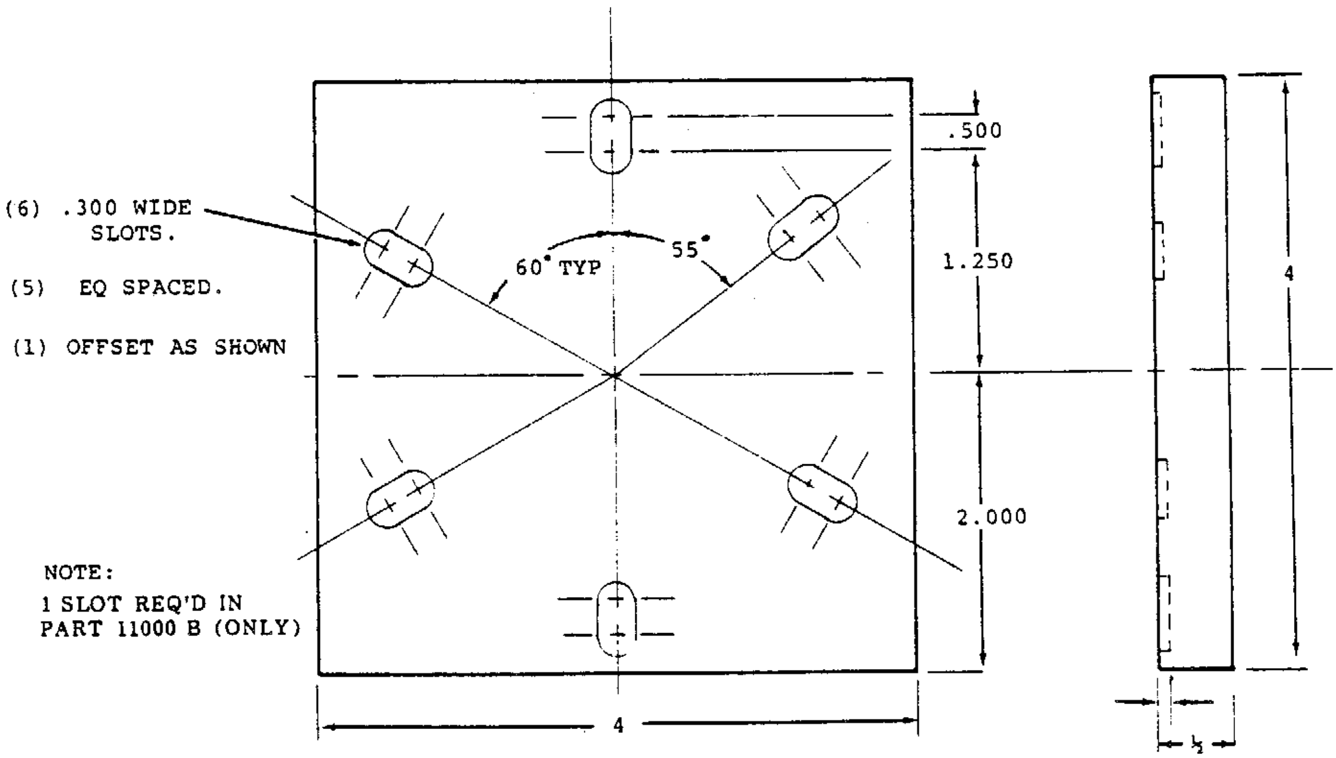
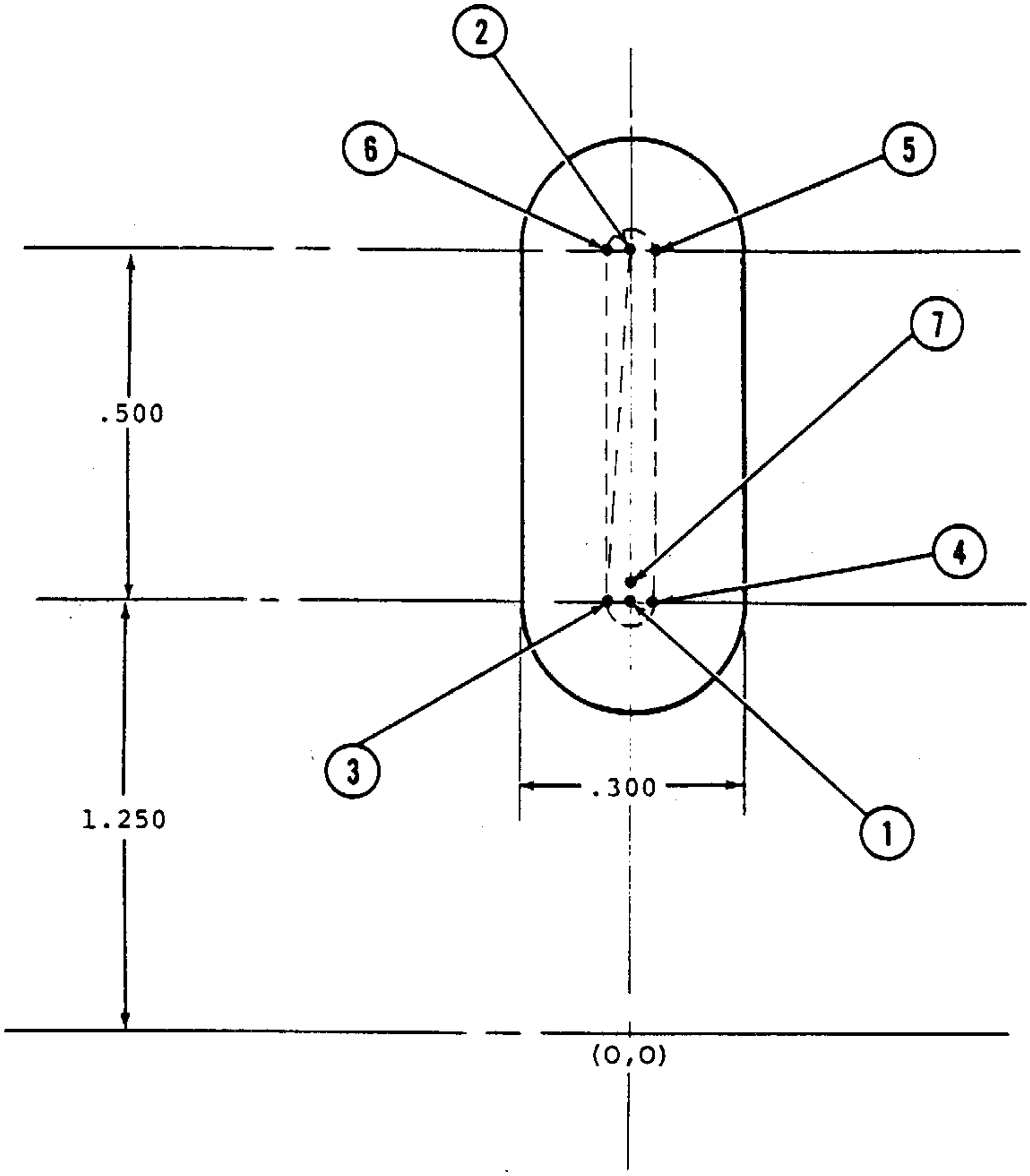


Figure 9-5. Plate (Sheet 1 of 2)

PROGRAM # 11



NOTE	X CO'ORD	Y CO'ORD
1.	XO.	Y 1.250
2.	XO.	Y 1.750
3.	X-.025	Y 1.250
5.	X.025	Y 1.250
4.	X.025	Y 1.750
6.	X-.025	Y 1.750
7.	XO.	Y 1.275

Figure 9-5. Plate (Sheet 2 of 2)

9.5.2. Scaling of a programmed part shape has a format:
G73 X (scale factor) Y (scale factor)

This denotes that it is permissible to have different values for X and Y, except when the program contains (G2,G3) circular data. The limits to the scale factor are practical values only (MIN. .0001 and MAX. 9999.).

Features:

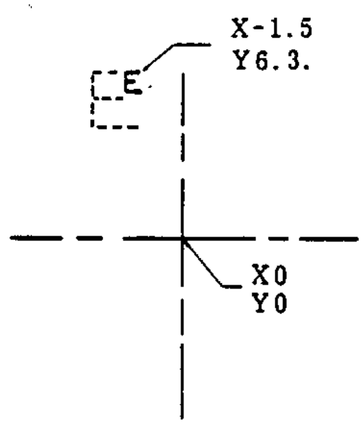
a) Scaling of scaled data will operate correctly if nested. The scale factor in the inner nest also gets scaled.

b) Scaling with Cutter Diameter Compensation is permissible. Note that the Cutter Diameter Compensation value for the tool does not get scaled.

c) When incremental data is to be scaled, the first block after transformation (G73) must be in absolute coordinates. This absolute coordinate input has to be the absolute unscaled dimension divided by the scale value.

For example:

```
N230G0G90X0Y0
N235G73X.433Y.433
N240X-3.4642Y14.5496
N245G91, etc.
```



The dimensions in block N240 are equal to 1.5/.433=3.4642 and 6.3/.433=14.5496

In this example, it was the programmer's intent to scale E to a value less than half size and place it at the known common point with coordinates X-1.5 Y6.3.

d) If the scale factor is to be changed within a program, a G72 must first be programmed followed by a G73 with the new scale factors.

Rules for Transformation:

1. The system must be in the multi-quadrant circular interpolation input mode (G75) when programming circular interpolation in order to have the Transformation mode (G73) operate correctly.

2. Polar coordinate input must not be used as transformation input.

3. Multihole row fixed cycle programming cannot be transformed.

4. Scaling and Rotation cannot be used simultaneously.

5. A cutter path with diameter compensation in effect may be transformed only if it is transformed (G73) before the compensation is invoked (G41/42). The compensation must be stopped (G40) before the transformation is ended (G72).

6. In the Transformation mode (G73) all X, Y, I, J data must be entered even though it may repeat a value previously entered. This applies for both linear (G1) and/or circular (G2,G3) data.

7. Transformation by rotation or by scaling takes place about XY absolute zero.

8. The first coordinate after invoking the scaling preparatory function (G73) must be in the absolute mode (G90).

9. Do not use scale factor as macro variable.

9.5.3 Scaling of Metric Data

The scale factor noted in previous paragraphs applies to that data in inch dimensions. For example, to cut a part one half the size programmed use G73X.5Y.5 or if metric use X(.5)*(25.4)   Y(.5)*(25.4)

If the part is programmed in metric, either of the following two methods may be used:

N20G70G73X.5Y.5   or   N20G71G73X12.7Y12.7
.N25G71

This will cause the part to be made one half the size of the metric program.