COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece

Dr. Dimitris Mourtzis

Associate professor

Patras, 2017

Chapter 11:
Cutter Diameter Compensation

Table of Contents

Objectives of Chapter 11

- Define **cutter diameter compensation**
- Describe **ramp off** moves and explain their importance
- List the **precautions** necessary when using cutter diameter compensation
- Write programs in word address that **utilize cutter diameter compensation**

Definitions and Codes

Programs presented in previous chapters required an **allowance** for the cutter radius in the programmed coordinates

- Most CNC machines have a built-in feature called **cutter diameter compensation (cutter comp)** that allows the part line to be programmed.

- (Confusion may be caused by use of the terms "**offset**" and "**compensation**’’) In this text, **"compensation"** refers to cutter diameter offset

- The term "**offset**" refers to **tool length offset** and the change in axis coordinates when programming arcs and angles.)

- **Cutter comp** is also called **Cutter Radius Offset (CRO)** by some controller manufacturers

Seams W., “Computer Numerical Control, Concepts & Programming”

Definitions and Codes

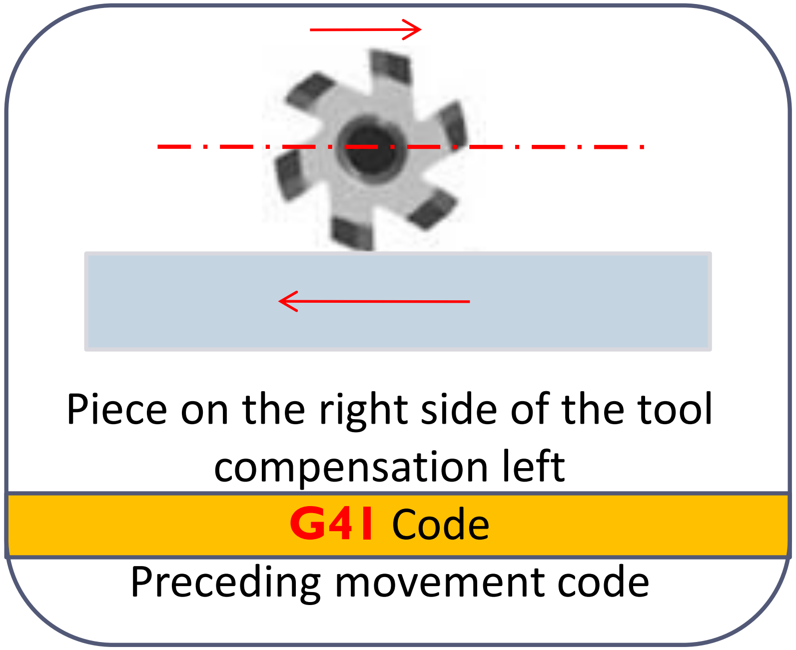
- In computer-aided programming languages (such as **Automatically Programmed Tool-APT**) and some **CAD/CAM systems** it is also called **cutcom**

- These terms all refer to the same thing: **a built-in cycle in the MCU** that, when activated, **alters the tool path** by an amount contained in the cutter comp register

The **value in the register is entered in by the setup person** when the job is being prepared

Definitions and Codes

- The path of the center of the cutting tool **is not identical** with the piece geometry
- **Compensation**: Is the **Automatic calculation** of the cutter path based on the diameter / radius of the cutting tool



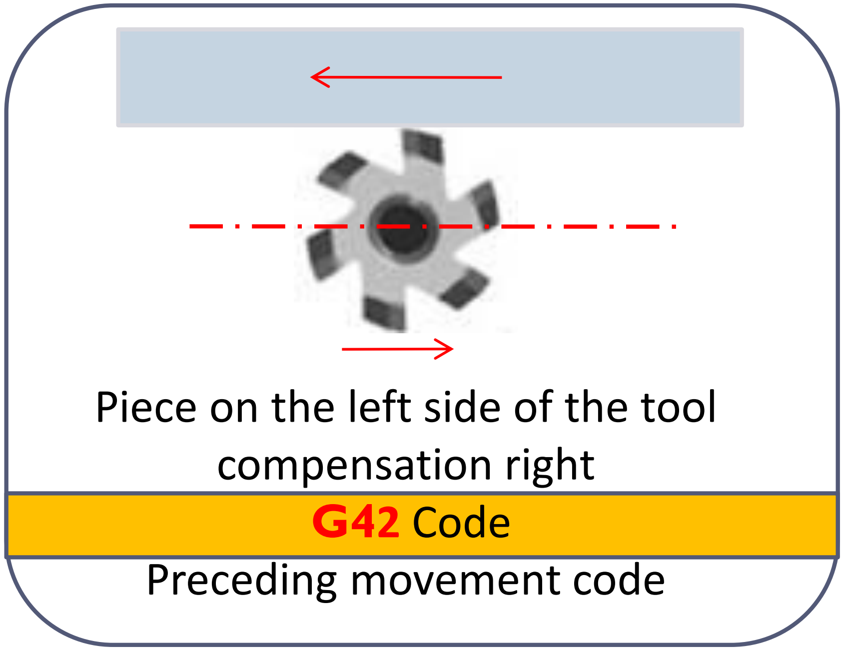


Figure 1:Cutter compensation through the use of G41 and G42

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Definitions and Codes

Cutter comp is accomplished through the use of G codes : G40, G41, G42

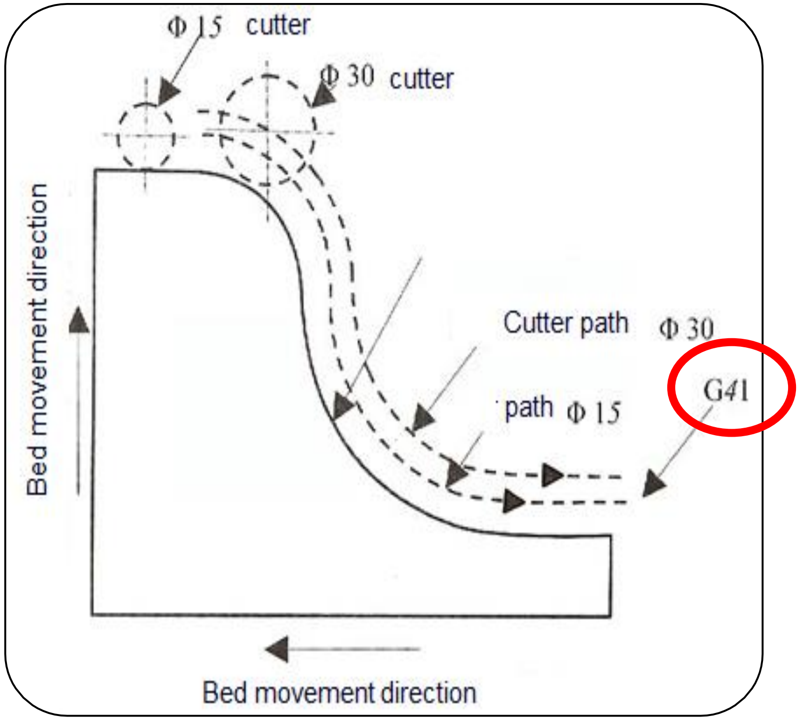
G40 – **Cutter diameter compensation cancel**. Upon receiving a **G40**, cutter diameter compensation **is turned off**. The tool will change from a compensated position to an uncompensated position on the next X, Y, or Z axis move

G41 – **Cutter diameter compensation left**. Upon receiving a **G41**, the tool will **compensate to the left** of the programmed surface. The tool will move to a compensated position on the next X, Y, or Z axis move after the **G41** is received

G42 – **Cutter diameter compensation right**. **Compensates to the right** of the programmed surface

Cutter Compensation

- **NOTE that there might be changes in cutter's diameter** due to:
  - Deterioration
  - Change cutter
  - Rounding of the edge radius of the cutting tool



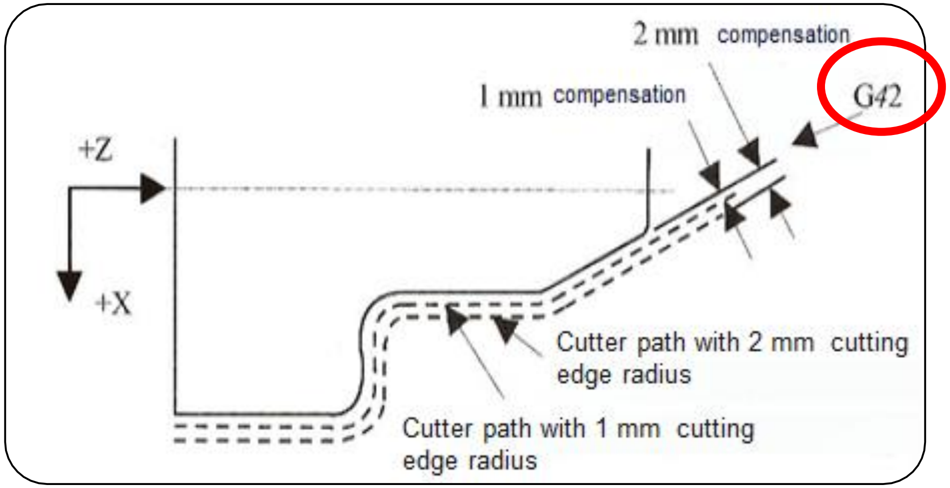


Figure 2:Cutter compensation G41,G42,
(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Codes G41, G42

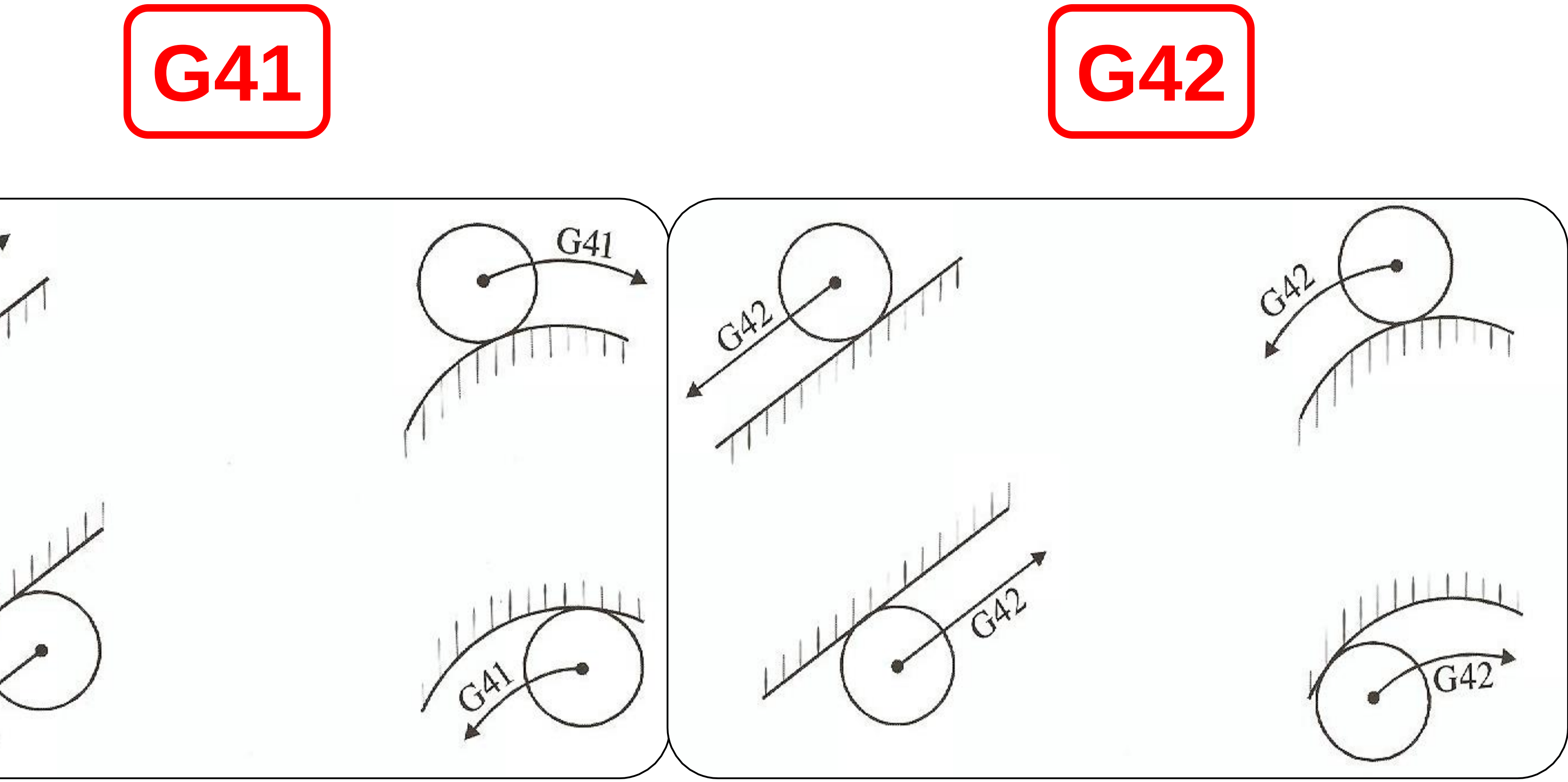


Figure 3:Example of G41,G42 codes

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Codes G41, G42

G41,G42

- Command format

N.. G01 G41 X.. Y.. D..

N.. G02 G41 X.. Y.. I.. J.. D..

Where:

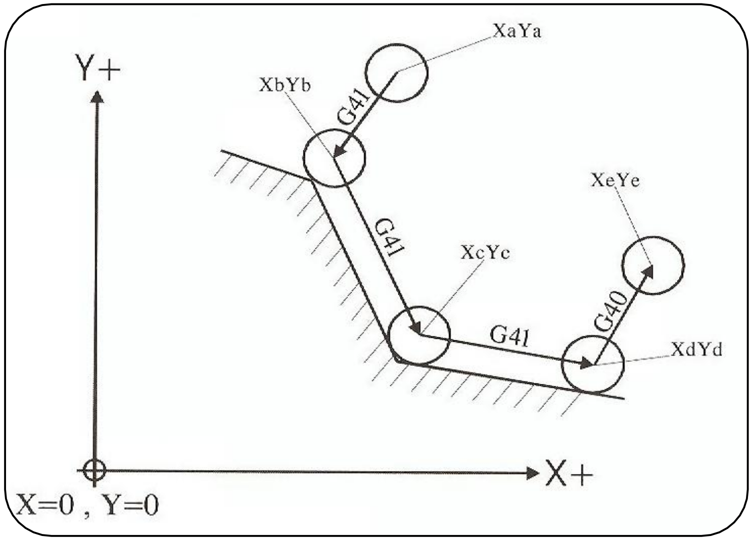
- **D** is the **memory address of machine's MCU** where the compensation value is registered

G40

- **Compensation cancel** (**G41** and **G42**) of cutter radius
- Activated **automatically** by machine at the beginning of each program
- «**Modal**» command

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Codes G41, G42 Examples



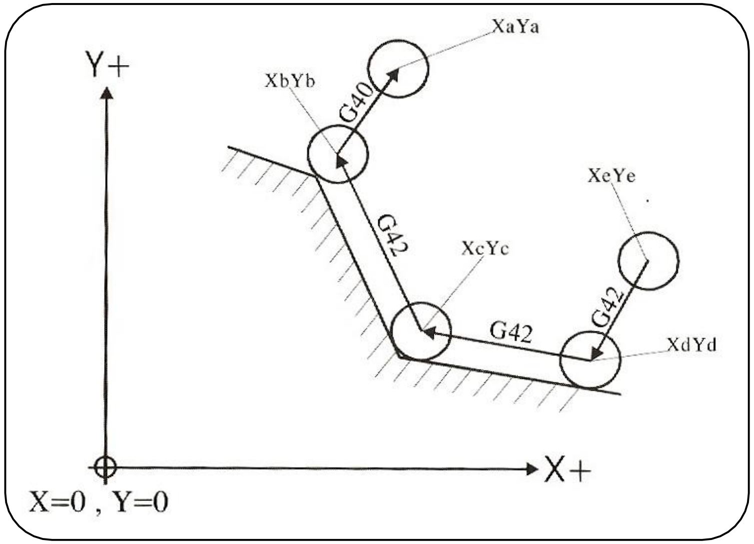


Figure 4:Correct use of G41,G42

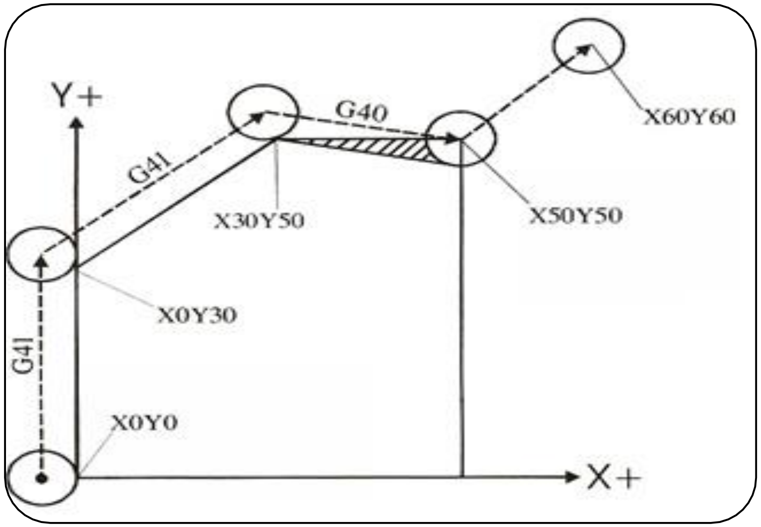
Correct use of **G41**,**G42** codes

N02	G00	XaYa
N04	G01 G41 Xb Yb F180 D1	
N06		XcYc
N08		XdYd
N10	G40 XeYe	

N02	G00	XeYe
N04	G01 G42 Xd Yd F180 D1	
N06		XcYc
N08		XbYb
N10	G40 XaYa	

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Codes G41, G42 Examples



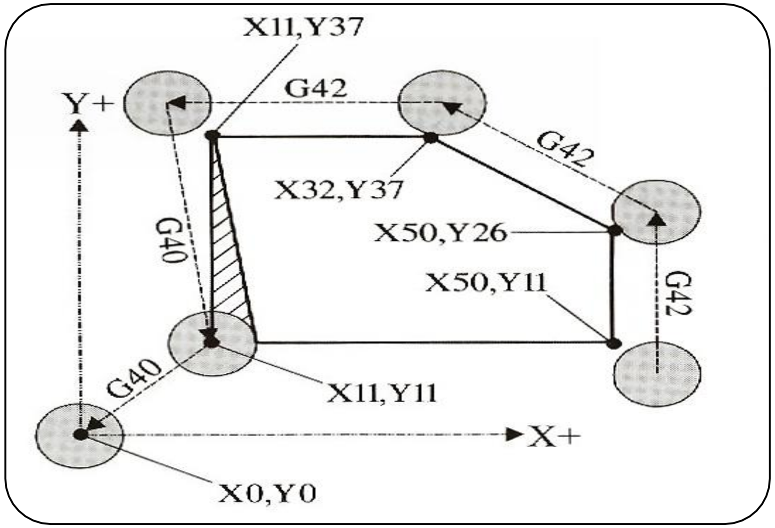


Figure 5:Incorrect use of G41,G42

Incorrect use of **G41**,**G42** codes

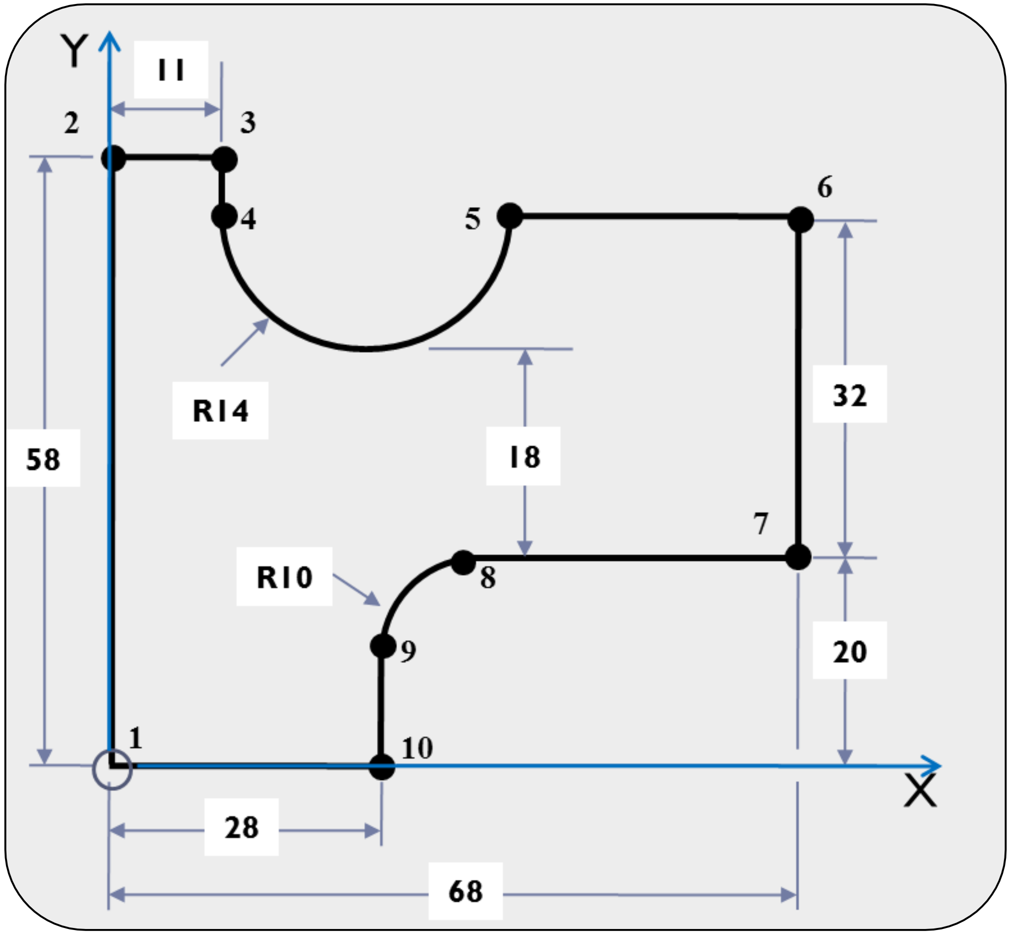
```
N02    G01 G41 X0 Y0 F220 D2
N04              X0  Y30
N06              X30 Y50
N08          G40 X50 Y50
N10          G40 X60 Y60
```

```
N02    G01 G42 X50 Y11 F220 D2
N04              X50  Y26
N06              X32 Y37
N08          G40 X11 Y11
N10              X0 Y0
```

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Codes G41, G42,G40 Examples

Example of peripheral milling using **end mill D8**



	X	Y
1	0	0
2	0	58
3	11	58
4	11	52
5	39	52
6	68	52
7	68	20
8	38	20
9	28	10
10	28	0

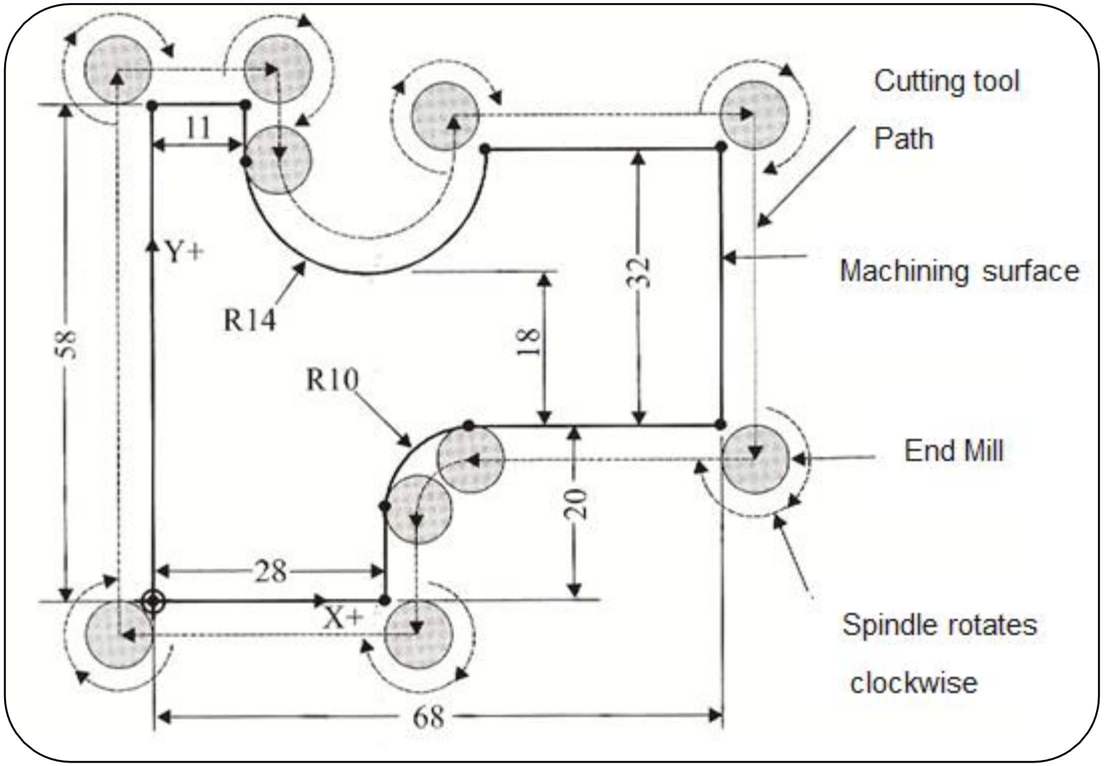
Figure 6:Part to be milled

(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

LAB

Codes G41, G42,G40 Examples

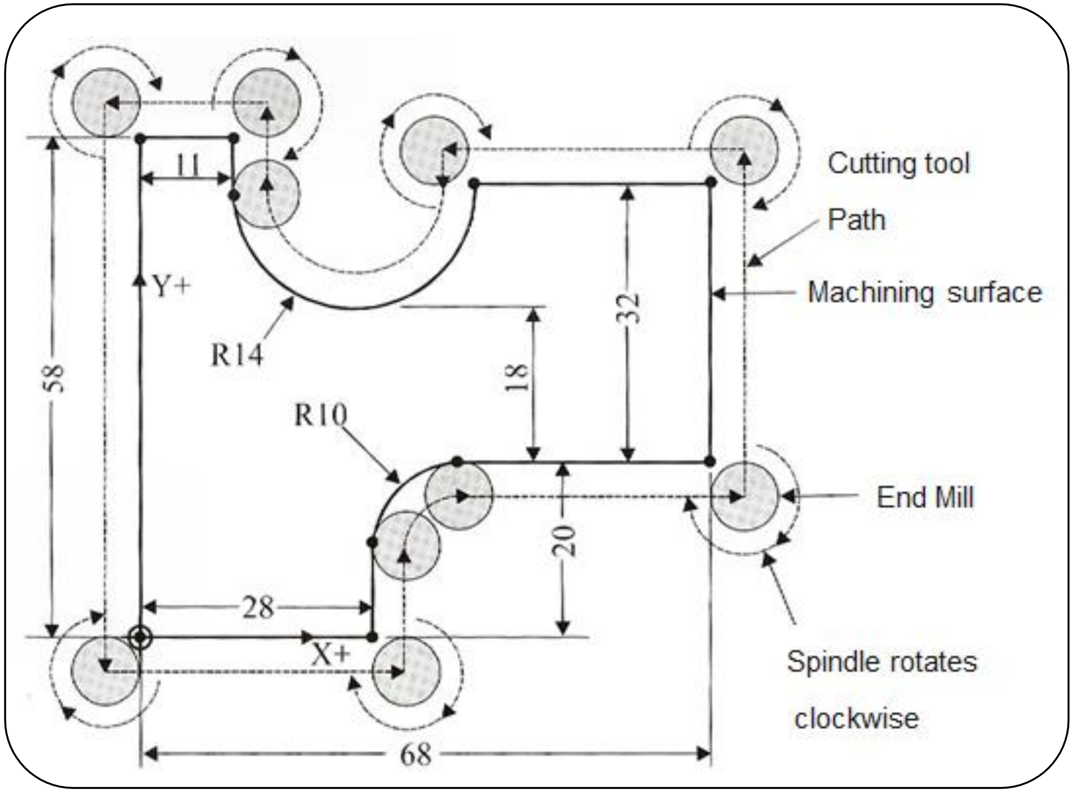
```
%
:004
N10 G21
N20 G91 G28 X0 Y0 Z0
N30 G40 G49 G80
N40 M06 T01
N50 S2000 M03
N60 G90 G00 G43 X-15 Y-15 Z40
N70 G01 Z0 F600 M08
N80 G41 X0 Y0 D01
N90 X0 Y58
N100 X11 Y58
N110 X11 Y52
N120 G03 X39 Y52 I14 J0
N130 G01 X68 Y52
N140 X68 Y20
N150 X38 Y20
N160 G03 X28 Y10 I0 J-10
N170 G01 X28 Y0
N180 X0 Y0
N190 G40 X-15 Y-15
N200 G00 Z50 M09
N210 G28 G91 X0 Y0 Z0
N220 M02
```



(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

LAB Codes G41, G42,G40 Examples

```
%
:005
N10 G21
N20 G91 G28 X0 Y0 Z0
N30 G40 G49 G80
N40 M06 T01
N50 S2000 M03
N60 G90 G00 G43 X-15 Y-15 Z40
N70 G01 Z0 F600 M08
N80 G42 X0 Y0 D01
N90 X28 Y0
N100 X28 Y10
N110 G02 X38 Y20 I10 J0
N120 G01X68 Y20
N130 X68 Y52
N140 X39 Y52
N150 G02 X11 Y52 I-14 J0
N160 G01 X11 Y58
N170 X0 Y58
N180 X0 Y0
N190 G40 X-15 Y-15
N200 G00 Z50 M09
N210 G28 G91 X0 Y0 Z0
N220 M02
```



(source: Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,κ.α.)

Ramp on and Ramp off

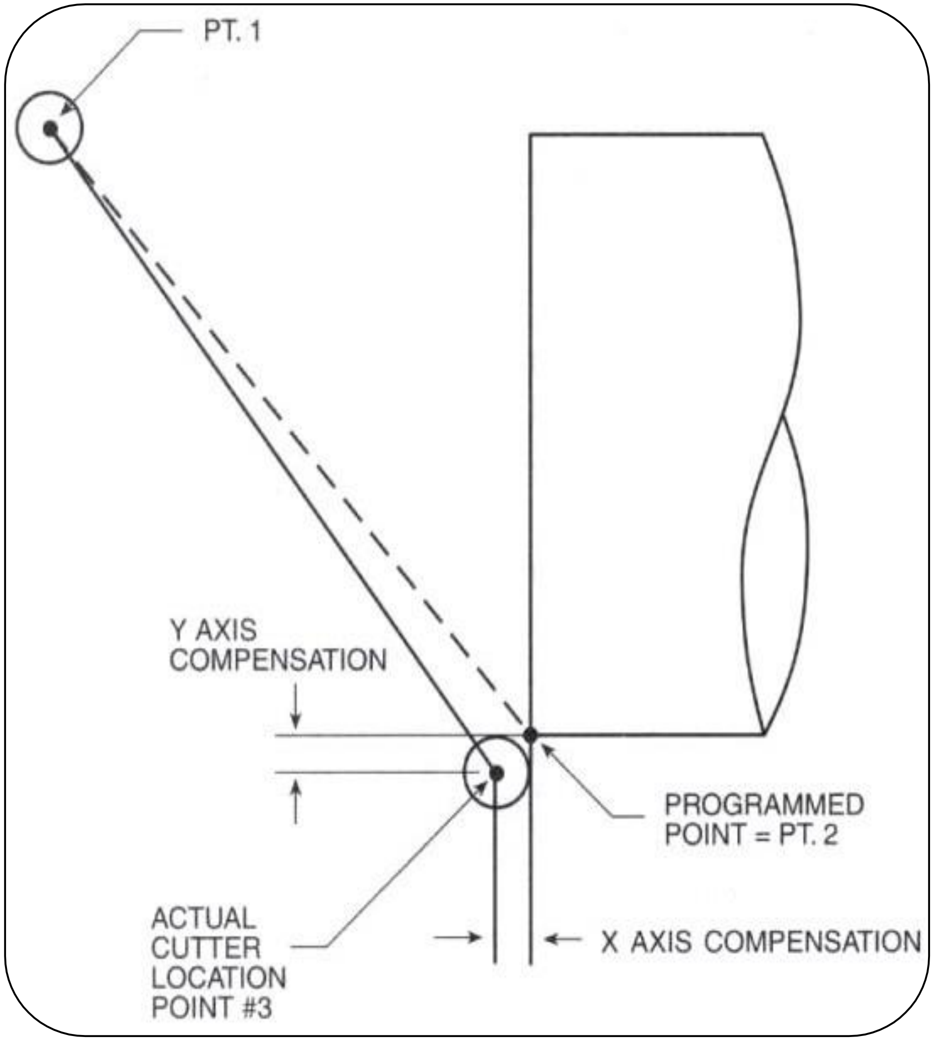


Figure 7: Ramp on move

Seams W., "Computer Numerical Control, Concepts & Programming"

- Two **terms** are important for understanding cutter diameter compensation:
  - **ramp on**
  - **ramp off**

- The tool is moved from **point #1 to point #2** following a **G41** command, the cutter will **compensate** in a plane perpendicular to the part surface and the spindle will move to point #3 rather than point #2
- This initial compensation move is called the ***ramp on* move**
- The machine is in the process of adjusting its path for the entire move from point # 1 to point #2. By the time it reaches point #2, it has fully ***compensated*** its path

Definitions and Codes

- Most controllers allow compensation to be performed on any two axes. A **G code** is used to determine **which axis combination is to be used**
  - If the part is to be machined using the X and Y axes, **compensation** is desired in the X/Y plane
  - If using the X and Z axes, **compensation** in the Z/X plane is needed
  - If using the Y and Z axes, **compensation** is needed in the Y/Z plane. The X/Y plane is used most commonly.

The G codes used to select the desired work plane are:

G17 – **X/Y** plane
G18 – **Z/X** plane
G19 – **Y/Z** plane

Definitions and Codes

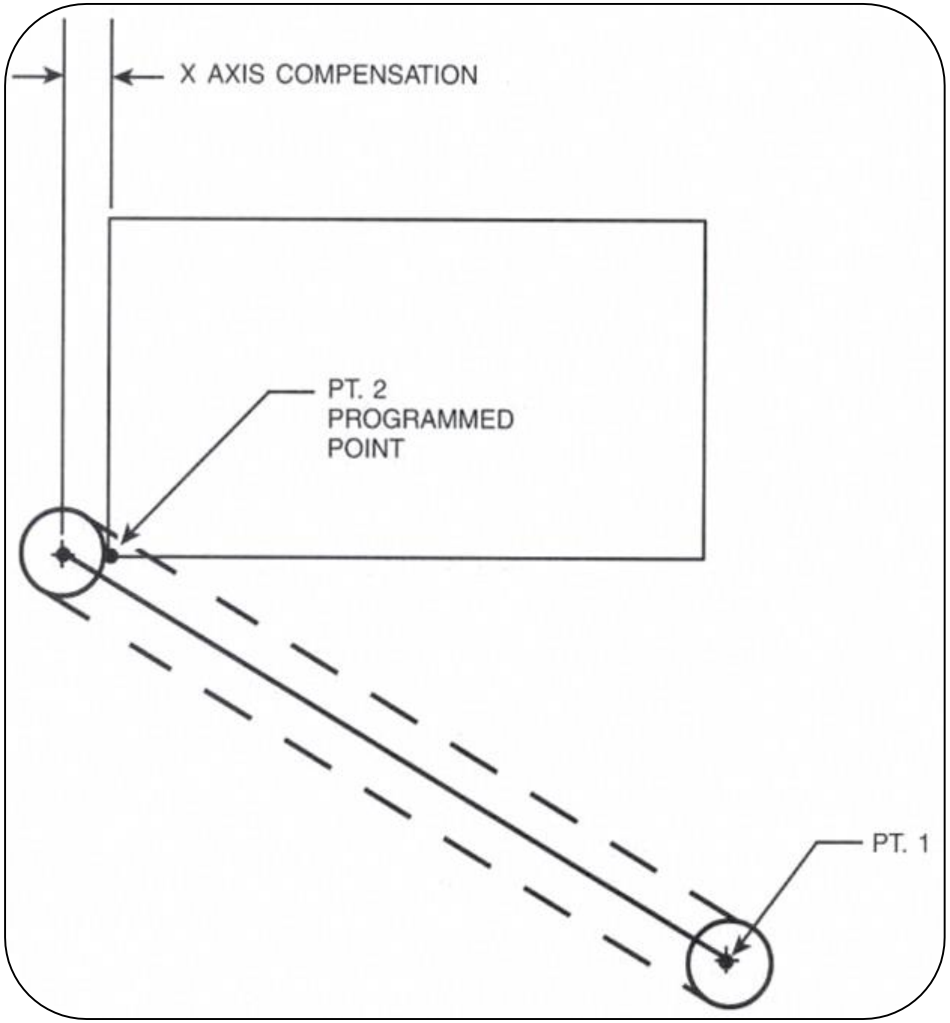


Figure 8:A situation where the corner of the part might be cut off

In Figure 8, **another type of situation** is demonstrated:

- The cutter starts at point #1 in this illustration, presenting the possibility that **the corner of the part might be cut off** in the process of moving to point #2 if the spindle moved downward or was already positioned there.
- If point #1 was the desired tool change location for this part, **the spindle would need to be fully retracted** and lowered after reaching the programmed location
- When **clamps or fixturing devices do not interfere**, it is not uncommon to **rapid the Z axis to depth** on the same move that positions X and Y

Seams W., "Computer Numerical Control, Concepts & Programming"

Definitions and Codes

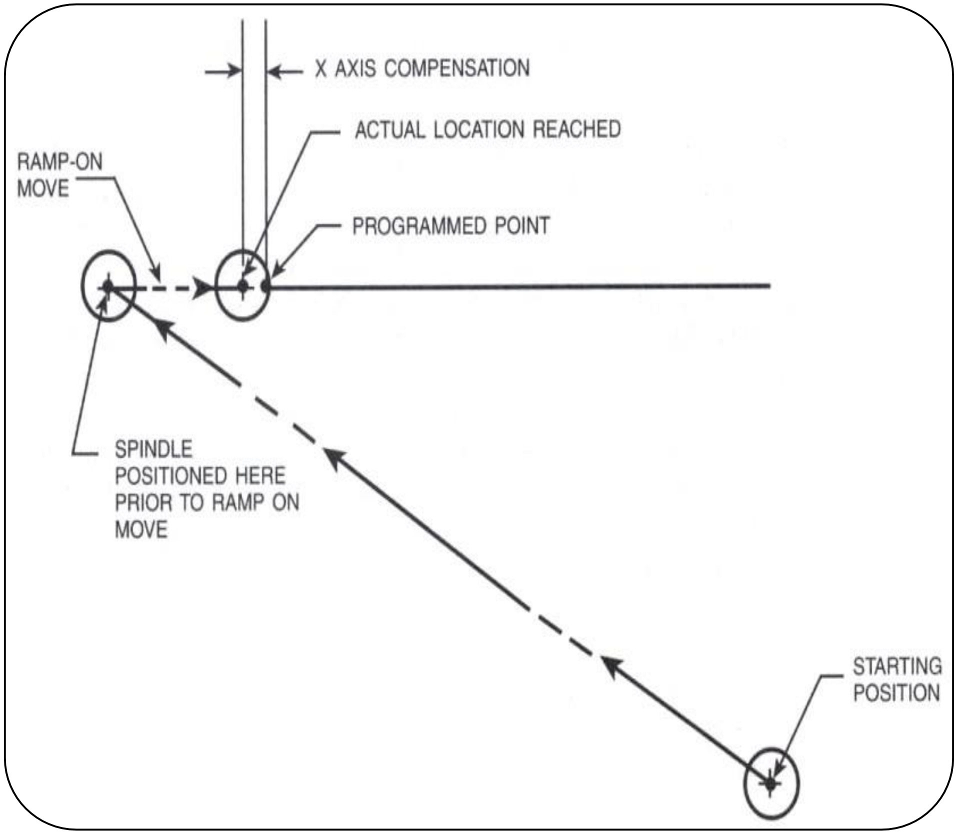


Figure 9: The most successful method of ramping on cutter compensation

- Some controllers do not allow cutter comp to be instituted in two axes **simultaneously**
- In these cases, it is necessary to program a location away from the part surface and **ramp on the compensation** 90 degrees from the desired part surface, as in Figure 9.
- Controllers are often particular about the manner in which cutter comp is **ramped on**
- It is advisable when programming several different controllers (or when there are controllers of different ages) to use the method in Figure 9

✓ This is the most successful method of **ramping on** cutter comp

Definitions and Codes

- The **ramp off move is the opposite of the ramp on move** and the **same precautions** are necessary
- In Figure 8, assume that **cutter comp is canceled** and a move is made from point #2 to point #1. In this case, the **corner** of the part may also be **cut off**
- Remember, the compensation is ***not turned off completely until the ramp off move is competed***

Two additional points should be **noted**:

1. With many controllers, cutter comp ***must be turned on*** after the length offset is initiated. Similarly, cutter comp ***must be cancelled*** prior to cancelling the tool length offset. Failure to do this will result in the **controller halting executing of the program, and an alarm signaling at the MCU console**
2. Cutter comp usually must be commanded **in rapid or feedrate modes**, **not** in a **circular interpolation mode**. If **G40**, **G41**, or **G42** are commanded after **G02** or **G03**, a machine controller alarm will result

Definitions and Codes

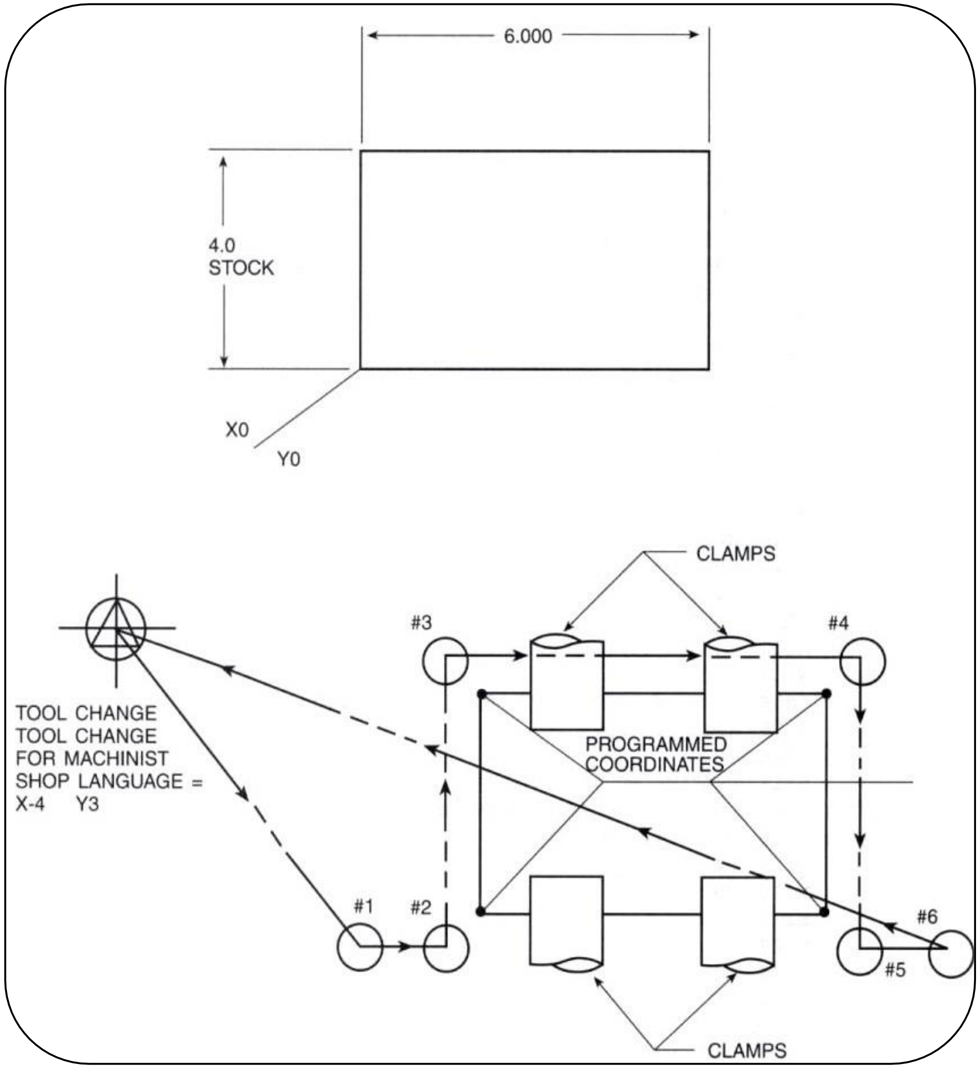


Figure 10:Part drawing and cutter path for cutter diameter compensation

Seams W., "Computer Numerical Control, Concepts & Programming"

- A short program to mill the part in Figure 10 is given in Figure 11. The program contains both a **roughing** and a **finishing cut**
- One way to accomplish this **without changing tools** or programmed coordinates is ***to program the diameter of the cutter as two separate diameters***
- This program uses a .500-inch-diameter end mill. By defining it as both a .520-inch diameter and a .500-inch diameter and using the same coordinates for both passes, the result ***is that .010 inch of stock is left for the finish pass***

Definitions and Codes

```
%
O1005
(* **********)
(* X0/Y0 = LOWER LEFT CORNER)
(* CLEARANCE ABOVE CLAMPS 3.000 MIN.)
(* TOOLS = 1.000 IN. 4FLT END MILL)
(* **********)
N001 G00 G40 G70 G90
N101 G91 G30 X0. Y0. Z0. M19
N102 T01 M06
(POSITION TOOL AND PICK UP LENGTH OFFSET)
N103 X-1. Y-.25 S2500 M03
N104 G43 Z0. H01 M08
N105 G01 Z-.62 F12.8
(RAMP ON CRO - USE REGISTER D11)
N106 G17 G41 X0. D11
N107 Y4.
N108 G00 Z3.
N109 X6.
N110 Z0.
N111 G01 Z-.62
N112 Y0.
(RAMP OFF CRO)
N113 G40 X7.
N114 G00 Z3.
```

```
(POSITION TO START OF NEXT SEQUENCE)
N115 X-1. Y-.25
N116 Z0.
N117 G01 Z-.62
(RAMP ON CRO - USE REGISTER D12)
N118 G17 G41 X0. D12
N119 Y4.
N120 G00 Z3.
N121 X6.
N122 Z0.
N123 G01 Z-.62
N124 Y0.
(RAMP OFF CRO)
N125 G40 X7. M09
N126 G00 Z0. M05
(CANCEL TOOL LENGTH OFFSET AND
RETURN TO TOOL CHANGE POSITION)
N127 G91 G30 X0. Y0. Z0. M19
N128 M30
%
```

Figure 11: Cutter diameter compensation program to mill the part in Figure 10

Seams W., "Computer Numerical Control, Concepts & Programming"

Program Example

Program Explanation

01005
Program number ("O" number).

N001 — *Safety line*
G00 — Commands rapid traverse mode.
G40 — Cancels any active cutter comp.
G70 — Selects inch input.
G90 — Selects absolute positioning mode.
Blocks N101 through N104 are the tool change sequence blocks **N101**
G91G30X0.Y0.Z0. —This command sends the tool to the automatic tool change position.
M19 —Orients the spindle.
N102
TOT — Places tool #1 tool change standby position.
M06 — Initiates an automatic tool change. The tool in standby position will be placed into the spindle. The tool currently in the spindle will *he* returned to the tool magazine.
N103
X/Y coordinates — Position the tool at location #1. **S2500** — Sets the spindle speed to 2,500 rpm.**M03**— Turns the spindle on clockwise.
N104
G43 — Turns on the tool length offset compensation.**Z0** — positions the Z-axis to the desired start point.
H01 — Commands the MCU to use the value in tool register #1 tor tool length compensation.**M08** —Turns on the flood coolant.
Blocks N105 through N126 *are* ***the*** *tool motion sequence*
N105
Positions the Z axis at feedrate to the proper cutting depth.

Seams W., "Computer Numerical Control, Concepts & Programming"

Program Example

N106

This block is the cutter comp ramp on move. The cutter will be compensated at the end of this block.

G17 — Selects the X/Y plane.

G41 — Turns on cutter diameter compensation left.

X0. — Moves the spindle from location #1 to #2.

D11 — Instructs the MCU to use the values in register 11 for cutter diameter compensation.

N107

Moves the spindle from location #2 to #3 at feedrate.

N108

Rapids Z axis safely above part in preparation for jumping over the clamps.

N109

Positions the spindle at rapid from location #3 to #4.

N110

Rapids the Z axis to .100 above the part.

N111

Feeds the Z axis to milling depth.

N112

Moves the cutter from location #4 to #5 at feedrate.

N113

This is the cutter diameter compensation ramp off move. The cutter will be fully decompensated at the end of this block.

G40 — Cutter compensation cancel code. **X7**. — Moves the spindle from location #5 to #6.

N114

Rapids the Z axis safely above the fixture clamps.

N115

Rapids the spindle from location #5 to #1

Program Example

N116
Rapids spindle to .100 above the part.
N117
Feeds the Z axis to the milling depth.
N118
This block is the cutter comp ramp on move. The cutter will be compensated at the end of this block.
G17 — Selects the X/Y plane.
G41 — Turns on cutter diameter compensation left.
X0. — Moves the spindle from location #1 to #2.
D12 — Instructs the MCU to use the values in register 12 for cutter
N119
Moves the spindle from location #2 to #3 at feedrate.
N120
Rapids Z axis safely above part in preparation for jumping over the clamps
N121
Positions the spindle at rapid from location S3 to #4.
N122
Rapids the Z axis to .100 above the part.
N123
Feeds the Z axis to milling depth.
N124
Moves the cutter from location #4 to #5 at feedrate.
N125
This is the cutter diameter compensation ramp off move. The cutter will be fully decompensated at the end of this block.

Program Example

G40 — Cutter compensation cancel code

X7. — Moves the spindle from location #5 to #6.

M09 — Turns off the coolant.

N126

Rapids Z axis to .100 above part and turns off the spindle.

Block N126 is the tool cancel sequence

N127

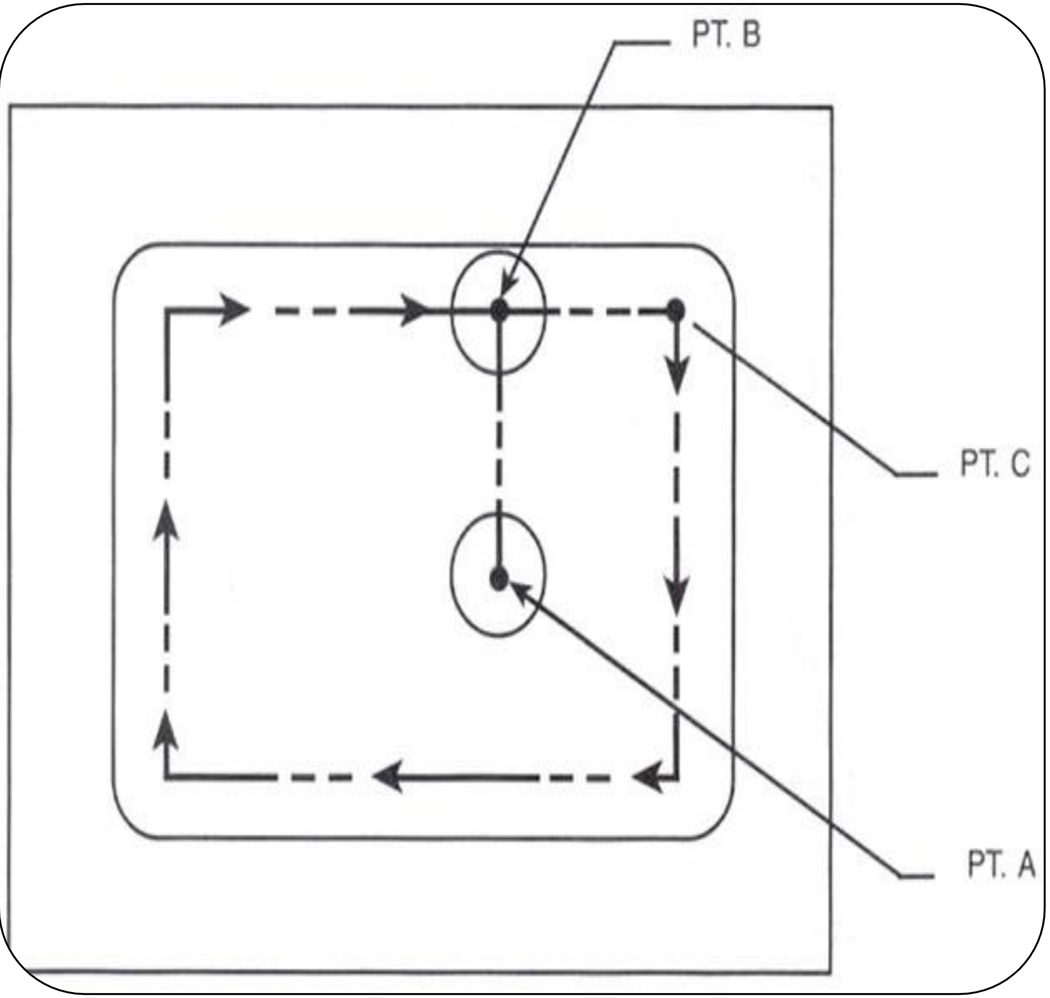
G91G30X0.Y0.Z0. — Commands the spindle at rapid traverse to the tool change position.

M19 — Orients the spindle.

N128

M30 — Signals end of program, memory reset.

Special Considerations



- Figure 12 illustrates the correct method for ***turning compensation on or off*** when machining an **inside pocket**
- **Point B** must be a **minimum** of one cutter **radius** away from the corner of the pocket.
- If **point C** was programmed as **the ramp on move**, the cutter would cut into the corner as in Figure 13

Figure 12:Turning compensation on or off when machining an inside pocket to prevent cutting into the corner *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Special Considerations

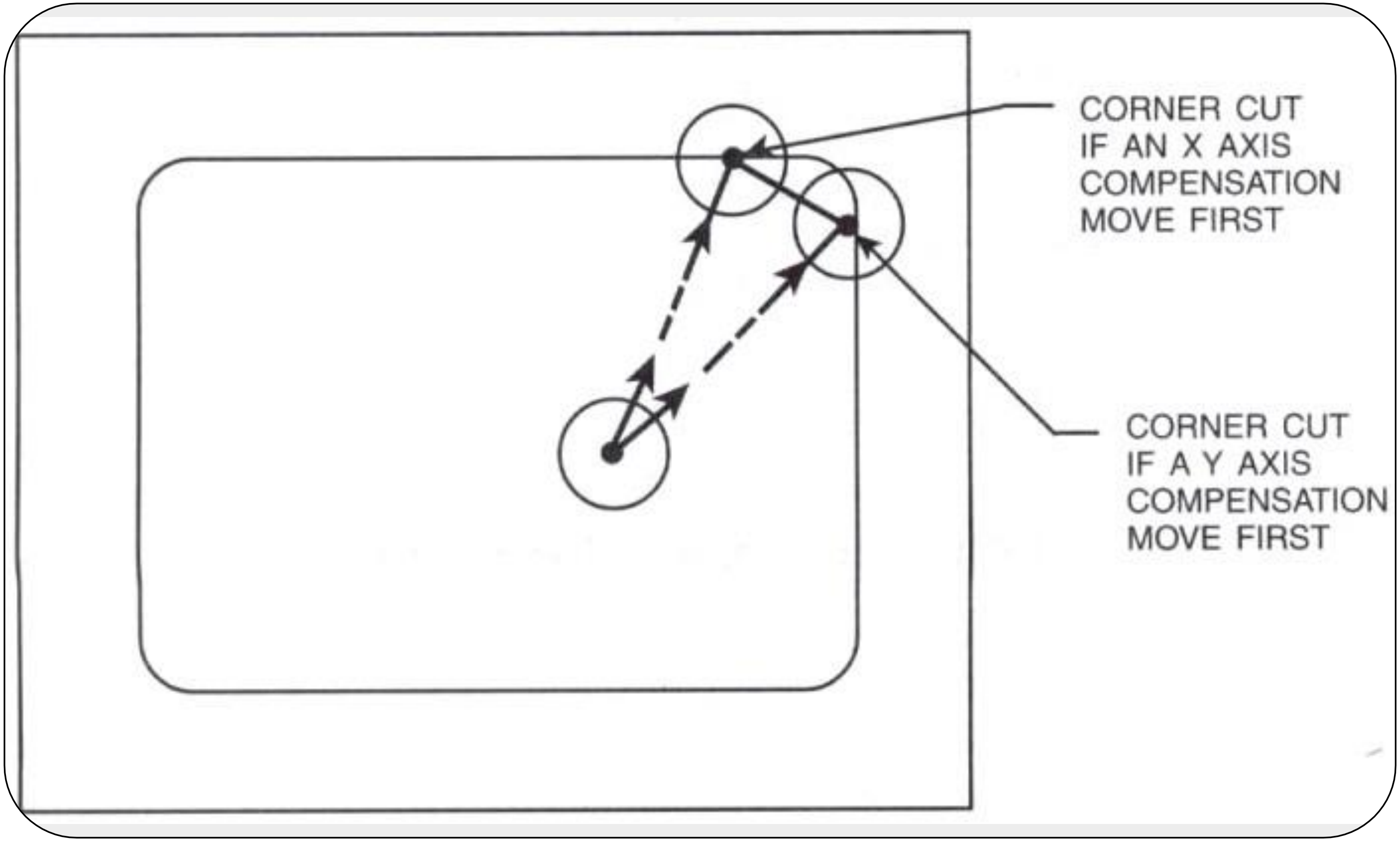


Figure 13:The direction of the cut depends on whether or not the X or Y axis is programmed *as the first move* following the G41 *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Special Considerations

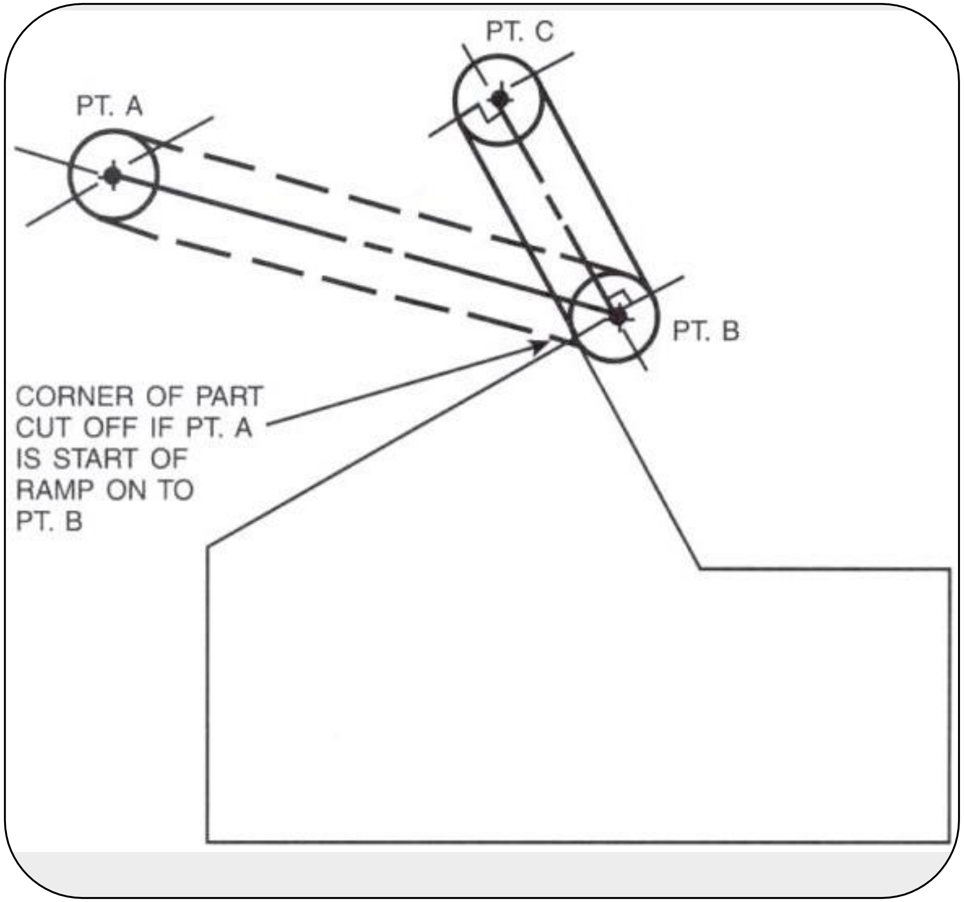


Figure 14: Ramping on or off an angle

Seams W., "Computer Numerical Control, Concepts & Programming"

- Figure 14 illustrates the **precautions** necessary when ***ramping on or off an angle***
- Point A should **not be used** for a ***ramp on or ramp off*** move since the corner of the angle will be cut off the part and there may be **damage** to the cutter
- Point C, or some other point roughly ***perpendicular to the angle***, should be used for the **ramp on or ramp off** move

Special Considerations

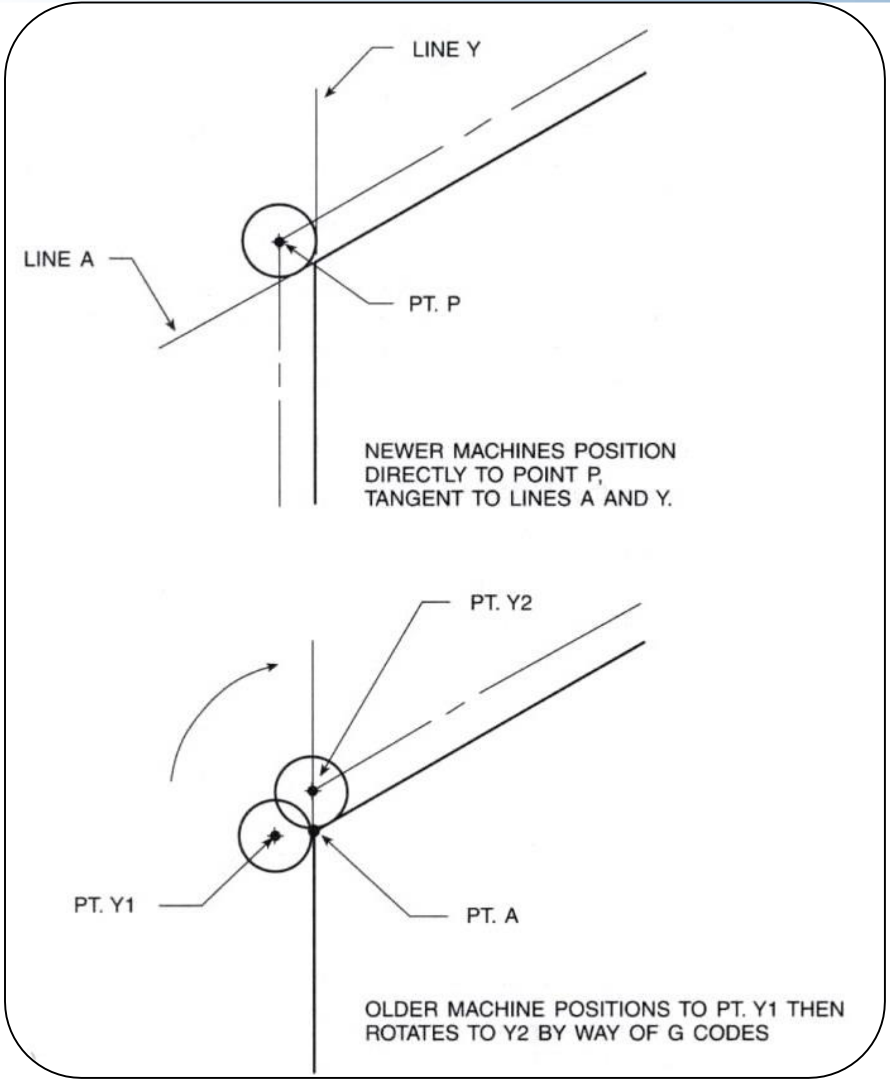


Figure 15 Cutter diameter compensation of angles

Seams W., "Computer Numerical Control, Concepts & Programming"

- Two different **methods of positioning** are used for cutter comp with respect to angles as demonstrated in Figure 15
- The machine positions the cutter **tangent** to point A on older CNC machines. A G code is then used to **initiate the rotation** from Y1 to Y2
- On newer machinery, the cutter is positioned directly to point P, **tangent** to both line A and line Y. No special G codes are necessary in this instance

NOTE

The programming manual for a particular machine will tell the programmer whether a G code is required

Special Considerations

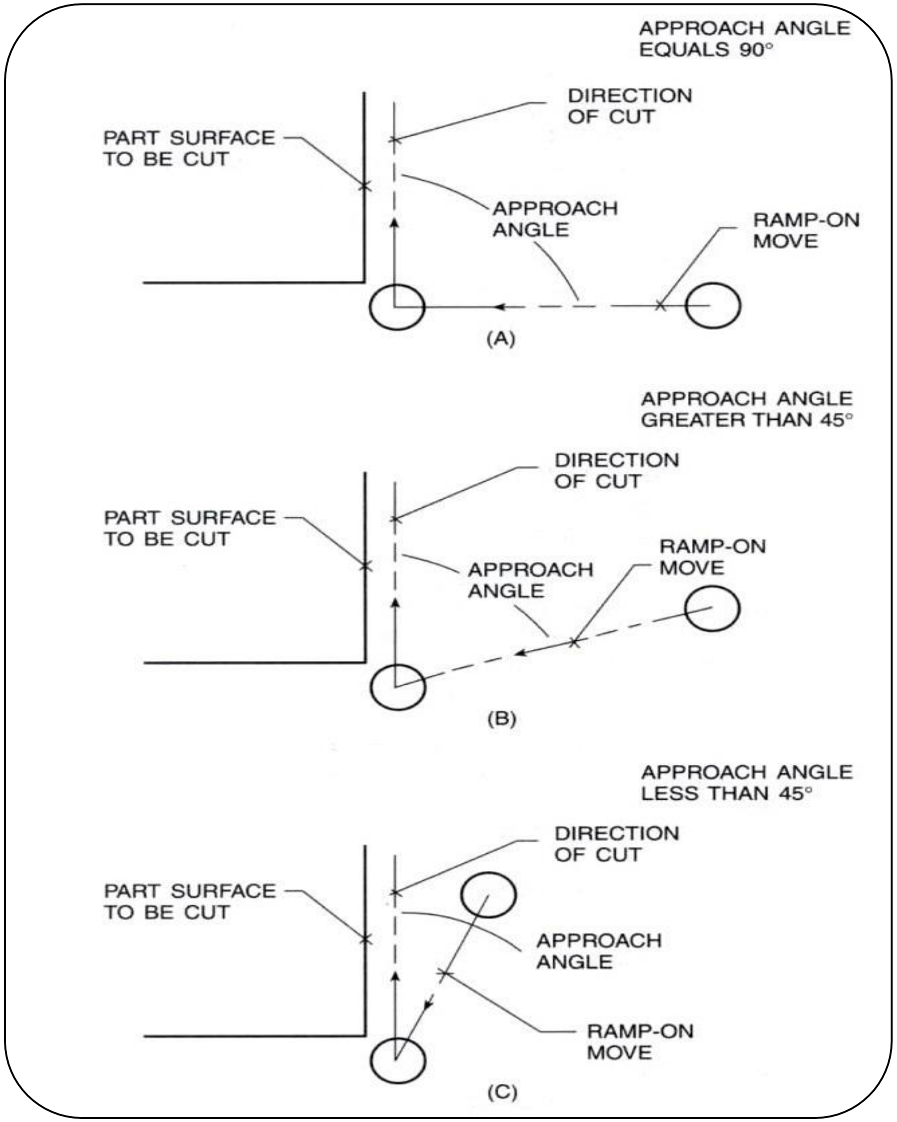


Figure 16: Cutter compensation approach angles

Approach Angles and Vectors

- Another **factor** to consider when using cutter diameter compensation is the **approach angle** used when **ramping on**
- As Figure 16 illustrates, there are **three possible angles** that can be used during a ramp-on move:
  - a) **90 degrees** to the next cut
  - b) **Less than 90** but greater than 45 degrees to the next cut
  - c) **Less than 45 degrees** to the next cut

Seams W., "Computer Numerical Control, Concepts & Programming"

Special Considerations

Approach Angles and Vectors

- **Some controllers will accept** any of these approach angles, **others will not**

- If an **unacceptable approach angle is used**, the cutter will move to the programmed coordinates, but **the cutter compensation will not take place**

- When programming a number of controllers, or if the NC program will be run on more than one type of controller, it is best to **use a 90-degree approach angle** to eliminate problems when ramping on cutter comp

Special Considerations

- Sometimes, a controller requires a **vector** to be commanded with the **G41** or **G42** to orient the cutter correctly prior to the ramp-on move

Technically, a vector is a geometric entity that has both magnitude (length) and direction

- In **NC programming**, vectors are simply mathematical arrows that point the cutter in a given direction

- To utilize a vector the **I** and **J addresses** are used

Seams W., "Computer Numerical Control, Concepts & Programming"

Special Considerations

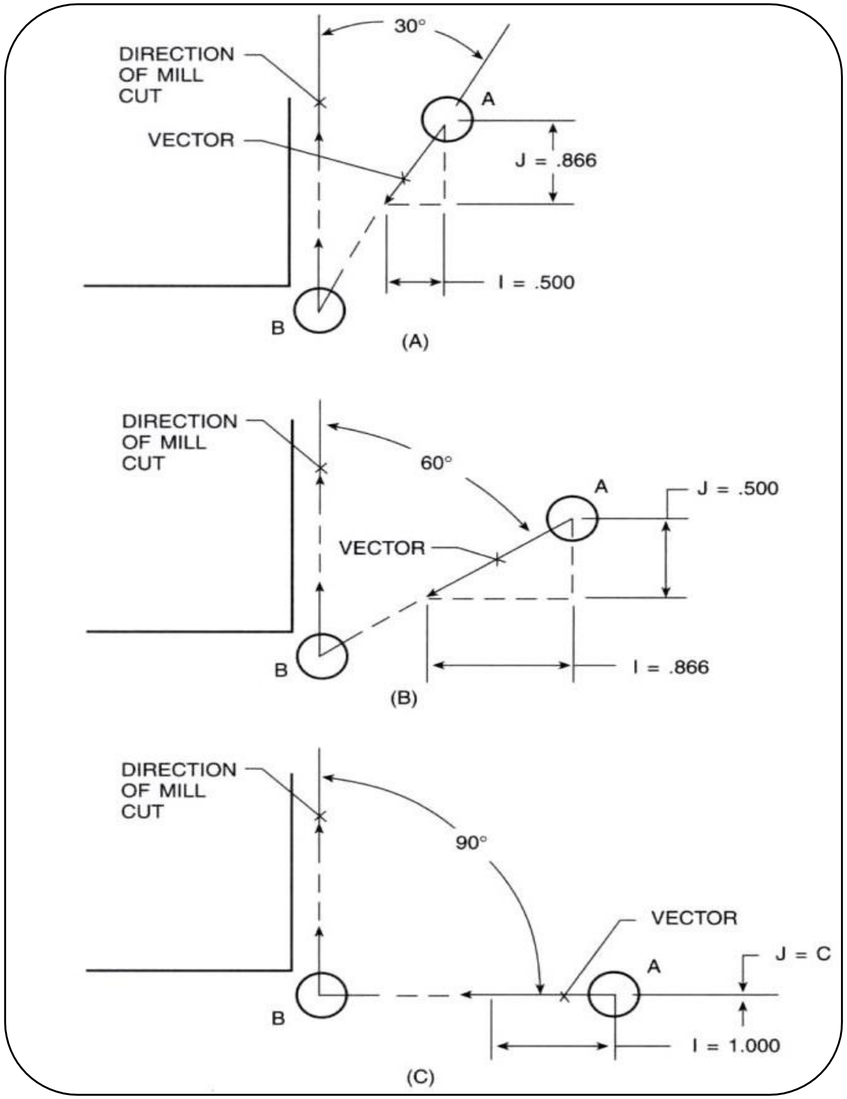
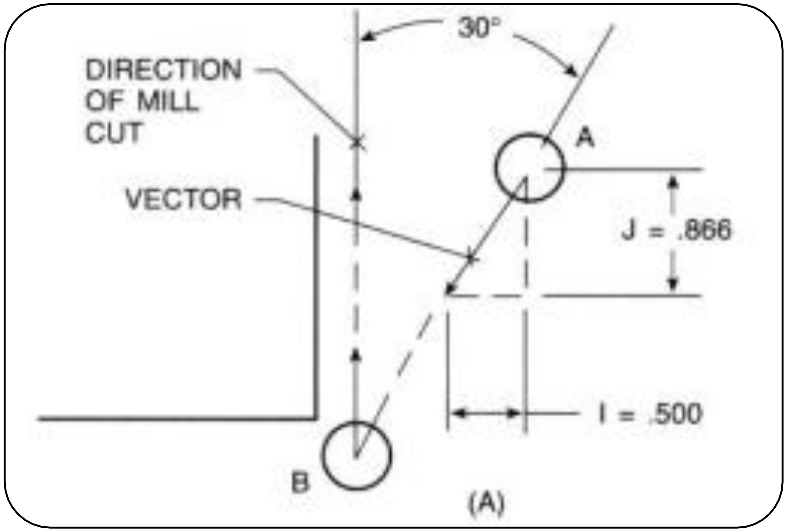


Figure 17: Cutter compensation vectors

- Figure 17 illustrates some **cutter comp vectors**
- If **cutter comp** was to be initiated from point A (Figure 17) and **ramped on** to point B, the following program blocks would be used:

Special Considerations



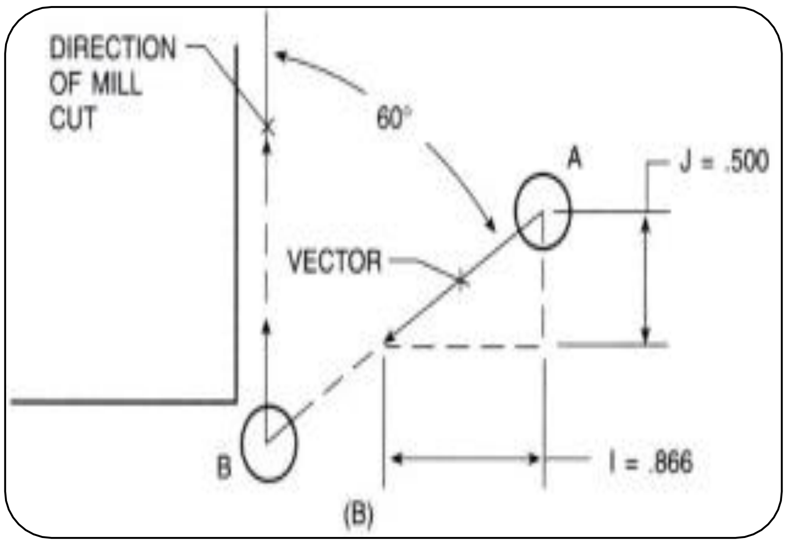


Figure 17 (a),(b) Cutter compensation vectors

Seams W., "Computer Numerical Control, Concepts & Programming"

- **For Figure 17 (a):**

  N010 G17 G42 1-.5 J-.866 D21

  N020 G00 X6.0 Y-.5

- **For Figure 17 (b):**

  N010 G17 G42 1-.866 J-.5 D21

  N020 G00 X6.0 Y-.5

Special Considerations

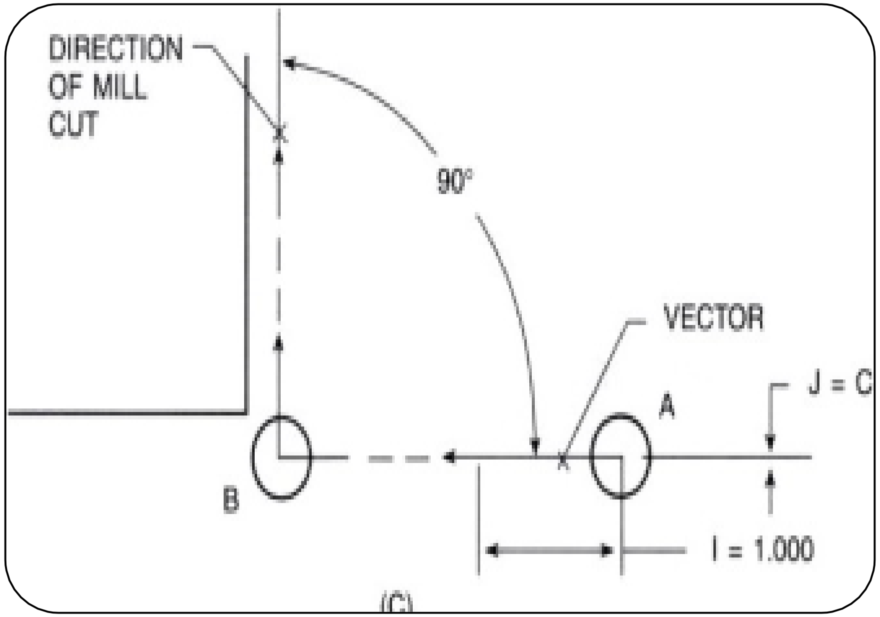


Figure 17(c) Cutter compensation vectors

Seams W., "Computer Numerical Control, Concepts & Programming"

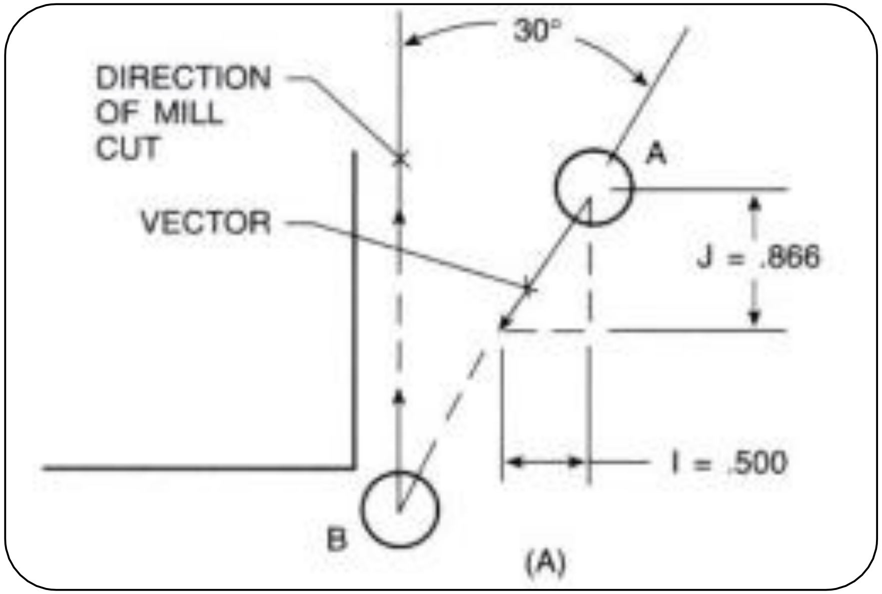
- **For Figure 17 (c):**

N010 G17 G42 1-1.0 J0 D21

N020 G00 X6.0 Y-.5

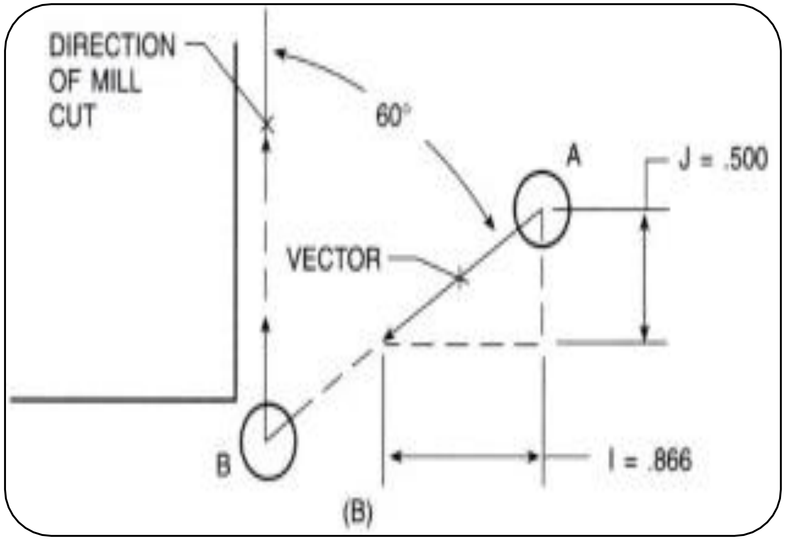
Special Considerations

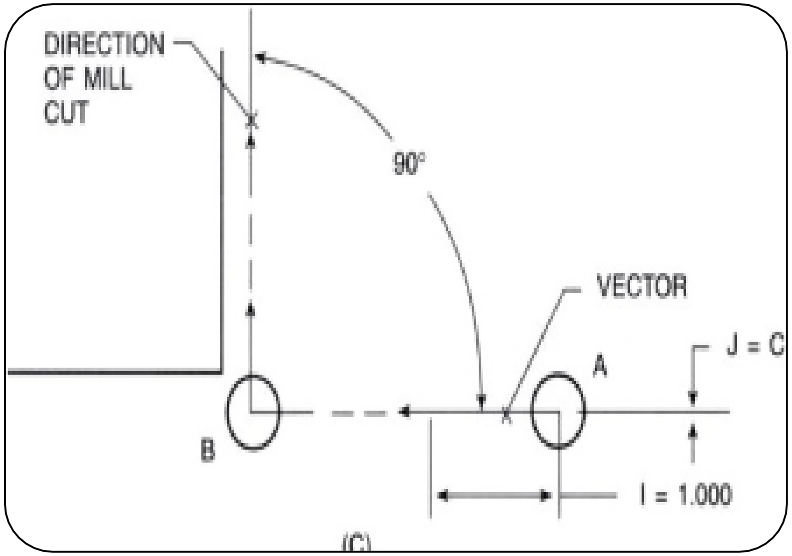
- In each of these cases, **I is the X-axis** component and **J is the Y-axis** component of a vector that is 1.0 inch long. This is called a **unit vector**



- In Figure 17(a), the approach angle is 30 degrees, therefore, **I equals the sine of 30 degrees**, while **J equals the cosine**

Special Considerations





- In Figure 17(b) the approach angle is 60 degrees. **I equals the sine of 60 degrees**, while **J equals the cosine of 60 degrees**

- Since the approach angle in Figure 17(c) is 90 degrees**, I simply equals 1.0** and **J equals zero**

Special Considerations

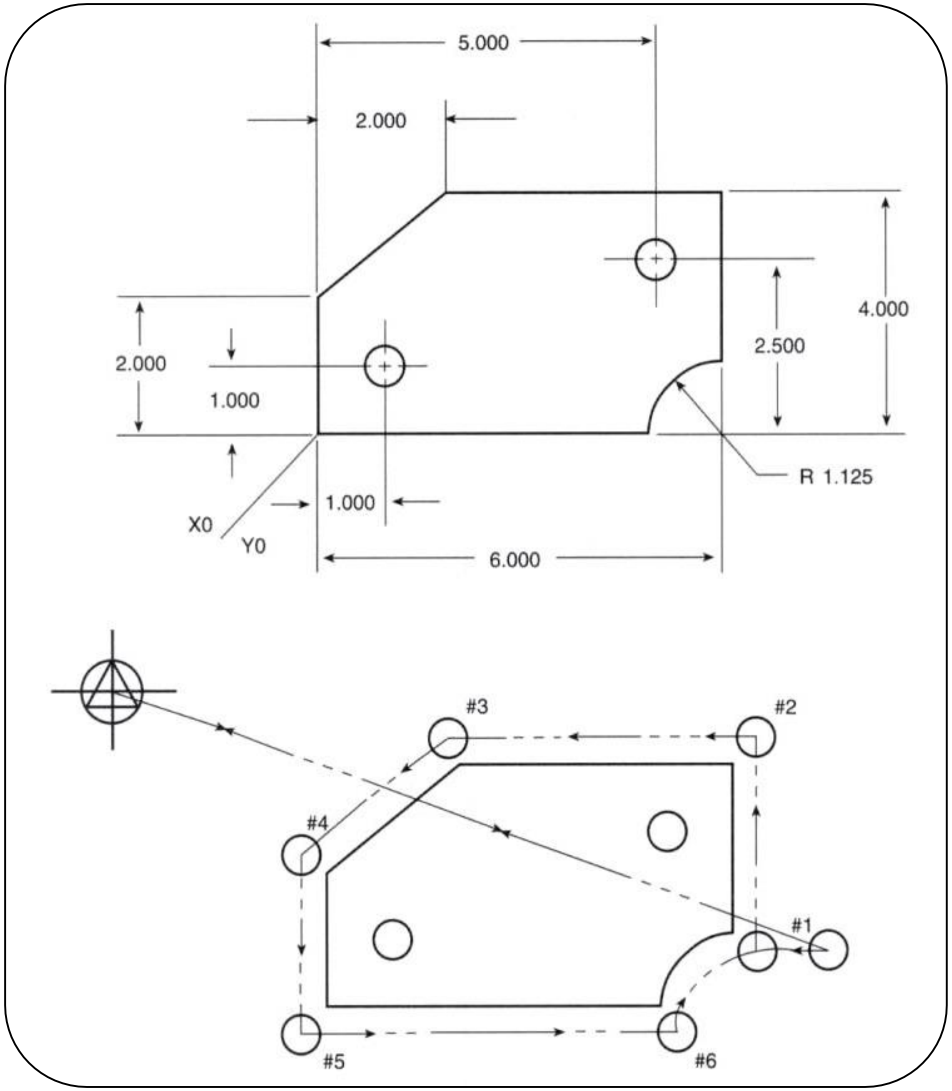


Figure 18 Part drawing and cutter path

- Figure 18 shows a part to be milled using **cutter diameter compensation**
- A program to mill the part is given in Figure 19. It is assumed that the part is **clamped** through two already existing **holes**

Special Considerations

```
%
O1013
(* **********)
(* X0/Y0 = LOWER LEFT CORNER)
(* TOOL  = 1.000 IN. 4FLT END MILL)
(* CLEARANCE ABOVE CLAMPS 3.000 MIN.)
(* **********)
N001 G00 G40 G90 G80
N100 T01 M06
(POSITION TOOL AND PICK UP LENGTH OFFSET)
N101 G00 X7. Y.875 S400 M03
N102 G43 Z0. H01 M08
N103 G01 Z-.89 F6.8
(RAMP ON CRO - USE REGISTER D11)
N104 G17 G42 X6. D11
N105 Y4.
N106 X2.
N107 X0. Y2.
N108 Y0.
N109 X4.875
```

```
N110 G02 X6. Y1.125 I0. J1.125
(RAMP OFF CRO)
N111 G00 G40 X7.
(RAMP ON CRO - USE REGISTER D12)
N112 G17 G42 X6. D12
N113 Y4.
N114 X2.
N115 X0. Y2.
N116 Y0.
N117 X4.875
N118 G02 X6. Y1.125 I0. J1.125
(RAMP OFF CRO)
N119 G00 G40 X7.5 M09
N120 G00 Z0. M05
(CANCEL TOOL LENGTH OFFSET AND
RETURN TO TOOL CHANGE POSITION)
N121 G91 G30 X0. Y0. Z0. M19
N122 M30
%
```

FIGURE 19: Program to mill part in Figure 10-12

Seams W., "Computer Numerical Control, Concepts & Programming"

Special Considerations

Program Explanation

N001

Safety line

N100

Tool change

N101-N103

Position tool to position #1

N104

Cutter comp ramp on move. The cutter comp is using register D11 that contains a value 0.020 diameter larger than the actual cutter. This will result in 0.10 per side of stock remaining on the part at the end of the first mill pass

N105-N110

Mills the part periphery

N111

Ramp off the cutter comp

N112

Ramp on cutter comp to begin second mill pass. Notice that the CRO register has changed. The value in this register will be the actual cutter size. At the end of this second mill pass, the part will be at finished size

N113-N118

Mills the part periphery

N119

Ramp off the cutter comp

N120-N122

Return spindle to the home zero location and terminate the NC program

Seams W., "Computer Numerical Control, Concepts & Programming"

Fine Tuning With Cutter Diameter Compensation

- Up to this point, **cutter diameter compensation** has been used to program the part line; the **program coordinates have matched the part dimensions**
- Another way **cutter comp** is employed is to **fine tune the cutter path**
- In this type of programming, the part is programmed using the parallel path method
- **Cutter comp** is used to compensate for the difference between the programmed and actual cutter diameter
  - For example, if a program is written for a .500-diameter end mill, but a re-sharpened end mill measuring .490 diameter is used, the .020 diameter difference can be compensated by using cutter comp.

Fine Tuning With Cutter Diameter Compensation

- In the **fine tune method**, cutter comp is usually used to compensate for a cutter which is smaller than the programmed diameter. When using the part line method exactly the opposite is the case

- **Cutter comp** is used to compensate for a cutter that is larger than the zero diameter cutter programmed (the part line)

- For this reason, it is necessary to **use a minus (-) value in the cutter comp register when using the fine tune method**

- Figure 20 is a word address program for the part in Figure 18, illustrating the fine tune method

Allowance is once again **being made for the cutter radius**. The cutter diameter compensation allows reground, undersize cutters to be used

Seams W., "Computer Numerical Control, Concepts & Programming"

Fine Tuning With Cutter Diameter Compensation

```
%
O1014
(* **********)
(* X0/Y0 = LOWER LEFT CORNER)
(* **********)
N001 G00 G40 G90 G80
N100 T01 M06
N101 G00 X7.5 Y.875 S400 M03
N102 G43 Z0. H01 M08
N103 G01 Z-.89 F6.8
N104 G17 G42 X6.51 D11
N105 Y4.51
N106 X1.7829
N107 X-.51 Y2.2171
N108 Y-.51
N109 X5.385
N110 G02 X6.5 Y.615 I1.635 J0.
N111 G01 Y4.5
N112 X1.7929
N113 X-.5 Y2.2071
N114 Y-.5
N115 X5.375
N116 G02 X6.5 Y.626 I1.626 J0.
N117 G00 G40 X7.5 M09
N118 G91 G30 Z0. M05
N119 G30 X0. Y0.
N120 M30
%
```

Figure 20: Program to mill the part in figure 10-12 using the ''fine tune'' method

Summary 1/2

The important concepts presented in this chapter are:

- **Cutter diameter compensation** is the **automatic calculation of the cutter path by the machine control unit**, based on the part line and cutter information contained in the program

- Cutter diameter compensation is **instituted** and **canceled** through use of the codes **G40**, **G41**, and **G42**

- **G41** is **cutter compensation left**, **G42** is **cutter compensation right**, and **G40** is **cutter compensation cancel**

Summary 2/2

- The "**ramp on**" move is the **initial compensation of the cutter**. The compensation occurs 90 degrees to the next axis movement following the **G41** or **G42**

- **Care** must be taken with the **spindle position prior to the ramp on move** to avoid cutting the part in the wrong area

- The "**ramp off**" move is the **opposite operation**. **Ramp off** will occur 90 degrees to the next axis movement following a **G40**. The compensation will be completely eliminated by the end of this move

Vocabulary Introduced in this Chapter

- Approach angle
- Cutter diameter compensation (cutter comp)
- Cutter radius offset (CRO)
- Ramp off move
- Ramp on move

References

1. Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006, Springer-Verlag
2. http://www.dptechnology.com/
3. Kalpakjian S., «Manufacturing Engineering and Technology», 2nd Edition, 1992, Addison-Wesley Publishing company
4. Mattson M., "CNC Programming, Principles and Applications", Delmar, 2002
5. Moriwaki T., "Multi-Functional Machine Tool", CIRP Annals Manufacturing Technology, Vol. 57/2, 2008, pp. 736-749
6. Seams W., "Computer Numerical Control, Concepts & Programming", 4th Edition, Delmar, 2002
7. Γ. Χρυσολούρης, «Συστήματα Παραγωγής Θεωρία και Πράξη» Μέρος I και II, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2001,
8. Γ. Χρυσολούρης, Δ. Μούρτζης, Κ. Τσίρμπας, Σ. Καραγιάννης, "Ορθογωνική Κοπή", Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2000
9. Γ. Χρυσολούρης, Δ. Μούρτζης, και άλλοι, "Εργαστήρια Μηχανουργικής Τεχνολογίας I και II"», Εκπαιδευτικές Σημειώσεις για το εργαστήριο του αντιστοίχου μαθήματος, Πανεπιστήμιο Πατρών, 2008 (4η Έκδοση)

References

10. Δ. Μούρτζης, “Αριθμητικός Έλεγχος Εργαλειομηχανών” Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2011 (3η Έκδοση)
11. Πετρόπουλου Π.Γ., «Μηχανουργική Τεχνολογία – ΙΙ. Τεχνολογία κατεργασιών κοπής των μετάλλων», 1998, Εκδόσεις Ζήτη
12. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ.Σαλωνίτης