CHAPTER 7

CUTTER DIAMETER COMPENSATION

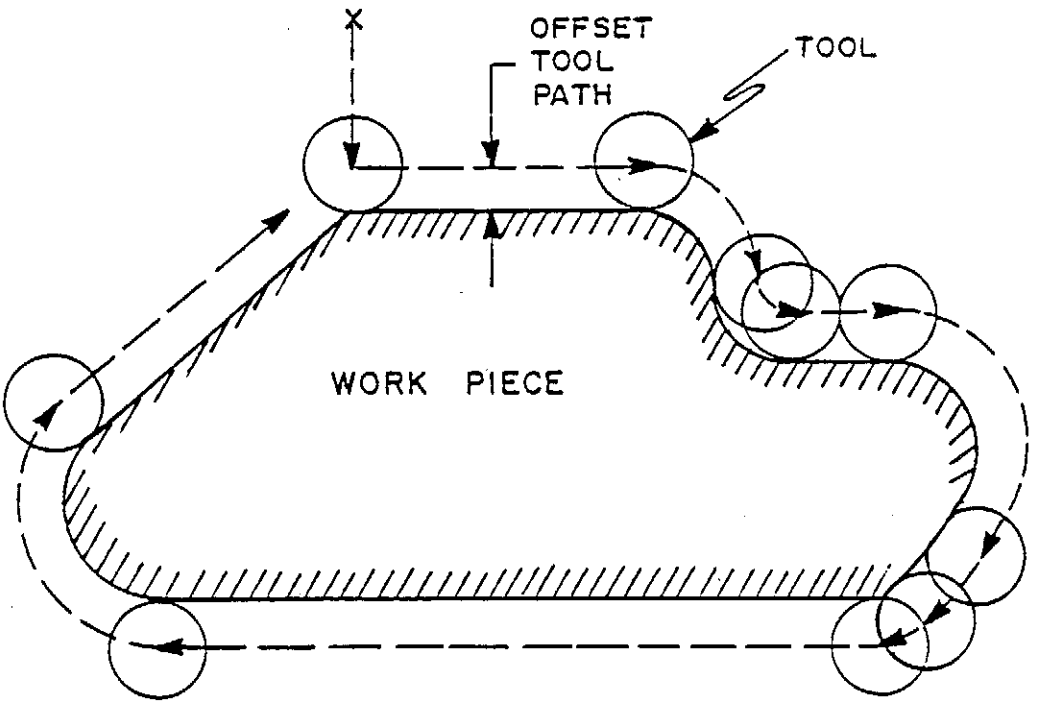
7.1 OVERVIEW

In order to machine the workpiece shown in the figure below the programmer must allow for the cutter diameter and calculate the offset path the tool must make to generate the desired shape. The cutter diameter compensation function provides means in the control for computing the offset path. This feature can be used three ways:

1. The workpiece shape can be programmed directly by the programmer. The cutter diameter to be used can be measured and in the Setup mode can be entered into the control. The control will automatically compute the offset tool path to generate the workpiece surface.

2. The input data to the control may already be the offset cutter path. In this case cutter diameter compensation may be used to compensate for the difference between the cutter diameter value assumed by the part programmer and the actual value. This cutter diameter compensation enables a given part program to be executed with cutters of different diameters and will produce the same part.

3. Two or more different diameter compensation values can be programmed for the same tool to take care of roughing, semi-finish and finish passes.



7.2 CUTTER DIAMETER DATA ENTRY

The following methods can be used for setting the cutter diameter value into the control:

1. Enter cutter diameter data via SETUP.

   Use the TOOL key to scroll through the tool data table, from tool #1 through #24. ENTER loads the numeric data keyed in and displayed on the screen (either T, TLO or DIA) into the system.

2. Enter cutter diameter data via part program text. The format is:

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

3. Maximum cutter compensation value is:

   + - 3.2768 inches
   (+ - 83.321 mm)

EXAMPLE:

```
T5//-.5;      For tool 5, DIA=-.5
T4/.75/.2;    For tool 4, TLO=.75,DIA=.2
```

If the program input data is already the offset cutter path, the input value is the actual cutter diameter minus the programmed cutter diameter. A positive cutter diameter value designates an oversize cutter, a negative cutter diameter value designates an undersized cutter.

The cutter diameter compensation value is activated when a G41 or G42 command is read. The tool number that selects the compensation value must be entered prior to the initiation of cutter compensation.

7.3 PLANE SELECTION

Compensation is only active in the XY plane. The system must be in the G17 mode.

Rules for Z moves with cutter compensation turned on:

1. A Z move may be programmed on the same block as a linear X, Y move. The Z move will not be compensated.

2. A Z move may be programmed without a linear X, Y move. However, this may not occur in the first three lines after cutter compensation is turned on, or in the last three lines before cutter compensation is turned off. Additionally if multiple Z only moves are used they must be separated by at least two blocks containing XY moves.

7.4 CUTTER DIAMETER COMPENSATION COMMANDS

The following commands are:

G40 Cutter compensation off
G41 Cutter compensation on, tool left of part
G42 Cutter compensation on, tool right of part

NOTE

Even though G41 or G42 is programmed, cutter compensation will not be turned on unless the current tool diameter is a value other than zero.

The system must be in either the rapid traverse (G0) or linear (G1) mode when cutter compensation is turned on. Tool left or tool right with respect to the part is detemined by looking from behind the tool in the direction of motion.

With standard right hand milling cutters:

G41 (Climb Milling)
G42 (Conventional Milling)

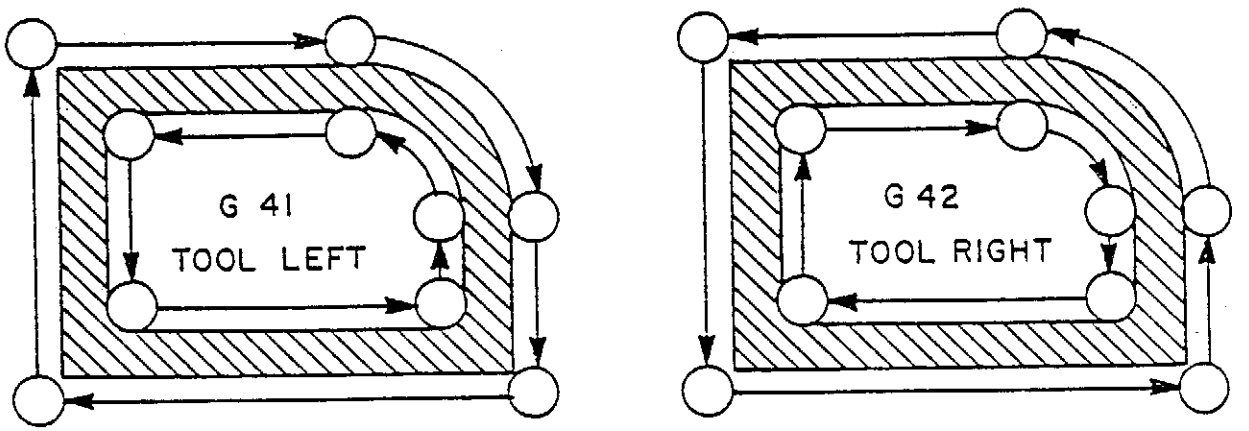


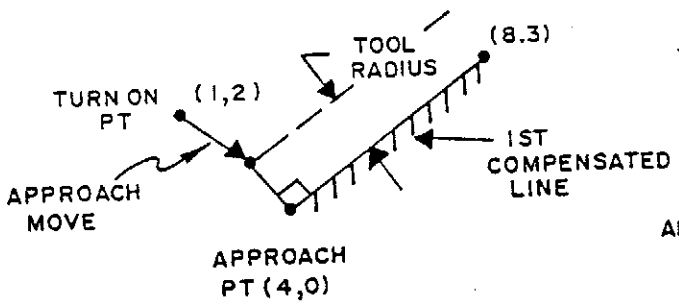
Figure 7-1: Programming Commands for Cutter Compensation

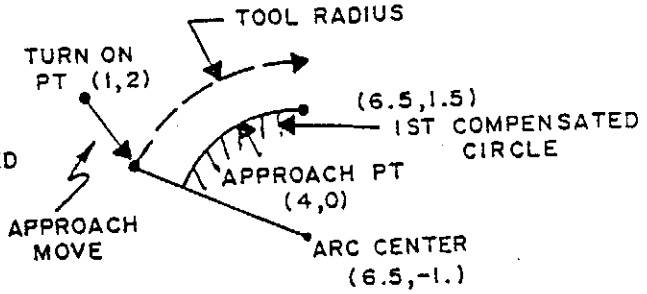
7.5 ENTRY INTO CUTTER DIAMETER COMPENSATION

The data block containing the G41 or G42 move is non-compensated. The tool will move directly to this point. The following sequence of commands must occur to turn on cutter compensation.

G41(G42)X___Y___;	TURN ON POINT
(G41)X___Y___;	APPROACH POINT
(G1,G2,G3)X___Y___....;	1ST COMPENSATED PATH

The approach point must be defined by a linear move. Before the approach move is made, the next input block will be read containing the definition of the first line or circle to be compensated. The approach move will then be made normal to the first line or circle to be compensated through the approach point at a distance equal to the stored tool radius.






```
G0G41X1.Y2.
G1X4.Y0.F20.
X8.Y3.
```

```
G0G41X1.Y2.
G1X4.Y0.F20.
G2X6.5Y1.5I6.5J-1.
```

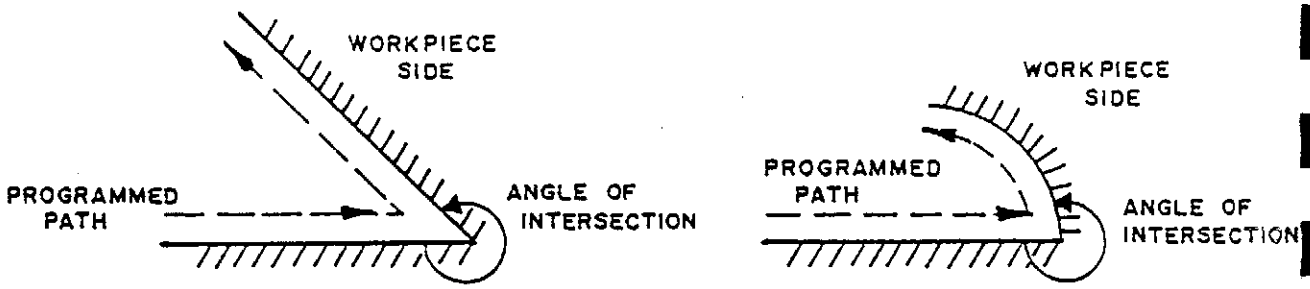
7.6 BLOCK INTERACTIONS

Cutter compensation handles the following types of interactions:

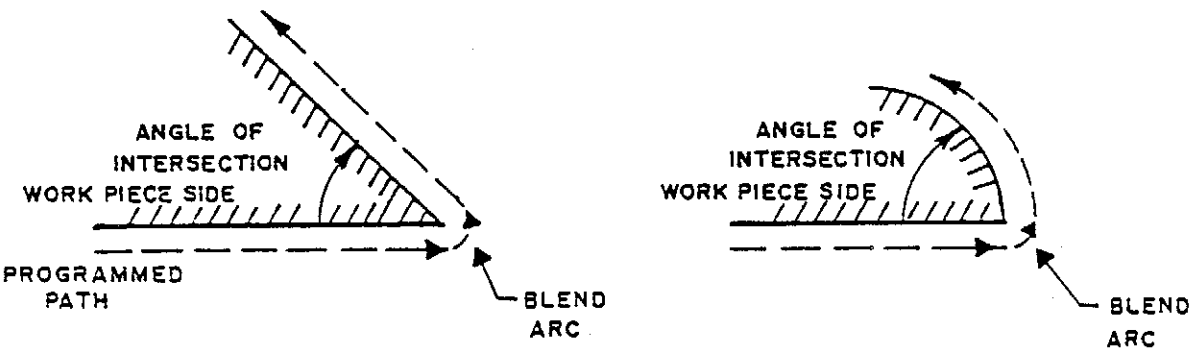
MOVE TO BE COMPENSATED	NEXT MOVE
Line	Line
Line	Circle
Circle	Line
Circle	Circle

Since the calculation of offset path for a particular move also depends on what the next move is, cutter compensation provides "look-ahead" ability, during cutter compensation 2 data blocks are read in advance normally. Thus inside the CNC there are three blocks, the block under execution, and the next two blocks which are used to calculate the next offset path.

The angle of intersection created by two blocks of move commands as measured on the workpiece side create an "inside-corner" when the angle is over 180 degrees, an "outside-corner" when the angle is less than 180 degrees or a "tangency" when the angle is equal to 180 degrees.



Inside corner (Angle of intersection greater than 180)



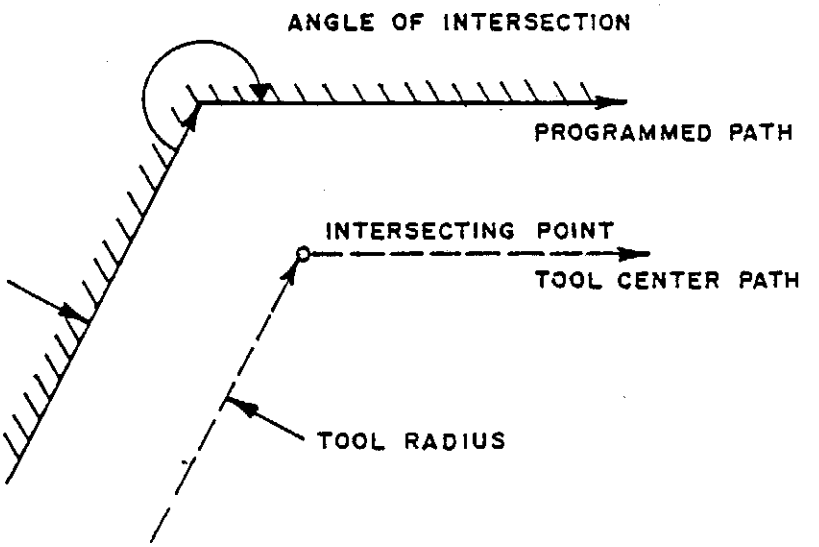
Outside corner (Angle of intersection less than 180)

Note that if an outside corner occurs an arc is automatically blended through the point of intersection tangent to the two paragrammed paths.

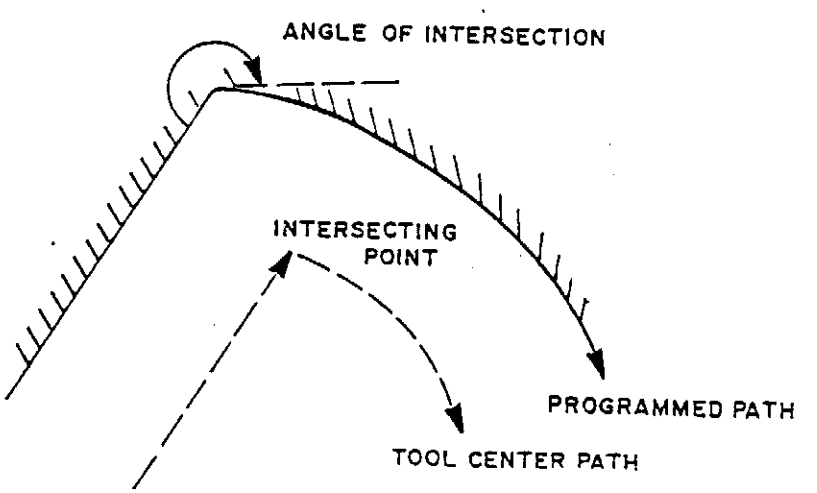
Examples of compensated cutter paths:

INSIDE CORNERS:

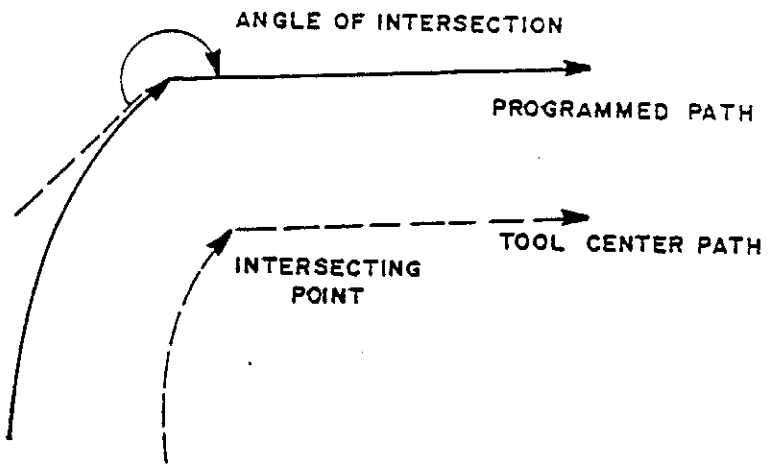
LINE-LINE



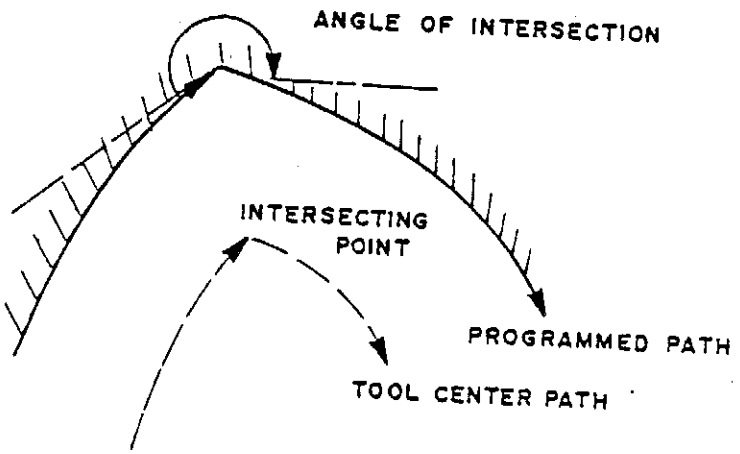
LINE-CIRCLE



CIRCLE-LINE



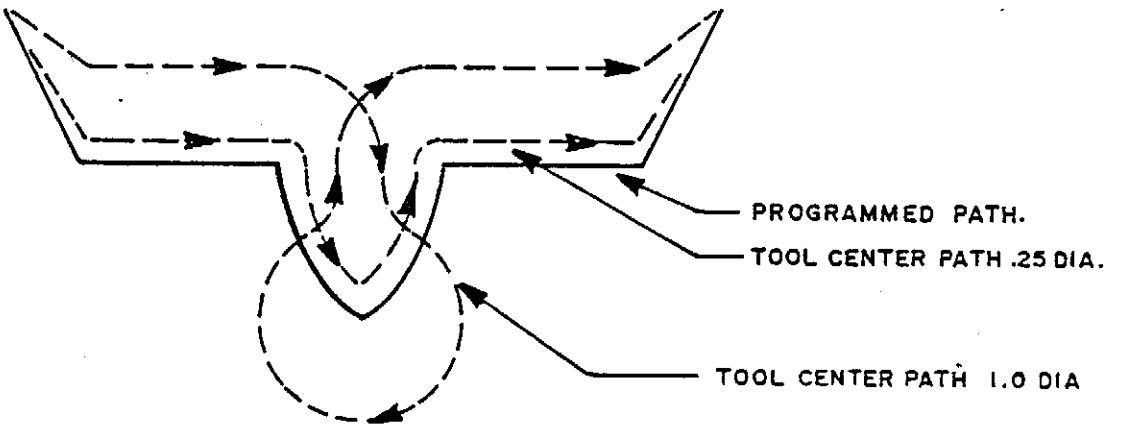
CIRCLE-CIRCLE



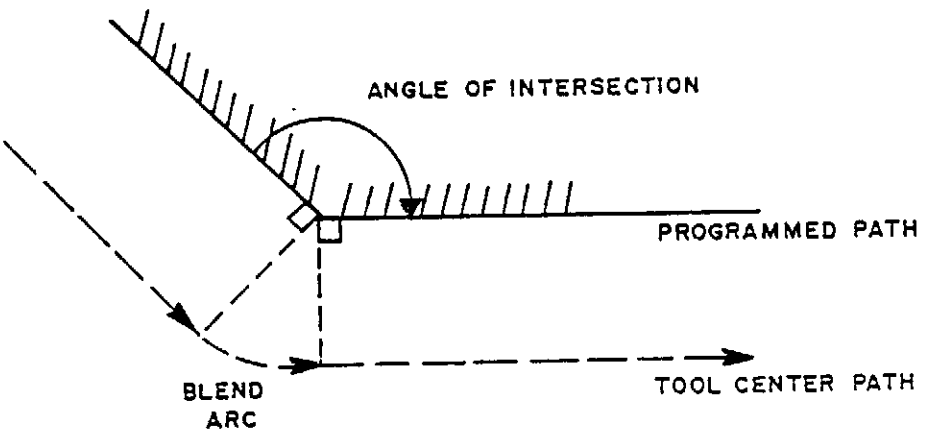
NOTE

If the part program calls for cutting a concave or notch-like feature in the part, the cutter diameter must be no greater than the diameter or width of the feature to be cut. If the cutter diameter is larger than the width of the feature to be cut, gouging will occur.

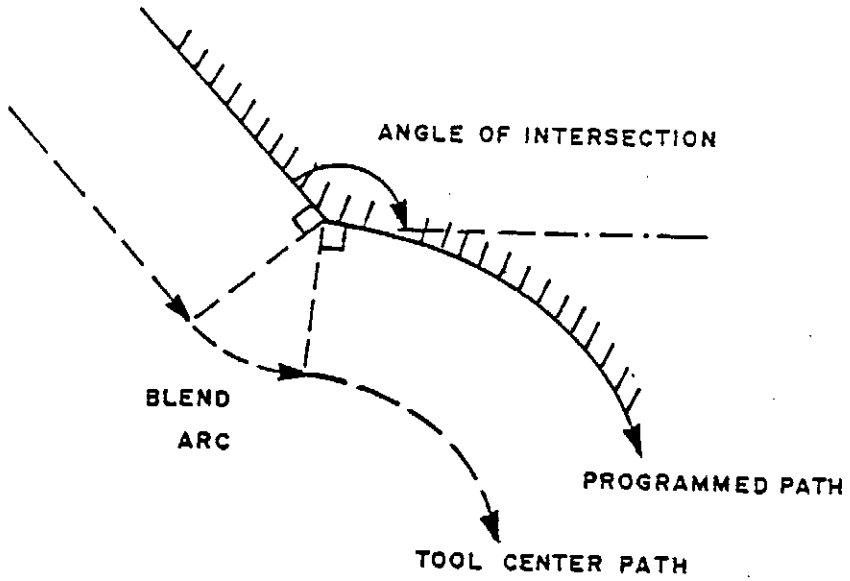
```
T1
G0G41X-.4Y.8Z-.1
G1X0Y0F20.
X1.
G3X1.4Y-.8I2.J0
X1.8Y0I.8J0
G1X2.8
G40X3.2Y.8
```



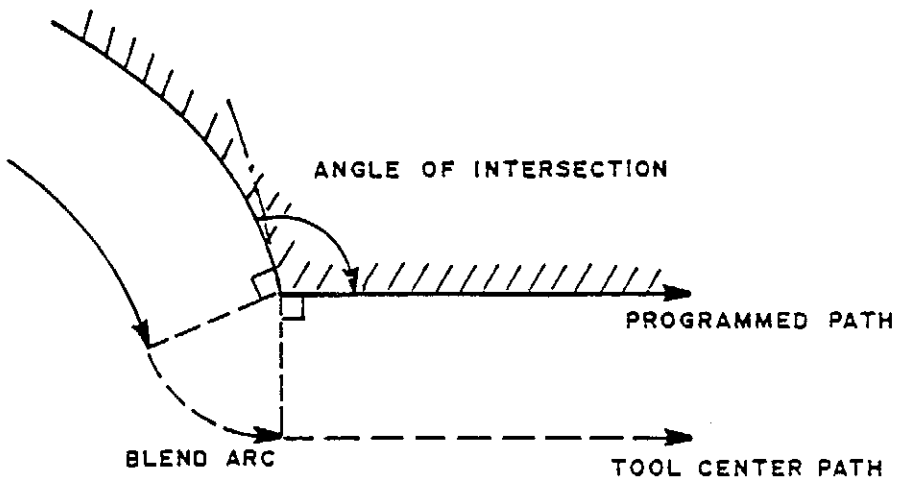
OUTSIDE CORNERS:



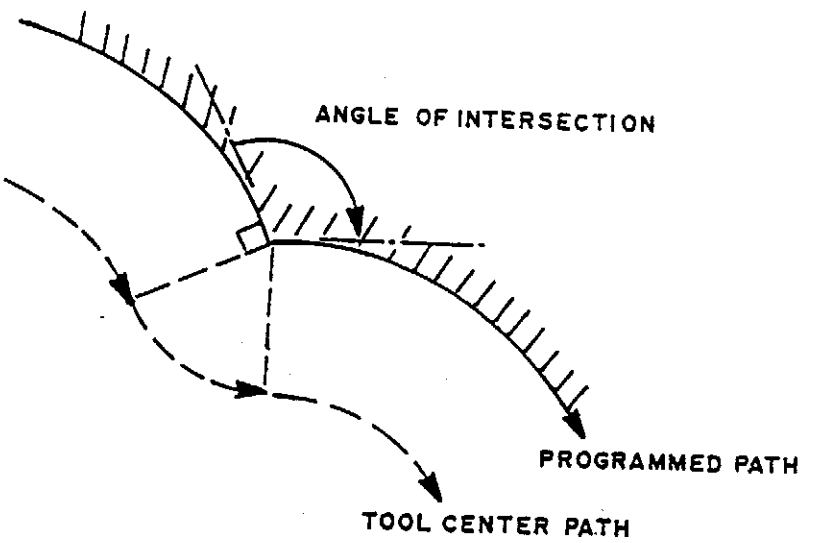
LINE-CIRCLE



CIRCLE-LINE

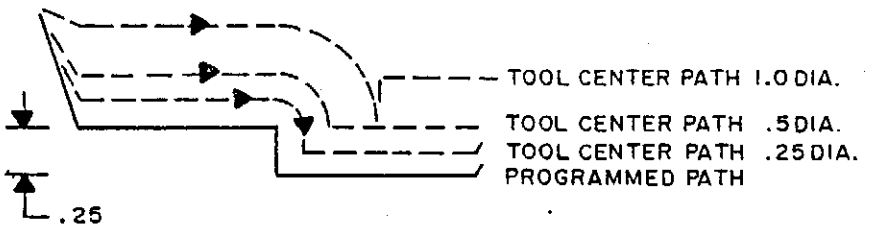


CIRCLE-CIRCLE



NOTE

If the part program calls for making a small step less than the cutter radius, gouging will occur.



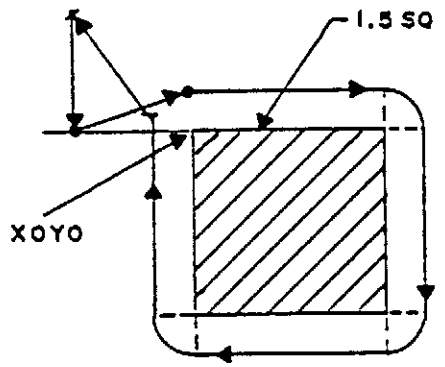

```
T1
G0G41X-.2Y.6Z-.1
G1X0Y0F20.
X1.0
Y-.25
X2.0
G40X2.2Y.6
```

7.7 EXIT FROM CUTTER DIAMETER COMPENSATION

Cutter compensation is turned off by a block containing a G40 command. The tool will move directly to this point. The following sequence of events must occur to turn off cutter compensation.

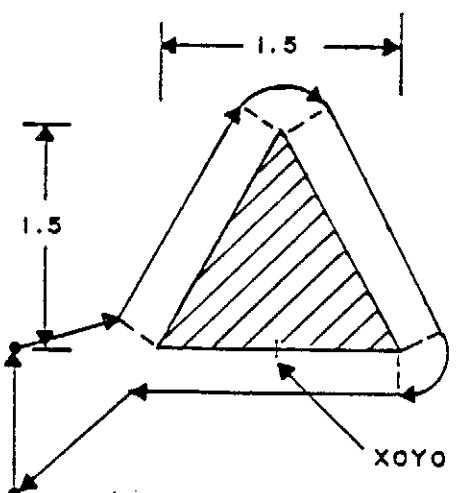
```
(G1,G2,G3)X___Y___....;      LAST COMPENSATED PATH
G40X___Y___;                 TURN OFF POINT
```

The exit cutter compensation move will be through the endpoint and normal to the last compensated path at a distance equal to the stored tool radius.

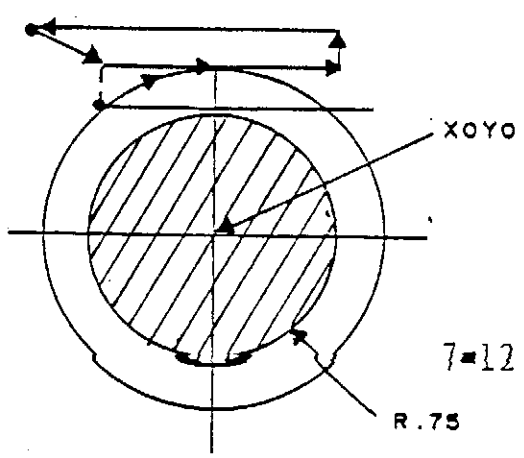


TOOL .5 DIA

```
N10G0G90X-1.Y1.T1M6
N20G41Y0Z0
N21G1X0Y0F20.
N22X1.5
N23Y-1.5
N24X0
N25Y.1
N26G0G40X-1.Y1.
N30T2M6
```




```
N10G0G90X-1.75Y-1.T1M6
N20G41Y0Z0
N21G1X-.75F20.
N22X0Y1.5
N23X.75Y0
N24X-.85
N25G0G40X-1.75Y-1.
N30T2M6
```




```
N10G0G90X-1.25Y1.25T1M6
N20G41Z0
N21G1X-.5Y.75F20.
N22X0
N23G2X0Y.75IOJO
N24G1X.75
N25G0G40X-1.25Y1.25
N30T2M6
```

FACTORS AFFECTING USE OF CUTTER COMPENSATION

1. The tool to part relationship (G41,G42) cannot be changed unless compensation is turned off (G40 mode). Also the amount of compensation cannot be changed while in G41 or G42.

2. In compensating for cutter diameter a G45 command can be used to adjust feedrates in accordance with the difference in raduis between the part surface and the tool path. The actual feedrate = input feedrate multiplied by tool path radius/part radius. For outside circles the feedrate will be increased, for inside circles the feedrate will be decreased (refer to Section 8.5).

3. A data block without a tool movement cannot be commanded with compensation turned on.

4. With cutter compensation in effect, some dwell will occur for blocks that are executed in less than .25 seconds.

5. Colinear moves cannot be programmed.

6. The minimum number of part program blocks that can be used in compensation is 5. The first block contains a G41 or G42, the next three blocks are compensated data points, the fifth block contains a G40 which turns compensation off.

7. Z axis moves will not be compensated. Z motion must not be programmed in either the first or last three compensated blocks. Multiple Z moves must be separated by at least two blocks of XY data.

8. M (Auxiliary function) codes cannot be used from the block after cutter compensation is turned on until the block after cutter compensation is turned off.

9. G99, deceleration override, is automatically generated when successive motion is tangential while cutter compensation is active. Any G99's input into the part program text are stripped out during cutter compensation and inserted only when successive moves are tangential.