CHAPTER 6

CANNED CYCLES

6.1 OVERVIEW

Canned cycles reduce programming time by allowing frequently used milling and drilling sequences to be programmed in a single data block.

6.2 Z AXIS CYCLES

The following lists the basic Z axis cycles:

G CODE	PLUNGE	OPERATION AT BOTTOM	RETRACT	APPLICATION
81	Feed		Rapid Traverse	Drilling
82	Feed	Dwell	Rapid Traverse	Spot facing
83	Peck		Rapid Traverse	Deep hole drilling
84	Feed		Feed	Tap
85	Feed		Feed	Bore
86	Feed	Stop-wait	Rapid Traverse	Bore
87	Peck		Rapid Traverse	High speed deep hole drilling
89	Feed	Dwell	Feed	Bore

6.2.1 Drill/Bore/Tap Cycles

The command format for the basic Z axis cycles is:

```
G81(82....)X___Y___Z___F___
G81(82....)A___Z___F___        (polar)
```

The Z axis canned cycles generally comprise a sequence of the following operations:

Operation [1] Position X and Y axes
Operation [2] Feed down (-Z)
Operation [3] Operation at the hole bottom
Operation [4] Retract to initial Z position

NOTES

1. The Z depth for a canned cycle is input as an incremental unsigned value equal to the depth of the hole plus the clearance distance desired.

2. An X or Y coordinate word is required for the drill cycle to occur. This may be U0 if the hole is to be drilled in place.

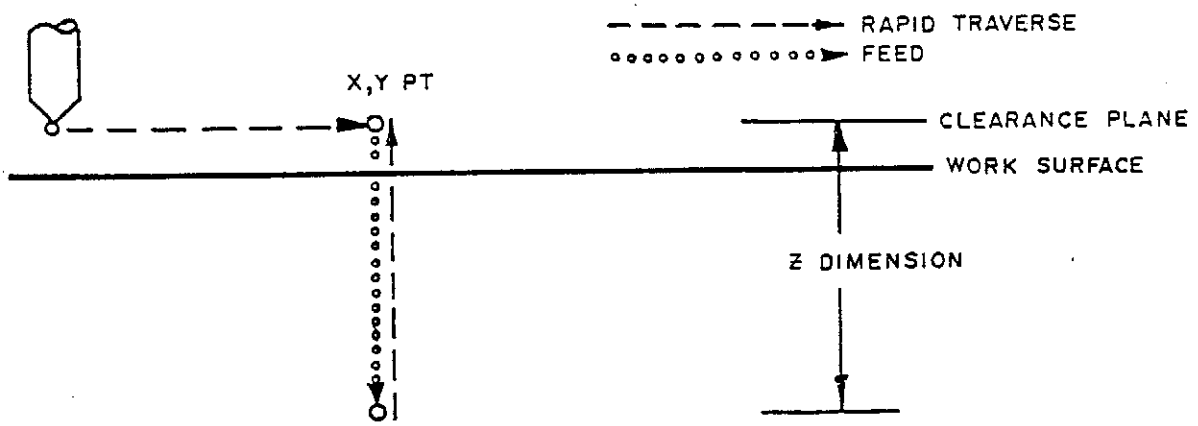
3. Once a Z axis canned cycle is input it remains in effect until it is cancelled by a G0 or G80 code. In every data block that contains an X or Y word, the specified Z axis cycle will occur.

4. To change the Z axis depth, the entire Z axis canned cycle format must be reentered with the new depth.

5. A rapid traverse Z move is not permitted within a fixed cycle. If required, the cycle must be terminated by a G0 move, then the cycle must be reinstated with the desired Z axis canned cycle code.

G81 Drilling Cycle

This command provides for a feed-in, rapid-out sequence suitable for drilling a series of holes.

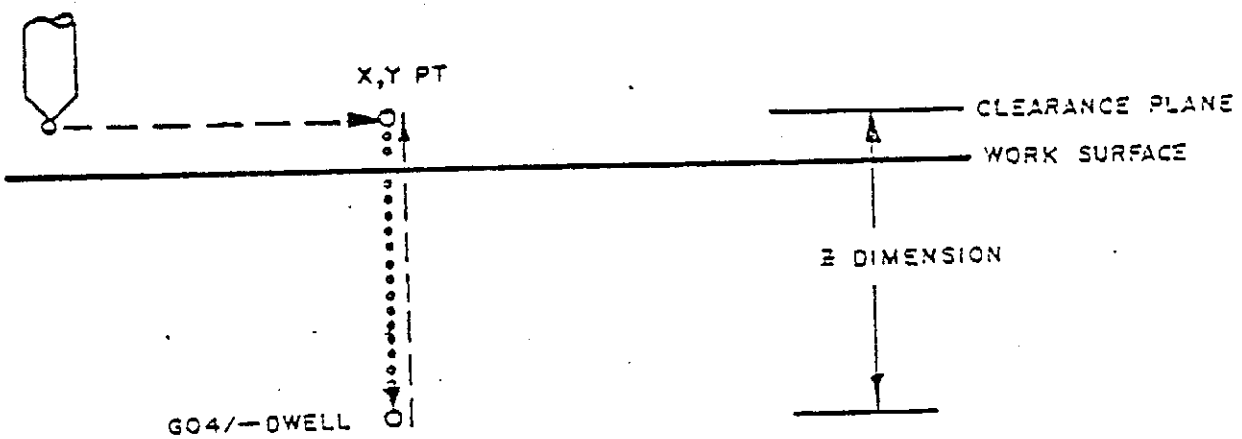


G82 Spot Facing Cycle

This command provides for a feed-in, dwell, rapid-out sequence suitable for spot facing and counter boring operations.

NOTE

This sequence is the same as G81 except for the delay that occurs when the Z axis reaches the programmed depth. The time of delay is set by a G4/___ command.



G84 Tapping Cycle

This command provides a feed-in, feed-out sequence suitable for a tapping operation with a tapping attachment for non-reversing spindles.

A chart of feed and speed values for tapping various pitches is given below. Feeds and speeds can be selected from this chart to program the desired thread pitch. Use of the designated speed will minimize the amount of tap holder compensation required.

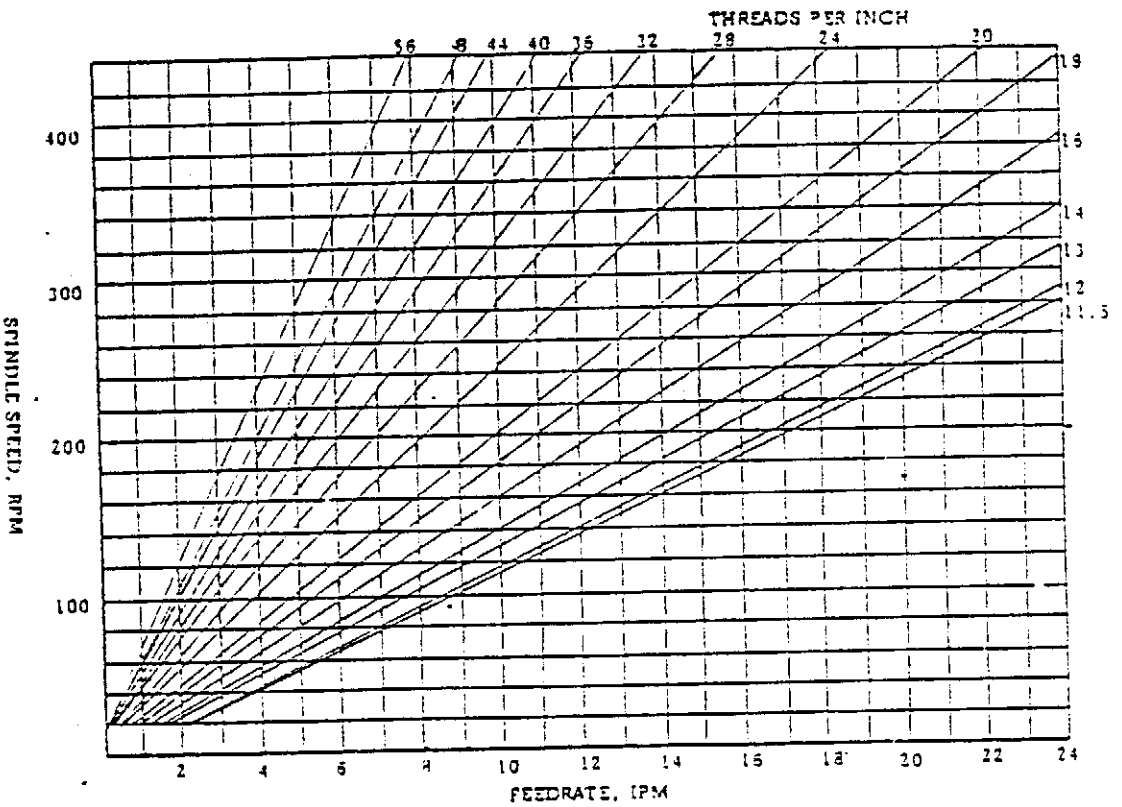
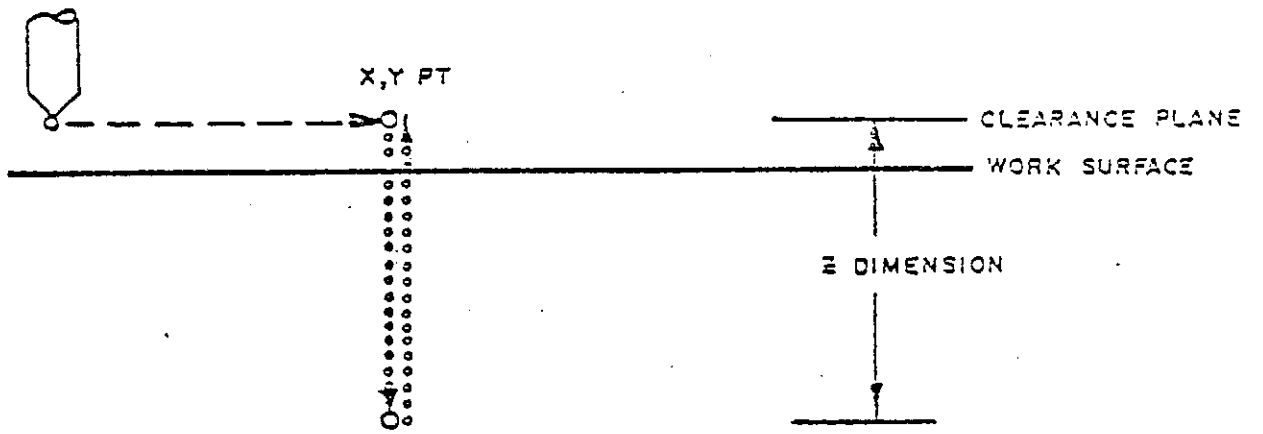


Figure 6-1: Feed and Speed Chart for Tapping

EXAMPLE:

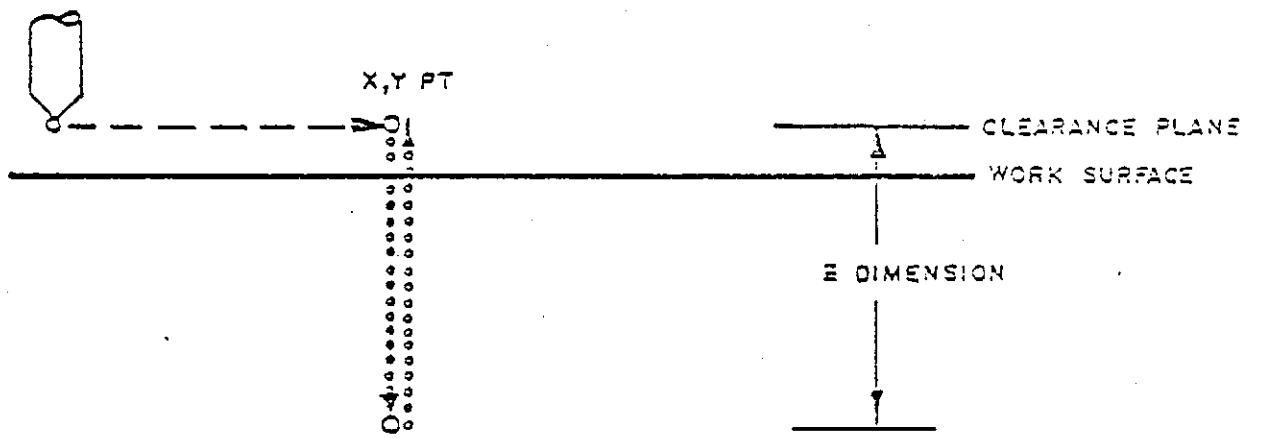
A 1/4-20 tap is to be programmed. A feedrate of 10.0 ipm is desired. From the chart a spindle speed of 200 rpm is selected.

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 Boring Cycle

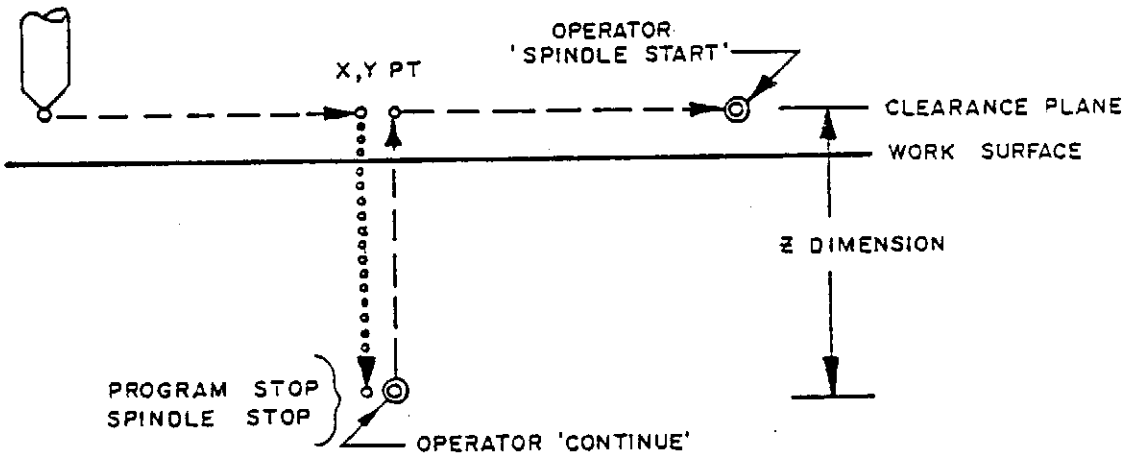
This command provides a feed-in, feed-out sequence suitable for boring or reaming operations.



G86 Boring Cycle

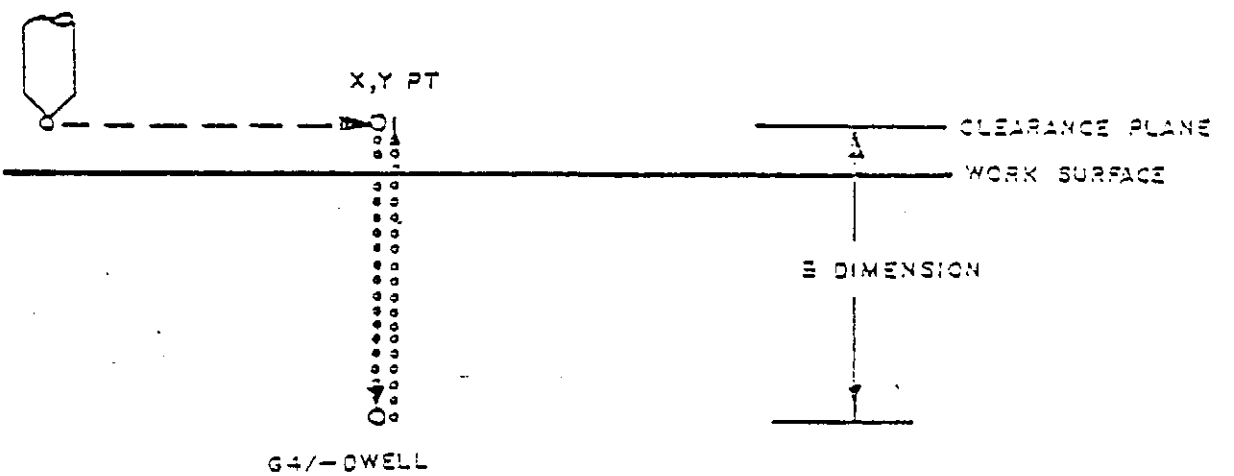
This command provides a feed-in, spindle stop to wait for operator CONTINUE command, then rapid-out. The axes will then rapid traverse to the next hole, if programmed, and wait for the operator to restart the spindle before feeding in.

The stop at the bottom of the hole enables the operator to orient the spindle if required.



G89 Boring Cycle

This command provides a feed-in, dwell, feed-out sequence suitable for boring. The time of delay is previously set by a G4/___ command.



6.2.2 Deep Hole Drilling Cycles

The format for the deep hole cycles is:

```
G83(G87)X___Y___Z___Z___Z___F___
G83(G87)A___Z___Z___Z___F___        (polar)
```

Where Z___Z___Z___ is the total Z depth, the first peck increment and subsequent peck increments.

NOTE

If the value for subsequent peck increments is omitted, the first peck increment is used for all peck distances.

The Z axis deep hole canned cycles comprise a sequence of the following operations:

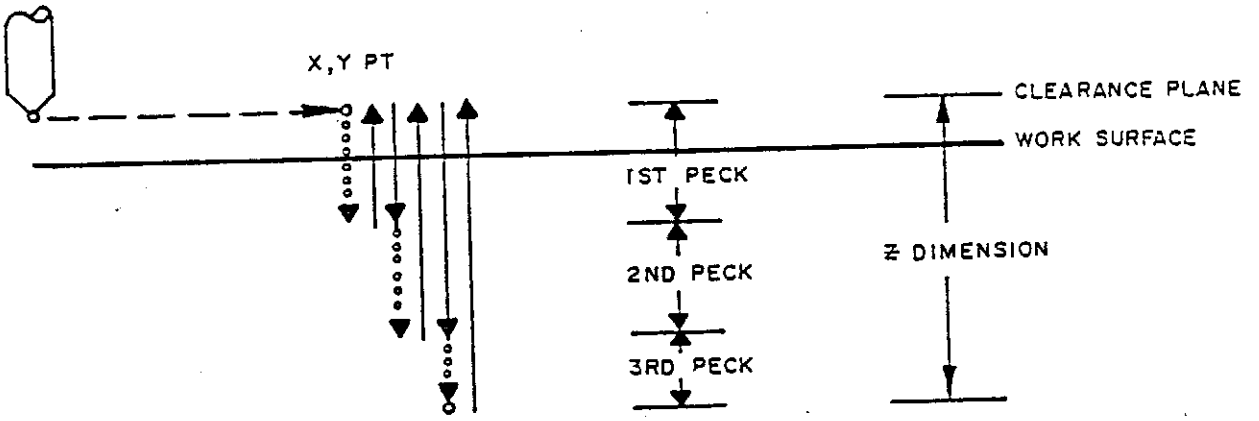
```
Operation [1] Position X and Y axes
Operation [2] Feed down 1st peck increment
Operation [3] Rapid traverse out
Operation [4] Rapid traverse back in to pervious peck depth
Operation [5] Feed down 2nd peck increment
          ... [3], [4], [5] are repeated until the total Z
              depth is reached
Operation [6] Retract to initial Z position
```

NOTE

All Z values are input as incremental unsigned values.

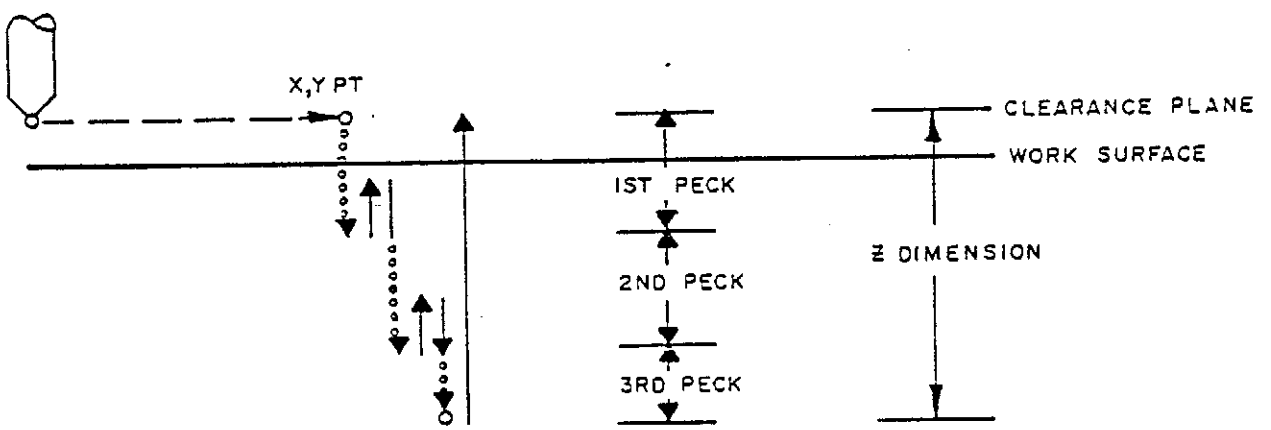
G83 Deep Hole Drilling Cycle

This command provides a deep hole drilling cycle as described above. After each peck increment the Z axis will retract back to the initial Z position.



G87 Chip Break Deep Hole Drilling Cycle

This command provides a chip break drilling cycle similar to the operation described previously. After each peck increment the Z axis will retract .01" and then rapid traverse back to the previous depth. The purpose of the G87 cycle is to break the chip rather than to withdraw the tool entirely from the workpiece as in a G83 cycle.



EXAMPLE:

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

6.3 MULTI-HOLE Z AXIS CYCLES

These cycles drill, bore or tap the hole patterns described below.

6.3.1 Row Drilling Bolt Circles - Incremental Hole To Hole Distance

This command provides means for drilling a row of holes along the X or Y axis or a bolt circle given the total distance from the first hole to the last hole and the incremental distance between holes. The format of this command is:

```
G81(G82....)X___X___Z___F___; X axis row
G81(G82....)Y___Y___Z___F___; Y axis row
G81(G82....)A___A___Z___F___; Bolt circle
```

Where X___X___,Y___Y___,A___A___ is the total hole distance, then the incremental hole distance. Z___F___ are the basic Z axis cycle parameters described in Sections 6.2.1 and 6.2.2 (Z___Z___Z___F___ are for deep hole drilling).

NOTES

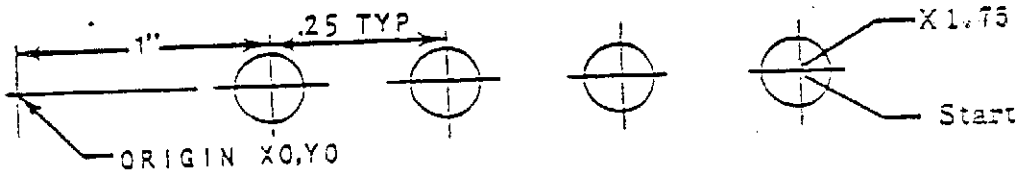
1. It is assumed for these cycles the programmer positions the axes over the first hole.

2. This format is for rows parallel to the X axis or parallel to the Y axis. However, these rows may be rotated using the transformation command.

3. The total distance may be either absolute or incremental (U and V input is also allowed). The incremental distance for X and Y rows is an unsigned value. The incremental distance for bolt circles is a signed value dependent upon the angular direction desired.

4. The last hole increment will be either the value input or the remaining distance, whichever is less.

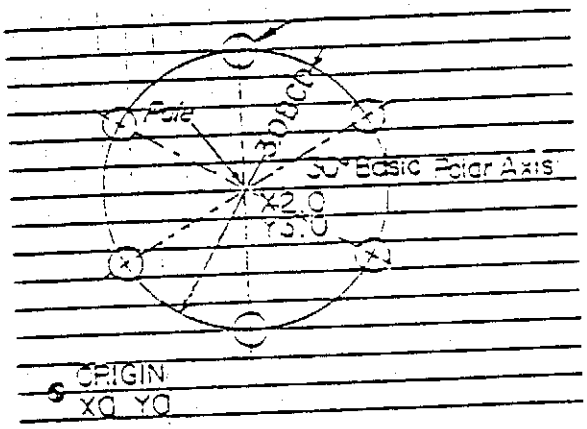
EXAMPLE: 4 deep drilled holes



Program either (A), (B) or (C) below:

(A) Abs: G90G87X1.X.25Z1.0Z.25Z.15F8.0;
(B) Incr: G91G87X-.75X.25Z1.0Z.25Z.15F8.0;
(C) Abs: G90G87U-.75X.25Z1.Z.25Z.15F8.0;

EXAMPLE: Bolt circle



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
```

1. Blocks N15 and N35 define the pole and radius.

2. Blocks N25 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.

6.3.2 Row Drilling And Bolt Circles - Number Of Holes

ROW DRILLING

These commands provide means for drilling a row of holes or a bolt circle given the total distance from the first hole to the last hole and the number of holes. The format of this command is:

G181(G182....)X___Y___Z___X___Y___Z___P___F___;

Where X___Y___Z___ is the location of the first hole in the row and the Z clearance plane, X___Y___ is the signed incremental distance from the first hole to the last hole, P___ is the number of holes to be drilled, Z___F___ are the basic Z axis cycle parameters described in 6.2.1 and 6.2.2 (Z___Z___Z___F___ for deep hole drilling).

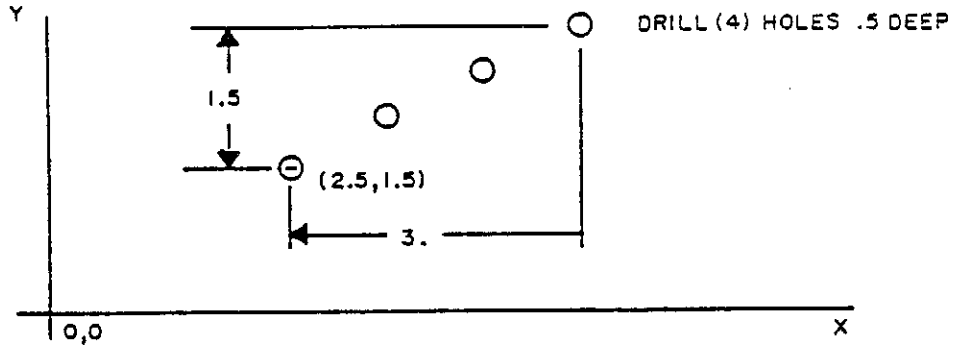
NOTES

1. This command will automatically position the axes over the first hole with a rapid traverse move.

2. Rows need not be parallel to the X or Y axis.

3. P, the number of holes, must be input with a decimal point.

EXAMPLE:



G181X2.5Y1.5Z.1X3.Y1.5Z.6P4.F20.;

BOLT CIRCLES

The format of the bolt circle commands is:

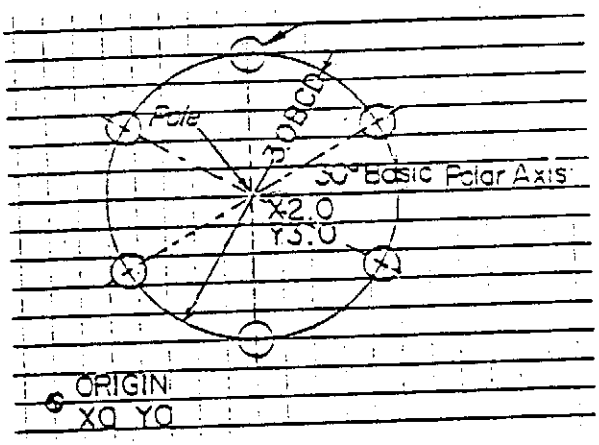
G181(G182...)A___Z___A___Z___P___F___;

Where A___Z___ is the location of the first hole in the bolt circle and the Z clearance plane, A___ is the signed incremental distance from the first hole to the last hole, P___ is the number of holes to be drilled, Z___F___ are the basic Z cycle parameters.

NOTE

R, I, and J data can be optionally included in this command.

EXAMPLE:



G181R1.5I2.J2.A30.Z.05A300.Z.2475P6.5F13.5;

This example will cause the same spot drilling sequence as the blocks N15, N20, N25 in the example in Section 6.3.1.

6.3.3 Frame Drilling - Number Of Holes

This command provides means for drilling a row of X and Y holes along the perimeter of a rectangular shape. The format of this command is:

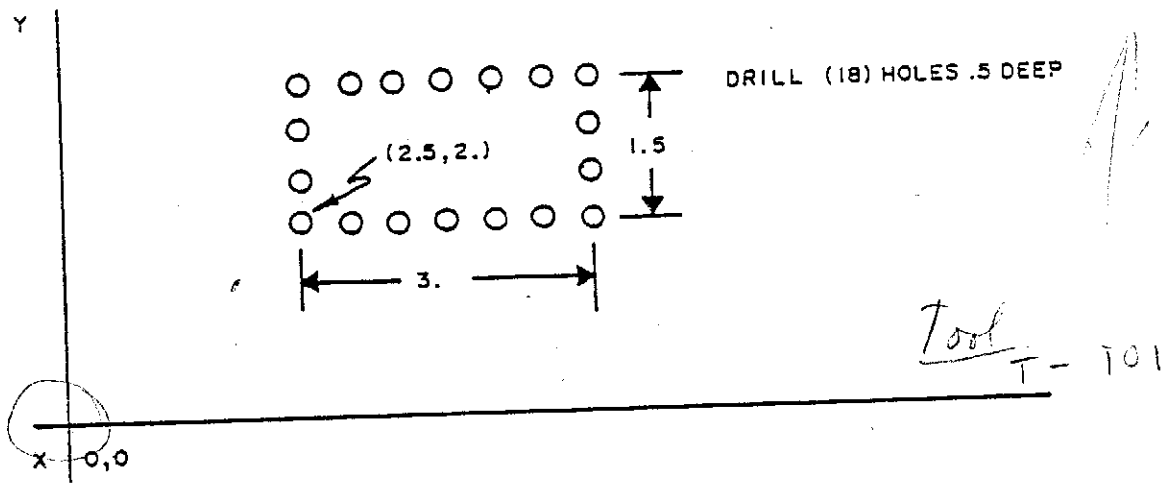
G191(G192...)X___Y___Z___X___Y___Z___P___P___F___;

Where X___Y___Z___ is the location of the first corner hole and the Z clearance plane X___Y___ are the signed values defining the incremental distance from the first hole to the last hole along the X and Y axes, P___P___ are the number of holes along the X and Y axis, and Z___F___ are the basic Z cycle parameters.

NOTE

The minimum value for P is 2.

EXAMPLE:



Tool T- 101,00

```
G91X2.5Y2.Z.05X3.Y1.5Z.55P7.P4.F20.;
```

6.4 MILL CYCLES

These functions enable machining frequently used shapes with one part program block. The following mill cycles are available:

G170: Outside Frame Mill
G171: Inside Frame Mill
G172: Pocket Frame Mill
G173: Outside Face Mill
G174: Inside Face Mill
G175: Outside Circle Mill
G176: Inside Circle Mill
G179: Slot Mill

The parameters for these cycles are:

	170,171	172	173,174	175,176	179
Center point	X	X	X	X	X
Center point	Y	Y	Y	Y	Y
Start point	Z	Z	Z	Z	Z
Length	X	X	X		P
Width	Y	Y	Y		P
Fillet Radius	R	R		R	
Radius					P
Angle of Rotation					
Full depth	Z	Z	Z	Z	Z
Step depth	Z	Z	Z	Z	Z
Step over,overlap		P	P		
Clearance	P	P	P	P	
Mill Feed	F	F	F	F	F
Finish allowance	P	P		P	
Finish feed	F	F		F	
Plunge feed	F	F	F	F	F

NOTES

1. The above mill cycles are cutter diameter compensated using the cutter diameter for the tool currently being used.

2. The mill cycles include an approach and departure more tangential to the part work surface, a Z axis step capability for deep cuts and roughing and finishing cuts.

3. Length, width, fillet radius, step depth, step over, clearance, finish allowance are all unsigned values.

4. Words may be addressed in any sequence except that words addressed by the same alphabetic character must be in the order shown.

5. All cycles except the G179 slot cycle may be transformed using rotation.

6. All parameters except step depth, finish feed and plunge feed must be entered even if they are 0. If finish feed is omitted, it will default to the mill feedrate. If plunge feed is omitted, it will default to 50% of the mill feedrate.

7. The Z start point must be at a clearance point above the work surface. Z cannot be at machine 0. It must be a minimum of .050" below Z home.

8. The dimension to the X, Y center point and the Z clearance plane may be incremental or absolute. All other dimensions are incremental as noted. The G179 slot mill cycle will set the control in absolute.

9. Variables may be substituted for canned cycle parameters. The variable to be used must not be in the parameter table on the previous page.

10. In G170-G179 cycles, the default values for fillet radius, Z depth and Z step are set0.

11. G170-G179 mill cycles end up at the input center point.

12. Milling cycles end execution at the center of the shape (X and Y center point defined in cycle) with the Z axis at the Z clearance plane.

13. A G179 cycle will put the system in absolute programming mode (G90) after execution. If the incrememtal programming mode (G91) was programmed prior to a G179, pressing the INCR button is necessary to remain in the incremental mode. (Item 9)

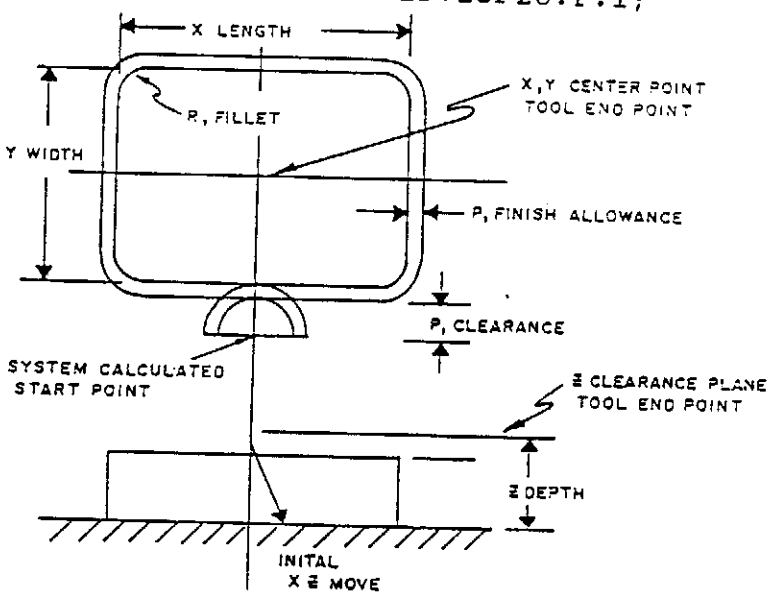
6.4.1 Outside/Inside Frame Mill

These cycles mill the outside or the inside of a rectangular shape. The format is this command is:

G170(G171)X___Y___Z___X___Y___R___Z___Z___P___F___
P___F___F___;

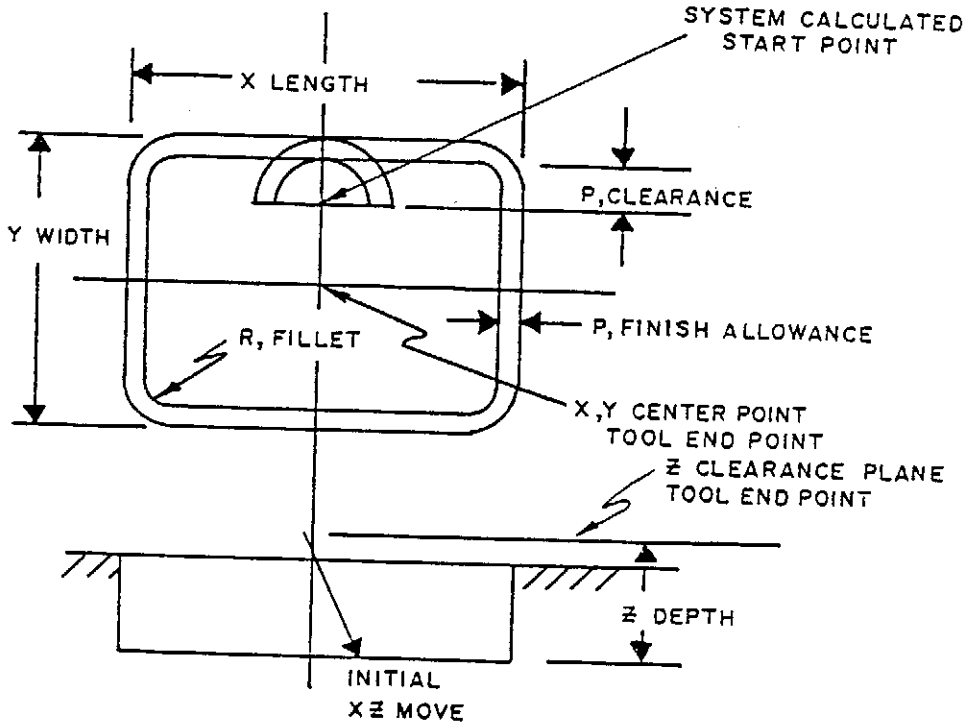
Where X___Y___ is the frame center point, Z___ is the clearance plane, X___Y___ is the unsigned length and width, R___ is the fillet radius, Z___Z___ is the full depth and the step depth, P___P___ is the initial entry clearance and the finish allowance, F___F___F___ is the mill feed, the finish feed and the plunge feed.

```
T1//0.;     TOOL DIAMETER =0
N10G170X3.Y4.Z-4.X2.Y1.5R.25Z.52P.25F20.P.1;
```



EXAMPLE:

```
T1//0.;     TOOL DIAMETER =0
N20G171X3.Y4.Z-4.X2.Y1.5R.25Z.52P.25F20.P.1;
```



6.4.2 Pocket Frame Mill

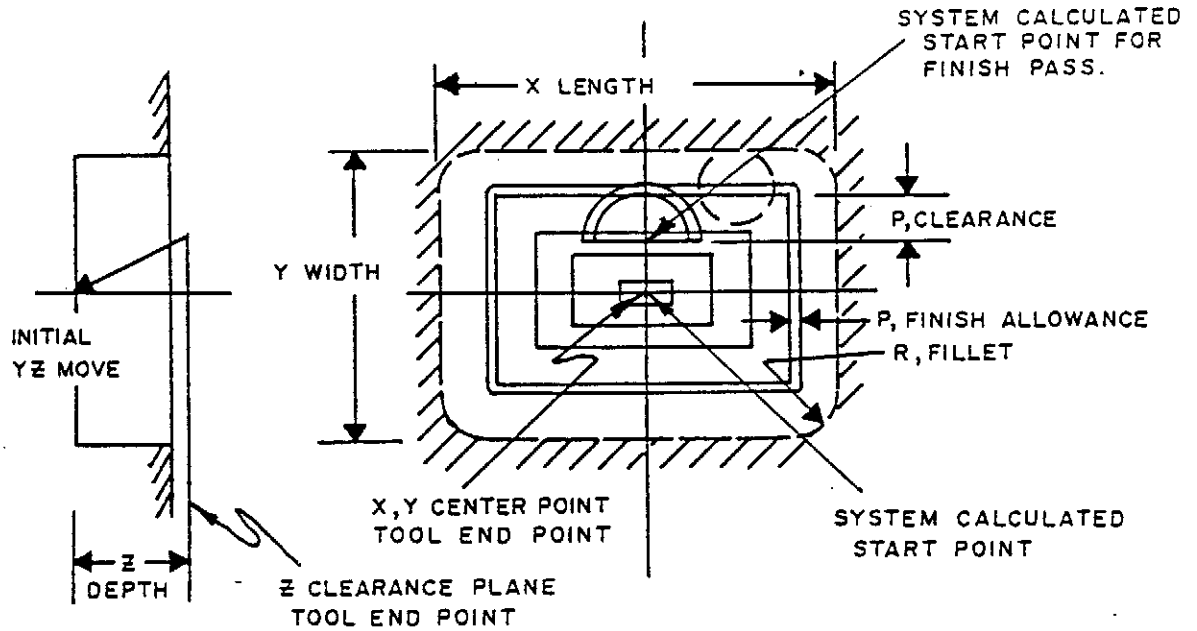
The pocket frame mill is a combination of an inside pocket mill routine and the inside frame mill routine. The format for this command is:

G172X___Y___Z___X___Y___R___Z___Z___P___P___F___
P___F___F___;

Where X___Y___ is the pocket center point, Z___ is the clearance plane, X___Y___ is the unsigned length and width, R___ is the fillet radius, Z___Z___ is the full depth and step depth, P___P___P___ is the step over, the clearance dimension for both the plunge to the center of the pocket and the final inside frame pass, and the finish allowance, F___F___F___ is the mill feed, the finish feed and the plunge feed.

EXAMPLE:

```
T1//.4;     TOOL DIAMETER =.4
N30G172X3.Y4.Z-4.X2.Y1.5R.25Z.52P.2P.25F20.P,05;
```



NOTE

The stepover on both the X and Y axes for the final pocket frame mill cut is equal to the input stepover value. If the pocket is not square, the stepover moves on the smaller of the axes to be moved will be less than the input stepover value.

6.4.3 Outside/Inside Face Mill

These cycles use zigzag moves to face the surface of a rectangular shape (G173) or to rough cut the material inside the boundary described by a rectangular shape (G174). The format for this command is:

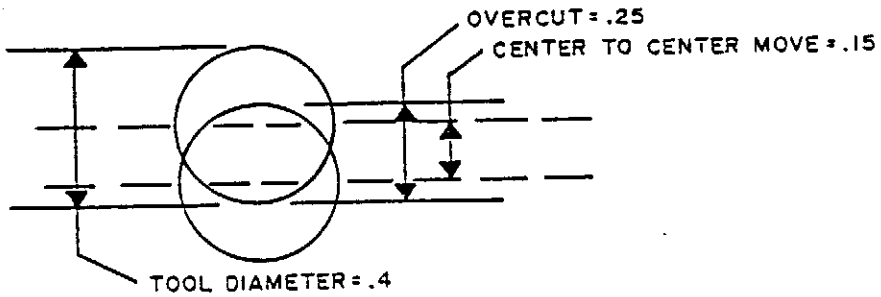
```
G173(G174)X___Y___Z___X___Y___Z___Z___P___P___
F___F___;
```

Where X___Y___ is the center point of the rectangle to be faced, Z___ is the clearance plane, X___Y___ is the unsigned length and width, P___P___ is the overlap and the clearance

for the plunge cut, F___F___ is the milling feedrate and the plunge feed.

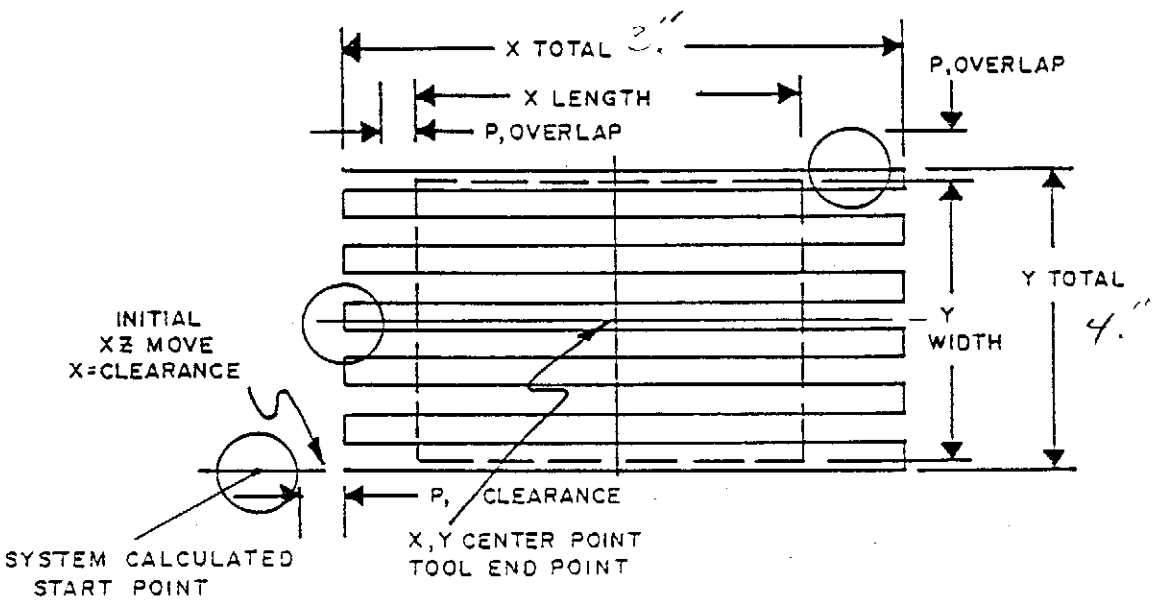
NOTE

Overlap is the amount of overcut caused by two consecutive passes. Thus, the center line to center line tool move is equal to the tool diameter minus the overlap. For example, if the tool diameter =.4 and the input overlap value =.25, then the center line to center line move =.15.



EXAMPLE:

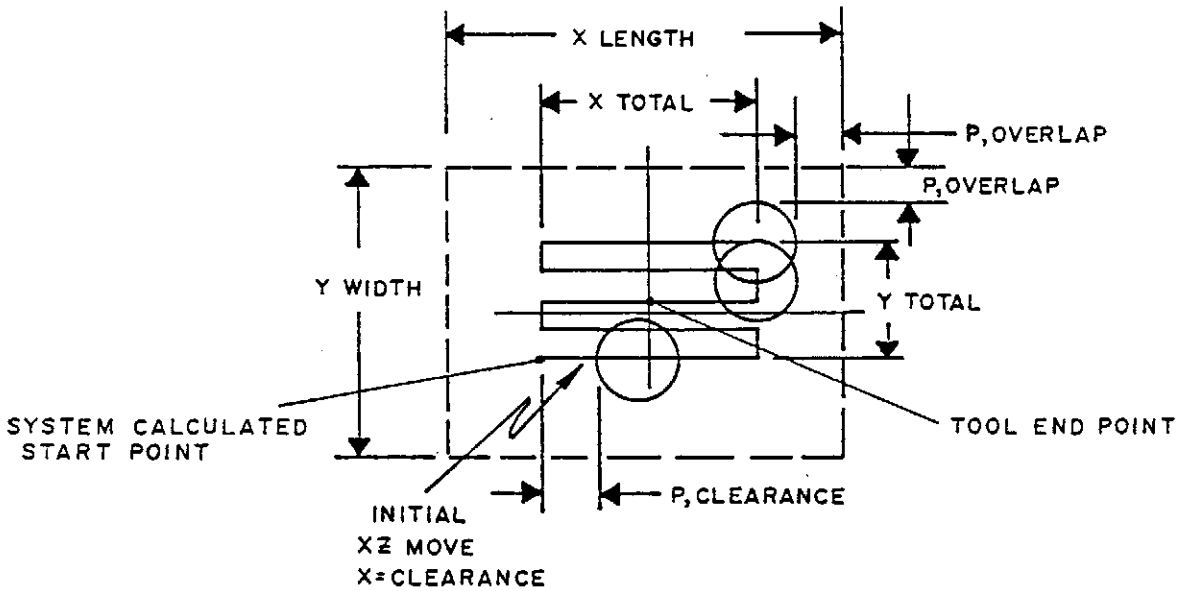
```
T1//.4;     TOOL DIAMETER =.4
N40G173X3.Y4.Z-4.X2.Y1.5Z.52P.25P.2F20.;
```



X Total = X Length + Tool Diameter + (2* overlap)
Y Total = Y Width + (2* overlap) - Tool Diameter

EXAMPLE:

```
T1//.4;     TOOL DIAMETER =.4
N50G174X3.Y4.Z-4.X2.Y1.5Z.52P.25P.2F20.;
```



X Total = Y Length - (tool diameter + (2* overlap)
Y Total = Y Length - (tool diameter + (2* overlap)

6.4.4 Outside/Inside Circle Mill

These cycles mill around the outside of a circular shape (G175) or the inside of a circular shape (G176). The format for this command is:

G175(G176)X___Y___Z___R___Z___Z___P___F___P___ F___F___;

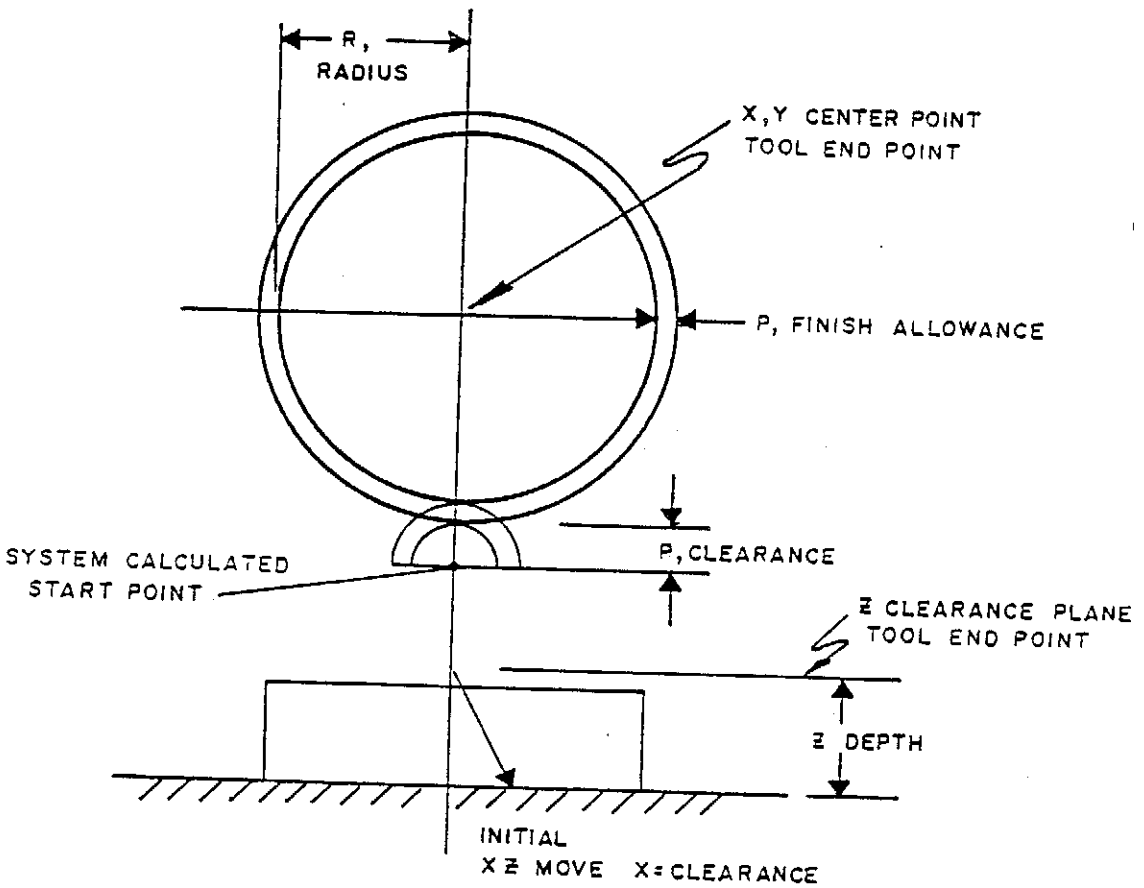
Where X___Y___ is the center point of the circle, Z___ is the clearance plane, R___ is the circle radius, Z___Z___ is the full depth and the step depth, P___P___ is the tool entry clearance and the finish allowance, F___F___F___ is the mill feed, the finish feed and the plunge feed.

NOTE

The sum of the entry clearance and the finish allowance must be smaller than the circle radius.

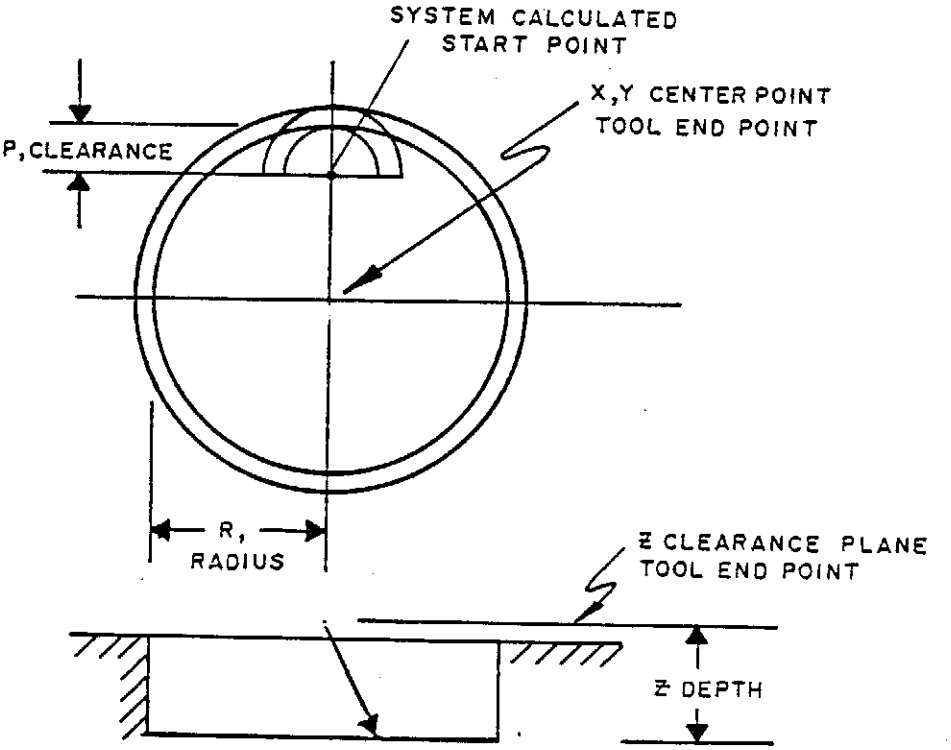
EXAMPLE:

```
T1//.0
N60G175X3.Y4.Z-4.R1.Z.52P.25F20.P.1;
```



EXAMPLE:

```
T1//.0
N70G176X3.Y4.Z-4.R1.Z.5P.25F20.P.1;
```



6.4.5 Slot Mill

This cycle mills a slot at any angle to the X axis. The format for this command is:

G179X___Y___Z___P___P___P___Z___Z___F___F___;

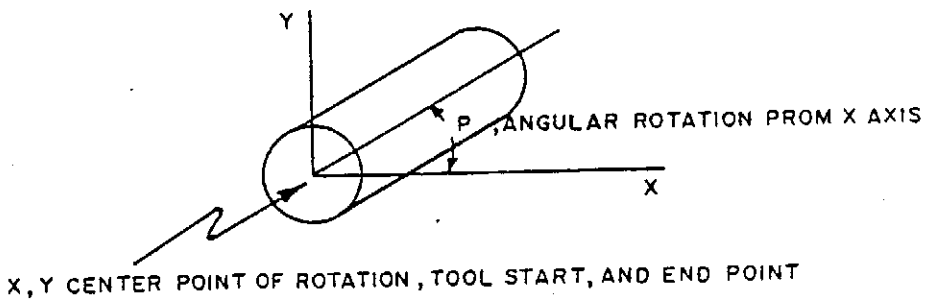
where X___Y___ is the center point of the arc boundry the slot about which rotation is to take place, Z___ is the clearance plane, P___ P___P___ is the out to out length of the slot, the slot diameter and the angular rotation from the X axis, Z___Z___ is the full depth and step depth, F___F___ is the mill feed and the plunge feed.

NOTE

The Slot Mill command cannot be transformed by rotation. If in incremental the system will be set to the absolute positioning mode G90 after execution of a G179 cycle. The next block must contain a G91 if incremental positioning mode is to be maintained.

EXAMPLE:

```
T1//.0
N80G179X3.Y4.Z 4.P1.5P.5P30.Z.52F20.;
```



EXAMPLE:

```
N90G179X3.Y4.Z-4.P1.5P.5P0.Z.52F20.;
```