CHAPTER 4

COORDINATE WORDS

4.1 OVERVIEW

A coordinate word specifies an axis movement and consists of the address of the axis to be moved and the value indicating the direction of motion and the distance.

ADDRESS OF COORINATE WORD		MEANING	
Basic axis	X	X axis move	Absolute or
	Y	Y axis move	Incremental
	Z	Z axis move	
	U	X axis move	Incremental
	V	Y axis move	
	W	Z axis move	
Parameters for circular interpolation (G75)	I,J,K	Absolute: Coordinates of center of circle. Incremental: Signed distance from arc start point to center of arc	
Polar coordinates	R	Pole radius	
	I,J	Absolute: coordinates of center of pole Incremental: Signed distance from pole start point to center of pole	
	A	Angular distance, degrees absolute/incremental	
	B	Angular distance, incremental	

Refer to preparatory function descriptions for additional coordinate words used in special control program features.

4.2 CONTROLLED AXIS

Three axes (X, Y and Z) are controllable by the system. For rapid traverse two axes (X, Y) are moved simultaneously. If a Z axis move is programmed in the same data block as X and Y, it will be moved either before or after X, Y motion. For linear interpolation all three axes are controlled simultaneously. For circular interpolation two axes are controlled simultaneously, either X, Y or X, Z or Y, Z dependent upon the plane designated by the programmer. For helical interpolation, circular interpolation in the X, Y plane will occur simultaneously with coordinated Z motion.

4.3 COORDINATE SYSTEMS

4.3.1 Machine Coordinate System

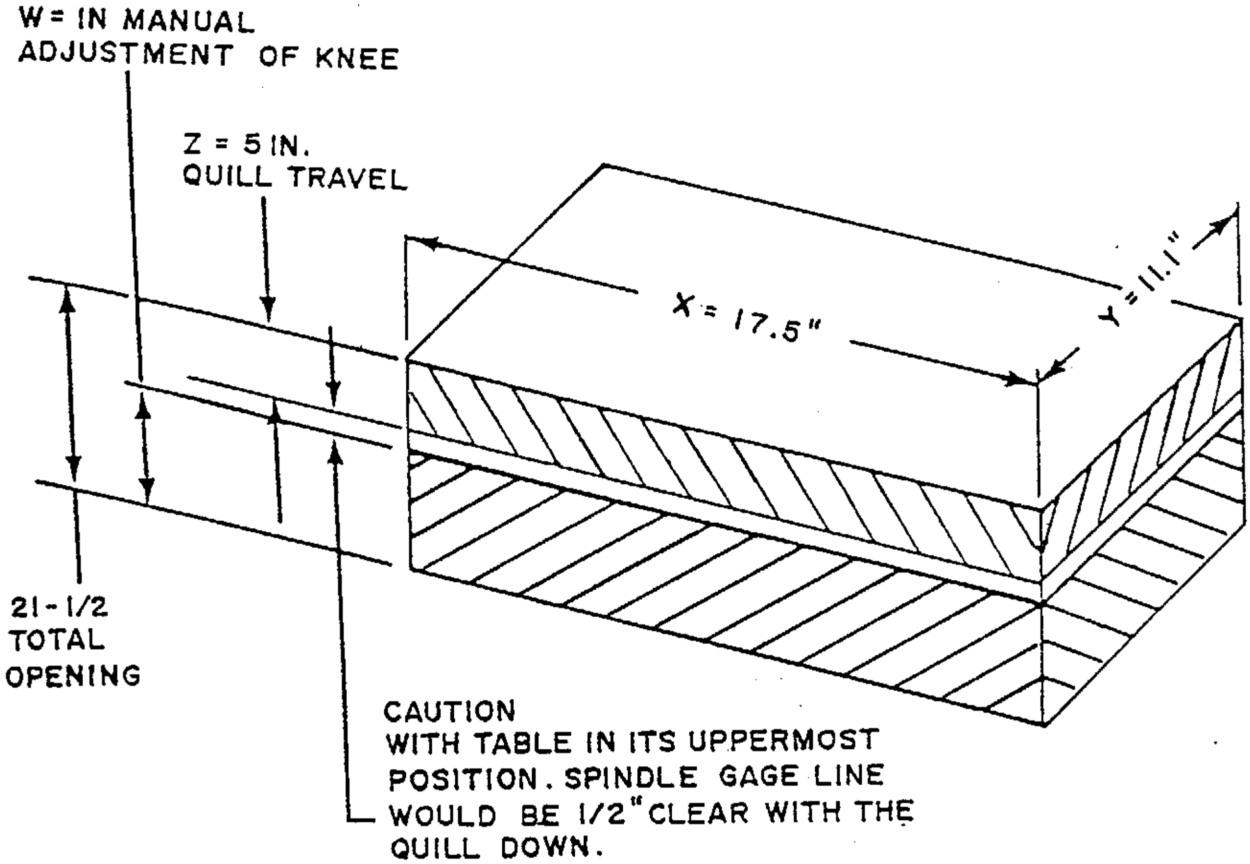
For the Bridgeport Series I Machine the axis travel limits are:

X axis (table)	17.5 inches
Y axis (saddle)	11.1 inches
Z axis (quill)	5.0 inches

During start up the axes are moved to the "Home" position, which is defined as the machine zero coordinate point. The quill will be in the uppermost position. The distance from the machine zero coordinate point to any point on the travel of the axes is called the machine coordinate point (or machine point).

NOTE

The "Home" position for all three axes is mechanically set by the position of the "Home" limit switches and a zero reference mark on the axis feedback encoders.



TOTAL MACHINE TRAVEL ENVELOPE

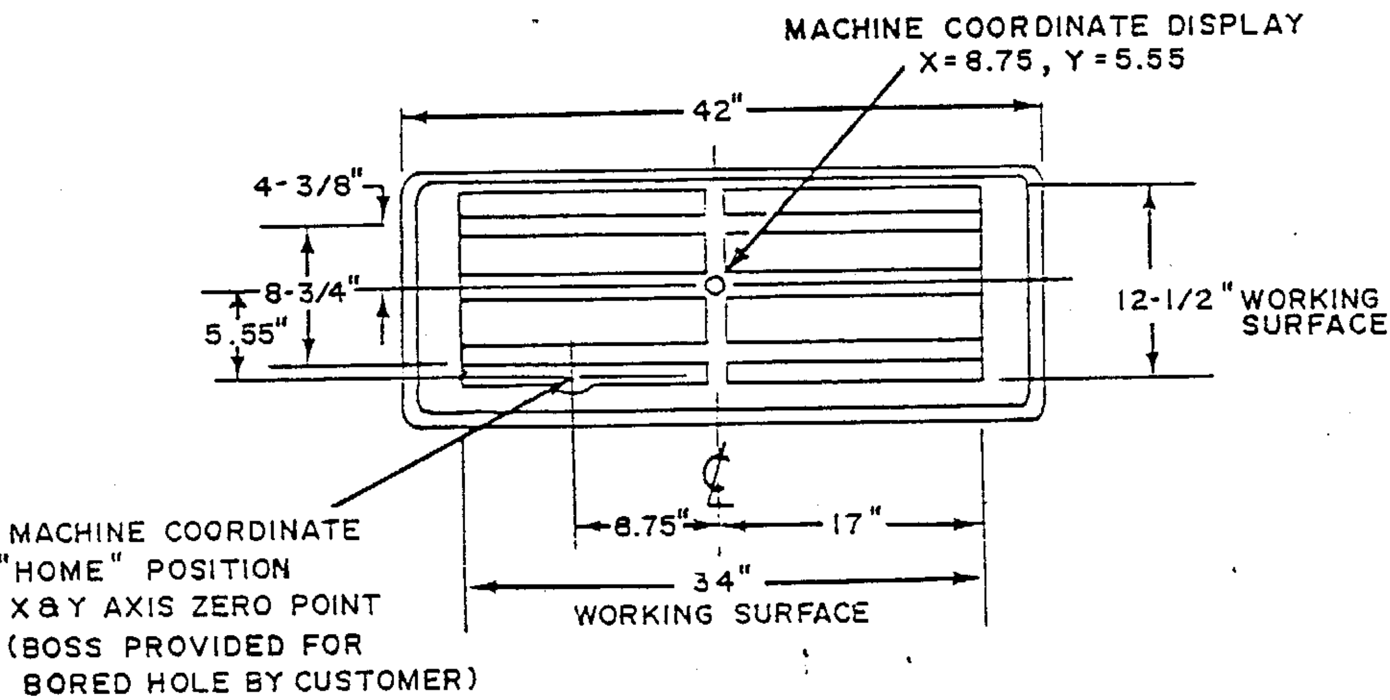


Figure 4-1: Machine Coordinate System

4.3.2 Part Program Coordinate System

When programming a part, the coordinate system must be established. Additionally, since the workpiece may be located at any arbitrary position on the table, the relationship between the machine coordinate system and the part program coordinate system must be established. This relationship is entered into the system by the operator during setup.

In the example shown next the following dimensions are known:

X-distance from bottom lefthand edge of part to X axis part program zero = 2.5
Y-distance from bottom lefthand edge of part to Y axis part program zero = 1.

Assume a .20" edge finder to locate the X coordinate for the bottom lefthand edge of the part. The X value of the machine coordinate point can be read as 1.90". Using the SET XYZ command, an X value of -2.60" is set into the control (distance along the X axis from the lefthand edge of the part to the part program zero point = 2.50" plus .10" to the center of the edge finder) to establish where the X axis part program zero is located on the table.

A similar process can be used to set the distance from the bottom lefthand edge to the Y axis part program zero. In this example, this value would be -1.10" using a .20" edge finder.

Once the location of the part program coordinate system is set it is stored in nonvolatile memory such that when power to the system is turned off, the value is retained. After homing the axes during start up, a MOV'XY command to X=0, Y=0 will cause the axes to move to part program zero.

The SET XYZ command is used to determine the location of the base part program coordinate system. A G92 command may be used within a part program to shift the location of part program zero. A G97 command allows establishing fixture offsets by shifting the part program coordinate system by an amount equal to the distance between the base part program coordinate system and the desired work part program coordinate system.

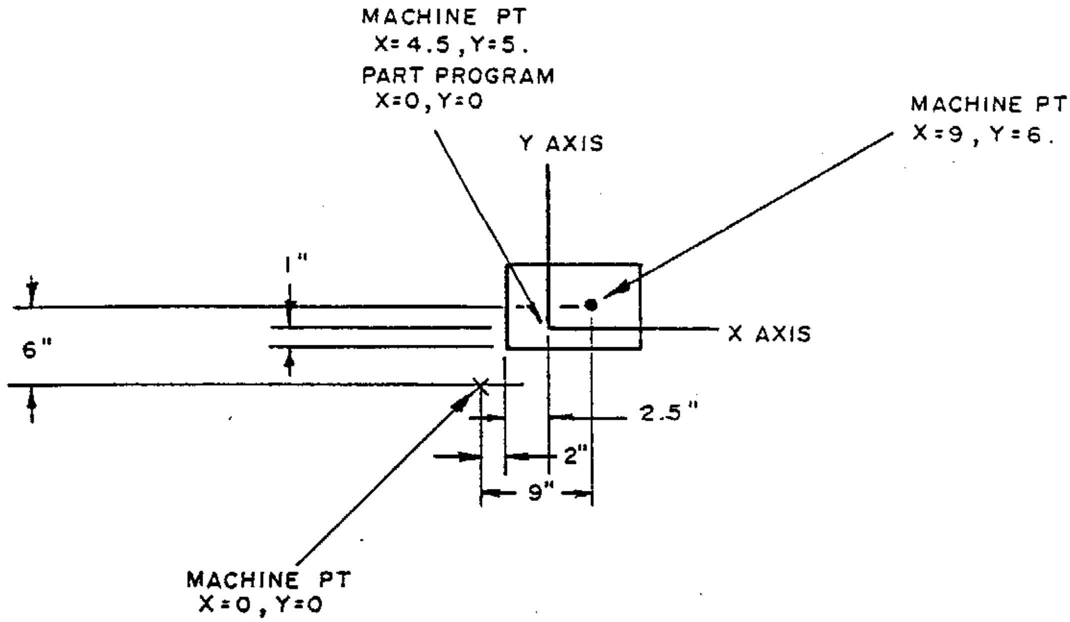


Figure 4-2: Part Program Coordinate System

Reference Point

A SET REF PT X and SET REF PT Y commands enable the operator to store a point on the table as a reference point. A MOV REF PT command causes the axes to move to the reference point. This feature can be used to retain the fixture location point. The SET REF PT X and SET REF PT Y commands can be used to store a point after the edge finder is used to pick up the bottom lefthand edge; first in the X axis and then in the Y, as shown in Figure 4-2. This enables rechecking the location of the fixture with the edge finder to determine if the setup dimensions have shifted over a period of time.

NOTE

The SET REF PT X and SET REF PT Y commands also reset the previously entered SET XY dimensions and the CLR PT. The distance from the REF PT to part program zero is reset to 0. The CLR PT location is reset to be the machine zero coordinate point (XY "Home"). Thus, if the SET REF PT command is used, it must be entered before the SET XYZ or SET CLR PT commands.

Clear Point

The SET CLR PT command enables storing the current XY axis position as a convenient point for changing tools. The M20, M21, M22, M26 commands halt part program execution and also sends the XY axes to this clear point. The quill is retracted prior to moving the X and Y axes.

NOTE

Both the REF PT and the CLR PT are stored as values with respect to the machine coordinate system.

4.3.3 Software Limits

In the control system all motion is based on the machine coordinate system. The part program interpreter adds an offset to all part program dimensions to transform them from the part program coordinate system to the machine coordinate system. A check is made by the part program interpreter before each move is made to determine if the move is within the limits of axis motion. These limits are set for the Series I machine as defined in Section 4.3.1. Exceeding these limits will cause an error message to be posted and the system will suspend part program execution. Refer to Appendix A in the R2E3 Series I Operating manual.

4.4 TOOL LENGTH OFFSET

Tool length offsets enable the operator to enter a value in the system so that adjustments can be made to allow for the difference between the tool length assumed by the programmer and the actual length of the tool used. Tool length offsets are useful when:

- Cutter preset dimension is not exact. This is particularly effective with end mills in end mill holders and also with tools which draw up into a collet.
- Change of reference plane.
- Broken cutter replacement.
- Dull cutter reground and replaced.

EXAMPLE:

Assume that Tool #1 has a travel of -2.000" from the upstop (Z axis home) and that Tool #2 has a travel of -2.000" + 1.500" from the same upstop. Further, consider the fact that the programmer requires both tools to do work on the workpiece at the same reference plane. It is evident that the programmer must know exactly where each tool tip is with reference to the only item that is common to both, the gage line. The programmer, must call for a specific Z axis motion on Tool #2 greater than Tool #1 if accuracies are to be achieved. Alternatively, a length offset capability in the control can take care of the difference.

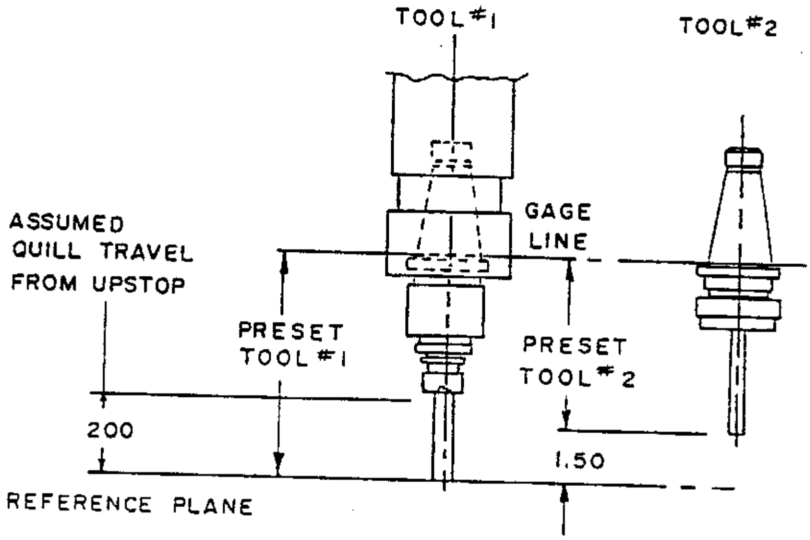


Figure 4-3: Tool Length Differences

The following methods can be used for setting the tool length offset.

a. TLO Touch off -

Use QUILL UP to move the quill to the up position. The display will show MZO. Jog the quill down until the tool just touches a convenient reference surface. Select the SET TLO=Z mode. The display will show the current tool number and the distance the quill is down from the mechanical quill zero reference point. Pressing EXECUTE will store this value as the offset for the designated tool. To check the value use QUILL UP to move the quill to the up position. QUILL UP will also cause the stored offset value for the current tool to be set as the current active tool offset value. Select the MOVE Z mode and enter 0. The quill will move down by an amount equal to the tool length offset.

b. Enter tool data via MDI -

Use the TOOL key to scroll through the tool data table, from tool numbers 1 through 24. ENTER loads the contents of the input screen (either T or TLO or DIA) as designated on the screen into the system.

NOTE

To skip over tool fields enter the desired T, tool number. For example: if currently positioned at tool #2 and tool #18 is desired, simply enter "18" and then press ENTER.

This method is useful when the measurement of the actual preset tip of the tool from a gage line can be compared with the required distance such that the difference can be input to the machine. The example below shows means for determining this offset distance:

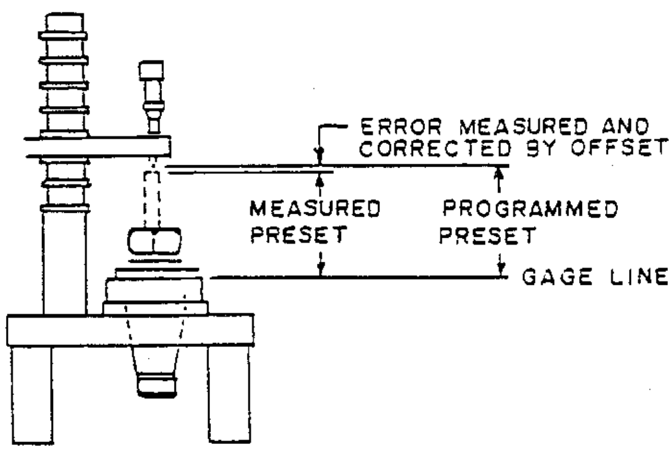


Figure 4-4: Measuring Actual Preset

c. Enter TOOL data via part program text.

The format is: T___/___/___

Where T___ is the tool number and /___/___ is the TLO value and the DIA value, respectively.

EXAMPLE:	T1/1.2;	FOR TOOL 1, TLO=1.2
	T5//-.5;	FOR TOOL 5, DIA=.5
	T4/.75/.2;	FOR TOOL 4, TLO=.75, DIA=.2

NOTES

1. Tool length offsets are unsigned numbers.
2. If a tool length offset already exists in the tool table, it cannot be overwritten by part program data.
3. The maximum tool length offset that can be input is 6.5536 inches (166.461mm).
4. 24 TLOs are allowed.
5. The tool length offset is automatically set into the system when a tool change (M6,M26) command is executed.
6. The diameter offset is automatically set in the system when a T (tool number) command is executed.

To set up the knee to use tool length offsets the following procedure is used:

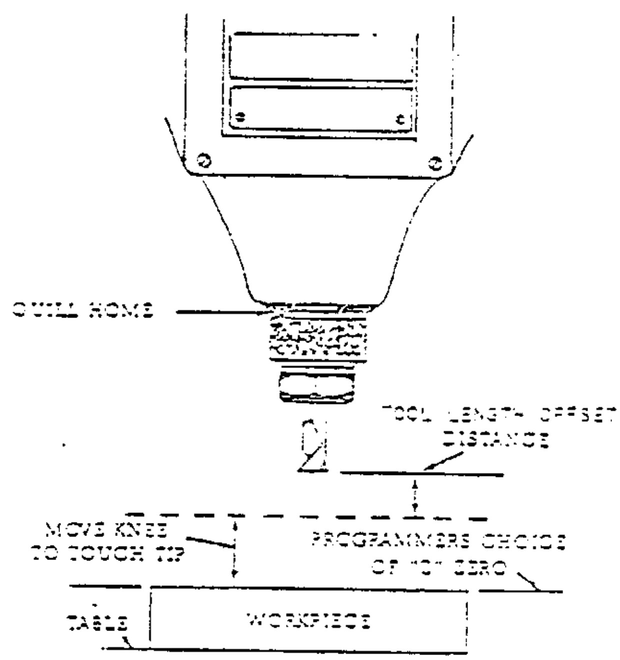


Figure 4-5: Translating The Z Coordinates

Place the longest tool that is to be used in the current job in the spindle. Call this tool out with its TLO value (which must be a minimum of .500"). Press GOTO. The quill will move down .500". Loosen the clamps on the knee and raise it to touch the "Z" Zero Plane on the workpiece to the tip of the tool. Lock the knee again; the machine Z axis and knee are now set for the job.

4.5 ABSOLUTE AND INCREMENTAL PROGRAMMING

The distance of tool travel in each axis may be input as either absolute or incremental data. Using incremental commands (G91) the data in a word is the distance along the designated axis from the existing position to the desired position. Using absolute commands (G90) the data in a word represents the coordinate value of the point from part program zero.

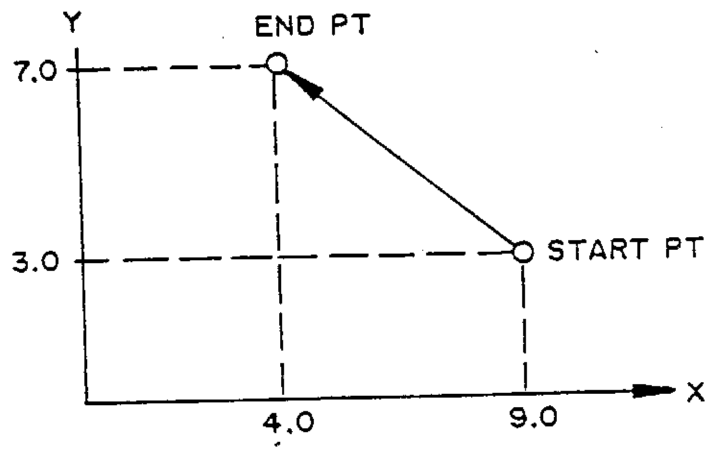


Figure 4-6: Incremental And Absolute Programming

For the example shown in Figure 4-6, programming in Incremental mode would be:

G91X-5.Y4.;

Programming in Absolute mode would be:

G90X4.Y7.;

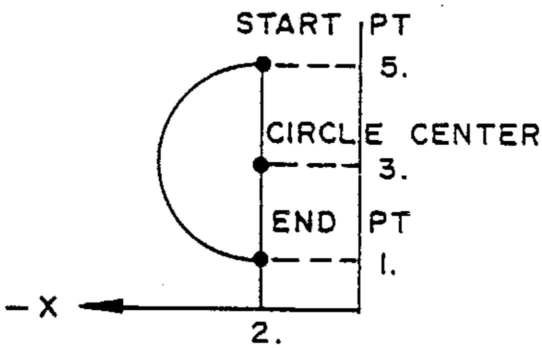
It is possible to change modes in a block of data, for example:

G90X4.G91Y4.;

U, V, W are incremental X, Y, Z moves, respectively. Thus, it is possible to program the above example as:

G90U-5.V4. G91U-5.V4. G90X4.V4.

In addition, the center point of an arc may be described as either an incremental or absolute dimension. In Incremental mode, the center of the arc is designated as the distance from the start point of the arc to the arc center. In Absolute mode, the center of the arc is designated by its coordinates from part program zero.



For example, in Incremental mode the arc would be programmed:

G91G3X0Y-4.I0J-2.F10.0

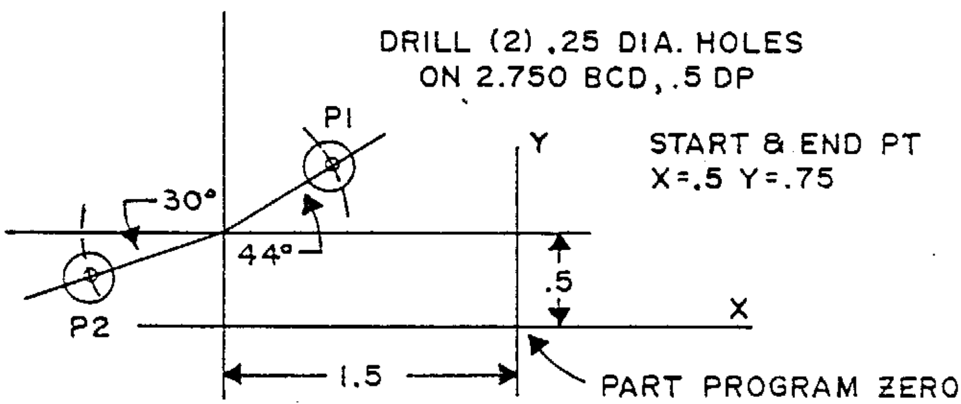
In Absolute mode the arc would be programmed:

G90G3X-2.Y1.I-2.J3.F10.0

4.6 POLAR COORDINATES

Besides rectangular coordinates, polar coordinates may be used to designate point locations. The location of a point is designated by the radius from the pole center and by the angle the radius makes with reference to the positive X axis.

EXAMPLE:



In Incremental mode this would be:

N1G92A0;	PRESET A=0.
N2G91R1.375I-2.J-.25;	DEFINE POLE CENTER
N3G81A44.Z.55F20.;	DRILL 1ST HOLE
N4A166.;	DRILL 2ND HOLE
N5G90G0X.5Y.75;	GOTO END PT

NOTES

1. Sequence numbers 2 and 3 could be combined:

   N2G91R1.375I-2.J-.25G81A44.Z.55F20.

2. I and J the pole center are defined incrementally (except if the system is in the BOSS 4-7 compatibility mode). In Absolute mode this would be:

```
N0G92A0
N1G90R1.375I-1.5J.5;   DEFINE POLE CENTER
N2G81A44.Z.55F20.;     DRILL 1ST HOLE
N3A210.;               DRILL 2ND HOLE
N4G0X.5.Y.75;          GOTO END PT
```

3. The range of A is +719.999 to -359.999 degrees.

4. Positive angles are measured CCW from the positive X axis, negative angles are measured CW from the positive X axis.

5. Polar coordinate data can be rotated, scaled and used in conjunction with cutter diameter compensation.

EXAMPLE:

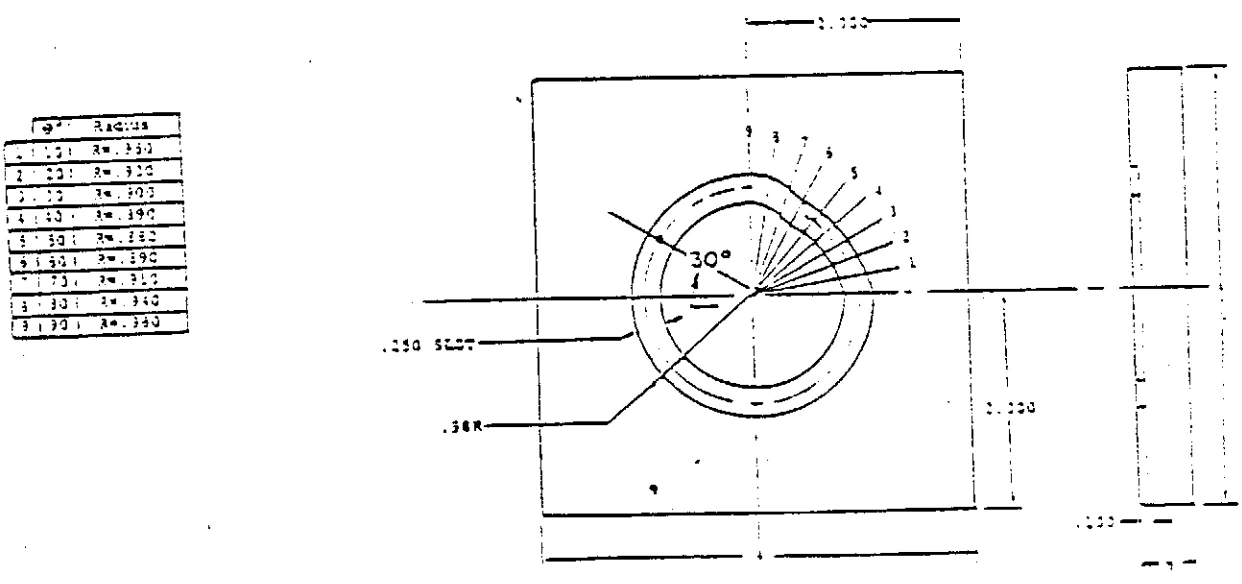


Figure 4-7: Cam Track

```
N1G0G90X0Y0T1M6;            START UP PT
N5G92A0;                    PRESET A=0.
N10R.98I0J0A150.;           GOTO A=150.
N15G13A30.Z-.12F12.         HELIX DOWN INTO PART
N20G3A0;                    CIRCLE CW TO 0 DEGR
N25G1R.95A10.;              PT1
N30R.93A20.;                PT2
N35R.9A30.;                 PT3
N40R.89A40.;                PT4
N45R.88A50.;                PT5
N50R.89A60.;                PT6
N55R.91A70.;                PT7
N60R.94A80.;                PT8
N65R.98A90.;                PT9
N70G3A185.;                 CIRCLE CW TO 185 DEGR
N75G13A30.,Z.135;           HELICAL OUT OF PART
N80G0X0Y0M2
```

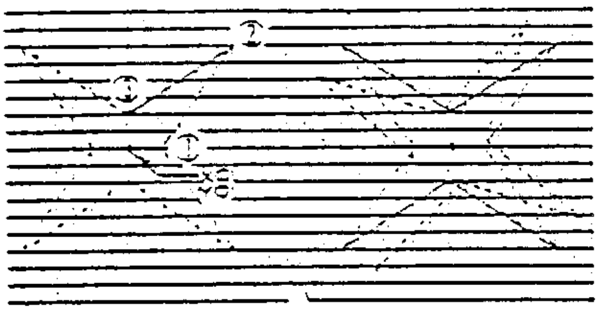
4.7 TRANSFORMATIONS

Under certain conditions it is useful to perform the following operations:

- Rotate the points on the cutter path through an angle in the XY plane.
- Scale the points on the cutter path by a given amount.
- Translate the cutter points to a different reference system. The transformation functions provide these capabilities.

4.7.1 Rotation

Rotation of a programmed part shape can be done by inserting the appropriate preparatory function 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 function code (G72) will turn transformation off.



```
(1)N20G0X.5Y0
   N25Z-.05
   N30G92A0
   =N60/4
(2)N40G90G1X1.5Y1.5F120.
(3)N50X0Y.5
   N60G91G73A90.0
   N70G0G90G72X-4.5Y3.75Z.05
```

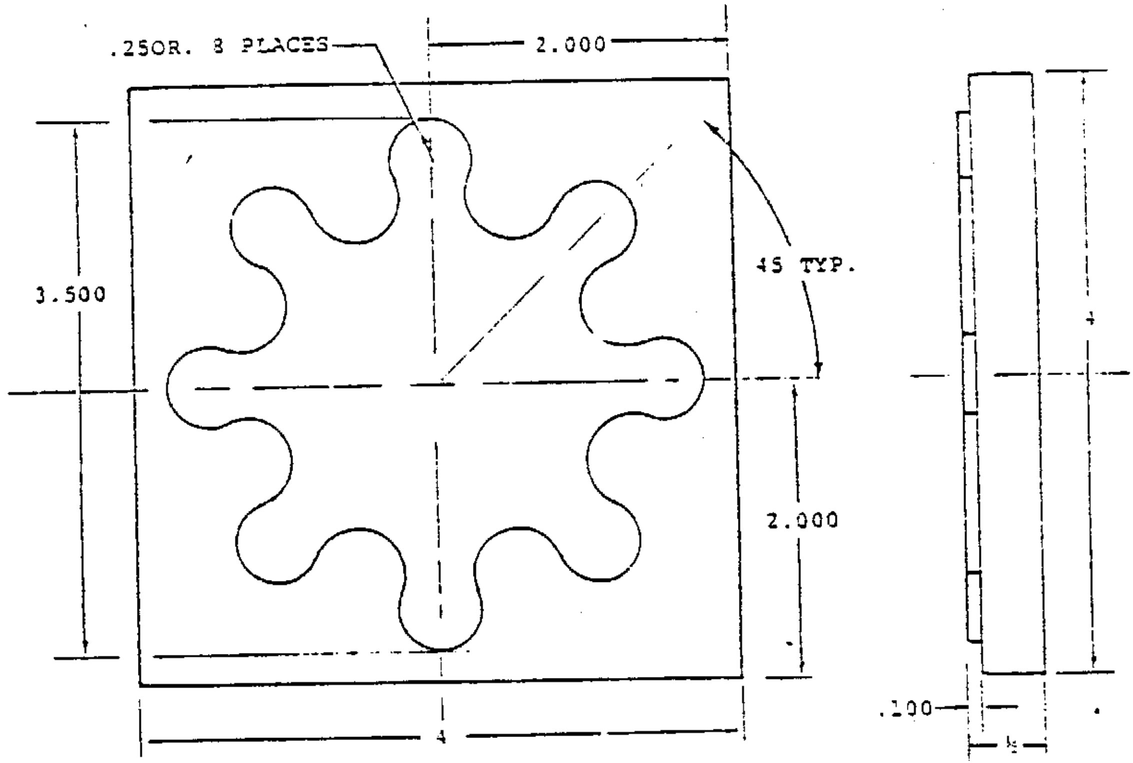
```
%N10G0G90X-4.5Y3.75T1M6
N15G92A0
#1
N20X.5Y0
N25Z-.05
=N60/4
N40G90G1X1.5Y1.5F120.
N50X0Y.5
N60G91G73A90.0
N65G0G90G72Z.05
$
=#1
N70G73A15.0
=#1
N80G0G90G72X-4.5Y3.75M2
E
```

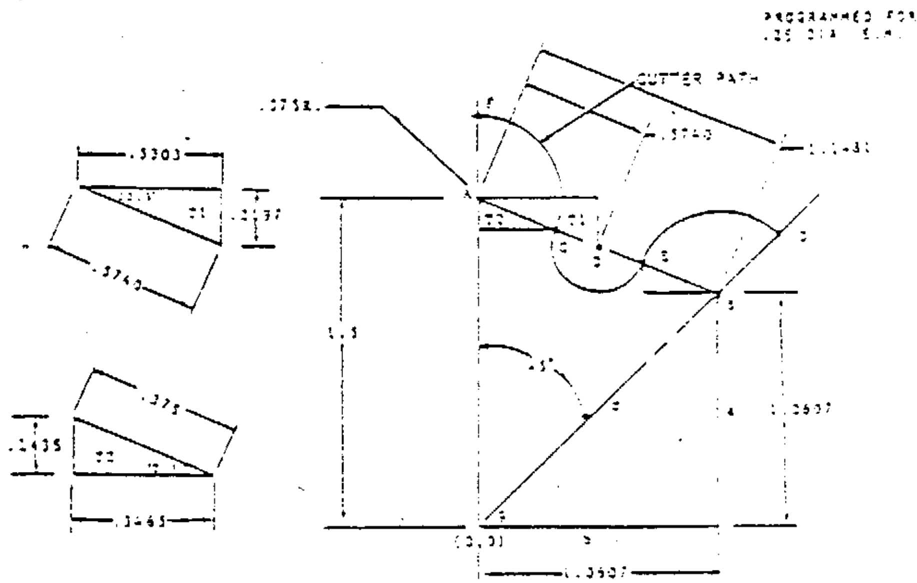
Features of Rotation:

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 routed 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 4-8 for a Geneva Mechanism Rotor.





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

Figure 4-8: Rotor, Geneva Mechanism

```
%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
```

NOTES

1. In the Transformation mode (G73) all X, Y, I, J data must be entered even though it may repeat a value previously entered.

2. Transformation by rotation takes place about XY absolute zero.

3. If rotation and scaling are used simultaneously, the order will be rotation first, then scaling.

4. A cutter path with diameter compensation in effect may be translated only if it is transformed (G73) before the compensation is turned on (G41, G42). The compensation must be stopped (G40) before the transformation is ended (G72).

4.7.2 Scaling

Scaling has the following format:

G73X___Y___Z___

Where X___ is the X scale factor, Y___ is the Y scale factor, Z___ is the Z scale factor. The scale factor has a range from .001 to 99.999.

NOTES

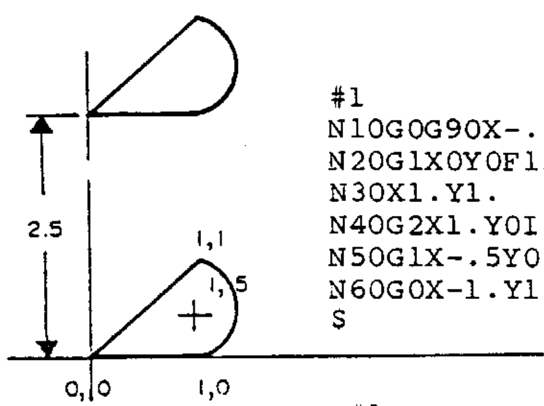
1. In the transformation mode (G73) all X, Y, Z data must be entered even through it may repeat a value previously entered.

2. If circle data is input, the X and Y scale factors must be the same.

4.7.3 Translation

The fixture offset (G97) command enables translating data from the base part program coordinate system to a designated work coordinate system. G96 re-establishes the base coordinate system.

EXAMPLE:




```
#1
N10G0G90X-.5Y0Z0
N20G1X0Y0F15.
N30X1.Y1.
N40G2X1.Y0I1.J.5
N50G1X-.5Y0
N60G0X-1.Y1.
S

=#1;      DO PART IN BASE COORD SYSTEM
G97Y2.5;      SETUP NEW COORD SYSTEM
=#1;  DO 2ND PART IN NEW COORD SYSTEM

G96G0G90X-1.Y1.M2; RETURN TO BASE
                   COORDS
```

NOTE

The G96 code may be placed in the last block.

The format of the G97 command is:

G97X___Y___Z___

Where X___Y___Z___ are the dimensions from the new work coordinate system to the base coordinate system. The G97 command can be used many times in a part program, each G97 command contains the dimension from the new coordinate system to the base coordinate system. G96 has the same effect as G97X0Y0, restoring the base coordinate system.

The translation function is useful in machining many workpieces placed on a work table sequentially.

NOTE

The first move after a G97 or G96 block must be an absolute poritioning move.

4.7.4 Mirror Image (G30,G31,G32)

These codes act to invert the direction of X, Y axis input commands and enable mirror images of the programmed part.

G30: Mirror Image Cancel
G31: Mirror X axis
G32: Mirror Y axis

NOTE

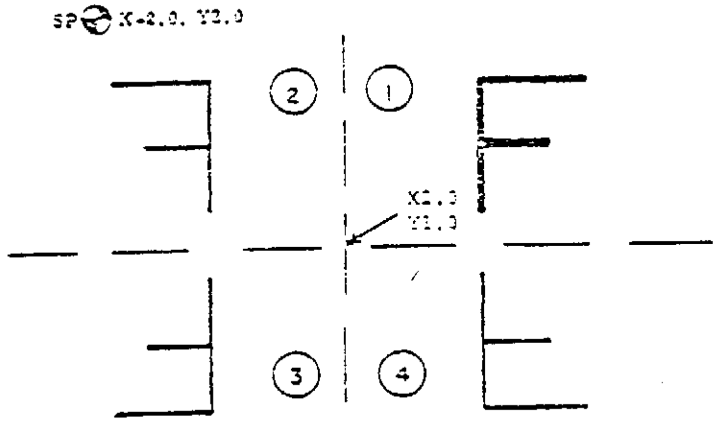
The point at which G31 is programmed is the axis of X symmetry. The point at which G32 is programmed is the axis of Y symmetry. The value of the X, Y output coordinate is:

X (coord) = X (symm) -X (input)
Y (coord) = Y (symm) -Y (input)

The order of transformation is rotation, mirror image, translation.

EXAMPLE of G30,G31,G32:

```
#1
N5G0G90X2.Y1.Z.05
N10X3.Y1.25
N15G1Z-.05F15.0
N20G91Y.25
N25X.25
N30X-.25
N35Y.25
N40X.3
```




```
N45G0G90X2.Y1.Z.05
$

   N100G0G90X-2.Y2.T1M6
(1)=#1
   N200G31
(2)=#1
   N300G32
(3)=#1
   N400G30G32
(4)=#1
   N500G0G30G90X-2.Y2.M2
```