SECTION V

PROGRAM ENTRIES DESCRIPTIONS

The following paragraphs contain descriptions, use and meaning of the program entries. Also covered in this section are various considerations to be taken into account and procedures to follow in programming a part. Character packing using the decimal is illustrated throughout thus dropping both leading and trailing zeros. This section covers basic Word Address programming; repetitive programming is covered in Section VI.

The first valid code on the control tape is the Rewind Stop. ASCII uses the percentage sign (%); it is ignored when reading in the forward direction. (Tape reader control only.)

5.1 DEFINITION BLOCK AND BLOCK DELETE

5.1.1 Definition Block (.)

When a decimal is the first character in a block of data, the information contained in that block will be executed during the search mode as well as during program execution. This feature should be used when a system mode is changed in a program, e.g. G90 to G91, G70 to G71, starting a Z axis canned cycle, etc.

WARNING: **Because the data contained in a definition block is executed during the search mode, a block causing slide motion must not be programmed.**

EXAMPLE:

%N1G0G90X0Y4.T1M6	
.N3G4/50	No motion - G4/50 sets Dwell
N5X1.Y2.Z.05	
.N10G82Z.55F80	No motion - Without X and/or Y coordinate canned cycle will not execute (See page 5-11)
N15X1.	
N20X0Y1.	
N25X1.Y0	
N30X2.Y1	
.N35G0G91	No motion - No X and/or Y coordinates
(etc.)	

5.1.2 Block Delete (/)

If it is desired to bypass certain portions of a tape program, a block delete (/) is entered at the beginning of each block of tape information which may require deletion. When the operator turns the DELETE ON-OFF switch, located on the control console, to the ON position, the control will read through all blocks of information which are preceded by the delete code, ignoring all commands in those blocks. (/) code must be the first character on a data block to be recognized as a delete code.

To cancel this function from the control, the DELETE ON-OFF switch must be placed in the OFF position to return to normal operation.

This feature could be used, for example, where a trial cut procedure is required. Each block of information within the trial cut sequence would be preceded by a delete code. The trial cut procedure could then be taken or by-passed at the operator's discretion, or as directed by written instructions from the programmer.

CAUTION: CARE MUST BE TAKEN TO AVOID DELETION OF INCREMENTAL DATA.

5.2 SEQUENCE NUMBER (N)

A programmed sequence number addressed by the letter N followed by up to 5 digits ranging from 0 to 19999 is used to identify blocks of tape information. As each programmed sequence number is read, the last 3 digits will be displayed on the 3 digit universal readout after the block in which it is contained is transferred to the active register. Thousandths and ten thousandths are not displayed.

NOTE: The Sequence Number code is also used as a checkpoint for Repetitive Programming. See Section VI.

5.3 PREPARATORY FUNCTIONS (G)

A preparatory function is required to change the programmed mode of operation of the control. The letter address G followed by two digits indicates the mode of operation. More than one preparatory function can be programmed in one block of information, however, caution must be exercised as the functions may be self-cancelling, e.g., G0 G1, in which case only the last function programmed would be in effect.

5.1.2 Block Delete (/)

If it is desired to bypass certain portions of a tape program, a block delete (/) is entered at the beginning of each block of tape information which may require deletion. When the operator turns the DELETE ON-OFF switch, located on the control console, to the ON position, the control will read through all blocks of information which are preceded by the delete code, ignoring all commands in those blocks. (/) code must be the first character on a data block to be recognized as a delete code.

To cancel this function from the control, the DELETE ON-OFF switch must be placed in the OFF position to return to normal operation.

This feature could be used, for example, where a trial cut procedure is required. Each block of information within the trial cut sequence would be preceded by a delete code. The trial cut procedure could then be taken or by-passed at the operator's discretion, or as directed by written instructions from the programmer.

CAUTION: CARE MUST BE TAKEN TO AVOID DELETION OF INCREMENTAL DATA.

5.2 SEQUENCE NUMBER (N)

A programmed sequence number addressed by the letter N followed by up to 5 digits ranging from 0 to 19999 is used to identify blocks of tape information. As each programmed sequence number is read, the last 3 digits will be displayed on the 3 digit universal readout after the block in which it is contained is transferred to the active register. Thousandths and ten thousandths are not displayed.

NOTE: The Sequence Number code is also used as a checkpoint for Repetitive Programming. See Section VI.

5.3 PREPARATORY FUNCTIONS (G)

A preparatory function is required to change the programmed mode of operation of the control. The letter address G followed by two digits indicates the mode of operation. More than one preparatory function can be programmed in one block of information, however, caution must be exercised as the functions may be self-cancelling, e.g., G0 G1, in which case only the last function programmed would be in effect.

The following is a list of these preparatory functions and a description of each. Subsequent paragraphs in this manual will describe the use of these functions in the context of a program.

G00 - Rapid traverse, positioning mode.

This function will cause the machine to operate in the rapid traverse positioning mode with absolute or incremental data input.

NOTE: Programmed feed remains in the feedrate register.

The XY axes involved will move towards the programmed point in rapid traverse at 45 degrees first followed by the continued motion of the major axis itself.

If the system is in rapid traverse and a Z move is programmed (and even if an X and/or Y move is concurrent), logic in the control console will split the move into XY motion as above and a separate Z motion. If the programmed Z motion requires the quill to go up, the Z motion will occur first then the X and/or Y move. If the programmed Z motion requires the quill to go down, the X and/or Y motion will occur first followed by Z motion.

G00 cancels G01, G02, G03, G18 and G19. It will also set the system in the G80 (fixed cycle G81, through G89 cancel) mode. Initializing the control with POWER ON, or RESET places the control in rapid traverse, positioning mode (G00).

G01 - Linear interpolation, feed.

This function will cause the machine to operate in the feed range (.2 to 32.0ipm)* as the axes travel along a straight line. The starting point of the path is defined by the X, Y, Z coordinates of the previous block. The end point is the programmed coordinates contained in the block of data. G01 cancels G00, G02 and G03.

G02 - Circular interpolation mode clockwise. (Figure 5-1)

This function indicates that the axes motions are to generate an arc in the clockwise direction. The arc radius is defined by the start point (the position at the end of the previous data block), the end point, and the arc center (I, J, K coordinates). Arc contouring motions are limited to movement in two out of three linear axes (X, Y, Z). G02 cancels G00, G01, and G03.

G03 - Circular interpolation mode counterclockwise. (Figure 5-1)

This function is the same as G02 except the axes motions generate an arc in the counterclockwise direction. G03 cancels G00, G01 and G02.

*(.2-51.IPM-BOSS 6.0)

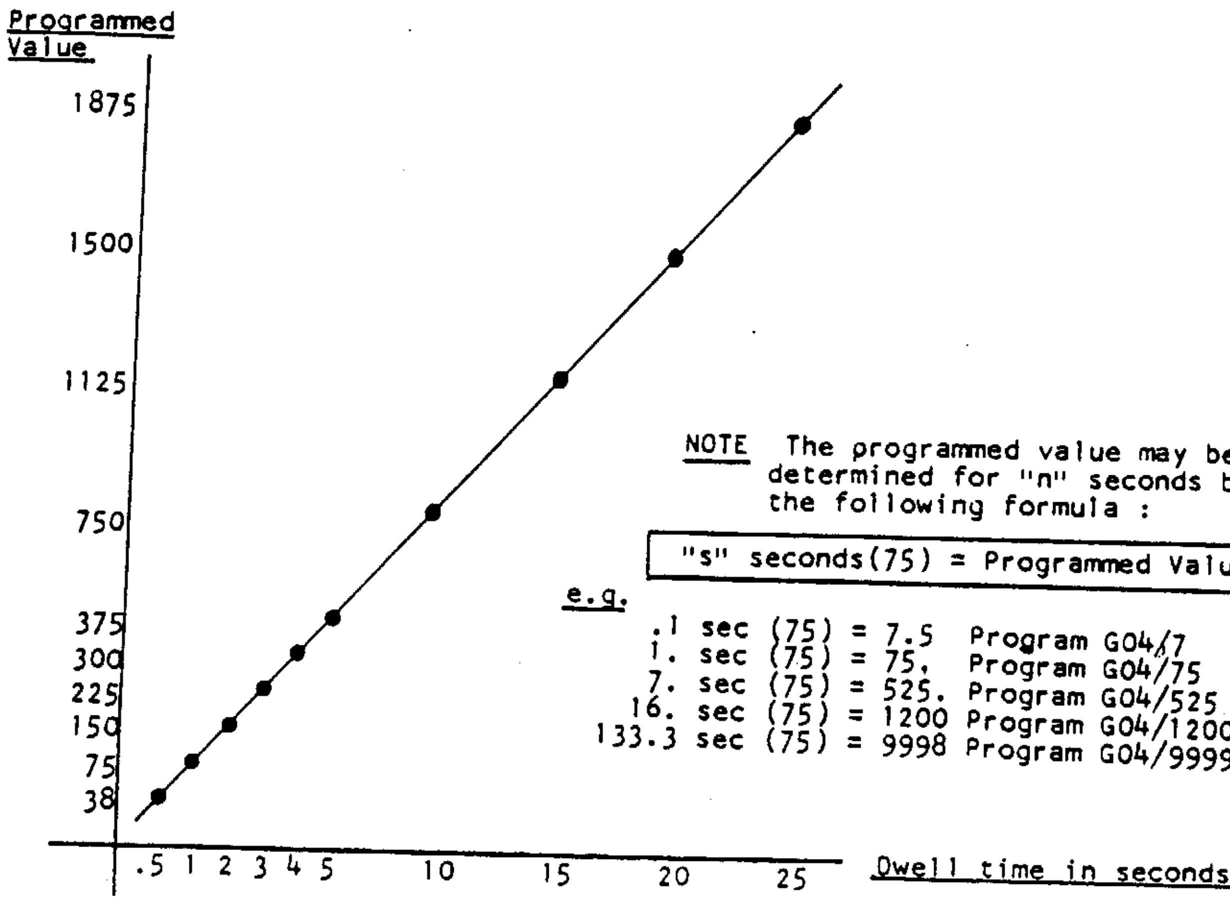
G04- Dwell

This preparatory function has the capability of setting a dwell time from .01 seconds to approximately 133 seconds. It may only be programmed for use with the following canned "Z" cycles:

G82 - Drill with dwell
G89 - Bore with dwell

When this function is read and executed on the machine, it will set an internal timer only, thus no dwell will occur in that block. The dwell will occur subsequently when either a G82 or G89 is executed.

The format is G04/n where n, the programmed value, is a number from 1 to 9999. This number is determined according to the following table and resultant formula:



NOTE The programmed value may be determined for "n" seconds by the following formula :

"s" seconds(75) = Programmed Value

e.g.

.1	sec (75)	=	7.5	Program G04/7
1.	sec (75)	=	75.	Program G04/75
7.	sec (75)	=	525.	Program G04/525
16.	sec (75)	=	1200	Program G04/1200
133.3	sec (75)	=	9998	Program G04/9999

NOTE: Round off to nearest whole number for input value.

With large production runs a fraction of a second reduction in cycle time can result in significant overall cost savings. It is important, therefore, to be able to calculate the dwell time based upon the desired number of revolutions of the cutting tool.

Consider the following formula and example:

Dwell in Seconds = Desired # of Rev./(Spindle RPM /60)

e.g.

Desired Dwell time 3 revolutions

Spindle Speed 1500 revolutions/min.

$$\# \text{ of Seconds} = 3 \text{ rev.} / (1500 \frac{\text{rev}}{\text{min}} / 60 \frac{\text{sec}}{\text{min}})$$

$$= \frac{3}{25}$$

$$= .12 \text{ sec.}$$

Referring to formula shown in Table 5-1

.12 sec(75) = 9

Programmed Block: G04/9

G17 - XY Plane selection. (Plan View) Figure 5-1

This preparatory function sets up the XY plane of operation for circular interpolation mode. This plane is the plane of operation in the control upon machine start-up. G17 cancels G18 or G19.

G18 - ZX Plane selection. (Front Elevation) Figure 5-1

This preparatory function sets up the ZX plane of operation for circular interpolation. This is used in conjunction with and programmed after the direction modifiers G02 or G03. G18 is cancelled by G00, G01, G17 or G19.

NOTE: Circular interpolation must be incremental input.(G74 Mode Only)

G19 - YZ Plane selection. (LH Side Elevation) Figure 5-1

This preparatory function sets up the YZ plane of operation for circular interpolation. This is used in conjunction with and programmed after the direction modifiers G02 or G03. G19 is cancelled by G00, G01, G17 or G18.

NOTE: Circular interpolation must be incremental input.(G74 Mode Only)

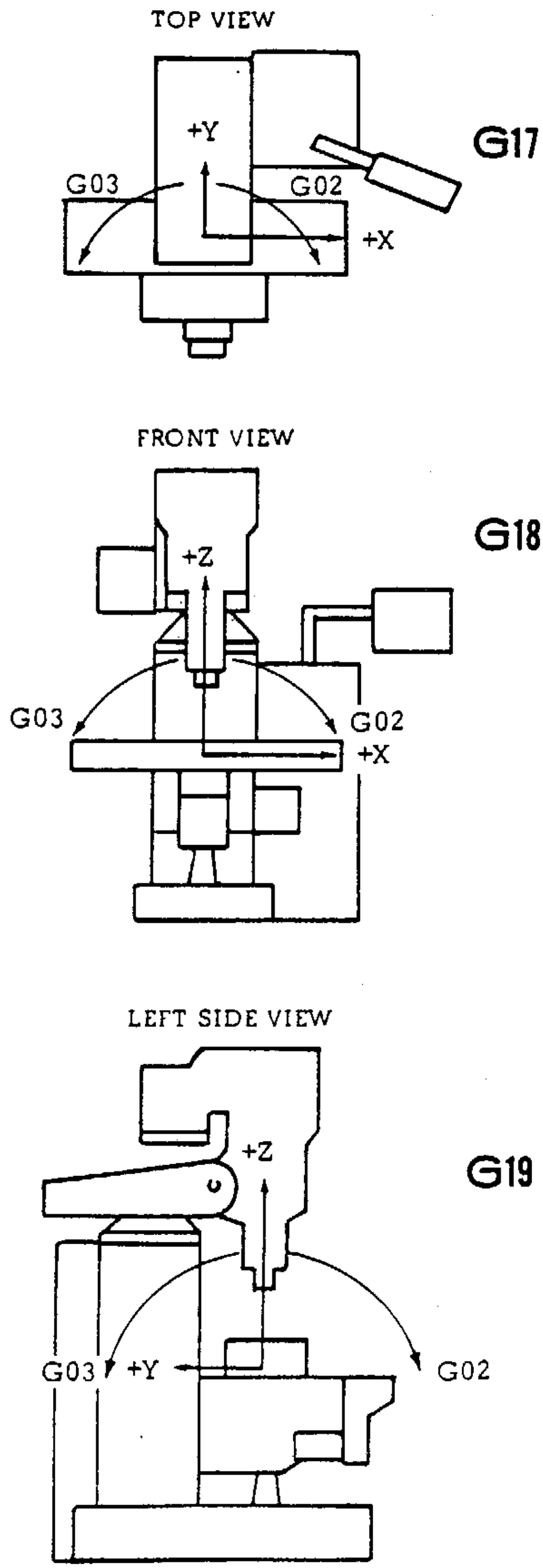


Figure 5-1. CW or CCW in Switchable Planes

G30 - Cancel Sign Reversal.

This is the normal POWER ON or RESET state for the control; it establishes plus and minus directions for the X and Y axes in accordance with EIA standard RS-267. Cancelled by G31 or G32.

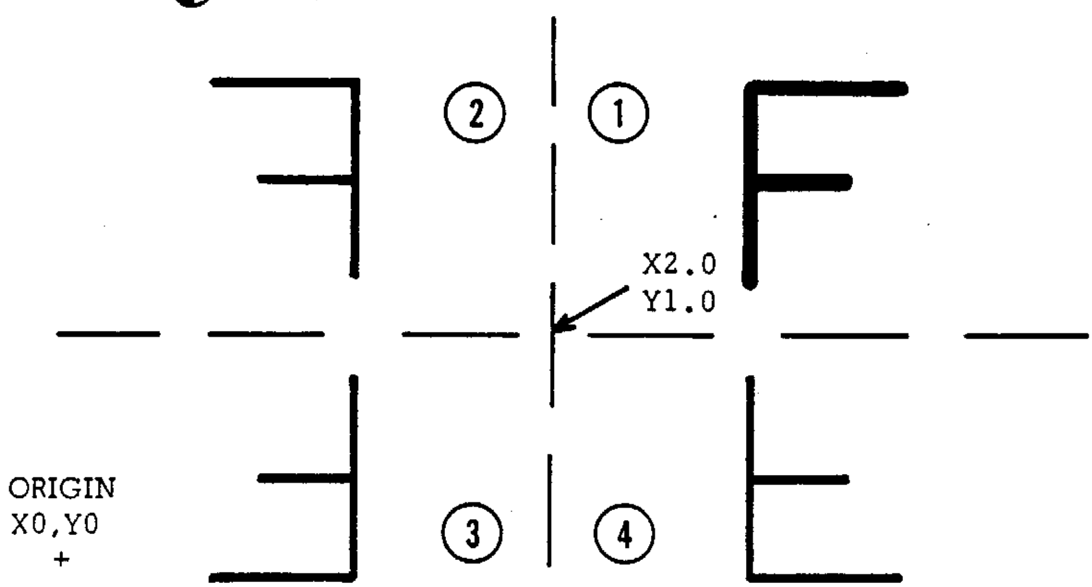
G31 - Reverses the direction signs for the X-axis.

G32 - Reverses the direction signs for the Y-axis.

The G30, G31 or G32 function must be programmed at the axis of symmetry. The absolute coordinates may be any value at the point when the function is programmed. All absolute, all incremental or a mixture of both coordinate systems may be programmed. The LED display in the operator's main panel will not show the correct absolute coordinates except at the axis of symmetry where the function is invoked or cancelled. G31 or G32 is cancelled by G30.

Example of G30, G31, G32:

SP X-2.0, Y2.0




```
#1
N5G0G90X2.Y1.Z.05
N10X3.Y1.25
N15G1Z-.05F150
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
```

G40, G41, G42 - Cutter Diameter Compensation. See Section 9.3

G70 - <u>Inch Dimension System Input</u>.

This is the normal POWER ON or RESET state for the control. TLO's and axis motion data are interpreted as being in the inch dimension system. Resolution is seven digits (000.0000"). G70 is modal and cancels G71.

G71 - <u>Metric Dimension System Input</u>.

After a G71 command, all TLO's and axis motion data are interpreted as being in the millimeter dimension system. Resolution is seven digits (0000.000mm). G71 is modal and cancels G70.

NOTE: Normally G71 would appear in front of any metric tape data and a G70 would appear in the program just before the M02. <u>Use Definition Block</u>.

SPECIAL MILL CYCLES (G77, G78, G79)

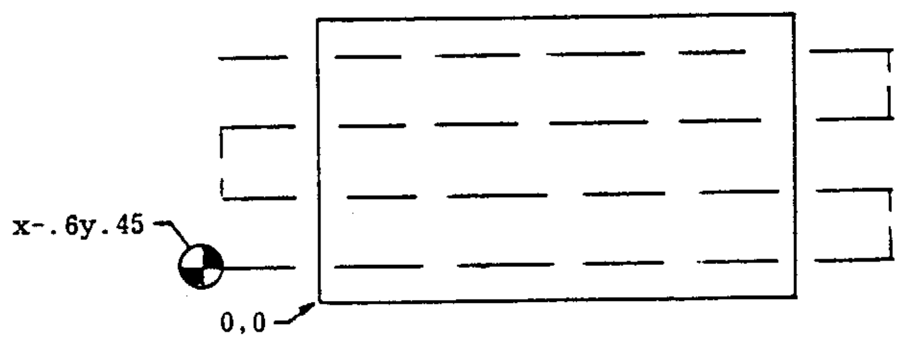
The control has three special mill cycles that reduce programming time and tape length for certain operations. These cycles require the tool to be at the start position and to depth <u>before</u> they are programmed.

G77 - <u>Facing Cycle</u>.

The format is: G77Xn1Yn2Yn3FV where

*n1 = signed incremental distance to be milled along X axis
*n2 = incremental distance to be milled along the Y axis
n3 = Y axis step over value
V = feedrate value *Not necessary to program G91 (incrementa

The G77 block must be followed by Absolute Coordinate data.



G77X7.2Y2.85Y.95F100

will face mill a 6.0" X 3.75" block as shown with a 1" dia. end mill. Absolute coordinates must not be used.

From the bottom left hand corner of the area to be milled, the first move will be an +X axis move equal to the X axis input value. The cutter will then

then make the last X axis move.

NOTE: 1. Tool must be at start position and to depth before initiating cycle.
2. Cutter Diameter Compensation cannot be used.
3. Next move after G77 must be absolute.

G78 - Pocket Milling Cycle.

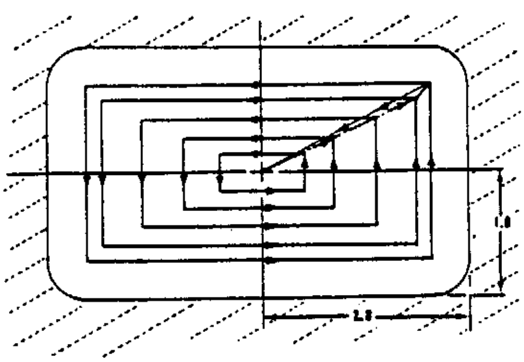
The format of this cycle is: G78Xn1Xn2Xn3Yn4Yn5Fv1Fv2 where:

n1 - the distance from the center of the pocket to the wall along the X axis less the cutter radius.
n2 - the X axis stepover value.
n3 - the optional distance for the final boundary cut. If n3=0, then the final boundary cut is set at the default value of .020".
n4 - the distance from the center of the pocket to the wall along the Y axis less the cutter radius.
n5 - the Y axis stepover value. If n5 is not programmed, the X and Y stepover distance will be equal to n2.
v1 - the feedrate value for all the roughing passes.
v2 - the optional feedrate for the boundary cut. If v2=0, the boundary cut will have a feedrate of 1.5 times v1.

For example: Mill the pocket shown using a .5" diameter cutter. For the most efficient result, using X .25" stepover maximum value, the minor axis (Y) stepover value should be .107"; calculated according to the following:

$$.25" \, (n4/n1) = .25 \, (.75/1.75) = .107$$

Program: G78X1.75X.25Y.75Y.107F100



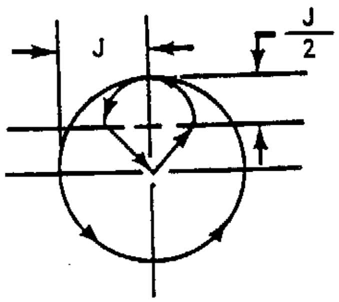
From the center of the pocket, the first move will be an X and Y axis stepover. The center will then move -X, -Y, +X, +Y at a value equal to twice the accumulated stepover distance. The cutter will then make another XY stepover move. This will continue until the cutter is within .020" of the pocket wall. NOTE: Once an axis reaches .020" from the pocket wall, it will no longer be incremented during the stepover move. The last stepover move will be .020" in both X and Y followed by a rectangular cut at a feedrate 50% higher than the input feedrate. The cutter will then feed to the center of the pocket to end the cycle.

Alternatively Program: G78X1.75X.25X.01Y.75Y.107F100F120

The roughing passes will proceed as before at 10 ipm and will continue until the cutter is within .010" of the pocket wall. The finishing +X+Y departures will then take place and the .010" perimeter pass will be made at 12 ipm. The tool returns to the center of the pocket at 12 ipm.

G79 - Internal Circular Milling Cycle

This cycle, consisting of only one block, may be useful in machining circular pockets, counterboring large diameters, rough hole boring, etc. It is the programmers responsibility to be at the center of the hole and to depth before calling the G79 cycle.



The format required is as follows:

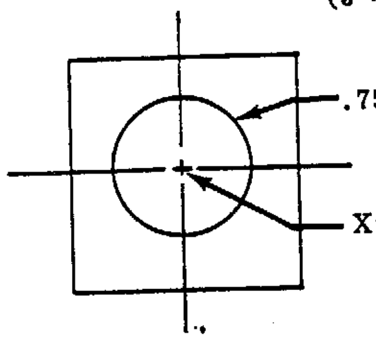
G79 J (value) F (value)

Where the J value is the radius of the hole to be machined minus the cutter radius. i.e.

J = (Part Radius) - (Cutter Radius)

Note: The accompanying sketch shows the actual cutter path for the G79 cycle. Note that the entering and leaving tangent small circle's radius is J/2.

Program #1 — Programmed for .5 Dia. End Mill (J = (.375)-(.25) = .125)




```
%N1G0G90X-3.Y3.T1M6
N5X0Y0Z.05
N10G1Z-.1F50
N15G79J.125F100
N20G0X-3.Y3.M2
E
```

It is important to note that G79 cannot be used with Cutter Diameter Compensation; however, the same cutter path may be programmed using G75 Multi-Quadrant Circular when cutter compensation is required. Program #2 below illustrates this method using the same cutter path shown in the sketch above.

Program #2

```
%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N15G1Z-.1F50
N20X.0625Y.0625F100
N25G3X0Y.125I0J.0625
N30G3X0Y.125I0J0
N35G3X-.0625Y.0625I0J.0625
N40G1X0Y0
N45G0X-3.Y3.M2
```

CANNED CYCLES Z-AXIS (G80-89)

Fixed cycles reduce programming time and tape length for certain operations. The basis for fixed cycle operation is that certain repetitive information is stored and retained in the control for use whenever it is recalled. The X-Y commands can be incremental (G91) or absolute (G90). The Z dimension within a fixed cycle will be entered into the system as an incremental unsigned value. The information that is stored is as follows:

a. The depth of Z motion to be under "fixed" cycle control. This is an incremental unsigned Z value. Once entered, the Z increment need not be re-entered unless it changes.

b. The feedrate desired for the depth motion.

c. (Deep Hole cycle G83 or Chip Break Cycle G87). The Z axis peck increments. These are entered as secondary Z axis commands. This is the depth the Z axis is to feed down for each peck cycle.

NOTE: The sum of the Z axis peck increments need not equal the total Z axis depth to be drilled.

After the fixed cycle (G81-G89) is programmed, the system must have an X and/or Y coordinate word. In Absolute Coordinates the word can be a repeat of the existing position. In Incremental,the word can be: X and/or Y Zero. The cycle functions as follows:

a. If not in position, the machine tool will first rapid traverse automatically to the X-Y coordinate as designated in the part program.

b. The Z axis cycle will occur.

The Z axis cycle will occur in every block starting with the data block which contains an X and/or Y word. This may be the block containing the Z axis cycle code: e.g.

N10G81X1.Z.31F80

or it may be the subsequent block: e.g.

.N10G81Z.31F80 (cycle will only be set in this block)
N15 X1. (Cycle will occur in this block).

This second method is strongly recommended to allow search execution. (See 5.1.1 Definition Block)
The fixed cycle is cancelled by either a G80 or a G00 code. The Z axis cycle will not occur after the move commanded by the data block which terminates the fixed cycle.

NOTE: If the Z depth changes, the fixed cycle code G81-G89 must be repeated with new Z depth.

NOTE: A rapid traverse Z move is not permitted within the fixed cycle. If required, the cycle must be terminated, the rapid traverse move inserted, then the cycle must be reinstated with fixed cycle code (G81-G89) and Z unsigned incremental value.

G80 - Fixed cycle cancel.

This function cancels all other fixed cycle G codes (G81-G89).

G81 - Drilling cycle. (Figure 5-2)

This function, when programmed with Z, F information, provides for a feed in - rapid out sequence suitable for drilling a series of holes that have the same diameter and depth. G81 remains in effect until a G00, G80 or another fixed cycle G code is programmed.

NOTE: The Z value is the incremental unsigned distance.

```
 %N1G0G90X0Y0T1M6
1 N5X1.Y.5Z.05
  .N10G81Z1.1F80
  N15X1.
2 N20X.5Y1.
3 N25G0X2.Z-.45
  .N30G81Z.6F80
  N35Y1.
4 N40X2.5Y.5
  N45G0X0Y0M2
  E
```

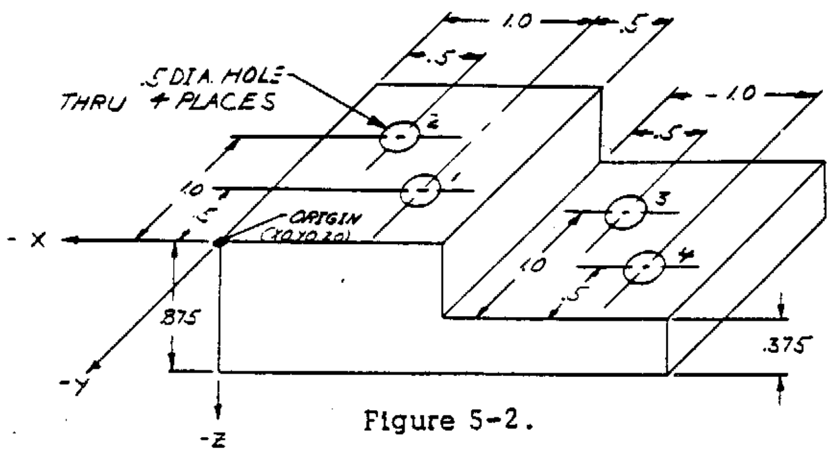


Figure 5-2.

N1 Block Set up the absolute (G90) and rapid traverse positioning (G0) mode. Move to the Origin X0, Y0; quill Home (M6). (operator put tool #1 into the spindle)

N5 Block Move over hole #1 and bring the tip of the tool to .05 clearance plane above the top of the work piece (Z0)

N10 Block In a definition block, set up a drilling cycle (G81) that when given a X or Y word will drill a hole 1.1" deep from the .05 clearance plane at a feed of 8. IPM

N15 Block Drill hole #1 using previous set information. (.N10 etc)

N20 Block Rapid traverse in X and Y to hole #2 then drill.

N25 Block Cancel the G81 cycle and rapid traverse in X and Z to .05 over hole #3 (X2.Z-.4 5)

N30 Block In a definition block set up a drilling cycle (G81) that when given an X or Y word will drill a hole .6" deep from the .05 clearance plane at a feed of 8. IPM

N35 Block Drill hole #3 using previous set information. (.N30 etc)

N40 Block Rapid traverse in X and Y to hole #4 then drill

N45 Block Cancel cycle (G0) drive quill home (M2) and move to (X0, Y0). End of Program Rewind Memory (M2).

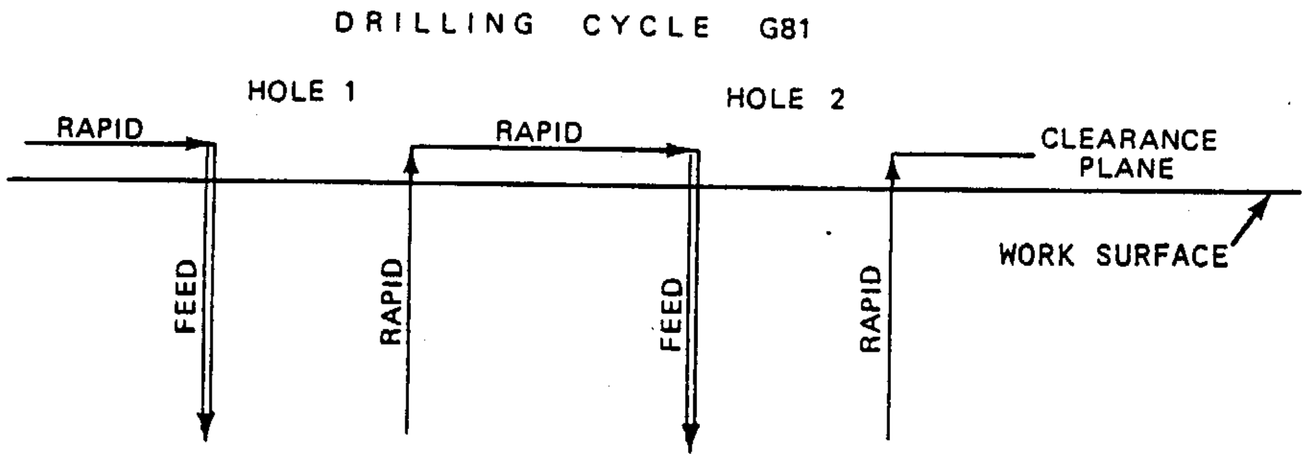


Figure 5-2. Drilling Cycle

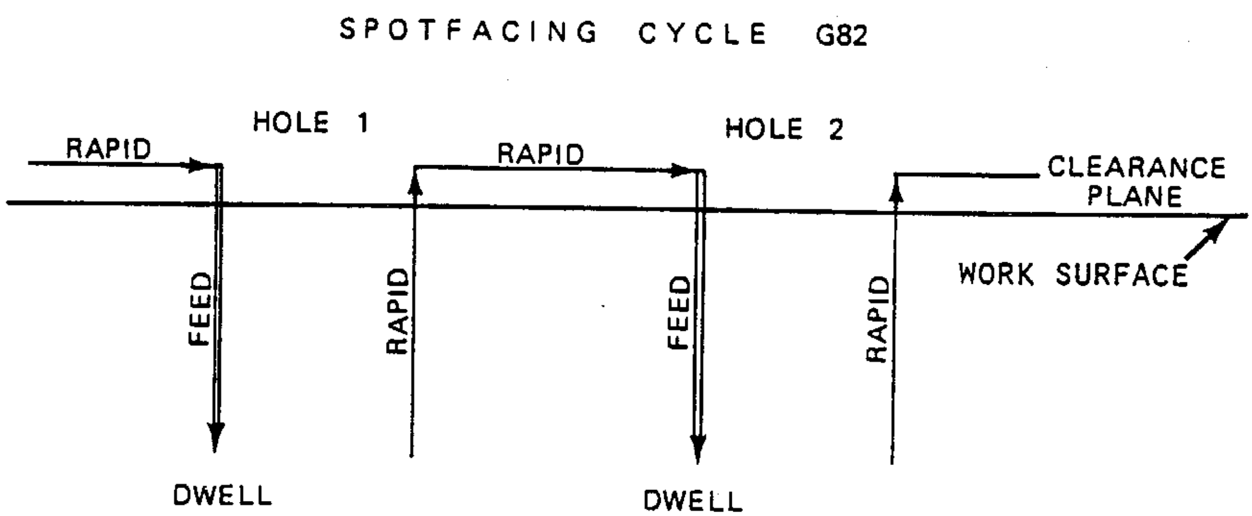


Figure 5-3. Spotfacing Cycle

G82 - Spotfacing cycle. (Figure 5-3)

This function, when programmed with Z, F information, provides for a feed-in-dwell-rapid-out sequence suitable for spotfacing, counterboring operations, etc. G82 remains in effect until a G00, G80 or another fixed cycle G code is programmed.

NOTE: Dwell time must be set by a previously programmed G04/n command. Z value is the incremental unsigned distance.

Example of G82 - Spotfacing cycle.

This sequence is the same as G81 except for delay which is forced when the Z axis reaches the programmed fixed cycle depth. The length of delay is set by the DWELL command G04/n previously programmed. (See Dwell page 5-4)

G83 - Deep hole drilling cycle. (Figure 5-4)

This function, when programmed with Z1, Z2, Z3, F information, provides a feed-in, rapid out, rapid in to bottom of hole, feed-in, rapid out, etc. sequence until the specified hole depth is reached. G83 remains in effect until a G00, G80 or another fixed cycle G code is programmed.

Z1 is the total Z depth. Z2 is the incremental distance for the first peck increment. Z3 is the peck distance for all subsequent increments.

NOTE: If Z3 is not input, Z2 becomes the peck distance for all increments. Z2 MUST be smaller than Z1.

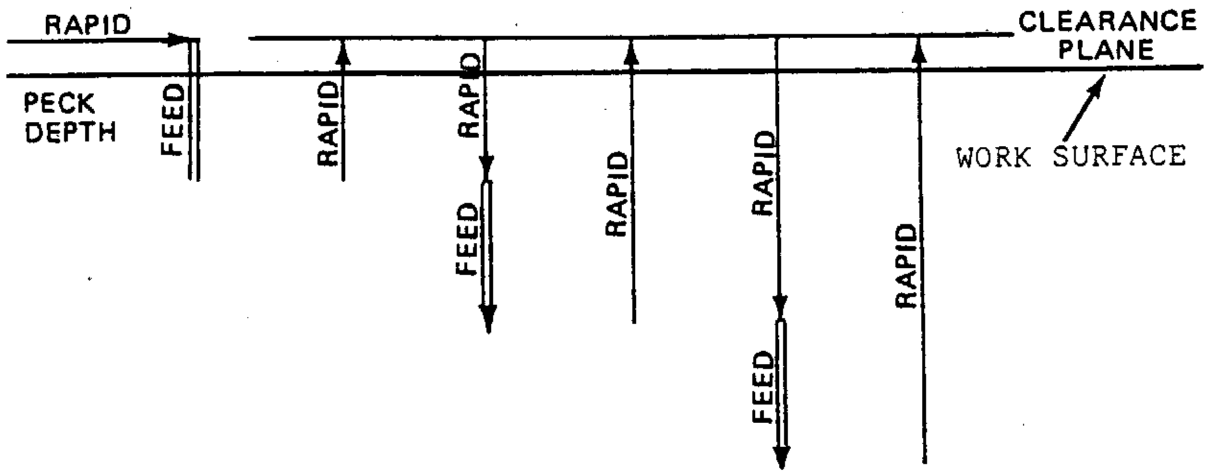
Example of G83 - Deep Hole Drilling Cycle

This sequence is programmed similar to G81 except that subsequent Z commands are used to specify the peck increment. For example, a hole to be drilled 3.0" deep with the first peck increment of .75" and subsequent peck increments of .4":

N4G83X8.0Y3.0Z3.0Z.75Z.4F80

After the X and Y axes have completed the move in rapid traverse, the Z axis will feed in .75", then rapid traverse out .75", rapid traverse in .75". The Z axis will feed in an additional .4", then rapid traverse out 1.150", rapid

DEEP HOLE CYCLE G83



DEEP HOLE CYCLE G87

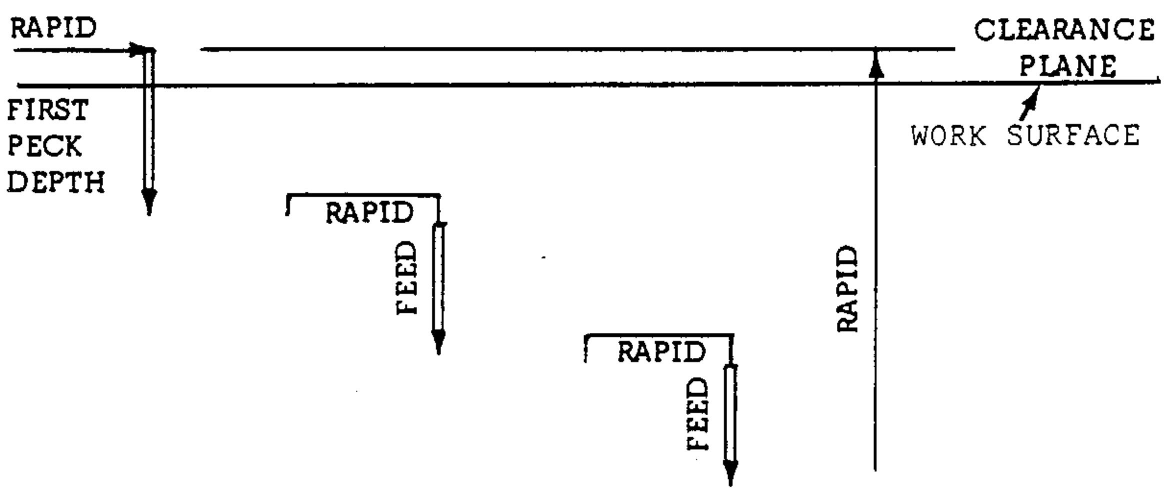


Figure 5-4. Deep Hole Drilling Cycles

TAPPING CYCLE G84

Must be used with tapping attachment for non-reversing spindles

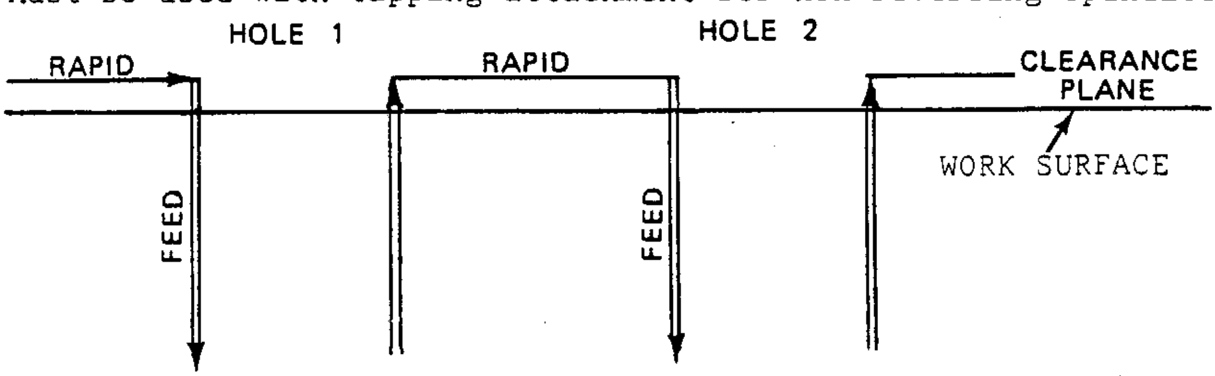


Figure 5-5. Tapping Cycle

traverse in 1.1500" and feed another .4". This cycle will be repeated until the total Z depth (3.0") is reached.

NOTE: The sum of the Z axis peck increments need not equal the total Z axis depth to be drilled. The last feed move in the cycle will be the Z peck increment or the depth remaining to be drilled, whichever is smaller. Further, the Z values are the incremental distance, the first being total depth and the second the first peck increment, all are unsigned including the third and subsequent increments.

G84 - <u>Tapping cycle.</u> (Figure 5-5)

This function, when programmed with Z, F information, provides a feed-in-feed-out sequence suitable for tapping operation with a tapping attachment for non-reversing spindles. G84 remains in effect until a G00, G80 or another fixed cycle G code is programmed.

NOTE: Z value is the incremental unsigned distance.

<u>Example of G84 - Tapping cycle.</u>

The tapping sequence follows the same programming format as G81. After the rapid X and Y axis move, the Z axis will feed in to the programmed depth and the Z axis will feed back out of the hole.

In the following example, a 1/4-20 tap is to be programmed. A feedrate of 10.0 ipm is chosen. The programmer must perform the following calculation to determine the tapping spindle speed.

Spindle rpm = ipm X threads/inch
= 10 X 20
= 200 rpm

Program recommendation is F99 with the spindle speed set to 200 RPM to force a condition where the tap extends from its holder. A hand held stroboscope is recommended to manually adjust for the exact speed.

A chart of feed and speed values for tapping various pitches is given in Figure 5-6. Select Feed & Speed values from this chart to program the desired Thread Pitch. Use of designated speed will minimize amount of tap holder compensation required.

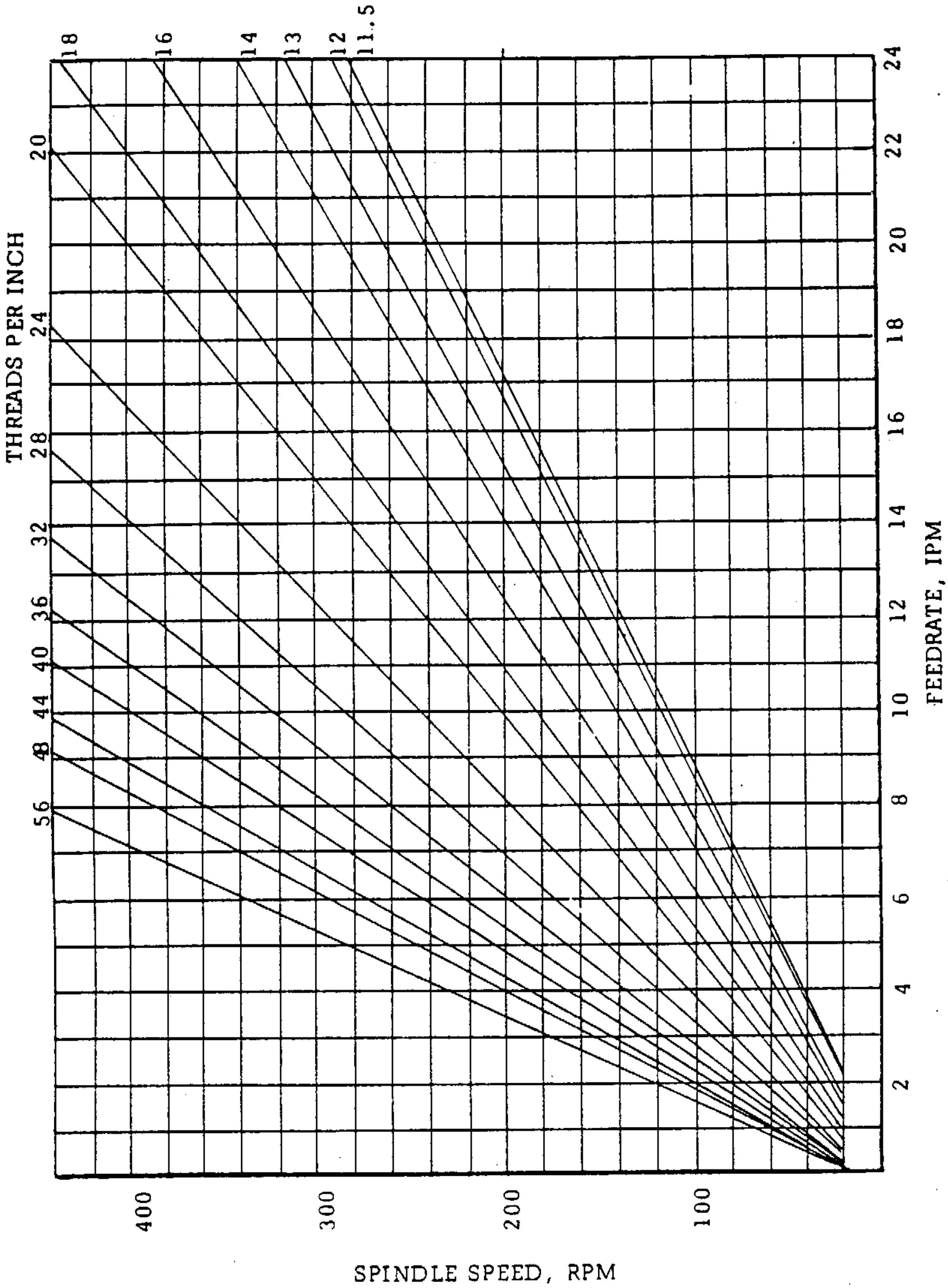


Figure 5-6. Feed and Speed Chart for Tapping

The external device required for performing tapping operations is a special tapping tool holder with built-in axial float allowance and a clutch device to reverse the tap when the feed stops at the bottom of the hole. The recommended axial float in this holder is 3/8" tension 3/8" compression, which compensates for any spindle speed - Z axis feedrate deviation from the actual tap lead.

G85 - <u>Boring Cycle.</u> (Figure 5-7)

This function, when programmed with Z, F information, provides a feed-in feed-out sequence suitable for boring or reaming operations. G85 remains in effect until a G00, G80 or another fixed cycle G code is programmed.

NOTE: Z value is the incremental unsigned distance.

<u>Example of G85 - Boring Cycle.</u>

This sequence is programmed using the same format as a G81 cycle. After the rapid X-Y axis move, the Z axis is fed at the programmed feedrate to the programmed depth and then retracted at the feedrate.

G86 - <u>Boring Cycle.</u> (Figure 5-8)

This function, when programmed with Z, F information, provides a feed in-feed stop-wait for operator command-rapid out-operator restart sequence suitable for boring operations. G86 remains in effect until G00, G80 or another fixed cycle G code is programmed.

NOTE: Z value is the incremental unsigned distance.

<u>Example of G86 - Boring Cycle.</u>

This sequence uses the same programming format as the G81. After the rapid X-Y axis move, the Z axis feeds to the Z depth at the programmed feedrate. The feed is then commanded to stop. The spindle is stopped by the operator and oriented if necessary. After pressing CONTINUE the Z axis is rapid traversed up. The spindle direction will be reinitiated by the operator. This cycle is input (typically) as follows:

```
N645 G0X0Y0Z.05
.N650G86Z.6F21
N655 X0
```

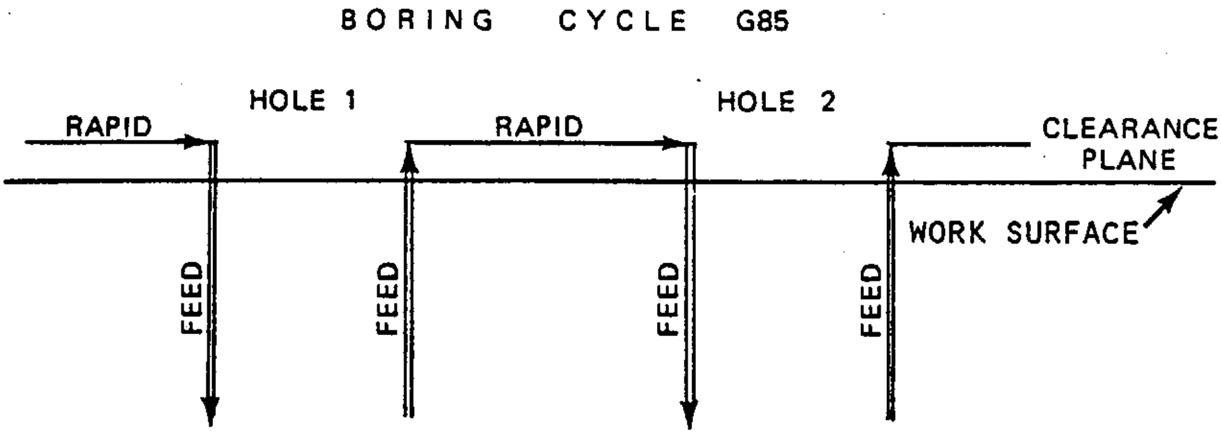


Figure 5-7. Boring Cycle

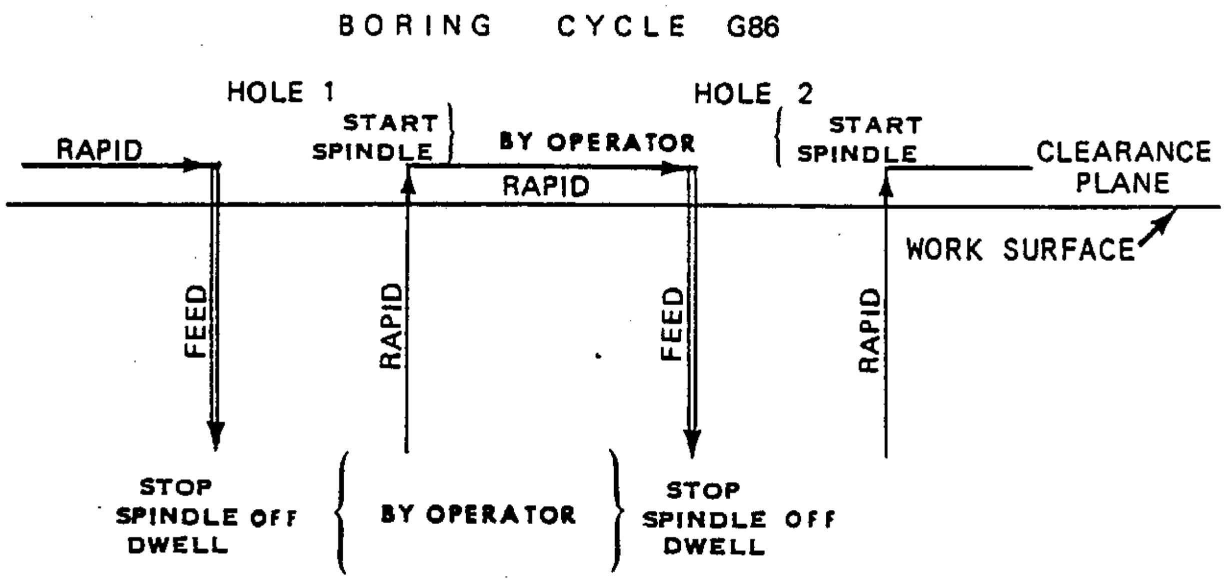


Figure 5-8. Boring Cycle

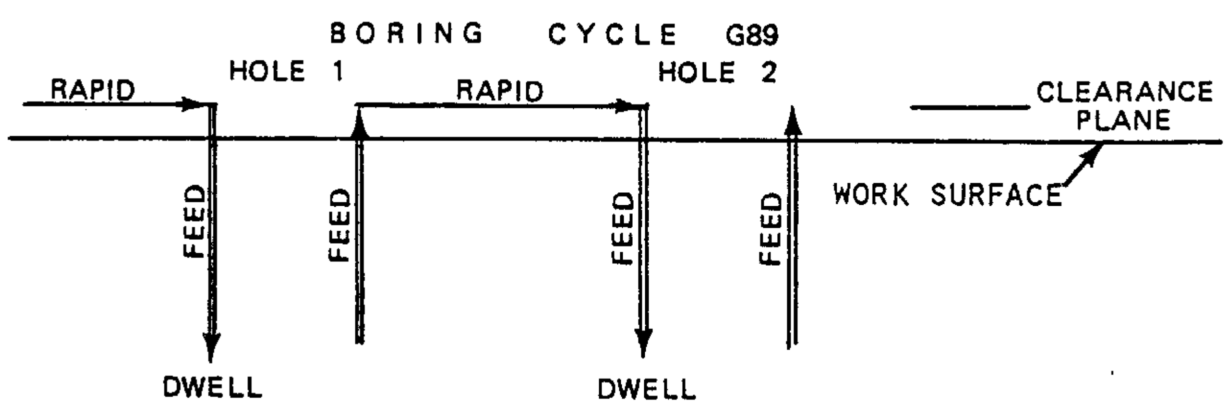


Figure 5-9. Boring Cycle

The same cycle with an important variation can be programmed thus:

```
N645G0X0Y0Z.05
N650G1Z-.6F21
N651M0
```

Operators stops spindle* and orients the cutting edge toward the +X direction. Operator depresses CONTINUE.

```
N652G0G91X-.01
N653G90Z.05
N654X0Y0
```

Operator starts the spindle.

*Spindle stops automatically if machine is equipped with the Auxiliary Control Group

The feature of this method is the ability to position the boring tool a suitable distance (in this example 0.010") from the finished bore before withdrawing the tool from the hole. Note that the orientation of the cutting edge relative to the part is controlled by the operator.

G87 - Chip Breaking Cycle. (Figure 5-4)

This sequence is programmed in exactly the same manner as the G83 - Deep Hole Drilling Cycle. G87 performs the same function but instead of rapid traverse out of the hole and back after a feed move, the quill will rapid traverse up and down by 0.050". The purpose is to break the chip rather than withdraw the cutting tool entirely from the work with each peck in the manner of the G83 cycle.

G89 - Boring Cycle. (Figure 5-9)

This function, when programmed with Z, F information, provides for a feed in-dwell-feed out sequence whenever the programmer requires such a sequence. The G89 remains in effect until a G00, G80 or another fixed cycle G code is programmed. Dwell time must be set by a previously programmed G04/m command. (See page 5-4)

NOTE: Z value is the incremental unsigned distance.

Example of G89 - Boring Cycle.

This fixed cycle is programmed in the same format as G81 and has the same feed in - feed out characteristics as G85, except that a dwell will occur between the feed in and feed out motions of Z axis. The dwell time must be previously set by a programmed G04/n command.

MULTIHOLE ROW FIXED CYCLE

This function executes the same output as if a row of holes was programmed using looping techniques (see Section VI). It applies to any one of the G81-89 Fixed Z axis cycles. Transformation of this cycle (G73) is not permitted.
NOTE: Do NOT use this feature in Metric Mode.

Example of Multihole Row Drilling

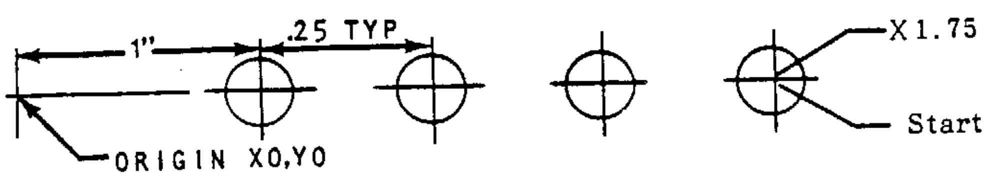
The format is (G81) Xn1Xn2 (Zn3FV)
or (G81) Yn1Yn2 (Zn3FV)

where n1=the total distance to be traveled along the X or Y axis (G91) or the absolute coordinate of the last hole (G90).
n2=the incremental distance between holes (must be in multiples of 0.0005")
n3=the Z axis incremental depth
FV=the Z feed rate
The bracketed data is optional depending upon whether a cycle has already been initiated or not.

A hole will be drilled in the start position, i.e. the existing position of the tool when this block was called. Successive holes will be drilled at the specified incremental distance, n2, until the last hole is drilled after the last move. The total distance n1 may be input after G90 or G91 codes denoting that n1 is the absolute coordinate of the last hole if in G90; or is the incremental total distance from first to last holes if in G91.

NOTE: Any G81-89 fixed cycle may be used.

For Example: to peck drill four holes with the breakchip cycle starting at an absolute coordinate of X1.75:



Program either (A) or (B) below:

(A) Abs: G90G87X1.X.25Z1.0Z.25Z.15F80
(B) Incr: G91G87X-.75X.25Z1.0Z.25Z.15F80

Note, If the block were programmed with "X" increment changed to .3:

Abs: G90G87X1.0X.3Z1.0Z.25Z.15F80
Incr: G91G87X-.75X.3Z1.0Z.25Z.15F80

the spacing between the first three holes would be .3" as programmed but the distance between the third hole and the last would be .15" in order to make the total span .75" as programmed.

SUMMARIZING THE DISCUSSION ON FIXED CYCLES:

Code	Movement In to Z Depth	Dwell	Spindle (1)	Movement Out
G81	Feed	No		Rapid
G82	Feed	Previous G04/n		Rapid
G83	Feed (Peck)	No		Rapid
G84	Feed	No		Feed
G85	Feed	No		Feed
G86	Feed	Wait for Operator	Stop (1)	Rapid
G87	Feed (Break Chip)	No		Rapid
G89	Feed	Previous G04/n		Feed

(1) The spindle ON/OFF is under the control of the operator.

NOTE: The system will be automatically set in rapid traverse for XY positioning by a G81-G89 fixed cycle code. A G80 or G00 will cancel any of the above cycles.

G90 - Absolute Input.

This preparatory function causes the control to accept all X, Y, Z entries in the contouring or positioning mode as an absolute coordinate in reference to a chosen absolute X, Y zero. The control will automatically compare the present absolute coordinate position with the new programmed absolute coordinate position and apply the difference as a motion command to the appropriate axis. The absolute input system will be in effect until a G91, incremental distance input command, is programmed. Initializing the control thru POWER ON or RESET places the control in Absolute Dimension Input (G90).

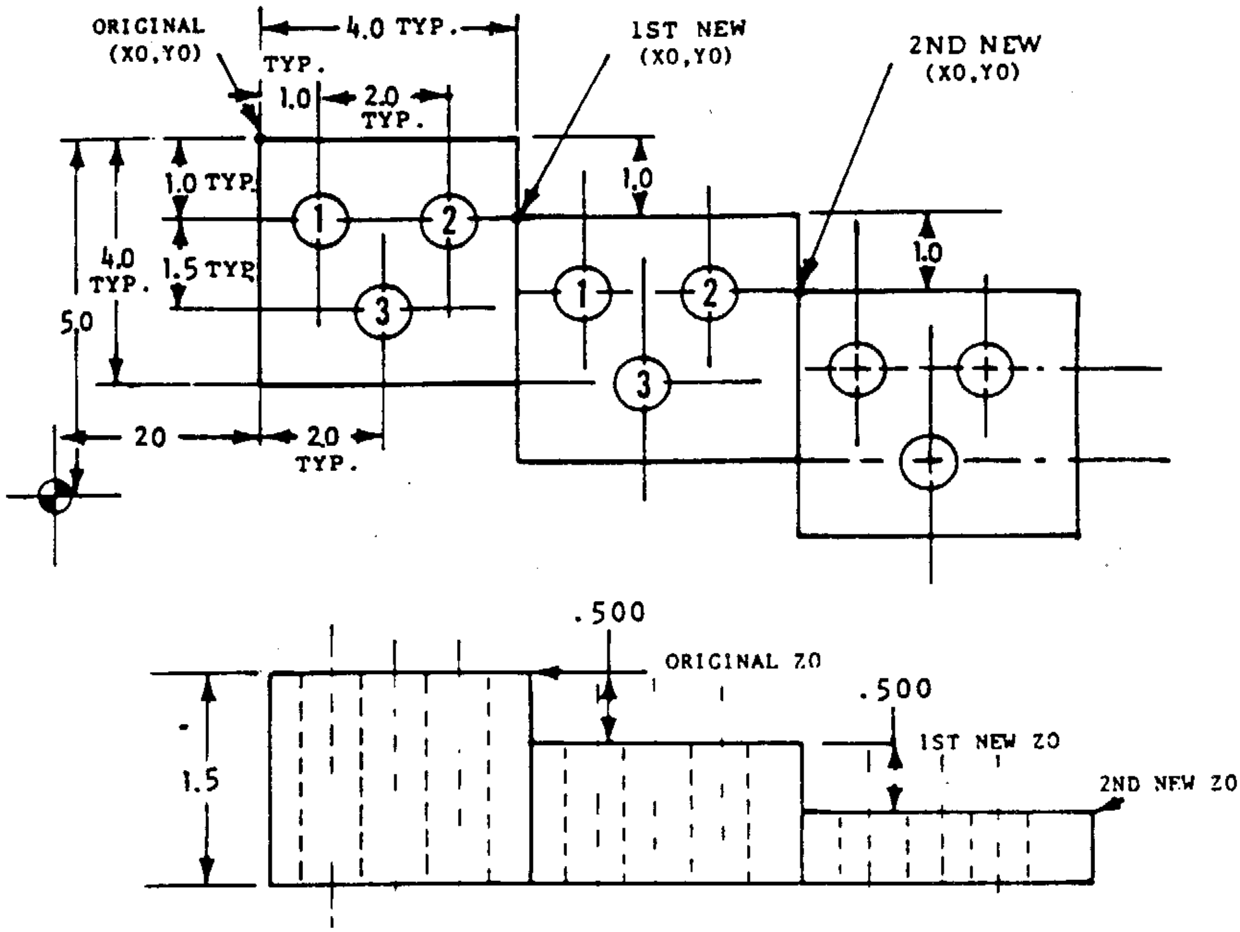
NOTE: The first Z value after a Z home command must be in G90 absolute coordinates.

G91 - Incremental distance input.

This preparatory function causes the control to accept all X, Y, Z entries in positioning or contouring mode as incremental distance commands. Thus, all linear axis motion entries will be interpreted as commands to move the same distance as that programmed, from the present position, and not to an absolute position. The incremental input system will remain in effect until changed by entering a G90, absolute input preparatory command, or initializing the control (See G90 above)

G92 - Preset absolute registers.

This preparatory function provides the ability to preset XYZ and A absolute position registers to any desired dimension. The machine slide will not move in




```
ST      %N1G0G90X-2.Y-5.T1M6
1        N5X1.Y-1.Z.05
         N10G81X1.Z.55F80
2        N15X3.
3        N20X2.Y-2.5
TNO      N25G0G92X-2.Y-1.5Z.55    --   TRANSLATING TO 1ST NEW ORIGIN
1        N30X1.Y-1.Z.05
         N35G81X1.Z.55F80
2        N40X3.
3        N45X2.Y-2.5
TNO      N50G0G92X-2.Y-1.5Z.55    --   TRANSLATING TO 2ND NEW ORIGIN
1        N55X1.Y-1.Z.05
         N60G81X1.Z.55F80
2        N65X3.
3        N70X2.Y-2.5
TOO      N75G0G92X10.Y-4.5Z-.95   --   TRANSLATING TO ORIGINAL ORIGIN
```

To find the value for the translation in X,Y and Z, assume you are at the location of the new origin looking back to where your tool is now. Whatever the values for X,Y and Z are at that time are your translation values for the G92 block.

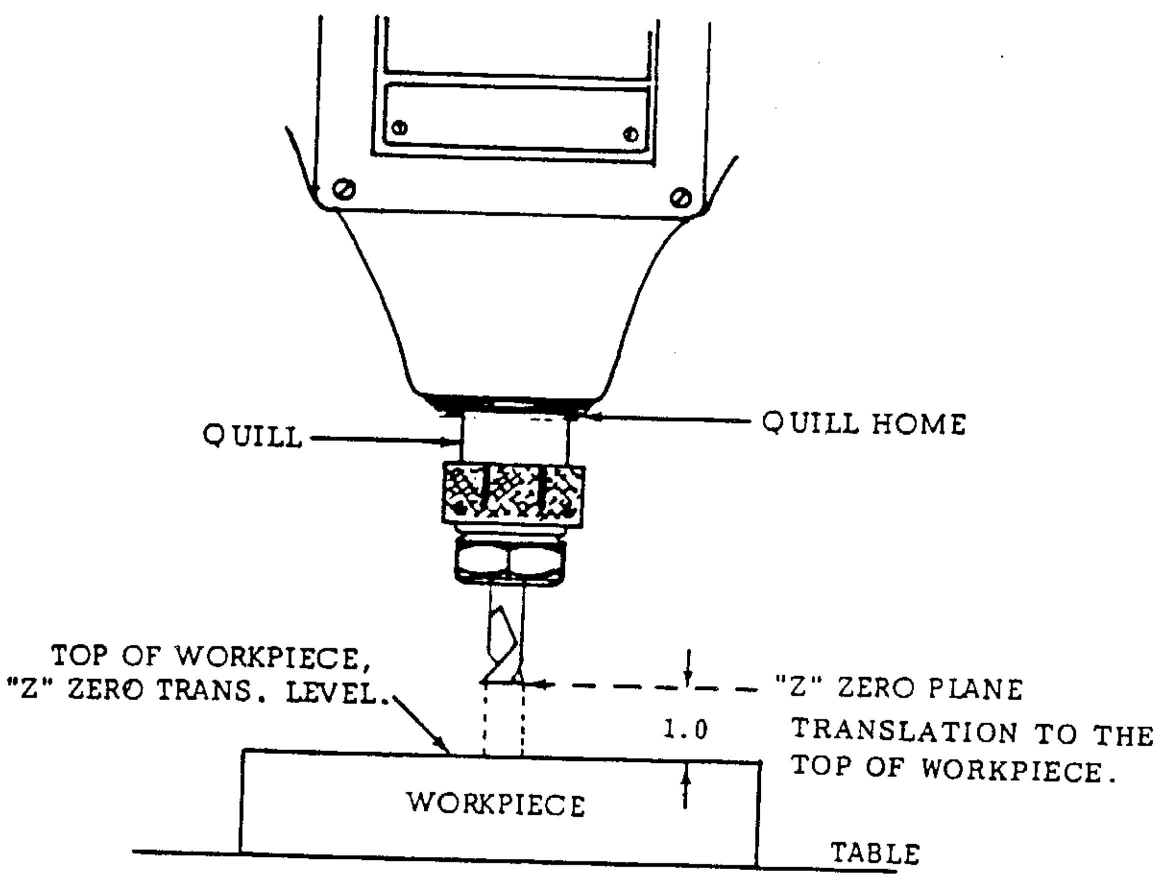
Figure 5-10. Translation of Part Coordinates response to this block which may be entered as part of the program or by MDI. This function allows programming from a starting point on a part which is defined by absolute coordinate values specified in the G92 block. This preparatory function establishes a convenient part programming reference coordinate at any point on the travel.

The G92 command also enables the translation of an original coordinate system to a new part coordinate system (see Figure 5-10). This is convenient if multiple parts on a fixture are made.

NOTE: Since the G92 preset command destroys the previous contents of the absolute register or registers, a new G92 preset command will be necessary to reestablish the original coordinate system. The system must be in G90 (Absolute) when calling G92.

Example: Preset the absolute registers to X = 2.5, Y = -.5, Z= 1.

G92X2.5Y-.5Z1.



Trans: G92 Z1.0

NOTE: After translating the Zero in Z, if a TnM6 is programmed to send the quill home, beyond the original Zero Plane, the previously programmed translation in Z will be cancelled and set back to the original Z Zero. If the translation is required again, it must be reprogrammed.

G99 - Deceleration override.

If the feedrate is greater than 2.8 ipm, the table accelerates from and decelerates to the programmed value during each block of information. Deceleration in a particular block of information can be reduced by programming a G99 code in the block. Some deceleration will still occur if the feed is over 8 ipm.

NOTE: Do not use G99 when in rapid traverse (G00). Do not use G99 when in feed if the next block is rapid traverse (G00).

Tangency Conditions

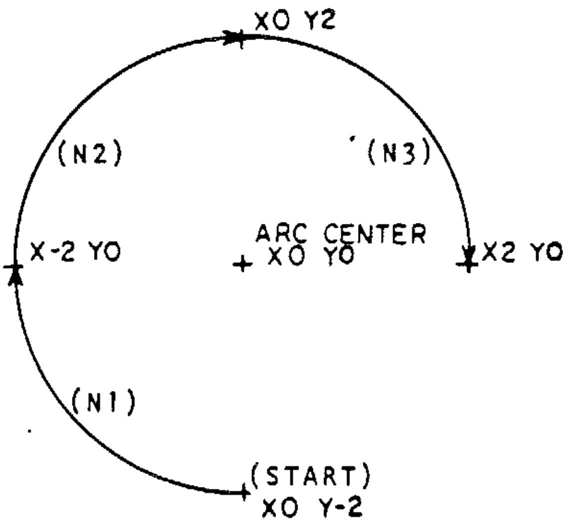
This technique is useful if subsequent motion is tangential as in a circle crossing quadrants.

Example #1

Move along a 2.0" radius circle clockwise from 270 to 0 degrees at 32 ipm. (The start point of the circle is X=0, Y= -2.0 with center at 0, 0.)

N1G91G2G99X-2.0Y2.0J2.0F320
N2G99X2.Y2.I2.
N3X2.Y2.J2.

The table will cross over from arc 1 to arc 2 (block number N1) and arc 2 to arc 3 (block N2) without decelerating. At the end of arc 3 (block N3) deceleration will occur.



Non Tangent Conditions

To determine if G99 can be used for non tangent conditions (approximately 5^{o}) proceed as follows:

1. The major axis in a block of information is the axis with the larger move, the other axes are in the minor axes. To use G99, the major axis in the next move segment must be the same axis and in the same direction as the major axis in the move segment being tested.

2. The minor axis velocity (MAV) can be calculated approximately by:

MAV = PFR* (MINOR AXIS MOVE/MAJOR AXIS MOVE)

where PFR = programmed feedrate
* = multiplied by

The change in minor axis velocities between the successive moves should not exceed 2.8 ipm (X, Y or Z).

For example if XD1 = .5, YD1 = .1, PFR1 = 25 ipm, XD2 = .5, YD2 = 0, and PFR2 = 25 ipm, the minor axis velocity for the first move is:

MAV 1 = 25* (.1/.5) = 5 ipm

The minor axis velocity for the second move is:

MAV 2 = 25* (0/.5) = 0 ipm

The change in velocity is 5 ipm, therefore, the first block of data output must not contain a G99, thus:

G91X.5Y.1

But, if XD1= 1.0, the MAV1 = 2.5 ipm with a change in velocity of 2.5 ipm. The first block of data could be:

G91G99X1.0Y.1

Minimum Radius

The last move in a sequence of linear moves in which deceleration override is used must exceed the minimum value given in the chart following, dependent on the feedrate used. The last move, which contains no G99 code, must be sufficient length to enable the control to decelerate to a smooth stop. The values following also represent the minimum radii which can be safely used with deceleration override for the various programmed feedrates:

Feedrate, ipm	Minimum distance (or radius)
0 - 6	.001
6 - 18	.025
18 - 32	.100
18 - 51 (BOSS 6.0)	.100

Miscellaneous Functions

The following list of Miscellaneous Functions demand either a slide hold or an external feedback switch closure. They are of the "Before Function" type denoting that the slide(s) must decelerate to a stop before the function takes place. A G99 must not be programmed in the block preceding this function.

M 00	Program Stop
M 01	Optional Stop
M 02	End of Program, Rewind
M 06	Tool Change
M 25	Z Axis Home

5.4 LINEAR AXIS COMMAND (X, Y, Z)

5.4.1 General

Programming axis motion can take the form of:

Rapid traverse, positioning mode,
Linear interpolation, feed mode,
Circular interpolation, feed mode,

Each of these modes can be programmed in either:

a. Absolute coordinates

b. Incremental coordinates

A decimal point may be included in motion data for programming convenience, enabling non-significant zeros both to the left and to the right of the decimal to be dropped.

e.g. The system will recognize:

X10000
X1. } - as the same input of one inch

X.0002
X2 } - as the same input of two tenthousandths of an inch

NOTE: Axis programming is limited to the fourth decimal place. (Tenths)
e.g. 9.4995

5.4.2 Absolute Input

When the "absolute input" system G90 is used, each linear axis command is composed of the letter address X, Y or Z and up to a seven digit number with 000.0001" (0000.001 mm) resolution. Leading zeros may be omitted, but trailing zeros are required unless a decimal point is included. After an axis has moved to a position, it remains there until a different position is commanded. A movement for each axis may be entered in the same block. The programmed position should not exceed the axis travel limit. Linear axis coordinate information (X, Y or Z) indicates the motion to be commanded and the distance the motion is to be moved with respect to a zero reference point. The zero reference point can be set so that axis motion may occur in either the plus or minus direction within the allowable machine travel limits.

For example, an absolute coordinate word for the X axis would be X-5. representing the absolute coordinate of -005.0000 inches.

Full four quadrant plus and minus Absolute Programming is standard.

5.4.3 Incremental Input

When the "incremental" distance input system (G91) is used, each linear axis command becomes an incremental or departure distance from the present position and not an absolute coordinate. Each axis command consists of a letter address, X, Y or Z, and algebraic sign to indicate direction, and up to a six digit number with 00.0001" (000.001 mm) resolution. Leading zeros may be omitted, but trailing zeros are required unless a decimal point is used.

NOTE: Six digit limitation is caused by maximum machine travel.

If the direction of the motion is minus, the incremental distance entry must be preceded by a minus sign. If the direction of motion is plus, no sign need be entered. Absence of a sign is interpreted to be a plus by the control. The maximum increment that can be programmed in one block is restricted by the travel limits of the machine.

5.4.4 Programming Rapid Traverse, Positioning Mode

The following problems will serve as examples to show how to program axis motion, where (a) describes the method to be used with absolute coordinates and (b) that to be used with incremental coordinates.

Example #2

	X	Y	Z
Initial Abs. Coord.	4.0	-2.0	6.0
Move 1	4.0	-2.0	4.5
Move 2	7.0	3.0	2.0

In rapid traverse move 2 is split into two rapid traverse moves: Since the Z motion direction is minus, the XY rapid occurs first then the Z by itself.

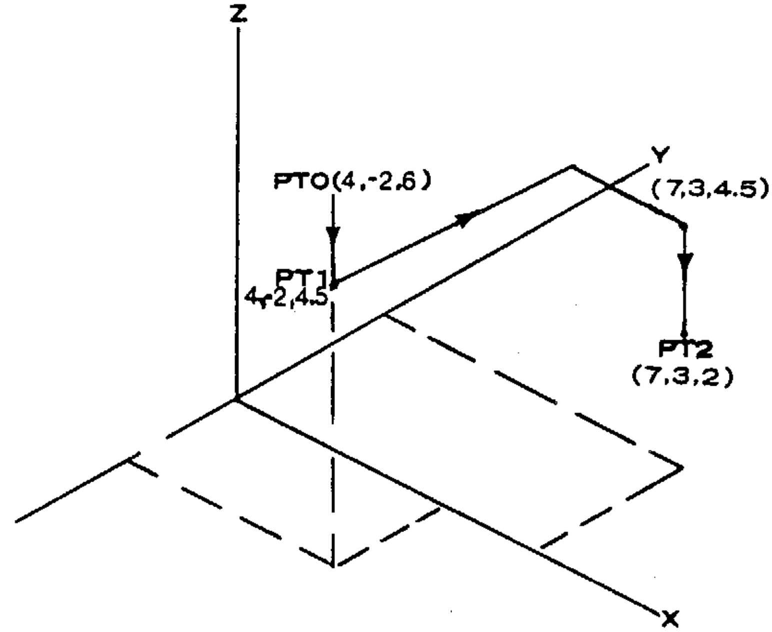
(a) G90G0Z45000
X70000Y30000Z20000

(b) G91G0Z-15000
X30000Y50000Z-25000

Format can also be:

(a) G90G0Z4.5
X7.0Y3.0Z2.0

(b) G91G0Z-1.5
X3.0Y5.0Z-2.5



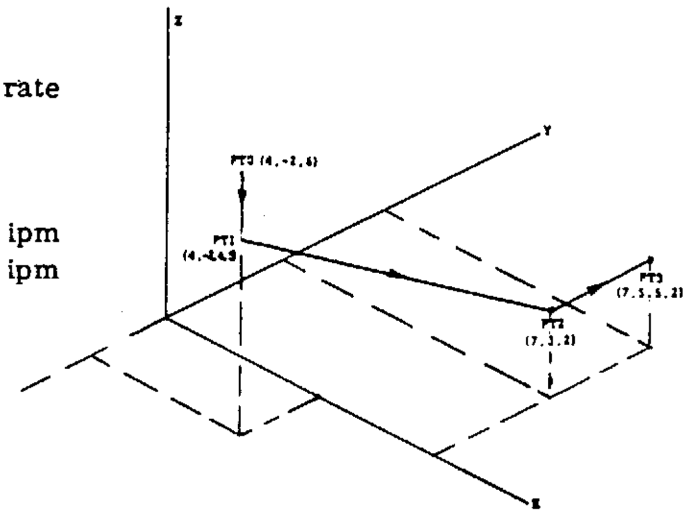
5.4.5 Programming Linear Interpolation, Feed Mode

Example #3

	X	Y	Z	feedrate
Initial Abs. Coord.	4.0	-2.0	6.0	
Move 1	4.0	-2.0	4.5	
Move 2	7.0	3.0	2.0	7.5 ipm
Move 3	7.0	5.5	2.0	12 ipm

(a) G90G0Z4.5
G1X7.0Y3.0Z2.0F75
Y5.5F120

(b) G91G0Z - 1.5
G1X3.0Y5.0Z - 2.5F75
Y2.5F120



5.5 ARC CENTER OFFSET COMMANDS (I, J, K)

5.5.1 (G74) - GENERAL DEFINITIONS (SEE Fig. 5-11)

The arc center offset entries are addressed by the letter, I,J and K followed by a six digit incremental distance with the leading zero suppresion where:

I - is the distance from the arc center to the starting position of the arc in the "X" axis.

J - is the distance from the arc center to the starting position of the arc in the "Y" axis.

K - is the distance from the arc center to the starting position of the arc in the Z axis.

SIGNS (+ or -) must not be entered with arc center offset entries.

5.5.2 Programming Axis Motion Using Circular Interpolation, Feed Mode

Any arc with radius of 999.9999" (9999.999mm) or less which falls in one quadrant can be programmed with a single block of data. Arcs which lie in more than one quadrant require two or more blocks of data (see exception in paragraph 9-2 [G75] Multiquadrant Circular Interpolation). Circular interpolation is possible in any of three planes defined by the X-Y-Z coordinate system.

The block contents are as follows:

G02 (G03)

Defines whether the arc is to be generated in the clockwise or counterclockwise direction respectively. Direction conventions are as shown in Figure 5-11.

G17 (G18 or G19)

This entry defines the plane of operation. G17 defines the XY plane, G18 defines the ZX plane, and G19 defines the YZ plane (Reference Figure 5-11). A plane once selected is modal and remains in effect until a complimentary cancelling plane is selected. The G17 plane is the turn-on or reset plane and is the plane in effect when power is turned on. Additionally, a G00 or G01 code will set G17 (XY plane).

The appropriate G18 or G19 code must be preceded by the circular interpolation direction modifier (G02 or G03) in G74 circular only, and all X, Y data must be incremental (G91).

X-Y-Z Dimensions

Similar to paragraph 5.4.1 X-Y-Z dimension commands are defined below as being programmed with the absolute input (G90 example [a]) or with the incremental input (G91 example [b]) mode in effect. In the XY plane with absolute dimensions the data on tape must be the end point of the arc. Incremental data requires the signed (±) X, Y or Z axis distance from the beginning of the arc to the end of the arc. Minus (-) sign must be entered; plus (+) sign is understood, it need not be entered and gets stripped by the control.

I-J-K Dimensions (G74 mode only - the initial POWER ON condition - See paragraph 9-2)

Defines the incremental unsigned distance parallel to the X, Y or Z axis from the center of the arc to the beginning of the arc. (Reference paragraph 5.5.1 above.)

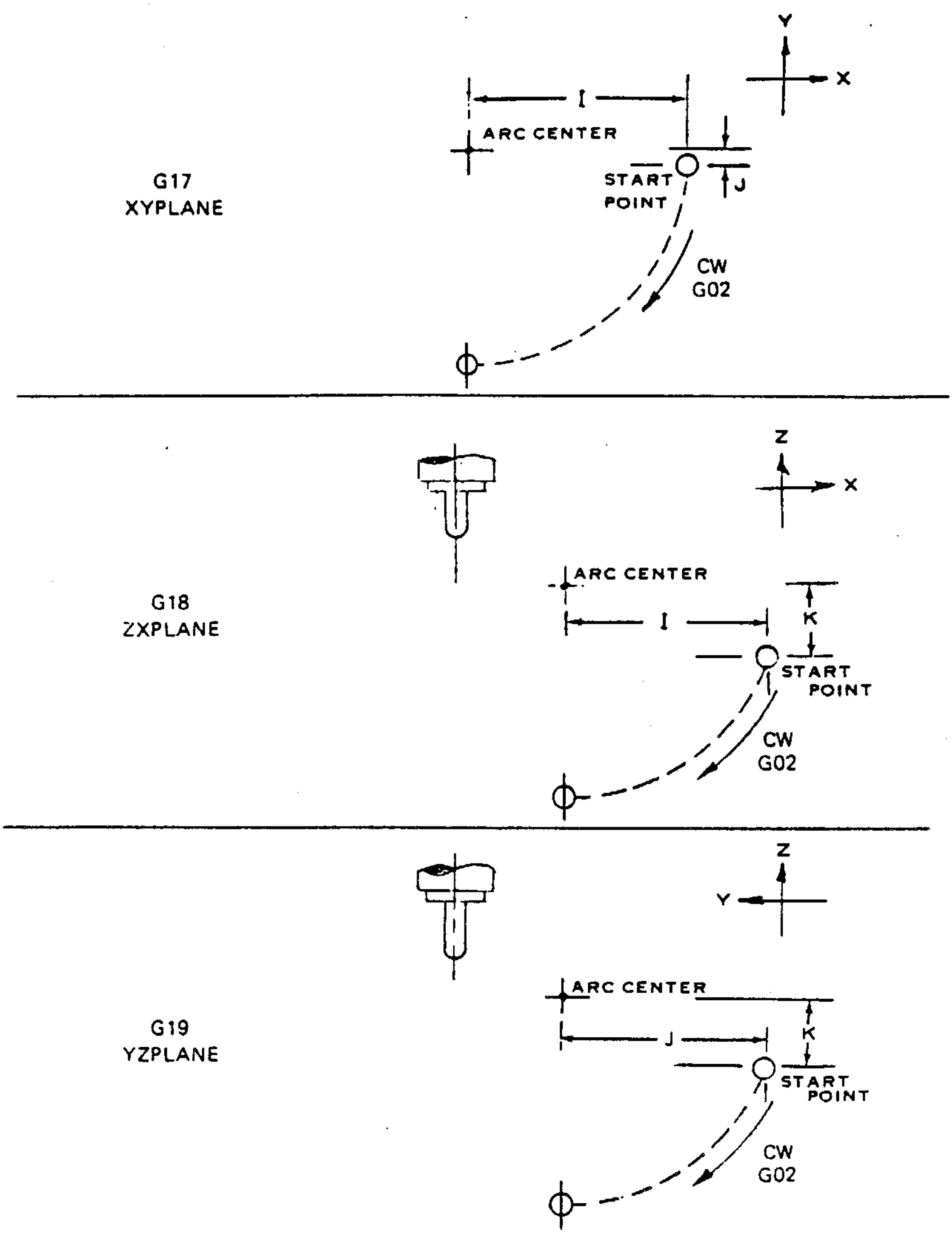
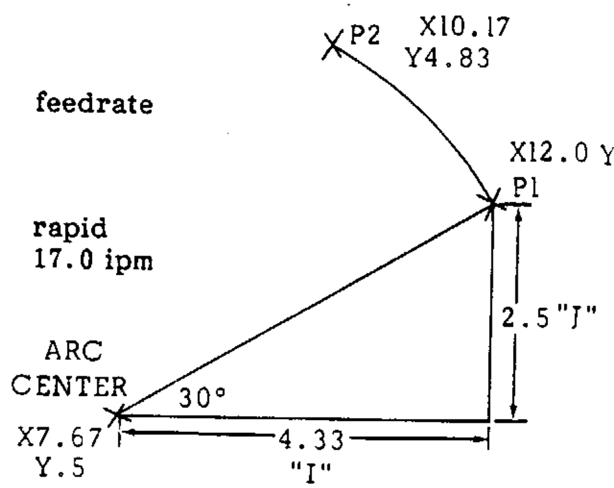


Figure 5-11. Arc Offsets

Example #4

Generate the arc shown: (P1 to P2)

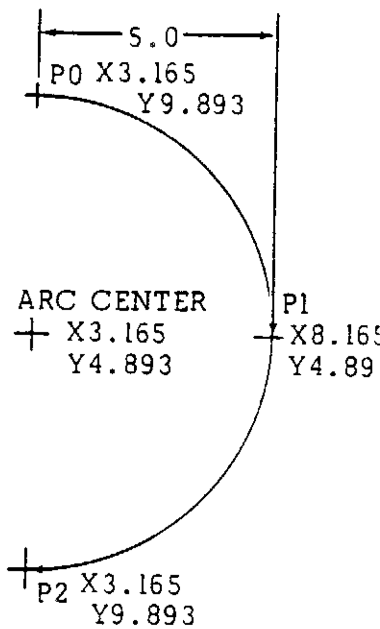


	X	Y	Z	feedrate
Initial Abs. Coord.	12.0000	.0	-2.0000	
PT. 1	12.0000	3.0000	-2.0000	rapid
PT. 2	10.1700	4.8300	-2.0000	17.0 ipm

(a) G90G0Y3.
    G3X10.17Y4.83I4.33J2.5F170
(b) G91G0Y3.
    G3X-1.83Y1.83I4.33J2.5F170

Example #5

Generate the arcs shown: (P0 to P1 to P2)



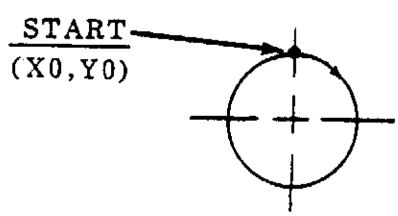
	X	Y	Z	feedrate
Initial Abs. Coord.	3.1650	9.8930	-2.0000	
PT. 1	8.1650	4.8930	-2.0000	7.5
PT. 2	3.1650	9.8930	-2.0000	7.5

(a) G90G2X8.165Y4.893I0J5.F75
    X3.165Y9.893I5.J0
(b) G91G2X5.Y-5.I0J5.F75
    X-5.Y-5.I5.J0

NOTE: The I, J or K values should be programmed when equal to Zero. G02 is modal and need not be programmed until the path mode is changed.

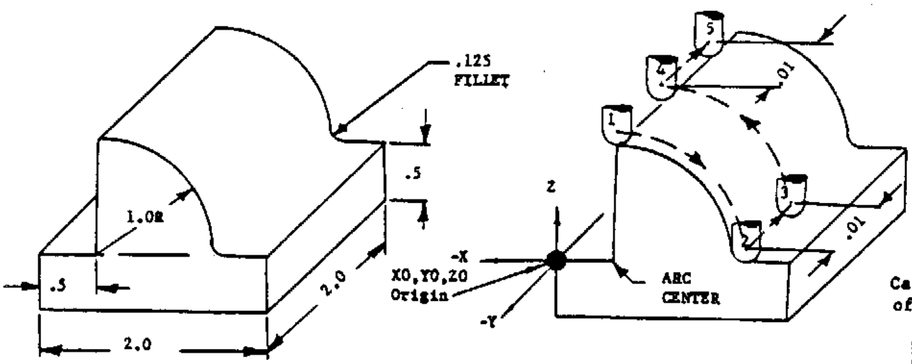
To program 4 quadrants of a circle with a 1.0" tool path radius beginning at 90 degrees and moving clockwise:

```
G1X0Y0
G2X1.Y-1.I0J1.F75
X0Y-2.I1.J0
X-1.Y-1.I0J1.
X0Y0I1.J0
```



For additional examples of contouring the XY plane, see Example #7, paragraph 5.11 and Section VII.

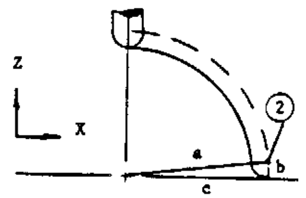
EXAMPLE #6   Program the part shown below using 1/4 Dia. Ball End Mill



ABSOLUTE COORDINATES

PT	X	Z
1	.5	1.125
2	1.6180	.125
3	1.6180	.125
4	.5	1.125
5	.5	1.125

Calculation Method to obtain "X" Coordinate of Points 2 and 3.



a= 1.125 b= .125

$c = \sqrt{a^2 - b^2}$

$c = \sqrt{1.125^2 - .125^2}$

c= 1.1180

Pt.2/3 = C+.5=1.6180

```
PT
    ZN1G0G90X-3.Y3.T1M6
1   N5X.5Y0Z1.175
    N10G1Z1.125F50
    =N30/100 ------------------------
2   N15G2G18G91X1.118Z-1.K1.125     |--- LOOP
3   N20G1Y.01                       |
4   N25G3G18X-1.118Z1.I1.118K.125F200
5   N30G1Y.01 -----------------------
    N35G0G90X-3.Y3.M2
    E
```

NOTE: The following procedures must be followed when programming XZ (G18) or YZ (G19) Circular.

a) G91 Incremental
b) G74 Circular
c) Directional modifier (G2 or G3) must come before plane selection (G18 or G19).
d) Program to the center of the radius of the Ball End Mill

N5 and N10 position the cutter to Pt. 1.
=N30/100 calls the Loop that will execute N15 thru N30 (100) one hundred times(see Loop Pg. 6.1).
N15 sets the system in circular (G2), selects the XZ plane (G18) and puts the system in incremental (G91). The X and Z values are the incremental distances from Pt. 1 to Pt. 2 and the K value is the incremental distance from the Arc Center to Pt. 1.
N20 moves an incremental distance (.01) from Pt. 2 to Pt. 3 in the Y axis.
N25 resets the system in circular (G3) and the XZ plane (G18). The X and Y values are the incremental distances from Pt. 3 to Pt. 4 and the I and K values are the incremental distances from the Arc Center to Pt. 3.
N30 is the last block of the loop which positions the cutter to Pt. 5 ready to begin the next execution of the loop.

5.6 FEEDRATE COMMANDS (F)

Feedrate commands are programmed directly in tenths of inches per minutes (or mm/min) using the letter address F and a three digit number ranging from:

F2 to F320 representing .2 to 32.0 IPM (5 to 812 mm/min)
—-Boss 3-5 Systems
F2 to F510 representing .2 to 51.0 IPM (5 to 1295 mm/min)
—-Boss 6 System

Once the feedrate is programmed for a linear (G1) or circular (G2, G3) move, it is modal, i.e., it remains in effect until reset or superseded by another feedrate number. G0 (rapid traverse) will override the set feedrate; however, if a subsequent G1, G2 or G3 block is programmed, the feedrate previously in effect will resume. Feedrates programmed higher than 32.0 IPM (51.0 Boss 6) will automatically default to the maximum value of the system (32.0 or 51.0 IPM).

The system will maintain constant vector velocity in the feed range for both 3-axis linear and 2-axis circular interpolation, regardless of the slope of cut.

NOTE: Feed is inhibited until the spindle has been turned ON.

5.6.1 Prorated Feed Rates

When programming feedrates for circular cuts, the ratio of the cutter path radius to the part surface radius affects the cutting rate, since the vector velocity is that at the center of the cutter, not at the surface of the material. This means that, in order to maintain a constant chip load when machining the outside of an arc, the feed rate should be increased according to the following:

$$\left(\frac{PR + CR}{PR}\right) \text{ multiplied by (Desired Feed Rate) = Programmed Feed Rate}$$

PR = Part Radius
CR = Cutter Radius

Referring to Example 7 below: Arc #1 (Block N11)

Desired Feed Rate = 10 IPM

PR=1.5"
CR= .5"
$$\left(\frac{PR+CR}{PR}\right)(10) \text{ IPM = Programmed Feed Rate}$$

$$\left(\frac{1.5 + .5}{1.5}\right)(10) = 13.3 \text{ IPM; Therefore Programmed Feed = F133}$$

Conversely when machining the inside of an arc, the feedrate should be decreased according to the following:

$$\left(\frac{PR - CR}{PR}\right) \text{ multiplied by (Desired Feed Rate) = Programmed Feed Rate}$$

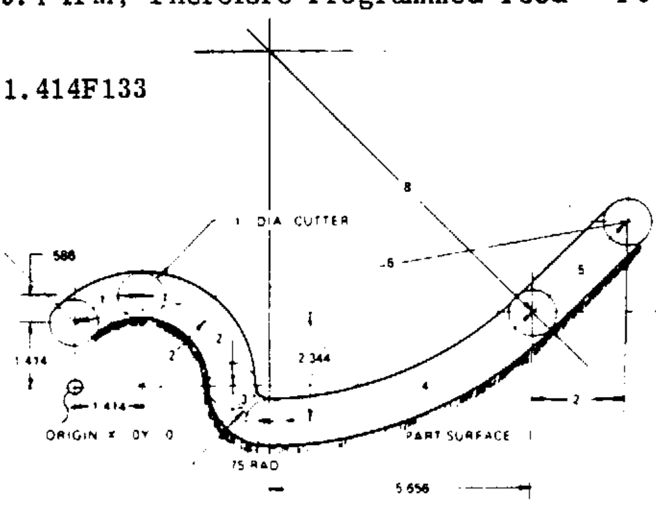
Referring again to Example 7 below: Arc #3(Block N13) and Arc #4 (Block N14)

Desired Feed Rate = 10 IPM

PR=1.25"
CR= .5"
$$\left(\frac{1.25 - .5}{1.25}\right)(10) = 6 \text{ IPM; Therefore Programmed Feed = F60}$$

PR=8.5
CR= .5
$$\left(\frac{8.5 - .5}{8.5}\right)(10) = 9.4 \text{ IPM; Therefore Programmed Feed = F94}$$

```
N11G2G90G99X1.414Y2.I1.414J1.414F133
N12G99Y0J2.
N13G3G99X4.164Y-.75I.75F60
N14G99X9.82Y1.594J8.F94
N15G1X11.82Y3.594F100
```



5.7 SPINDLE SPEED COMMANDS (S)

5.7.1 General

The letter address S programmed with a four digit number is ignored by the system but may be used to instruct the operator as to what speed to set manually for the operation involved. The display will show this programmed value of spindle speed without the tens digit. For example, S1234 will display 123, signifying 1230 RPM.

When starting, the feed is inhibited until the spindle has been turned ON.

5.7.2 Spindle Drive

The 2 HP spindle drive motor is a constant speed (1750 RPM) AC induction motor driving a transmission consisting of a varispeed conical sheave arrangement. The output of the varispeed belt drive can be directly coupled to the spindle or be clutched to engage an 8.3:1 reduction gear to the spindle.

HP versus speed characteristics are shown in Figure 5-12.

5.8 TOOL SELECT COMMANDS (T)

The tool select code consists of the letter T programmed with a two digit number ranging from 1 to 24.

5.8.1 Tool Length Offset

When an M06 (tool change) command is received by the control, the Tool Length Offset value previously stored in the system for that particular tool will be transferred into the Z absolute register.

The T command can also be used within the program to load values into the Tool Length Offset storage registers. The format for this command is:

Ta/b where a is the Tool Number (from 1 to 24) and b is the Z axis offset value.

For example:

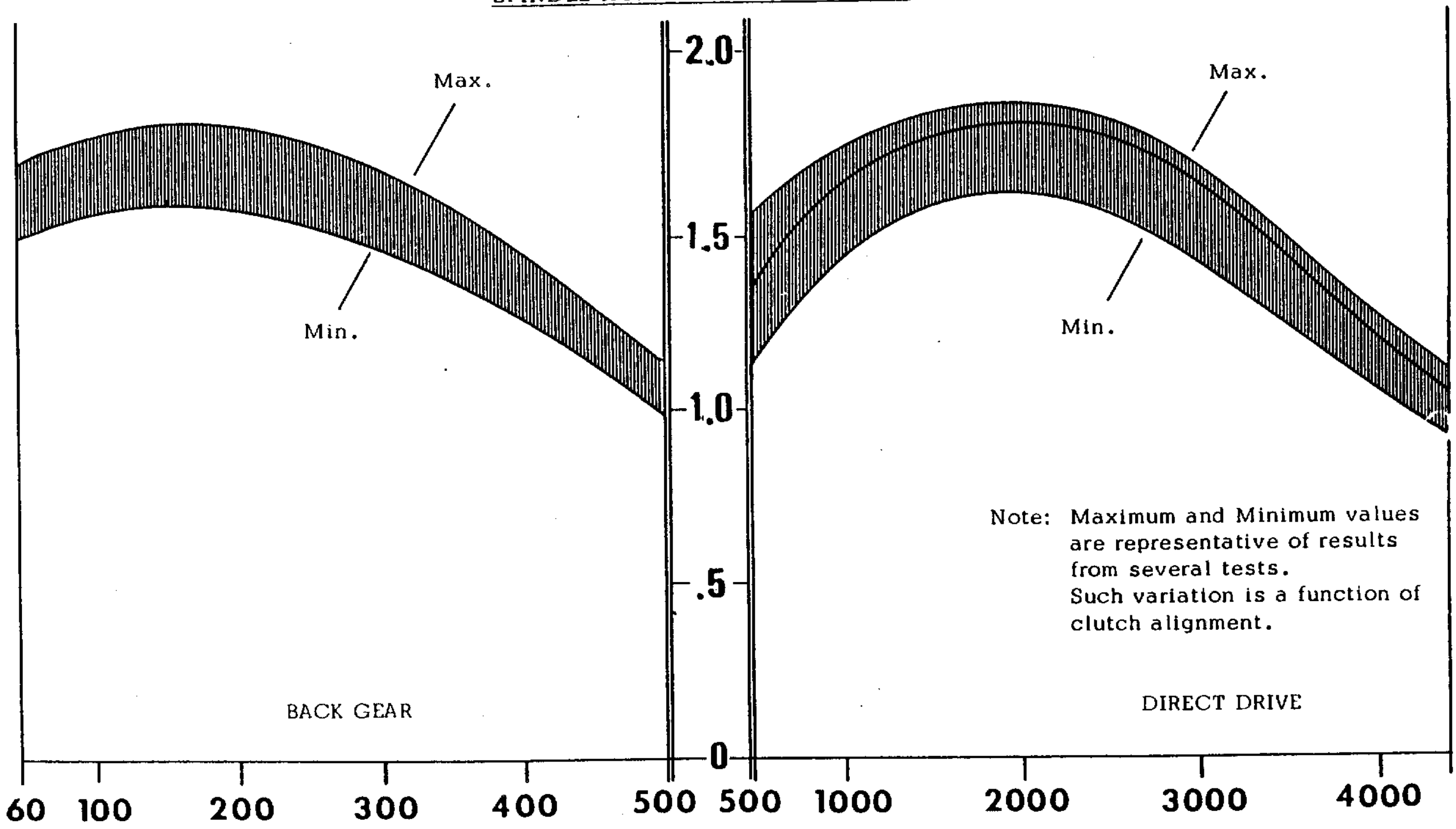
T1/1.0	will load offsets of 1.0", .45" and 1.25" into tool
T2/.45	locations 1, 2 and 12 respectively.
T12/1250	See full description under Tool Offset paragraph 3.2

CAUTION: TLO must be entered only to the third decimal place, e.g. T14/.4995 will be stored as 4.995.

Tool holders must be loaded in the spindle by Tool Number to match the Offsets.

MODEL (2J) 2HP MILLING HEAD

SPINDLE HORSEPOWER VERSUS RPM



Note: Maximum and Minimum values are representative of results from several tests. Such variation is a function of clutch alignment.

5.8.2 Tool Diameter (Also see Section 9.3.7.1)

The tool diameter may be input to storage either by MDI or by tape. It is used by the operator only for display purposes.

The T command identifies the reference diameter with the following format:

Tn//d	where n is the Tool Number (1-24) and d is the Tool Diameter

For example:

T1//.5
T2//.375
T3//.125

This data may be coded at the beginning of the program text or it may be embedded later in the program. If at the beginning and the d value has to be changed, this can be done with MDI (see M-128). If, however, the Tn//d is embedded, a change to this value can only be made using the EDITOR (see M-128). Two different values for the same tool number must not be used.

5.9 MISCELLANEOUS FUNCTION COMMANDS (M)

The miscellaneous function code consists of letter address M followed by two digits. It is used to program special machine functions. Only one miscellaneous function may be entered per block of data.

The following is a list of these functions and a description of each along with a description of their use in the context of a program.

NOTE: In the following miscellaneous function descriptions, the notations (B), (A) are to be recognized as:

B - Activated before axis motion occurs.

A - Activated after axis motion is complete.

M00 - (B A) Program Stop

A program stop is used whenever the programmer requires the program cycle to stop and allow the operator to perform some manual function such as tool transfer (without changing the offset), inspection, fixture adjustment, etc. Adequate operator instructions should accompany the program. XY motion data programmed in rapid traverse (G0) may be in the same block as the program stop (M0), however, the M0 must not be programmed in G1, G2 or G3 blocks as the spindle will be stopped before axis motion occurs.

To continue operation depress START/CONTINUE button.

M01 - <u>(B A) Optional Program Stop.</u>

This code stops the machine in the same manner as a program stop (M00) except that the operator must elect to do so. This is done by turning the OP STOP ON-OFF switch on the control console to the ON position sometime prior to the time this code is executed by the program. If the OP STOP ON-OFF switch is positioned to OFF, this command will be ignored.

M02 - <u>(B,A) - End of Program.</u>

This function has the same effect as M00 except the quill is commanded to retract to the Home position before axis motion occurs. The M02 command will set the sequence number to 0 and reset the part program counter to 0 thus enabling the operator to repeat the part program by depressing START/CONTINUE.

M06 - <u>(B,A) Tool Change.</u>

This function denotes that a tool change is requested.

The tool change command (M06) is activated immediately upon transfer into active storage.

The following sequence occurs:

a. After the Z axis retracts to its UP position, then X and Y motion will occur, if programmed.
b. The Z absolute register is automatically set to the value of the Tool Length Offset addressed by the tool select number.
c. Operator stops the spindle.*
d. Operator changes tools.
e. Operator restarts the spindle, sets the correct speed for the new tool and depresses the START/CONTINUE button.

NOTE: It shall be the programmer's responsibility to position the applicable axis (axes) for part, fixture, and/or tool clearance prior to programming an M06.

*Spindle stops automatically if the machine is equipped with the Auxiliary Control Group.

<u>In BOSS 5 and BOSS 6 systems the first Z value after a tool change command must be in absolute coordinates.</u>

M25 - <u>(B) Z Axis to "Home"</u>

This function when programmed will cause the quill to move to the UP position at the Z axis rapid traverse rate. When the quill reaches the home position, X and Y axis motion, if programmed, can occur.

NOTE: M codes not defined above but present in the program text will be ignored by the system.

** In BOSS 5 and BOSS 6 systems the first Z value after a Z home command must be in absolute coordinates.

5.10 END OF BLOCK COMMAND (CR)

The code in use is the Carriage Return (CR). The (CR) code is essential for each block of data and it is this code that causes the control function contained within that block to be initiated.

In the MDI mode, the black key (the EOB code) is essential for each block of data and it is this code that causes the control function contained within that block to be initiated. The last valid block on a control tape is the letter E (CR). This is used by the tape reader control only for the purpose of rewinding the punched tape after loading the program to storage.

5.11 DEVELOPING A CONTINUOUS PATH PROGRAM

We have chosen an item to be manufactured from aluminum plate as shown in Figure 5-13. Some basic decisions associated with the part are immediately apparent: First, that most dimensions are established from the lower left hand corner which lends itself easy to programming in absolute coordinates from that point as the(X0,Y0)origin. Secondly, since the tool should be clear of the workpiece at the end of the program, to allow the operator to change parts, the start point or SET POINT is located at the top left hand corner.

Consideration involved in the holding of the part on the table are not included here but some thoughts must be explored. If two finished edges are available then the part can be banked against pins and clamped over these points. Further, the part can be raised on 1/8" parallels to allow the cutter to project below the bottom edge and not machine the fixture.

If, however, there are no finished starting edges, two approaches can be taken: First, make a 2 station fixture, the first to finish two edges and the second to proceed as above. Secondly, program the part in such a way that at possibly two convenient times there is a program stop in order to remove clamps from unmachined portions and insert clamps over the machined edges.

Feed and speed considerations will be evaluated along with the possible need for rough and finish passes but some fundamental consideration follows: With metal removal rates being of prime consideration and variable amounts up to full cutter width to remove, use a 2 flute (for chip clearance) HSS end mill of 1" diameter.

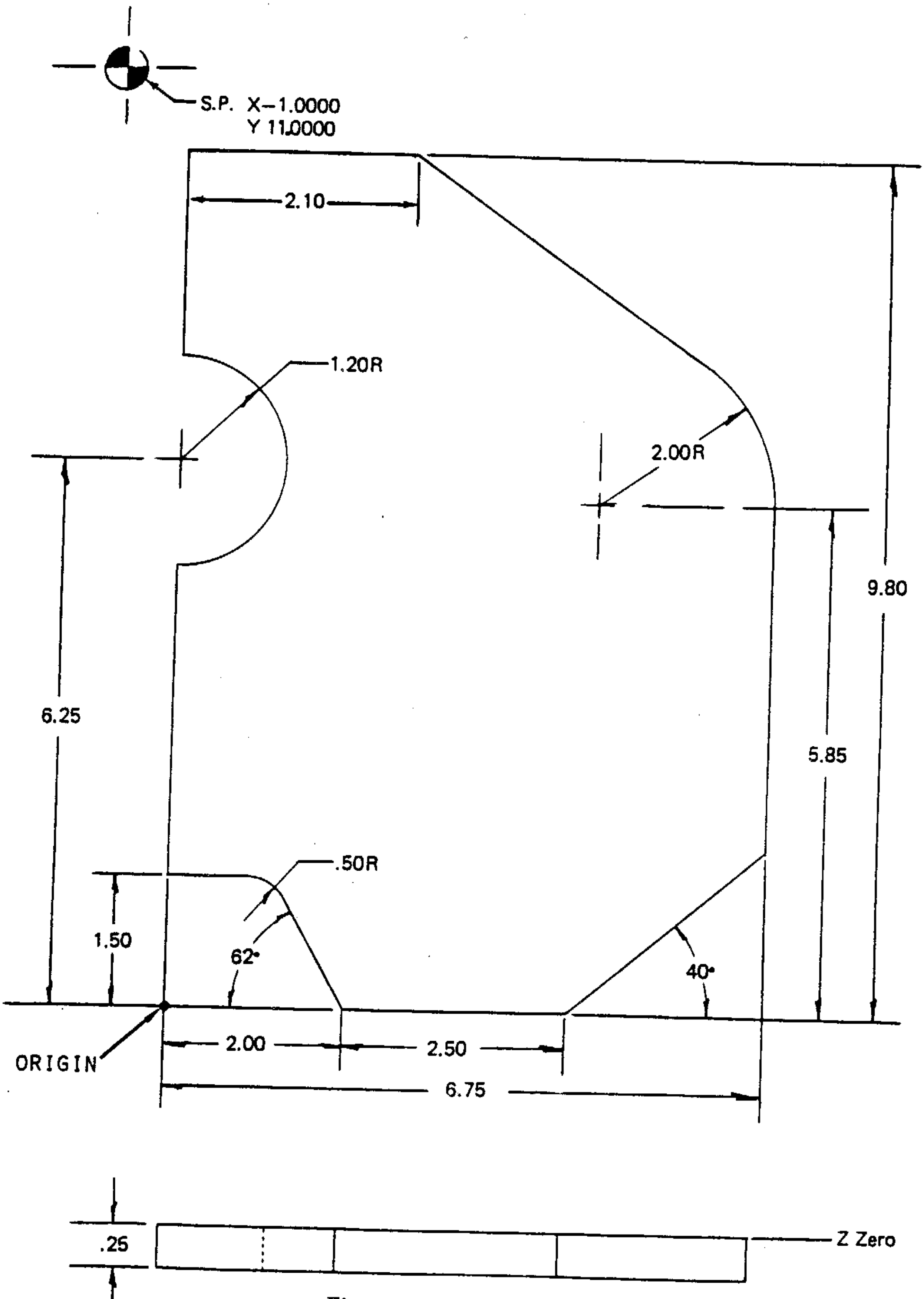


Figure 5-13. Cover Plate

SFM = 500, SPINDLE SPEED - 1910 RPM

Chip Load = .005, Feedrate = 19.1 ipm

Metal Removal = 4.8 ins.^{3}/min = 1.6 HP

Since the spindle speed is in the most efficient HP region, these figures will be satisfactory.

If the part is to be programmed manually, proceed with an N/C layout with calculations as shown in Figure 5-14. The following calculations are pertinent:

$a = \text{Tan}^{-1}\ (2.15/3.95)$	=	$28^\circ 33.7'$
$b = \sqrt{(3.95)^2 + (2.15)^2}$	=	4.4972
$c = \text{Sin}^{-1}\ (2.0/b)$	=	$26^\circ 24.5'$
$d = a + c$	=	$54^\circ 58'$
$e = .5 * (90 - d)$	=	$17^\circ 31'$
$f = .5 * (\text{Tan}(e))$	=	.1578

Thus the following absolute coordinates can be used:

Point 2 : X ABS = 2.6+f	=	2.7578
Point 3: X ABS = 4.75+ (2.5* Cos(d))	=	6.1852
Y ABS = 5.85+ (2.5* Sin(d))	=	7.8970
Point 4: X ABS = 6.75+.5	=	7.2500
Y ABS = Drawing	=	5.8500
I (X) = 2.5* Cos(d)	=	1.4352
J (Y) = 2.5* Sin(d)	=	2.0470
Point 5: Y ABS = (2.75-h)*.5 Tan(40)	=	1.6548
Point 6: X ABS = 4.5 + .5 Tan (20)	=	4.6820
Point 7: X ABS = 2.0 - i	=	1.6996
Point 8: X ABS = 2.0 - i - j	=	0.9020

*= multiply

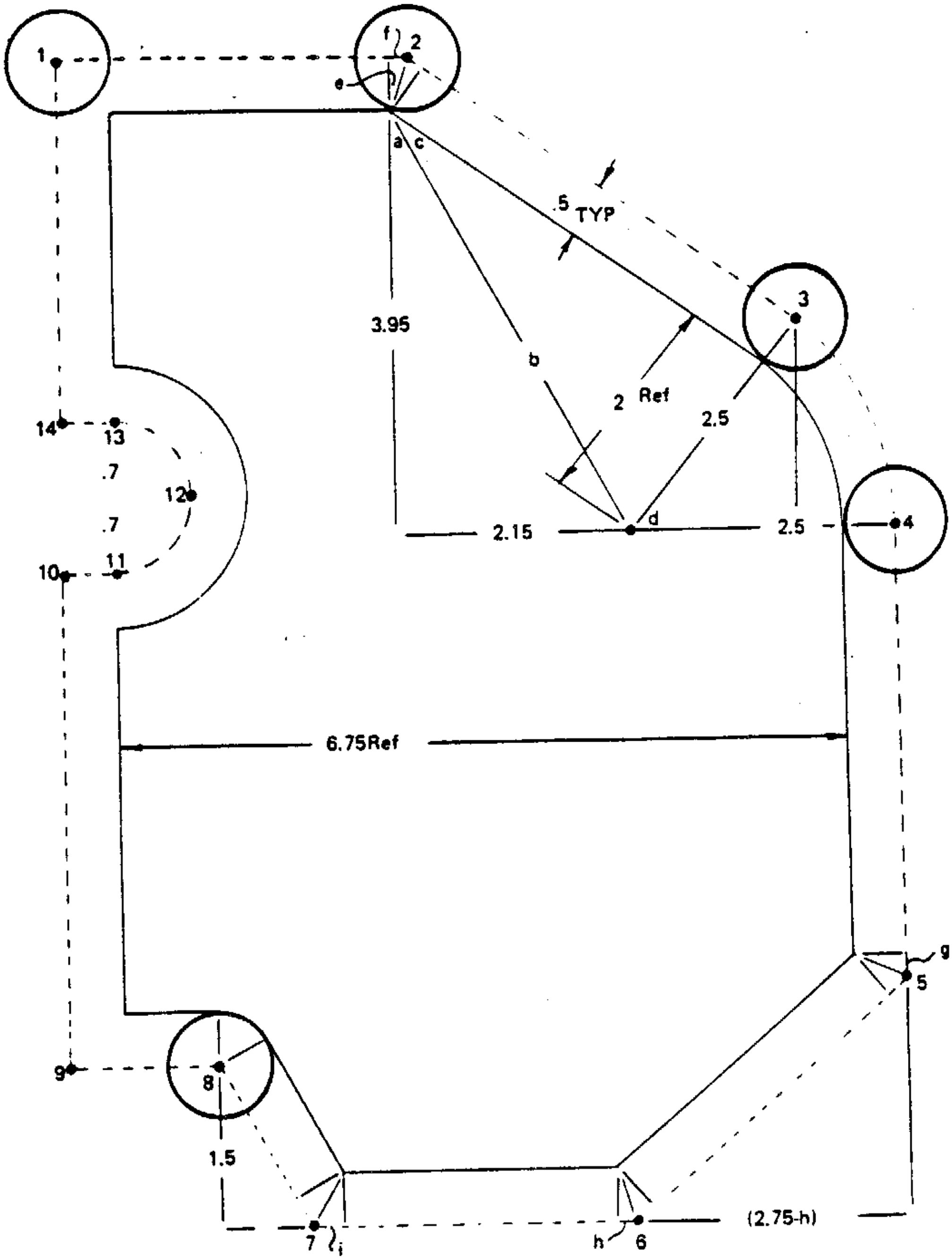


Figure 5-14. Cover Plate N/C Layout

COVER PLATE
<u>ABSOLUTE COORDINATE LIST</u>

1.)	X-.55	Y10.3
2.)	X2.7578	Y10.3
3.)	X6.1852	Y7.897
4.)	X7.25	Y5.85
5.)	X7.25	Y1.6548
6.)	X4.682	Y-.5
7.)	X1.6996	Y-.5
8.)	X.902	Y1.
9.)	X-.5	Y1.
10.)	X-.5	Y5.5
11.)	X0	Y5.5
12.)	X.7	Y6.25
13.)	X0	Y6.95
14.)	X-.5	Y6.95
15.)	X-.5	Y10.3
16.)	X-1.	Y11.

The reader should verify the methods used and figures obtained.

The writing of the program manuscript can now be started. The completed program is shown in Figure 5-15. An explanation of the data follows by sequence number:

<u>SEQ. NO. 5</u> Position to X-1.0Y11.0 for loading tool number 1 (T1M6). Start the spindle at 1910 RPM clockwise.

<u>SEQ. NO. 10</u> Position over location 1 in X and Y followed by a quill move of minus Z the tool length offset value and the Z value.

<u>SEQ. NO. 15</u> Set the system in linear interpolation feed mode (G1). Mill top of part at a feed rate of 19.1 IPM (LOC 2).

<u>SEQ. NO. 20</u> Mill angle surface to tangency of 2.00 Radius (LOC 3). Call feedrate Deceleration Override (G99).

<u>SEQ. NO. 25</u> Set the system in circular interpolation clockwise mode (G2). Call feedrate Deceleration Override (G99). Input the X and Y absolute coordinate for the end point of the 2.00R (LOC 4) and input the I and J values of the unsigned distance from the arc center to the start point of the arc. Modify the feedrate to 24.0 IPM as follows:

((2 + .5)/2) * 19.1 = 23.875 = 24 IPM

((PR + CR)/PR) * DF = IF

PR = Part Radius

CR = Cutter Radius

DF = Desired Feed

IF = Input Feed

<u>SEQ. NO. 30 Through 60</u> mill remaining sides of the part up to the start of 1.20 radius in linear interpolation feed mode (G1). Set the feedrate to 19.1 IPM (LOC 5, 6, 7, 8, 9, 10 & 11).

<u>SEQ. NO. 65</u> Sets the system in circular interpolation counter clockwise mode (G3). Call feedrate Deceleration Override (G99). Input the X and Y absolute coordinates for end point of the first quadrant of the 1.20 radius (LOC 12) and input the J value (arc offset). Modify the feedrate to 11.1 IPM. ((1.2-5)/1.2) * 19.1 = 11.1, ((PR-CR)/PR) * DF = IF.

<u>SEQ. NO. 70</u> Complete the 1.20 radius (LOC 13).

PREPARED BY	RLP	DATE	2/1/78
CHECKED BY		DATE	
SHEET 1	OF 1	TAPE NO	J8650

Bridgeport. TEXTRON

Bridgeport Machines Division of Textron Inc.

PART NO.	123456
PART NAME	COVER PLATE
COMPANY NAME	

NUMERICAL CONTROL PROGRAM

NOTES:	N	G	X/Y	X/Y/Z	Y/Z/I/J	Y/Z/J/K	F	S	T	M
%		-LOAD	1.000 D	IA. END	MILL -					
START	N5	G0 G90	X-1.	Y11.				S1910	T1	M6
1	N10		X-.55	Y10.3	Z-.35					
2	N15	G1	X2.7578				F191			
3	N20	G99	X6.1852	Y7.897						
4	N25	G2 G99	X7.25	Y5.85	I1.4352	J2.047	F240			
5	N30	G1		Y1.6548			F191			
6	N35		X4.182	Y-.5						
7	N40		X1.6996							
8	N45		X.902	Y1.						
9	N50		X-.5							
10	N55			Y5.5						
11	N60	G99	X0							
12	N65	G3 G99	X.7	Y6.25	I0	J.7	F111			
13	N70	G99	X0	Y6.95	I.7	J0				
14	N75	G1	X-.5				F191			
1	N80			Y10.3						
START	N85	G0	X-1.	Y11.						M2
	E									

F-1378

Figure 5-15. Completed Program Manuscript

SEQ. NO. 75 and 80 Complete the remainder of the part (LOC 14, 1).

SEQ. NO. 85 Set the system in rapid traverse mode (G0). Return the tool to the starting coordinates. The Z axis will retract to the home position before the X and Y axes move because the X and Y motion was programmed with the rewind command (M2).