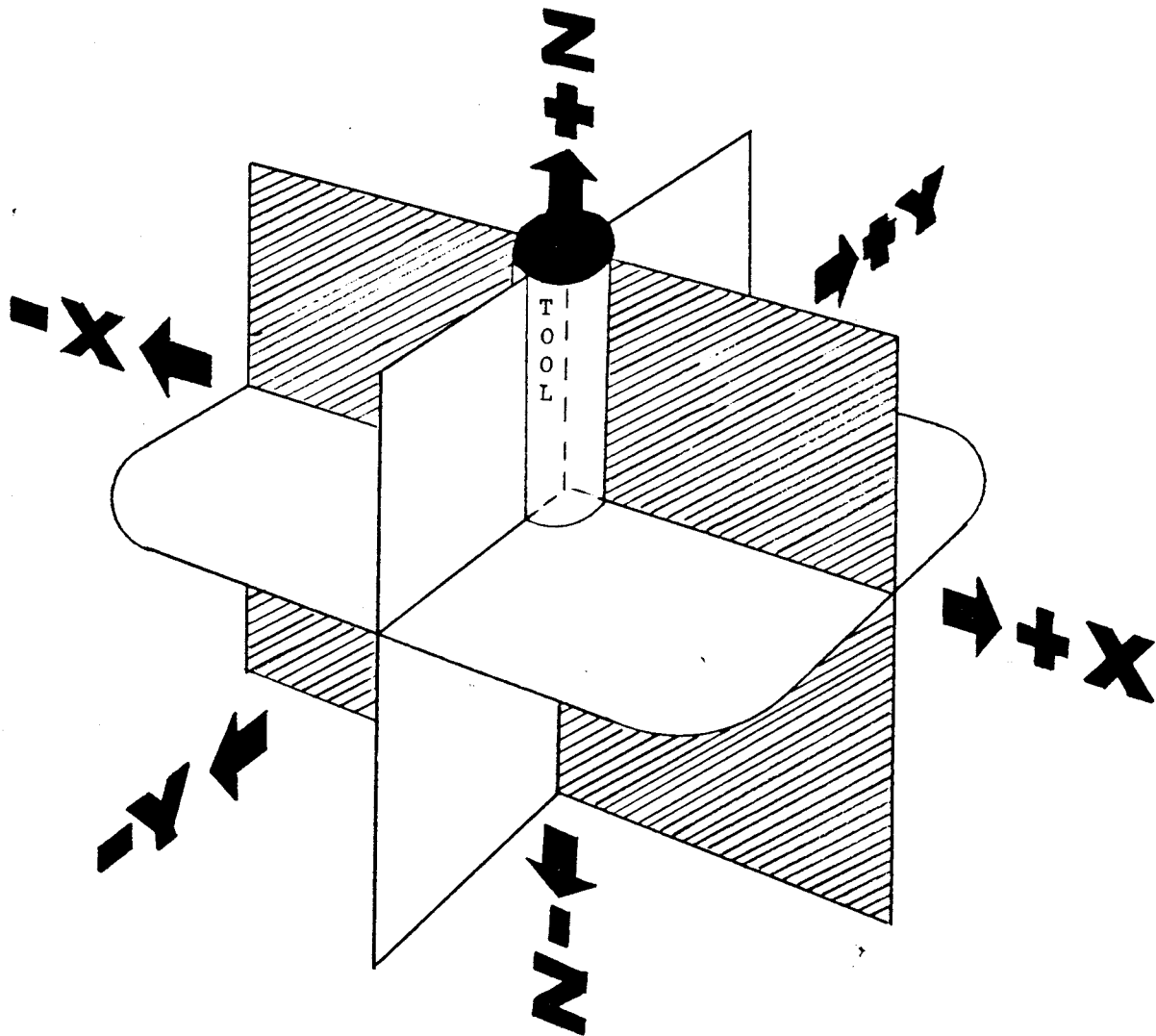


PROGRAMMING AIDS

Bridgeport **TEXTRON**

Bridgeport Machines Division of Textron Inc.

TOOL MOTION



RECTANGULAR COORDINATES

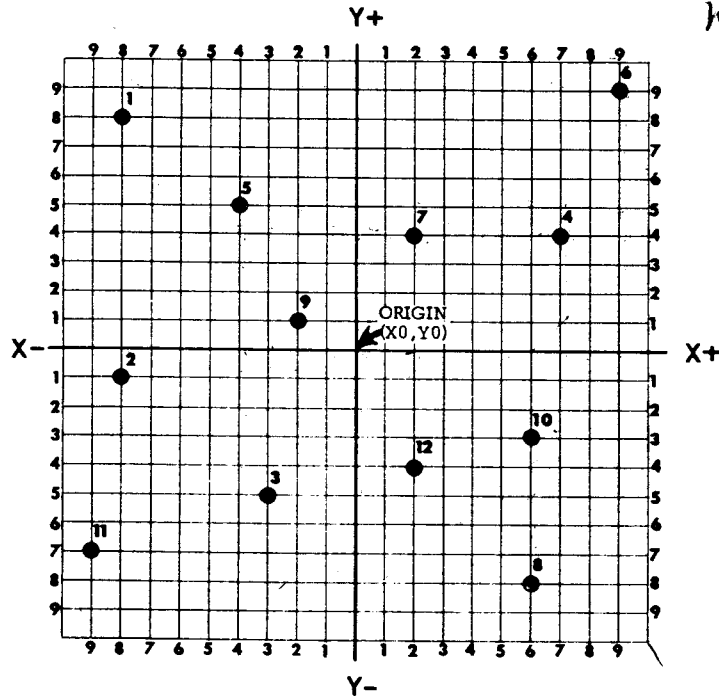
M26 clear point
M22 Rewind

MDI

TLO

AUX

M26



ABSOLUTE

- 1 X -8. Y 8. ✓
- 2 X -8. Y -1. ✓
- 3 X -3. Y -5. ✓
- 4 X 7. Y 4. ✓
- 5 X -4. Y 5. ✓
- 6 X 9. Y 9. ✓
- 7 X 2. Y 4. ✓
- 8 X 6. Y -8. ✓
- 9 X -2. Y 1. ✓

INCREMENTAL

- 6 to 1 X -17. Y -1. ✓
- 5 to 4 X 11 Y -1. ✓
- 8 to 3 X -9. Y -3. ✓
- 2 to 4 X 15 Y 5. ✓
- 4 to 5 X -11 Y 1. ✓
- 5 to 6 X 13. Y 4. ✓
- 12 to 7 X 0. Y 8. ✓
- 10 to 8 X 0 Y -5. ✓
- 4 to 9 X -9. Y -3. ✓

Exercise -- Find the absolute and incremental coordinates of the points listed above.

Absolute System: A numerical control system in which all coordinate locations are programmed from a fixed or absolute zero point. (origin)

Incremental: A numerical control system in which each coordinate location is given in terms of distance and direction along rectangular axes from the previous position and not from a fixed zero location.

16 inch maximum for series of holes in a straight line.

CNC PROGRAM ENTRIES DESCRIPTION
PER EIA-RS - 274C

You can leave this out

*

- / SLASH - Block delete or value separation
- N Sequence Number 1 - 19999 (5 digits)
- G Preparatory functions (sets mode of operation, 2 digits) (3) New with 3 digits
- G00 Rapid traverse (programmed feedrate remains in register)
- G01 Feed in a linear path
- G02 Circular interpolation, CW. Arc
- G03 Circular interpolation, CCW. Arc
- G04 Dwell - will occur with G82 & G89 functions
- G17 X, Y circular interpolation
- G18 X, Z circular interpolation
- G19 Y, Z circular interpolation
- G30 Cancel symmetry
- G31 X symmetry
- G32 Y symmetry
- G40 Cancel compensation
- G41 Cutter compensation left
- G42 Cutter compensation right
- G70 Select Inch dimension system -
- G71 Select Metric dimension system - *Feed Rate will have to be converted to millimeters.*
- G72 Cancel transformation (Rotation, Scaling)
- G73 Transformation (Rotation, Scaling)
- G74 Standard Circular (In effect until changed by G75)
- G75 Multi-Quadrant Circular Interpolation
- G77 Facing cycle
- G78 Pocket milling cycle
- G79 Internal hole mill cycle
- ~~G80~~ Fixed cycle cancel (*NOT NECESSARY*)
- G81 Drill cycle
- G82 Spot facing cycle (with dwell set G04/N) *Feeds in dwells for a moment & Rapids out*
- G83 Deep hole drill cycle, Z, Z, Z, F (Z total depth, 1st peck incr., all other incr.)
- G84 Tapping cycle, (need reversible tapping head)
- G85 Boring cycle, (feed in, feed out) - *USE FOR REAMING*
- G86 Boring cycle, (feed in, manual spindle stop, rapid out)
- G87 Chip breaking cycle, (backs up .05 then returns .05)
- G89 Boring cycle, (feed in, dwell, feed out).
- G90 Absolute input, (in effect until changed by G91)
- G91 Incremental input, (in effect until changed by G90, power off, or reset button pushed)
- G92 Zero Shift *up to Boss 7 Do NOT need with Boss 8 use G97 and G96 to cancel*
- G99 Deceleration override. (over 2.8 I.P.M.)
- X, Y, Z Axis command (.0001 to 999.9999 inches)
- I, J, K Arc center offsets (.0001 to 999.9999 inches)
- M MISCELLANEOUS functions
- M00 Program stop - *STOP and make adjustment.*
- M01 Optional program stop (cont. by oper., OP-STOP ON-OFF switch)
- M02 Rewind
- M06 Tool change *will shut off machine & drive spindle home.*
- M25 Z axis home, (X, Y axis move, if any, will occur after)
- *M51 Indexer
- F Feedrate 3 digits
- S Spindle speed (ref.) 4 digits
- T Tool select, 2 digits 01-24
- A Angle decimal degrees
- R Radius vector distance

*G82) use the G04/
G89) PRIOR to*

G04/1.00 will give you 1 sec. of dwell

*Z depth UNASSIGNED
depth*

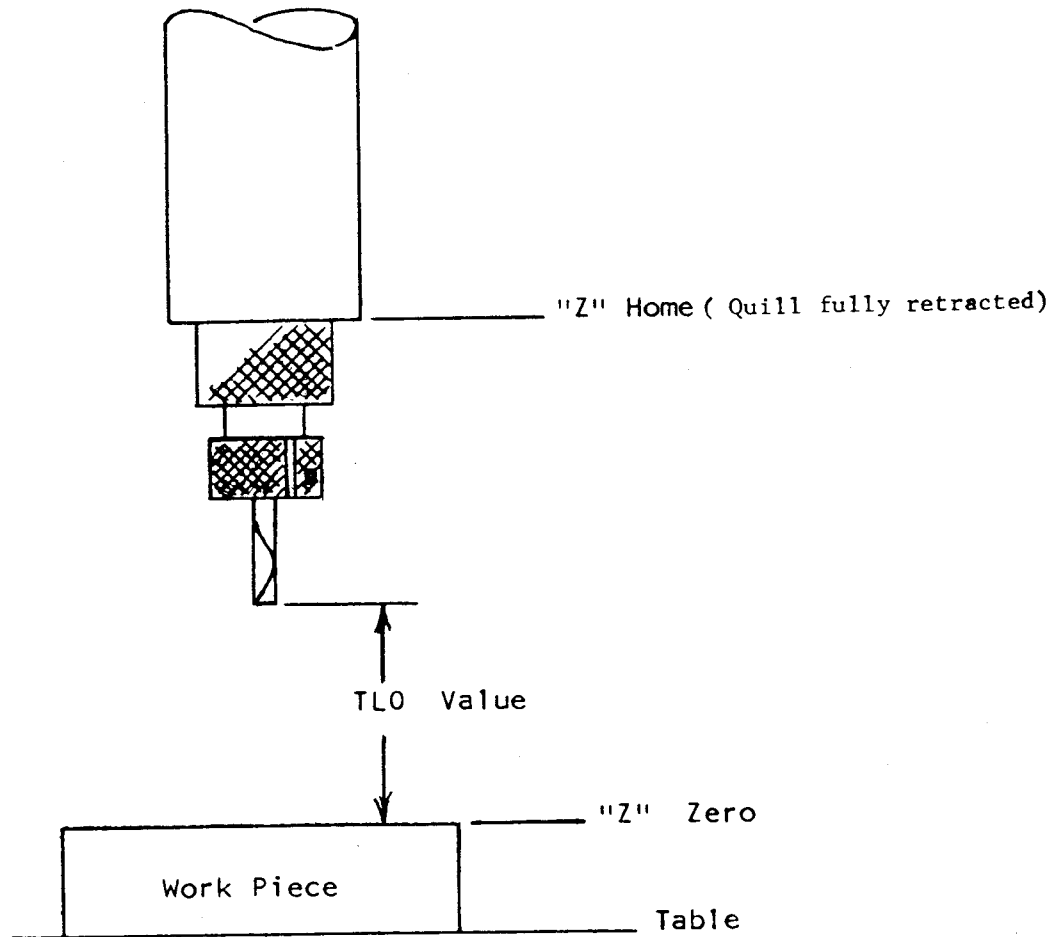
length of Drill point = .3 X Dia of DR.

*Spot facing G04/1.00
has to come before
G82 (spot facing)*

*Optional

G97 How far is new zero from ORIGINAL.
G96 Cancel zero shift will go back to original zero.

TOOL LENGTH OFFSET (TLO)



Tool Length Offset:

TLO is a multi-functional device used to: Simplify "Z" axis programming; reduce tooling set-up procedures; and adjust minor inaccuracies in depth for tight tolerance part control.

Tool Length Offset Value:

The length from the tip of the tool, with the spindle in the home position, to the "Z" zero plane. Wherever the "Z" zero plane has been selected to be.

G92 FLOATING ZERO SHIFT

G92 is a preparatory function that provides the ability to preset the X,Y,Z absolute position registers to any desired dimension.

G92 is frequently used thru MDI as a set-up aid as in example #1 below. It is important to remember, however, that when programmed for multiple parts or secondary operations as in example #2 below, you must program the last G92 to return to your original origin.

NOTE The following three examples all require the machine be set in G90 Absolute Coordinates before invoking the G92 Preset command.

#1 MDI (Set-Up)

Operations:

Example #1A

Touch off the workpiece in the "X" axis. (P1)

Next, in MDI enter:
(G90) G92x-2.25

Touch off the workpiece in the "Y" axis (P2)

Next in MDI enter
G92Y2.25

You have told the machine where the tool is physically from the origin; thereby establishing your part origin (X0, Y0).

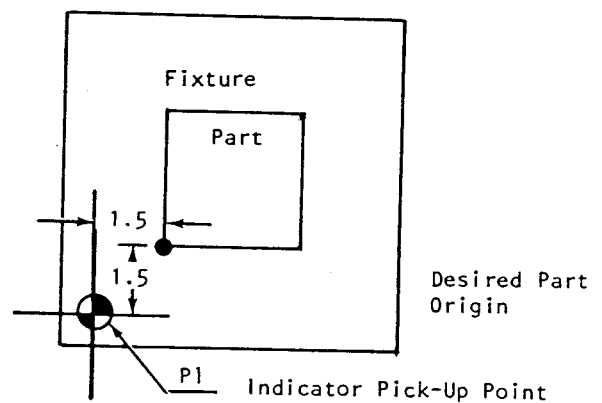
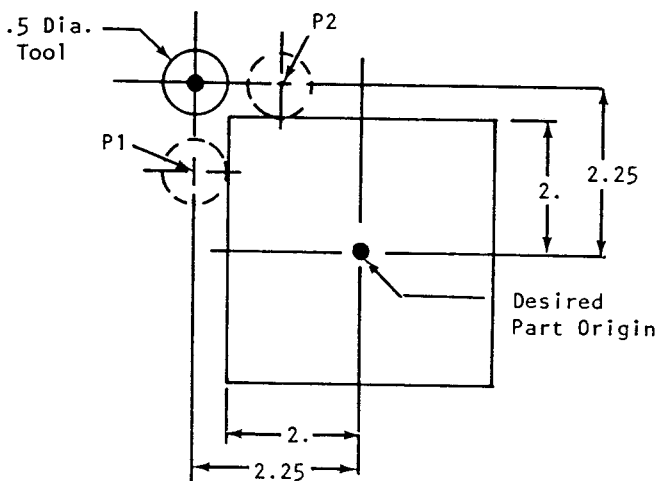
Example #1B

Position centerline of spindle over P1.

Next, In MDI mode, enter:
(G90)G92X-1.5Y-1.5

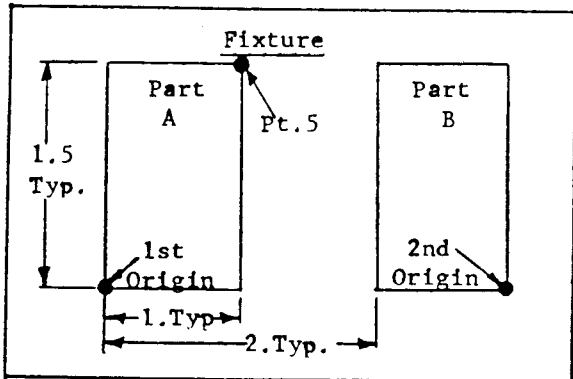
Results:

You have told the machine where the tool is physically from the origin, thereby establishing your Part origin (X0, Y0).



G92 Floating Zero Shifts Continued

#2 Programming G92



Example #2

TOOL POSITION	DESIRED ORIGIN	G92 PRESET BLOCK
ORIGIN 1	ORIGIN 2	G92X-3. (Y0)
POINT 5	ORIGIN 2	G92X-2.Y1.5
ORIGIN 2	ORIGIN 1	G92X3. (Y0)

**** NOTE: INFORMATION IN () OPTIONAL

G91G92 Incremental Zero Shift

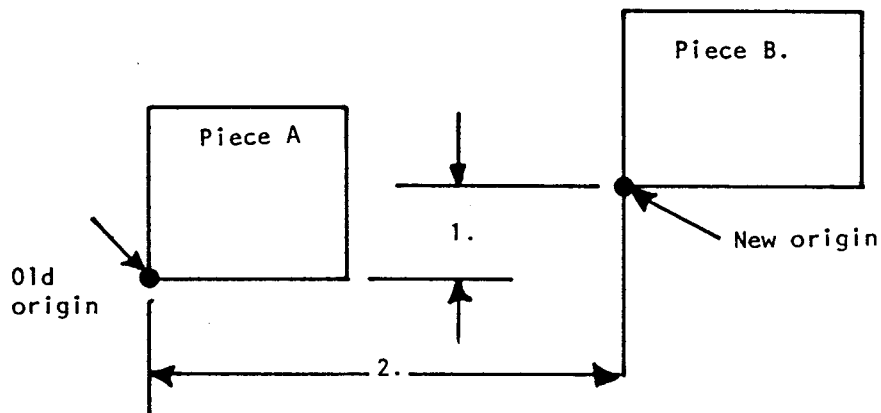
It is often desirable to be able to shift the part "Zero" location without the necessity of knowing the exact current location of the tool. This particular feature can be accomplished by using a G91G92 incremental zero shift.

The example shown below will illustrate this method of programming. To shift the part "Zero" location from piece "A" to piece "B", the following block must be programmed:

```
N200G91G92X-2.Y-1.
```

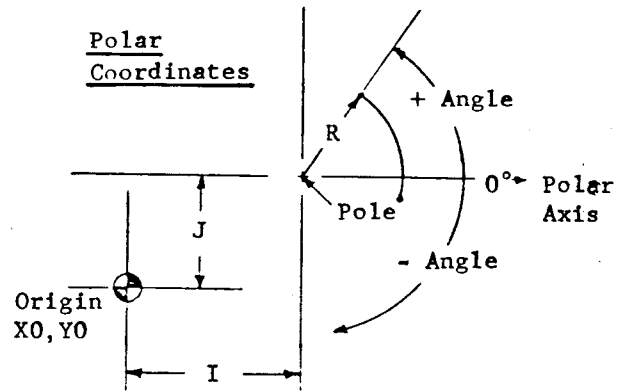
You will note, the (X,Y) coordinate values that are programmed with a G91G92 incremental shift, are the signed incremental values from the new origin to the old origin.

** NOTE -- The tools current position at the time the G91G92 block is executed is not relevant.



POLAR COORDINATES

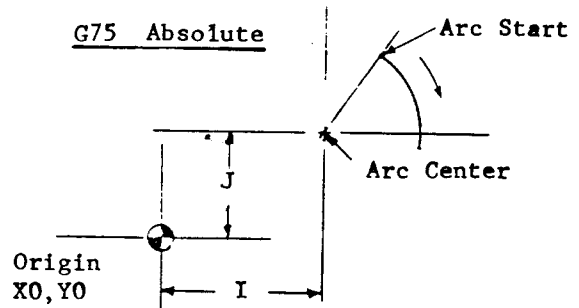
Both in point to point operations, (drilled holes on a bolt circle) and in continuous path milling, "I" is the absolute distance along the "X" axis from the chosen origin (X0, Y0) to the center, or POLE which you are revolving around.



Multi-Quadrant Circular Interpolation

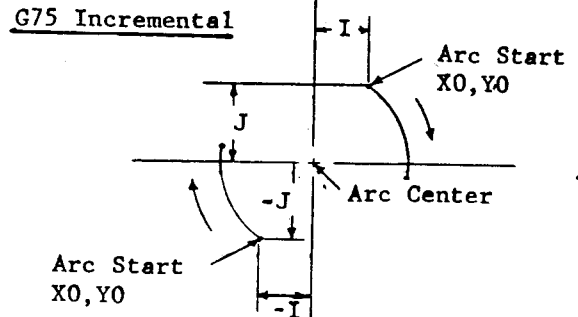
G75 ABSOLUTE

"I" is the Absolute distance along the "X" axis from a chosen origin (X0, Y0) to the arc center.
 "J" is the Absolute distance along the "Y" axis from a chosen origin (X0, Y0) to the arc center.



G75 INCREMENTAL

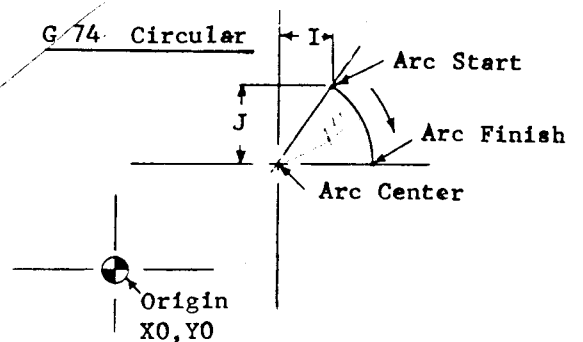
"I" is the SIGNED distance from the arc center to the start point along the "X" axis.
 "J" is the SIGNED distance from the arc center to the start point along the "Y" axis.



Standard Circular Interpolation

G74 (Turn on mode)

"I" is the Incremental UNSIGNED distance along the "X" axis from the center of the arc to the start point of the arc.
 "J" is the Incremental UNSIGNED distance along the "Y" axis from the center of the arc to the start point of the arc.
 "Start point" is the point where the arc begins.



NOTE** I0 and J0 must always be programmed.....

THE FOLLOWING RULES MUST BE FOLLOWED WHEN STARTING COMPENSATION:

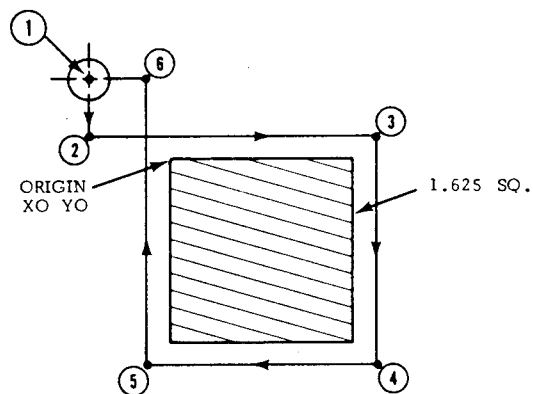
1) When starting Cutter Diameter Compensation G41 or G42 must be programmed in a block containing X and Y data. The system requires the XY coordinates in order to establish a position to start the compensation. The G41 or G42 and X and Y data may be programmed in a Rapid Positioning block, a linear feed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

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

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

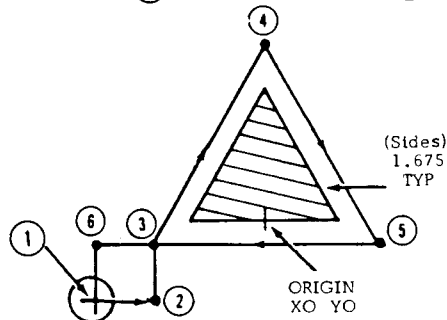
The three examples below illustrate a correct starting procedure:

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

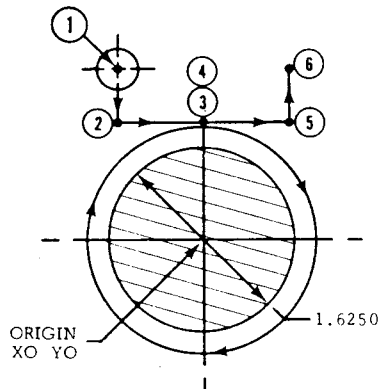


3/8 Diameter End Mill (all examples)

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



```
%N1G0G90X-1.Y1.T1M6
N5G41X-1.6623Y-.6875Z-.1
N10G1X-1.1623F200
N15Y-.1875
N20X0Y1.8256
N25X1.1623Y-.1875
N30X-1.6623
N35G40Y-.6875
N40G0X-1.Y1.M2
E
```



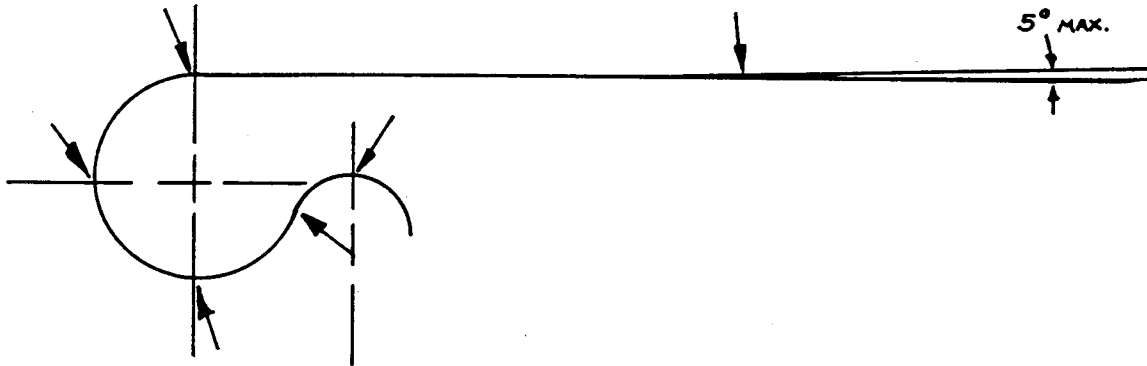
```
%N1G0G90X-1.Y1.T1M6
.N5G75
N10G41X-.75Y1.5Z-.1
N15G1Y1.F100
N20X0
N25G2X0Y1.I0J0
N30G1X.75
N35G0G40Y1.
N40X-1.M2
E
```

FEED DECELERATION ELIMINATION

Use G99 in every line where you wish to eliminate deceleration at the end of the travel of that line.

When using G99 the change of direction must not be more than 5° of being tangent.

Arrows in drawing below show where feed deceleration override could be. (Example assumes G74 circular)



NOTE Below 2.8 ipm is not effective
Over 8.0 ipm some deceleration will still occur

PRORATED FEEDRATES FOR INSIDE OR OUTSIDE RADI

OUTSIDE FORMULA: $\frac{(\text{PART RADIUS}) + (\text{CUTTER RADIUS})}{(\text{PART RADIUS})} = \text{FACTOR}$

(FACTOR) x (STRAIGHT LINE FEEDRATE) = NEW FEEDRATE

INSIDE FORMULA: $\frac{(\text{PART RADIUS}) - (\text{CUTTER RADIUS})}{(\text{PART RADIUS})} = \text{FACTOR}$

(FACTOR) x (STRAIGHT LINE FEEDRATE) = NEW FEEDRATE

BASIC PROGRAMMING EXAMPLES

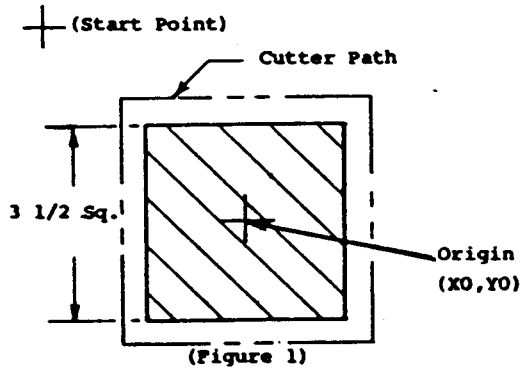
Three simple examples follow illustrating how to program cutter compensation.

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

Program "A"
Program Cutter Path:
No Cutter Compensation

```

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



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

```

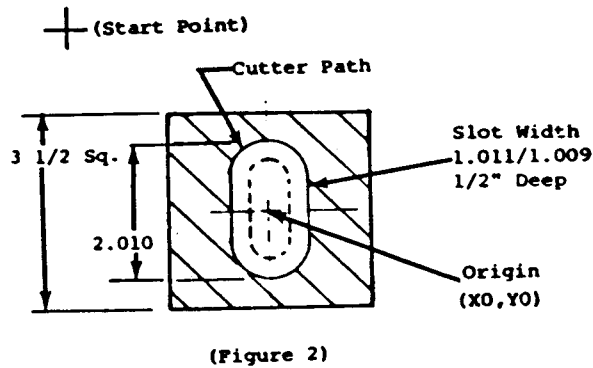
%N1G0G90X-3.Y3.T1M6
N5G41X-2.26Y2.5Z-.1
N10G1Y2.F100
N15X2.
N20Y-2.
N25X-2.
N30Y2.26
N35G0G40X-2.5Z.05
N40X-3.Y3.M2
E
    
```

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

Program "A"
Program Cutter Path:
No Cutter Compensation

```

%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N12G1Y-.5Z-.1F50
N13Y.5F100
N15X.255
N20G3X-.255Y.510J.5F50
N25G1Y-.5F100
N30G3X.255Y-.510J-.5F50
N35G1Y.5F100
N40G0X-3.Y3.M2
E
    
```

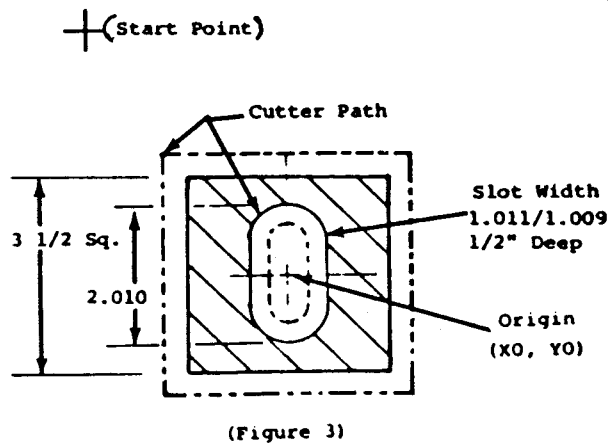


Program "B"
Program Cutter Path:
Cutter Comp.-invoked (N15)
revoked (N45)

```

%N1G0G90X-3.Y3.T1M6
.N5G75
N10X0Y0Z.05
N12G1Y-.5Z-.1F50
N15G41X0Y.5F100
N20X.255
N25G3X-.255Y.510J.5F50
N30G1Y-.5F100
N35G3X.255Y-.510J-.5F50
N40G1Y.5F100
N45G0G40X0Y0Z.05
N50G0X-3.Y3.M2
E
    
```

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



```

%N1G0G90X-3.Y3.T1M6
.N5G75
N10G41X-2.26Y2.5Z-.1
N15G1Y2.F100
N20X2.
N25Y-2.
N30X-2.
N35Y2.26
N40G0G40X0Y0Z.05
N42G1Y-.5Z-.1F50
N45G41X0Y.5
N50X.255
N55G3X-.255Y.510J.5F5
N60G1Y-.5F100
N65G3X.255Y-.510J-.5F
N70G1Y.5F100
N75G0G40X0Y0Z.05
N80X-3.Y3.M2
E
    
```

EDITOR SUMMARY

- B - Set line pointer after the last line of program
- C/old text/new text - Change character or character string
- D - Delete current line
- Dn - Delete n lines starting at current line (n=0-999)
- F text string - Search for first occurrence of text string starting next line
- * I - Insert text in front of current line (Text Mode - CR, CR to Exit)
- * K - (Kill) Deletes entire program and enters next mode
- L - List entire text
- Ln - List entire text and output n inches of leader with % and E for n 3
- N - Move line pointer forward one line
- Nn - Move line pointer forward n lines (n=2 to 999)
- N-n - Move line pointer backward n lines (n=1 to -999)
- P - Print current line
- Pn - Print n lines starting at current line (Note: Line pointer does not advance)
- R text - Replace current line with new text
- T - Set line pointer to top of program
- Ctrl O - Abort listing or printout
- Ctrl Q - Input tape text (Can only be used in conjunction with I or K)
- Ctrl R - Print current line before storing (NO CARRIAGE RETURN)
- Ctrl U - Delete current line being typed
- Ctrl Z - Exit the editor
- CR - Advance line pointer one line and print (Use LF - BOSS 4 only)
- Rubout - Erase last character typed (consecutive rubouts may be used)

* Note: Text Mode Insertion Characters
I add K are inserted by editor keyboard and exited by cr cr

Note: Ctrl Q will insert data thru editor tape reader only.
Keyboard is locked out.....

CNC & BTC I MILL MAINTENANCE

LUBRICATION - BTC I & CNC I & II

1. Reservoir on left side of column.

Capacity: Cap. 946cc = Aprox. 80 - 90 Hrs. spindle run time

Recommended Lubricants:

- A) Sunoco - Way lube #1180 oil - Part # 1523132
- B) Exxon - Febis K53
- C) Mobil - Vactra oil #2
- D) American - Waytac oil #31
- E) Texaco - Way Lubricant D
- F) Shell - Tonna 33

Instructions:

- A) Float switch prevents restart of spindle if oil is low...Maintain proper level in reservoir.
- B) Pull pump by hand at start of day.

BTC I Arm Reservoir: - Mobile velocite #6 or equivalent.

Note

Spindle bearings are packed with lifetime grease, therefore they require no maintenance.

FILTERS

- 1. Logic Console: One disposable type Part #1506551. This filter is located on the main logic cabinet (R.H. side of machine). Change regularly according to your shop conditions.
CNC/BTC I
- 2. Nema Box: One disposable type Part #1507452. This filter is located on the bottom R.H. side of the nema electrical box at the back of the machine. Change as above.
CNC/BTC I
- 3. BTC I only: Spindle enclosure - Pn 1504114

Pneumatics CNC & BTC I

- 1. Main regulator: Micro-fog Lubricator-
Lubrication: Cappella "A" Wax Free Oil Part #2652006
Air Pressure: Maintain regulator set to 80 psi - left side of column.
BTC I Requirements: - 4 CFM @ 80 PSI

Tape Reader

Run new tape back and forth, then clean head.
Clean lamp and lens.
Clean read head and sprocket with brush. Recommended cleaners may be used...Miller Stephenson M5-200 tape head cleaner.
Isopropyl Alcohol may be substituted.
System is sensitive.....Use caution.

CLEAN SHOP.....CLEAN MACHINES = Minimum Down Time.....