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 13:
Advanced CNC Features

Table of Contents

Objectives of Chapter 13

- **Explain** the concept of **mirror imaging**
- **Decide** when the use of **mirror imaging** is appropriate
- Write simple programs in word address that **employ** **mirror imaging**
- **Explain** the concept of **polar rotation**
- **Decide** when the use of **polar rotation** is appropriate
- Write simple programs in word address that **employ** **polar rotation**
- Write simple programs in word address that **employ** **polar rotation used in a loop**

Objectives of Chapter 13

- **Explain** the concept of **helical interpolation**
- **Decide** when the use of **helical interpolation** is appropriate
- Write simple programs in word address that **employ** **helical interpolation**

Mirror Imaging

Mirror Imagining is used to reduce programming time when there is symmetry

- Depending on the **quadrant** of the mirror imaging some or all of the factors below will be affected :
  - Sign of the axis
  - Milling direction(up- or down- milling)
  - Arc rotation direction (clockwise or counterclockwise)

Mirror Imaging

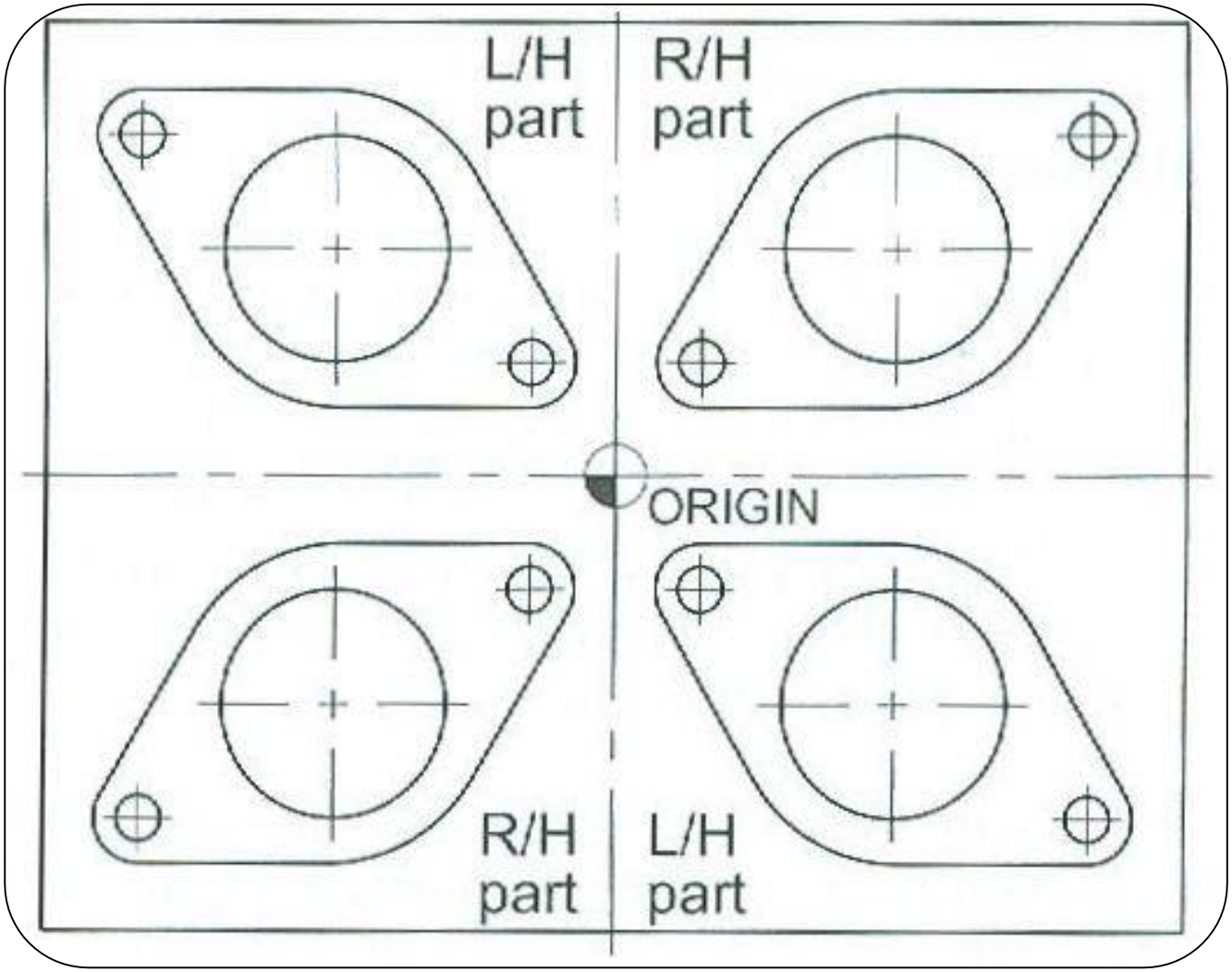


Figure 1:Mirror Imaging

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

Mirror Imaging

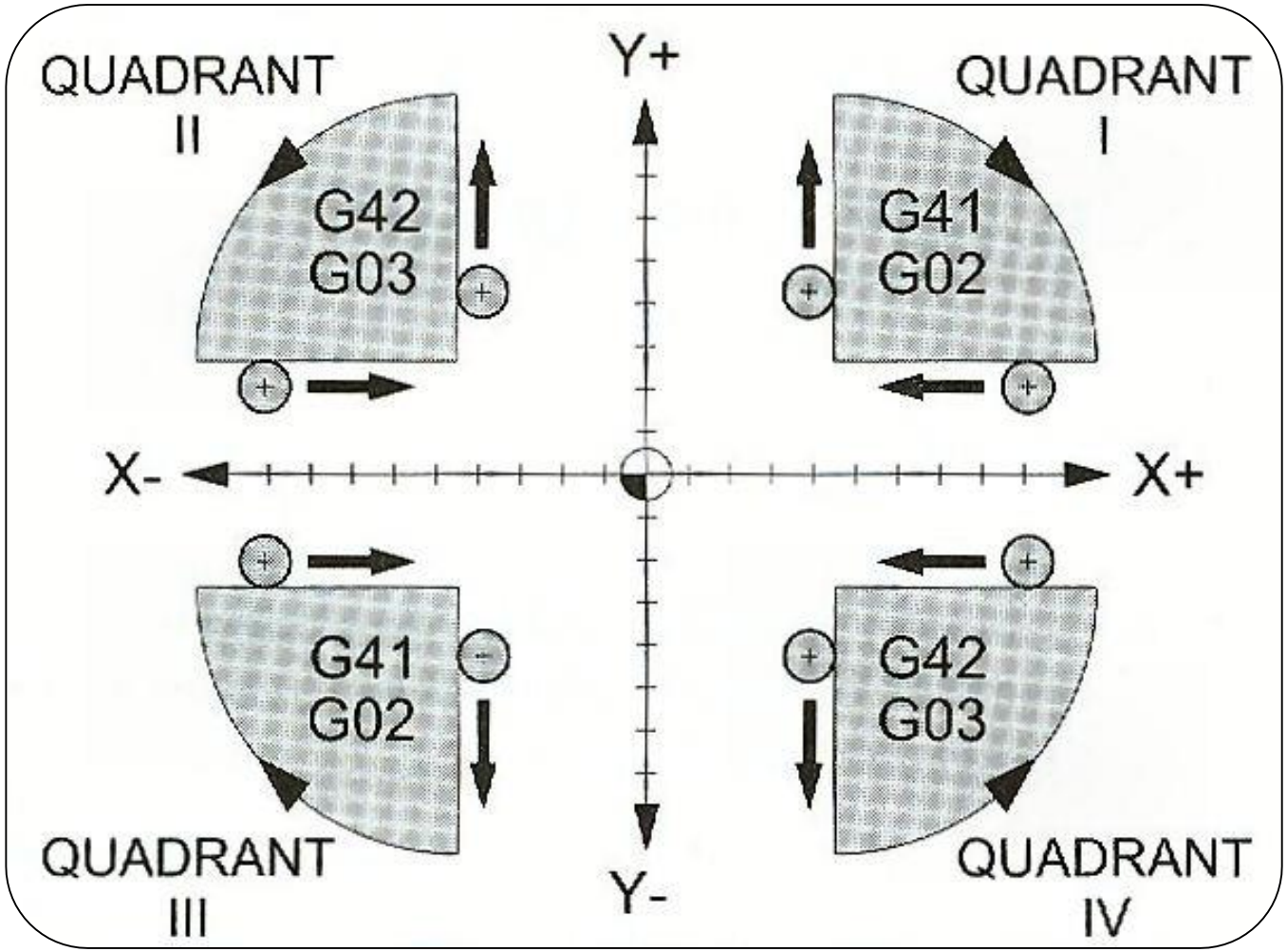
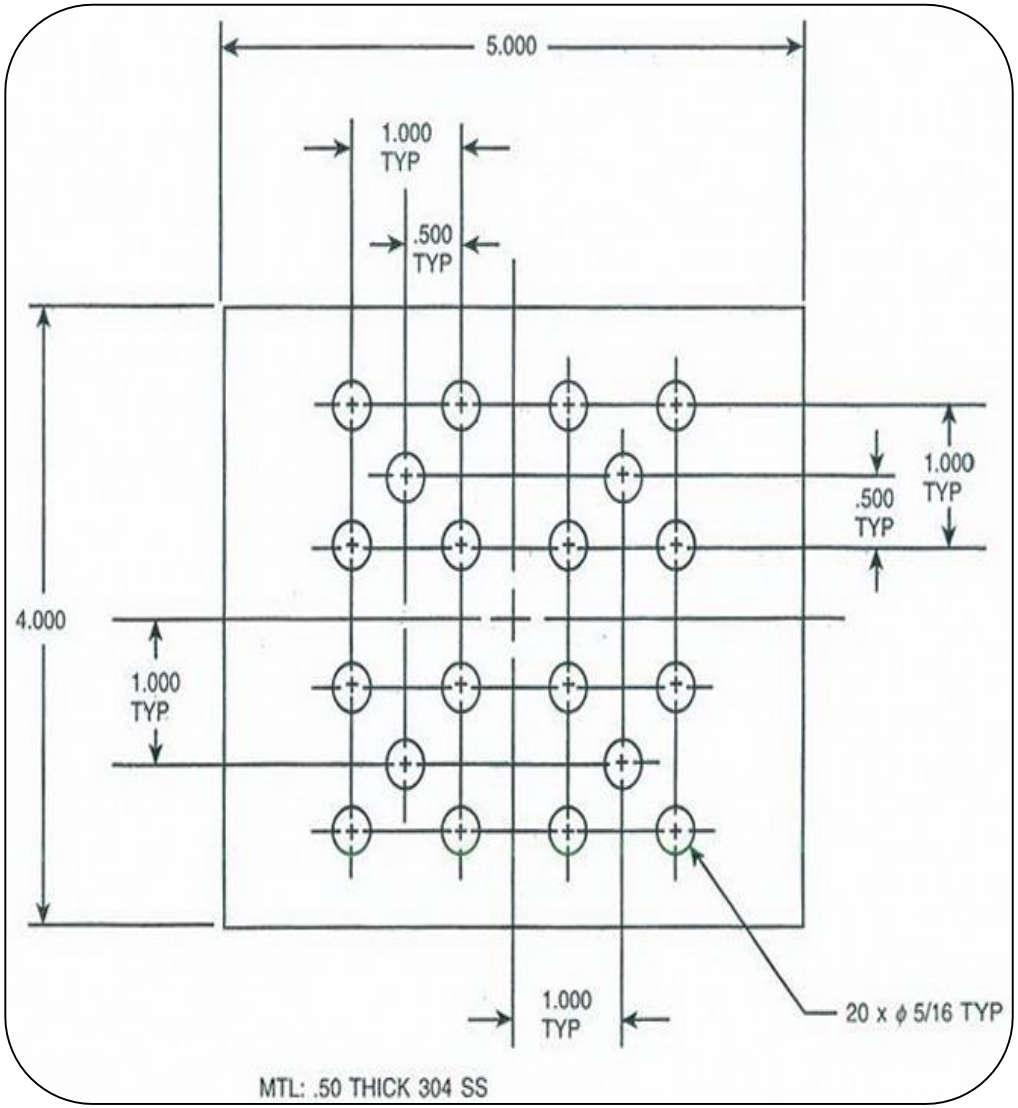


Figure 2: Depending on the quadrant important factors of the machining are affected

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

Mirror Imaging



- **Program Format**

M21 – Mirror image X axis

M22 – Mirror image Y axis

M23 – Mirror image off

Figure 3: Part drawing *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Mirror Imaging

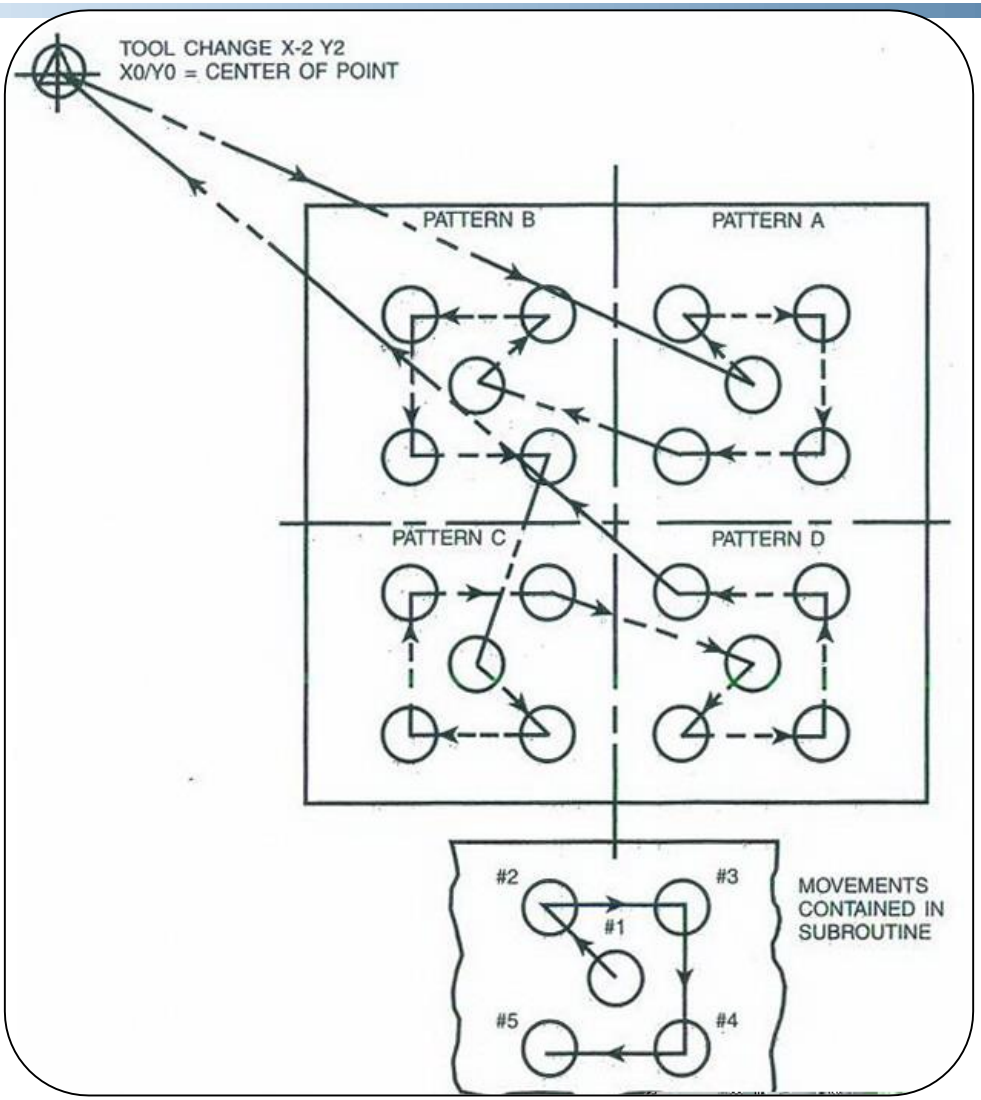


Figure 4: Tool path for the part shown in Figure 3 *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Mirror Imaging

```
%
O1203
(* **********)
(* X0/Y0 = CENTER OF PART)
(* THIS PROGRAM CALLS SUBPROGRAM 1000)
(* **********)
N001 G00 G40 G90 G70                    (SAFETY LINE)
N10 1G00 X0. Y0. S641 M03               (POSITION OVER PART CENTER)
N102 G45 Z0. H01 M08                    (PICK UP LENGTH OFFSET)
N103 M98 P1000 L1                       (CALL SUBPROGRAM)
N104 M21                                (MIRROR X-AXIS)
N105 M98 P1000 L1                       (CALL SUBPROGRAM)
N106 M22                                (MIRROR Y-AXIS)
N107 M98 P1000 L1                       (CALL SUBPROGRAM)
N108 M23                                (TURN OFF AXIS MIRROR)
N109 M22                                (MIRROR Y-AXIS)
N110 M98 P1000 L1                       (CALL SUBPROGRAM)
N111 G91 G28 Z0. M09                    (Z AXIS TO HOME ZERO)
N112 G28 X0. Y0. M05                    (X/Y TO HOME ZERO)
N113 M30                                (END OF PROGRAM)
01000
(* **********)
(* SUBPROGRAM FOR PROGRAM 1203)
(* **********)
N001 G81 G90 G99 X1. Y1. Z-.7 R0. F5.   (DRILL CYCLE - DRILL HOLE 1)
N002 G91 X-.5 Y-.5                      (DRILL HOLE 2)
N003 X1.                                (DRILL HOLE 3)
N004 Y-1.                               (DRILL HOLE 4)
N005 X-1.                               (DRILL HOLE 5)
N006 G80 G90                            (CANCEL DRILL CYCLE)
N007 M99                                (RETURN TO MAIN PROGRAM)
%
```

Figure 5: Program to machine part in Figure 3

Mirror Imaging

Program Explanation

N001
Safety line
N101
G00X0.Y0. — Positions spindle over the center of the part. **S641M03** — Turns on the spindle clockwise at 641 rpm.
N102
G45ZO.H01 —Turns on tool length offset using register 1. **M08** — Turns on the coolant
N103
M98P1000L1 — Executes subprogram 1000 one time. The part located in the first quadrant will be drilled.
N104
M21 —Turns on mirror imaging on the X axis.
N105
M**98P1000L1** — Executes subprogram 1000 one time. Since the X axis is mirrored, the part in the second quadrant will be drilled**.**
N106
M22 —Turns on mirror imaging on the Y axis.
N107
M98P1000L1 — Executes subprogram 1000 one time. Both the X and Y axes are mirrored. Therefore, the part in the third quadrant will be drilled.
N108
M23 —Turns off all mirror imaging.
N109
M22 — Turns on the Y axis mirror image. Only the Y axis is now mirrored.

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

Mirror Imaging

N110
M98P1000L1 — Executes subprogram 1000 one time. The part in the fourth quadrant will now be drilled.
N111
G91G28Z0. — Returns the Z axis to the home zero position. **M09**—Turns off the coolant.
N112
G28X0.Y0. — Returns the X and Y axes to the home zero position.
N113
M30 — End of program code.
Subprogram Explanation
N001
G81 —Turns on the drill cycle. **G90** — Places the MCU in absolute positioning mode. **G99** — Specifies a return to reference level when the Z axis retracts after each cycle iteration.
X1 .Y1. — Coordinates of hole #1.
Z-.7 — The final drill depth.
R0. — Specifies Z0. as the feed engagement point.
F5. — Sets the drilling feedrate at 5.0 inches per minute.
N002
G91 — Places the MCU in incremental positioning mode.
X-.5Y-.5 — Coordinates of hole #2.
N003
X1. — Coordinates of hole #3. The MCU is still in incremental mode.
N004
X-1. — Coordinate of hole #4.
N005
M99 — Returns the MCU to the main program.

Polar Rotation

Despite the differences in controllers, there is **certain information** that every MCU needs in order to carry out a **polar rotation**:

I. The **X axis coordinate** of the center of rotation

II. The **Y axis coordinate** of the center of rotation

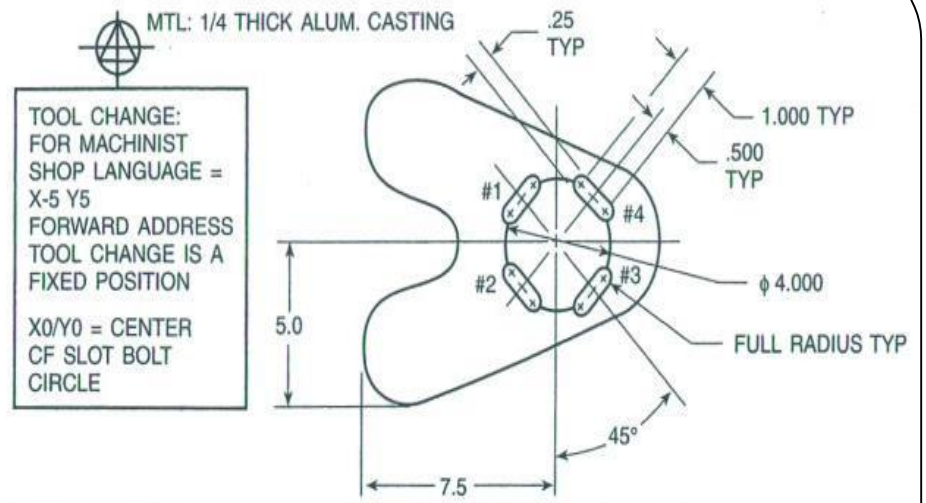
III. The **index angle**, or the ***angle as measured counterclockwise*** from the + X axis to the start of the rotation

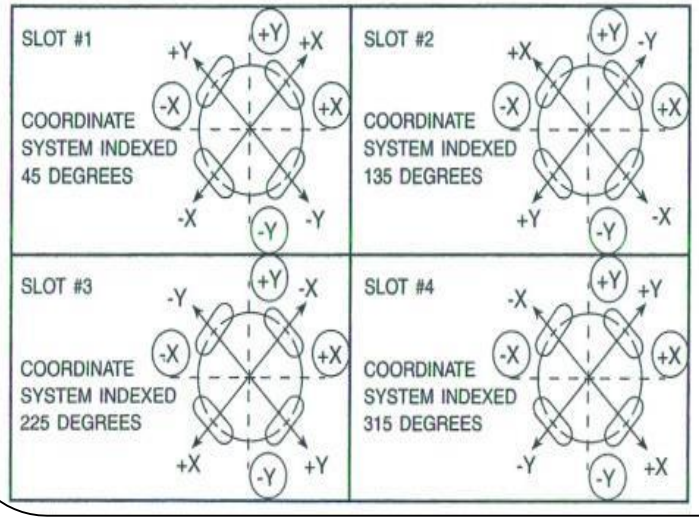
IV. **The amount of the rotation**. Following the initial rotation to the **index angle**, subsequent rotations may be specified ***as some angular value other than the index angle***. The rotations will occur in a **counterclockwise direction**

V. A **code to initiate** polar rotation

VI. A **code to cancel** polar rotation

Polar Rotation



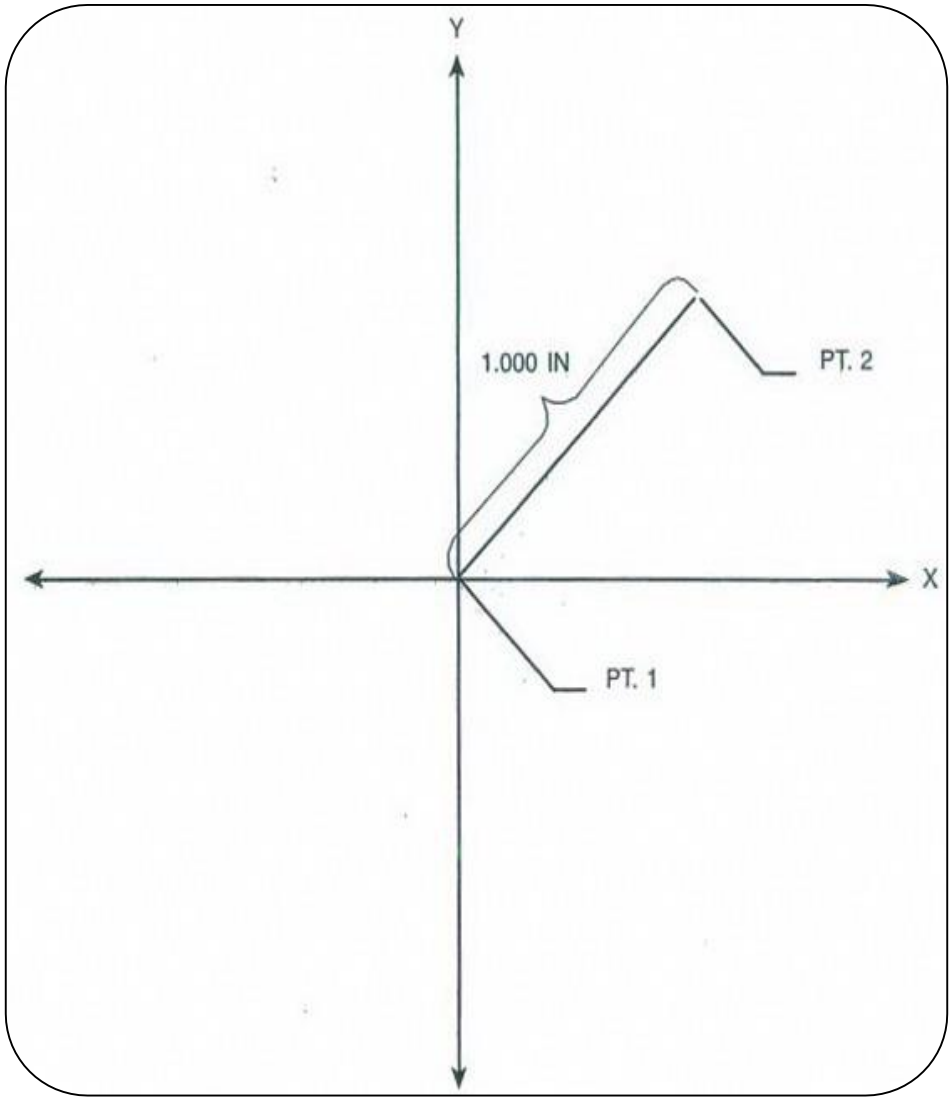


The amount of the rotation

- In the case shown in Figure 6, this amount is **90 degrees**

Figure 6: Part drawing *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Polar Rotation



- In other words, following the **initial index** of the coordinate system **45 degrees**, subsequent rotations will be **90 degrees** until the ***cancel command is given***

Figure 7:Polar Rotation *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Polar Rotation

```
%
O1206
(* **********)
(* X0/Y0 = CENTER OF PART)
(* THIS PROGRAM CALLS SUBPROGRAM 2000)
(* **********)
N001 G00 G40 G90 G70              (SAFETY LINE)
N101 G00 X0. Y0. S3500 M03        (INITIAL MOVE TO CENTER OF PART)
N102 G45 Z0. H01 M08              (PICK UP LENGTH OFFSET)
N103 G61 X0. Y0. A45 D90 L4       (INVOKE POLAR ROTATION)
N104 M98 P2000 L1                 (CALL SUBPROGRAM)
N105 G61                          (INVOKE POLAR ROTATION)
N106 M98 P2000 L1                 (CALL SUBPROGRAM)
N107 G61                          (INVOKE POLAR ROTATION)
N108 M98 P2000 L1                 (CALL SUBPROGRAM)
N109 G61                          (INVOKE POLAR ROTATION)
N110 M98 P2000 L1                 (CALL SUBPROGRAM)
N111 G60                          (CANCEL POLAR ROTATION)
N112 G00 G91 G28 Z0. M09          (Z AXIS TO HOME ZERO)
N113 G28 X0. Y0. M05              (X/Y TO HOME ZERO)
N114 M30                          (END OF PROGRAM)

02000
(* **********)
(* SUBPROGRAM FOR PROGRAM 1206)
(* **********)
N001 G00 X-.5 Y2.                 (POSITION TO THE SLOT)
N002 G01 Z-.36 F2.8               (FEED TO MILLING DEPTH)
N003 X.5                          (MILL SLOT)
N004 G00 Z0.                      (RAISE THE SPINDLE)
N005 M99                          (RETURN TO MAIN PROGRAM)
%
```

Figure 8: Program to machine part in Figure 6

Polar Rotation

Program Explanation

N001
Safely line.
N101
G00X0.Y0. — Positions the spindle at the center of the part.
S3500M03 —Turns on the spindle clockwise at 3,500 rpm.
N102
G45Z0.H01 — Turns on the tool length offset compensation. **M08** —Turns on the coolant.
N103
G61 — Polar relation on code.
X0.Y0. — Center point of the rotation.
A45 — Starting index angle of rotation.
D90 — Amount to rotate coordinate system with each **G61** call
L4 — Tells the MCU that 4 rotations will occur.
N104
M98P2000L1 — Jumps to subprogram O2000.
N105
G61 — Causes the next polar rotation to occur.
N106
M98P2000L1 — Jumps to subprogram O2000.
N107
G61 — Causes the next polar rotation to occur.
N108
M98P2000L1 — Jumps to subprogram O2000.
N109
G61 — Causes the next polar rotation to occur.

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

Polar Rotation

N110
M98P2000L1 — Jumps to subprogram O2000.
N111
G60 — Turns off the polar rotation.
N112
G00G91G28Z0. — Sends the Z axis to home zero.
M09 — Turns oft the coolant.
N113
G28X0.Y0. — Sends the X/Y axes to home zero
M05 —Turns off the spindle.
N114
M30 — End of program code.

Subprogram Explanation N001

G00X-.5Y2. — Positions spindle to the start of the slot.
N002
G01Z-.36F2.8 — Feeds the spindle to milling depth at a feedrate ■ inches per minute.
N003
X.5 — Positions the spindle at feedrate to the end of the slot.
N004
G00Z0. – Retracts the cutter from the slot
N005
M99 – Returns the MCU to the main program

Helical Interpolation

- **Helical interpolation** is **circular interpolation** with two axes while **simultaneously feeding** at a linear rate with the third. The result of this type of operation is a **helix**

Care must be taken in ***calculating the number of turns*** and the lead of a **helix**, be it a thread or other type of part

- **Helical interpolation** may be used inside of or in conjunction with **do loops** and **subroutines**

Helical Interpolation

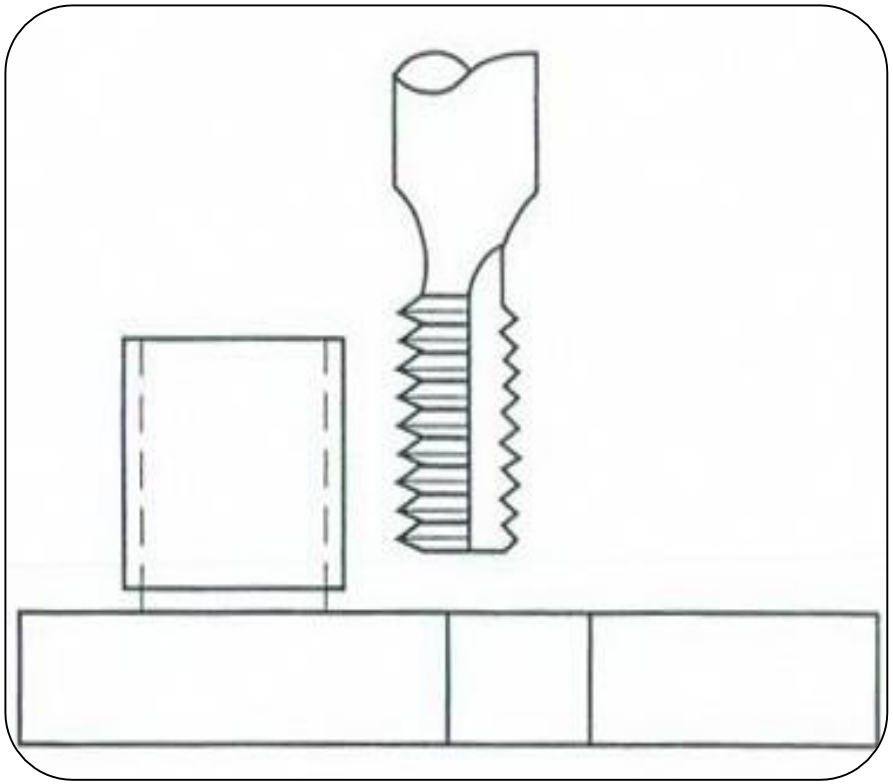


Figure 9: Thread milling cutter setup

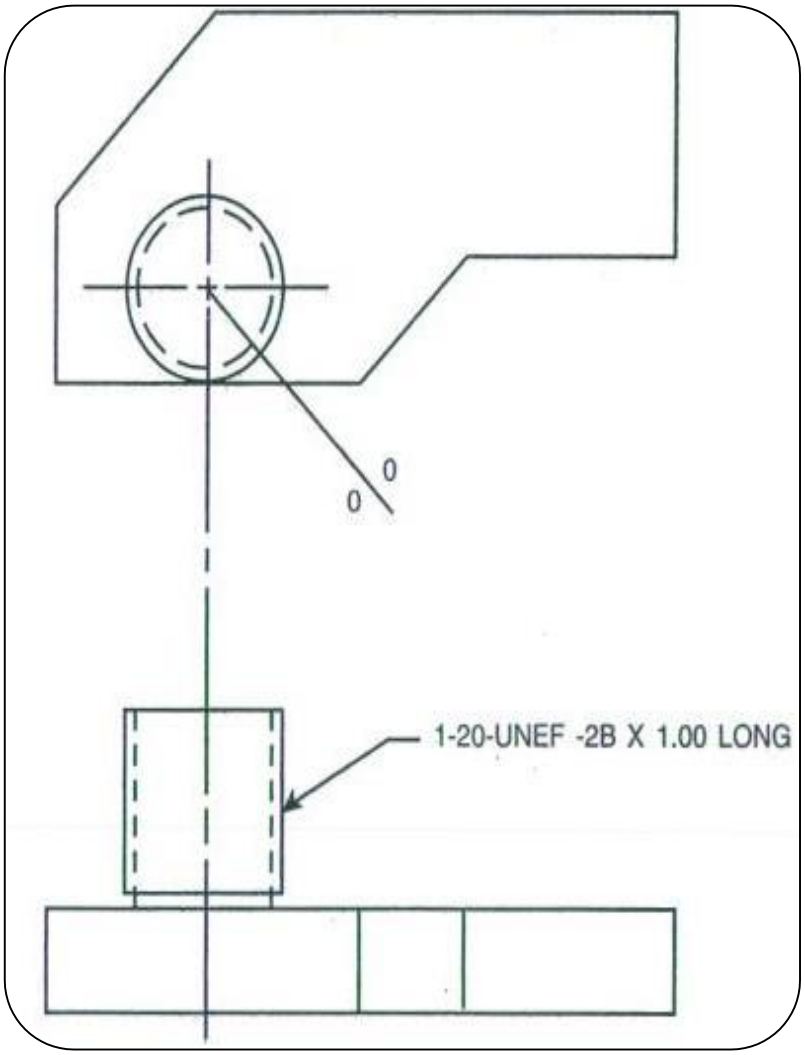
(Seams W., "Computer Numerical Control, Concepts & Programming")

Program Format

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

- For the X/Y plane – **G17 G02/G03** X…Y…I…J…Z…F…
- For the X/Z plane – **G18 G02/G03** X…Y…I…J…Z…F…
- For the Y/Z plane – **G19 G02/G03** X…Y…I…J…Z…F…

Helical Interpolation




```
%
O1209
N001 G00 G80 G90 G40                    (SAFETY LINE)
N101 G00 X1.6 Y0. S800 M03              (POSITION TO START POINT)
N102 G45 Z-.818 H01 M08                 (FEED TO START DEPTH)
N103 G01 X.9694 F7.                     (FEED INTO WORKPIECE)
N104 G17 G02 X.9694 Y0. Z-.868I-1. J0.  (FIRST HELICAL TURN)
N105 G02 X.9694 Y0. Z-.918 I-1. J0.     (SECOND HELICAL TURN)
N106 G02 X.9694 Y0. Z-.968 I-1. J0.     (THIRD HELICAL TURN)
N107 G01 X1.6                           (FEED OUT OF WORKPIECE)
N108 G00 G91 G28 Z0. M09                (Z-AXIS TO HOME ZERO)
N109 G28 X0. Y0. M05                    (X/Y AXES TO HOME ZERO)
N110 M30                                (END PROGRAM)
%
```

Figure 10(a): Part drawing,(b) Program to mill part in Figure 10

Helical Interpolation

Program Explanation

N001
Safety line
N101
G00X1.6Y0. ~ Positions spindle to the start point.
S800M03 — Turns on spindle clockwise at 800 rpm.
N102
G45Z-.818H01 — Rapids spindle to starting depth and turns on the tool length offset.
M08 — Turns on the coolant
N103
G01X.9694F7. — Feeds the cutter into the workpiece at 7.0 inches per minute.
N104
G17 — Specifies the X/Y plane as the circular interpolation plane.
G02 — Initiates circular interpolation clockwise.
X.9694Y0. — The end coordinates of the circular cut (in this case 360 degrees}.
2-.918 —The Z coordinate for the helical cut. The Z axis will feed downward to this coordinate during the circular X/Y motion.
I-1 J0.—The X/Y vector from the center of the cutter to the center of the arc.
N105
G02 — Circular interpolation cods.
X.9694Y0. — The end coordinates of the circular cut (in this case 360 degrees).
Z-.918 — The Z coordinate for the helical cut.
I-1 J0.—The X/Y vector from the center of the cutter to the center of the arc.

Helical Interpolation

N106
G02 — Circular interpolation code.
X.9694Y0. — The end coordinates of the circular cut (in this case 360 degrees)
Z-.968 — The Z coordinate for the helical cut.
I-1 JO. — The X/Y vector from the center of the cutter to the center of the ate.
N107
G01 — Puts the control back in linear interpolation mode.
X1.6 — Feeds the spindle clear of the part.
N108
G00G91G28Z0.M09 — Sends the spindle to Z home position and turns off the coolant.
N109
G28X0.Y0.M05 — Sends the spindle to the X/Y home position and turns off the spindle.
N110
M30 — End of program code.

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

Helical Interpolation

```
%
O1211
N001 G00 G80 G90 G40                    (SAFETY LINE)
N101 G00 X.9694 Y0. S2000 M03           (POSITION TO START POINT)
N102 G45 Z.1 H01 M08                    (FEED TO START DEPTH)
N103 G17 G02 X.9694 Y0. Z.05 I-1. J0. F25.   (TURN1)
N104 G02 X.9694 Y0. Z.0 I-1. J0.              (TURN 2)
N105 G02 X.9694 Y0. Z-.05 I-1. J0.            (TURN 3)
N106 G02 X.9694 Y0. Z-.1 I-1. J0.             (TURN 4)
N107 G02 X.9694 Y0. Z-.15 I-1. J0.            (TURN 5)
N108 G02 X.9694 Y0. Z-.2 I-1. J0.             (TURN 6)
N109 G02 X.9694 Y0. Z-.25 I-1. J0.            (TURN 7)
N110 G02 X.9694 Y0. Z-.3 I-1. J0.             (TURN 9)
N111 G02 X.9694 Y0. Z-.35 I-1. J0.            (TURN 10)
N112 G02 X.9694 Y0. Z-.4 I-1. J0.             (TURN 11)
N113 G02 X.9694 Y0. Z-.45 I-1. J0.            (TURN 12)
N114 G02 X.9694 Y0. Z-.5 I-1. J0.             (TURN 13)
N115 G02 X.9694 Y0. Z-.55 I-1. J0.            (TURN 14)
N116 G02 X.9694 Y0. Z-.6 I-1. J0.             (TURN 15)
N117 G02 X.9694 Y0. Z-.65 I-1. J0.            (TURN 16)
N118 G02 X.9694 Y0. Z-.7 I-1. J0.             (TURN 17)
N119 G02 X.9694 Y0. Z-.75 I-1. J0.            (TURN 18)
N120 G02 X.9694 Y0. Z-.8 I-1. J0.             (TURN 19)
N121 G02 X.9694 Y0. Z-.85 I-1. J0.            (TURN 20)
N122 G02 X.9694 Y0. Z-.9 I-1. J0.             (TURN 21)
N123 G02 X.9694 Y0. Z-.95 I-1. J0.            (TURN 22)
N124 G02 X.9694 Y0. Z-1. I-1. J0.             (TURN 23)
N125 G01 X1.6                           (FEED OUT OF WORKPIECE)
N126 G00 G91 G28 Z0. M09                (Z AXIS TO HOME ZERO)
N127 G28 X0. Y0. M05                    (X/Y AXES TO HOME ZERO)
N128 M30                                (END PROGRAM)
```

Figure 11: Program to mill part in Figure 10

Helical Interpolation

Program Explanation

N001
Safety line.
N101
G00 X.9694 Y0. — Positions spindle to the start point
S2000M03 — Turns on spindle clockwise at 2,000 rpm
N102
G45 Z.1 H01 M08 — Rapids spindle to the starting depth (.100 a top of the part) and turns on the tool length offset code.
N103
G17 — Specifies the X/Y plane as the circular interpolation plane. **G02** — Initiates circular interpolation clockwise
X.9694 Y0 — The end coordinates of the circular cut (in this case, 360 degrees)
Z.05 — The Z coordinate for the helical cut
I-1. JO—The X/Y vector from the center of the cutter to the center of the
F25. — Specifies a feedrate of 25 inches per minute for the motion
N104 through N124
Helical motion lines to make turns 2 through 23 of the cutter. Notice the Z axis feed downward .050 inch with each turn. Since the **G17** plane designation command is modal, it is not necessary to specify a **G17** with each line. On most CNC controls, the circular interpolation codes **G02/G03** are also modal and could be omitted as well. Many programmers, however, prefer to specify a **G02/G03** at each circular motion to help ***clarify the program's Intent***
N125
G01 X1.6 — Moves the cutter away from the workpiece at feedrate
N126 through N128
Sends the machine to **home zero** and **terminates** the program

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

Summary 1/2

- **Mirror imaging** means changing the **sign (+ or -)** of an axis movement
- **Mirror imaging** is used in a program to ***save repetitive programming*** when the direction of movement is the only difference between part features
- **Mirror imaging** is normally used in conjunction **with subroutines** or **do loops**
- **Polar rotation** is an **indexing** of the NC machine's Cartesian coordinate system to some **angle** other than its normal state
- **Polar rotation** may be used to perform operations that otherwise would require the use of a **rotary axis** or lengthy coordinate calculations

Summary 2/2

- **Polar rotations** may be used in conjunction with **do loops** or **subroutines**
- **Helical interpolation** is **circular interpolation** with two axes while **simultaneously feeding** at a linear rate with the third. The result of this type of operation is a **helix**
- **Care** must be taken in ***calculating the number of turns*** and the lead of a **helix**, be it a thread or other type of part
- **Helical interpolation** may be used inside of or in conjunction with **do loops** and **subroutines**

Vocabulary Introduced in this chapter

- Helical interpolation
- Mirror imaging
- Polar axis system
- Polar rotation
- Tread lead

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 Ι και ΙΙ, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2001,
8. Γ. Χρυσολούρης, Δ. Μούρτζης, Κ. Τσίρμπας, Σ. Καραγιάννης, “Ορθογωνική Κοπή”, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2000

References

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)

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