COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

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

LMS

Dr. Dimitris Mourtzis

Associate professor

Patras, 2017

Chapter 15:
Programming CNC Turning Machines

Table of Contents

Objectives

- Write simple **turning** and **facing** routines

- Write simple **taper turning** routines

- Write simple routines to perform **circular interpolation** using programmed arc centres and programmed radius value methods

- Write simple **thread-turning** routines using single-pass and multi-pass threading

Introduction

- CNC lathe controllers **vary in their coding** to an even greater extent than mill controllers. It is, therefore, **difficult to discuss programming practices**
- **EIA standards specify axis movement**, for example, but some lathes use a ***left-hand coordinate system***, with the X and Z-axes **reversed** from the standard configuration
- Other lathes **reverse the X-axis** direction and not the Z. On lathes using twin turrets, the **X-axis is often reversed**
- The uses of coding and the cycles available also **differ** to a large extent. The EIA codes pertaining to lathes are generally used, but many other codes may be added
- This chapter will discuss basic lathe programming routines for **turning, facing, taper turning, circular interpolation, and thread cutting**
- Each routine is placed in a **miniprogram**. ***Each program can be thought of as a building block***: To machine a complete part, these building blocks can be **linked together** in one program as will be demonstrated

Machine Reference Point

- A machine ***reference point*** is a fixed position on the machine. Upon receiving the proper **G code**, the machine ***automatically returns to the reference point*** location
- This point is often the **home zero location** used for tool changing and as a **park position** at the end of the program
- Often it is necessary to send the tool back to the reference point by way of another point, called an ***intermediate point***
- The code used in this chapter to ***return the tool to reference*** is **G28**. U- and W-axis coordinates are specified along with the **G28**. A command of "**G28 UO. WO**." returns the tool to the ***home zero location***
- A value other than "**UO WO**" specifies the **intermediate point** the tool must pass through on its return to the home zero location
- A command of "**G28 U.5 W1**." returns the tool to ***home zero***, passing through a point located at X.500 and Z1.000

Diameter Versus Radius Programming

- The difference between ***radius programming*** and ***diameter programming*** is an important one
- ***Diameter programming*** references the X-axis coordinate to the diameter of the workpiece
- This means that ***every .001 inch programmed moves the tool .0005 inch*** as measured radially
- If the ***X axis advances .500 inch into the part, .500 inch is removed from the diameter***. To accomplish this, the X axis moves only .250 inch, or half the programmed amount
- In ***radius programming***, the X axis moves the **programmed amount**
- If .500 inch of movement along the X axis is programmed, the tool advances .500 inch. When the Z-axis move is made, 1.000 inch of material is **removed** from the part

Diameter Versus Radius Programming

- The MCU on a CNC lathe expects the X-axis coordinates to be entered in either **diameter coordinates** or **radius coordinates**
- The two methods ***cannot be mixed*** in one program. Diameter coordinates ***cannot be used on a control set up for radius coordinates*** and vice versa
- The machine manual must be consulted to determine the type of coordinate expected
- The coordinates may be either **incremental** or **absolute**, depending on whether **G90** or **G91** is active
- As **in milling**, **G90** selects ***absolute positioning*** and **G91** selects ***incrementa***l
- Other controllers use a "**W**" address for ***incremental X*** and a "**U**" address for ***incremental Z***

Turning and Facing

- Figure 1 shows a part to be **turned** and **faced** in a lathe. Note that the position of the tool turret relative to the X0/Y0 location and the machine origin is given

- The ***machine coordinate system may be transferred to the work-piece*** either within the program by use of G codes or by the operator during machine setup

- It is usually more **efficient** to ***define the work coordinate system during setup***

- For routines in this chapter, this will be assumed. Figure 2 shows a part similar to the one in Figure 1 but with metric dimensions.

- Figure 3 presents a short program to turn and face the part drawn in Figure 1. Figure 4 presents a metric version

Turning and Facing

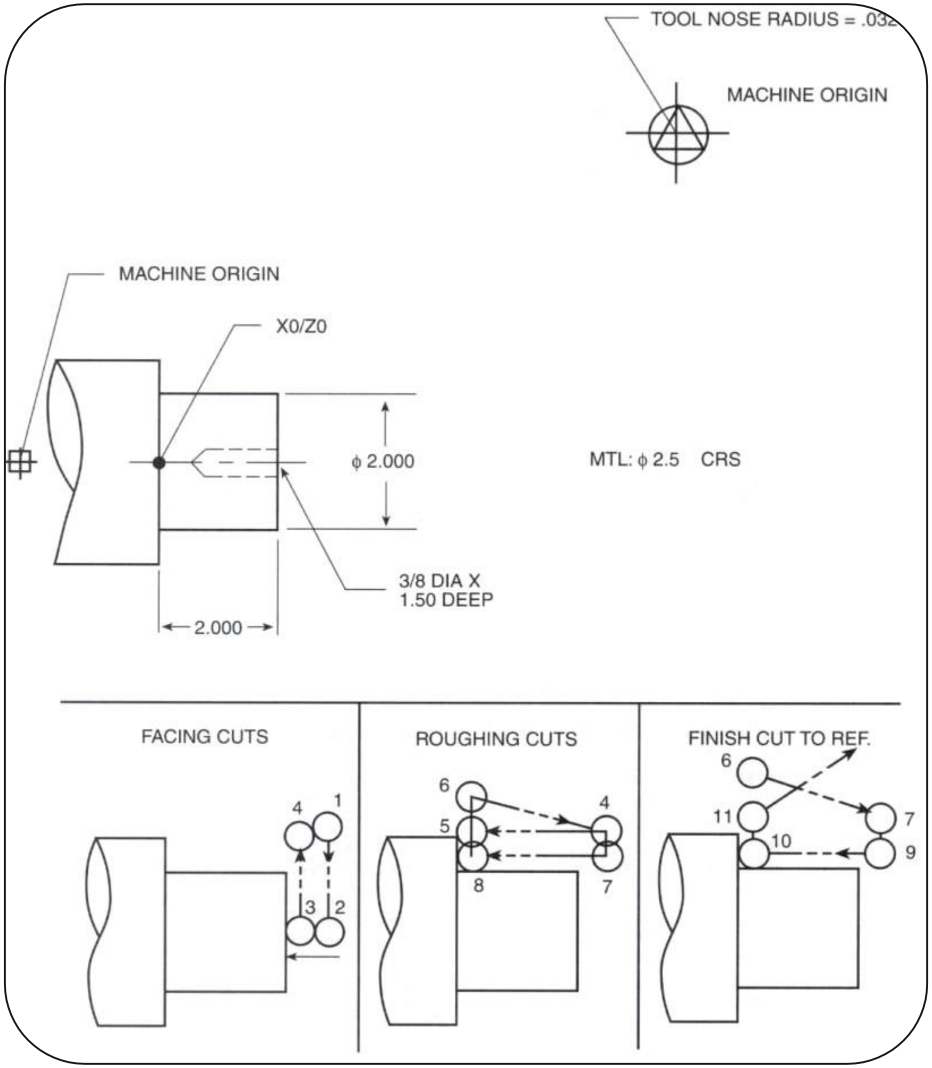
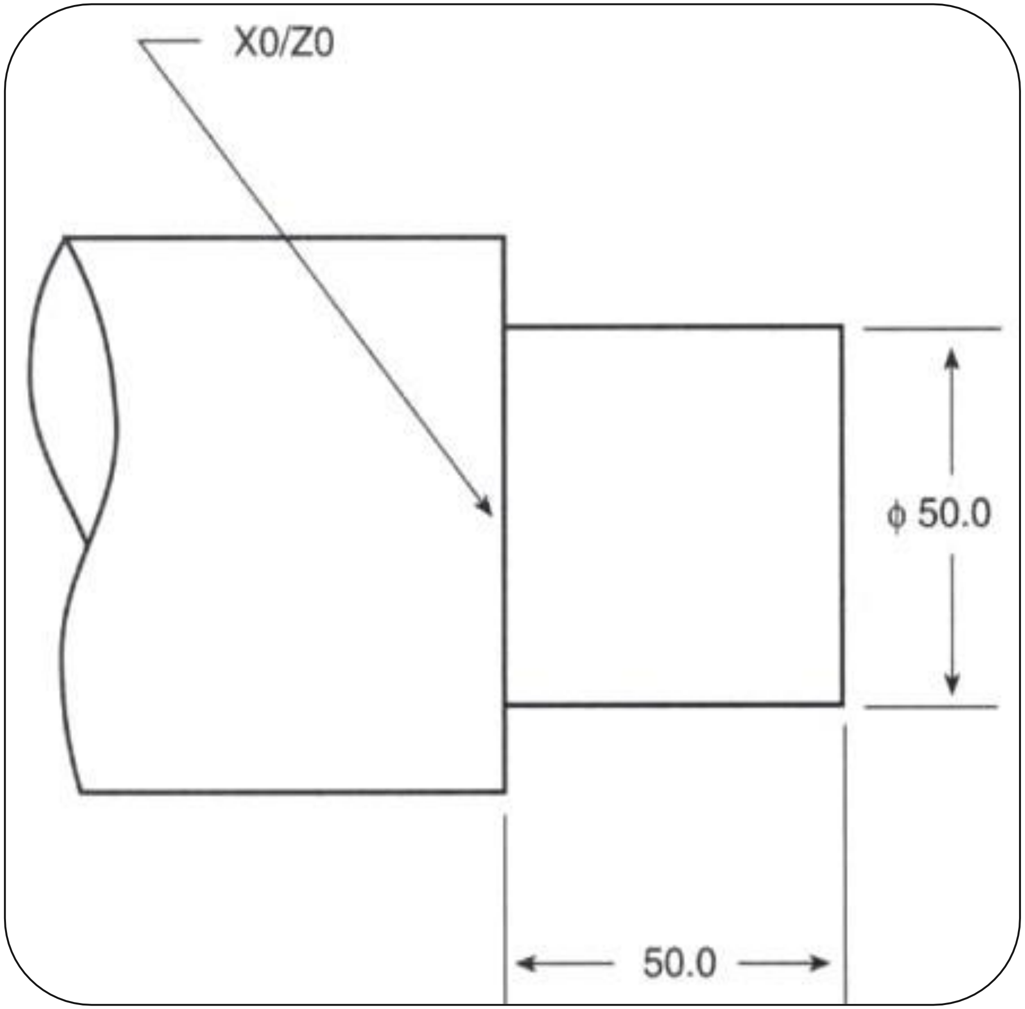


Figure 1 : Part to be turned and faced in a lathe

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

- Figure 1 shows a part to be **turned** and **faced** in a lathe
- Note that the position of the ***tool turret*** relative to the X0/Y0 location and the ***machine origin*** is given

Turning and Facing



- Figure 2 shows a part similar to the one in Figure 1 but with ***metric dimensions***

Figure 2: Part from figure 1 with metric dimensions

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

Turning and Facing

```
%
O1403
(* **********)
(* X0 = CENTERLINE OF SPINDLE)
(* Z0 = PART SHOULDER)
(* **********)
N010 G00 G99 M08              (SAFETY LINE, COOLNT ON)
N020 T0101 M42                TURRET POS, HIGH RANGE)
N030 S1200 M03                (SPINDLE ON)
N040 X2.6 Z2.042              (POSITION TO #1)
N050 G01 X0. F.007            (FEED TO #2)
N060 Z2.032                   (FEED TO #3)
N070 X2.314 F.003             (FEED TO #4)
N080 Z.042 F.007              (FEED TO #5)
N090 X2.6                     (FEED TO #6)
N100 G00X2.320 Z2.132         (RAPID TO #4)
N110 G01 X2.0840              (FEED TO #7)
N120 Z.042 F.003              (FEED TO #8)
N130 X2.6                     (FEED TO #6)
N140 G00 X2.084 Z2.132        (RAPID TO #7)
N150 G00 X2.062               (FEED TO #9)
N160 Z.032 F.003              (FEED TO #10)
N170 X2.55                    (FEED TO #11)
N180 G00 G28 U0. W0. M09      (RETURN TO HOME/COOLNT OFF)
N190M05                       (SPINDLE OFF)
N200M30                       (END PRGM)
%
```

Figure 3 Program to turn and face part in Figure 1

- Figure 3 presents a short program to **turn** and **face** the part drawn in Figure 1

Turning and Facing

```
%
O1404
(* **********)
(* X0 = CENTERLINE OF SPNIDLE)
(* Z0 = PART SHOULDER)
(* **********)
N010 G00 G99 M08             (SAFETY LINE)
N020 T0101 M42               (TURRET POS, HIGH RANGE)
N030 S1200 M03               (SET SPEED)
N040 X67. Z52.               (POSITION TO #1)
N050 G01 X0. F.5             (FEED TO #2)
N060 Z51.                    (FEED TO #3)
N070 X60. F.13               (FEED TO #4)
N080 Z2. F.5                 (FEED TO #5)
N090 X67.                    (FEED TO # 6)
N100 G00 X60. Z101           (RAPID TO #4)
N110 G01 X53.                (FEED TO #7)
N120 Z2 F.13                 (FEED TO #8)
N130 X67.                    (FEED TO #6)
N140 G00 X53. Z101.          (RAPID TO #7)
N150 G01 X51.                (FEED TO #9)
N160 Z1.                     (FEED TO #10)
N170 X66.                    (FEED TO #11)
N180 G00 G28 U0. W0. M09     (RETURN TO REF & COOLNT OFF)
N190 M05                     (SPINDLE OFF)
N200 M30                     (END PGRM)
%
```

- Figure 4 presents a **metric version** of a short program to **turn** and **face** the part drawn in Figure 1

Figure 4: Program to turn and face part in Figure 2

Turning and Facing

- **G00**—As in milling programs, **G00** puts the machine in **rapid traverse mode**

- **G01—Linear interpolation**. As with milling, the machine will position the tool to the programmed coordinates at feedrate, in a straight line

- **G28—Return to reference point**. A **G28** is programmed with a U and W coordinate. Upon receiving the **G28**, the machine **positions the tool at the fixed machine reference point**

- **G99—Selects inches per revolution or millimeters per revolution feed-rates**. The **feedrates** are the programmed value per revolution of the spindle. A **G95 F.01** advances the tool .010 inch for every revolution of the spindle

> - **M40—**Selects the **low gear range.**
> - **M41—**Selects the **middle gear range.**
> - **M42—**Selects the **high gear range**.

Turning and Facing

N010

G00 — Selects the rapid traverse mode. **G99** — Selects per revolution feedrate. **M08** —Turns on the coolant.

N020

T0101 —Selects a tool number and calls the tool offset in register #1.

M42 — Selects high gear range.

N030

S1200 — Sets the spindle speed to 1200 rpm.

M03 — Turns on the spindle.

N040

X/Z coordinates — Rapid the tool to location #1, Figure 1 .The X-axis coordinate is diameter programmed, as are all the X coordinates in this program.

N050

G01 — Selects feedrate movement.

X0 — Feeds the tool to location #2. This is the rough facing cut.

F.007 — Sets the feedrate to .007 inch per spindle revolution (.5 mm metric)

N060

Z **coordinate** — Feeds the tool from location #2 to location S3. This sets the Z axis depth for the finish facing cut.

N070

X coordinate — Feeds the tool from location #3 to location #4. The coordinate is diameter programmed.

F.003 (F.13 metric) — Sets finish feedrate.

Program Explanation

Turning and Facing

N080

Z coordinate — Feeds the tool from location #4 to location #5. This is the first roughing pass.

F.007 (F0.5 metric) — Sets the roughing pass feedrate.

N090

X coordinate — To feed from location #5 to location #6. This cut rough faces the shoulder of the part and retracts the tool for the return move.

G00 — Selects rapid traverse. This is a return to start of cut move. No feedrate is necessary.

X/Z coordinates — Move the tool at rapid from location #6 to location #4.

N110

G01 — Selects linear interpolation (feedrate mode).

X coordinate — Feeds the tool from location #4 to location #7. This move could also have been made in rapid traverse. Using a feedrate here eliminated the possibility of chipping the tool cutting edge on the corner of the stock.

Z coordinate — Feeds the tool from location #7 to location #8. This is the second rough turning pass.

F.003 (F.13 metric) — Sets finish feedrate.

N130

X coordinate — Rough faces the shoulder, retracting the tool.

N140

G00 — Selects rapid traverse.

X/Z coordinate — Positions the tool to location #7.

Turning and Facing

N150

G01 — Selects feedrate movement.

X coordinate — Feeds the tool from location #7 to location #9. This positions the X axis depth for the finish pass.

N160

Z coordinate — Feeds the tool from location #9 to location #10. This completes the turning.

N170

X coordinate— Feeds the tool from location #10 to location #11.This move finish faces the part shoulder.

N180

G00 — Selects rapid traverse.

G28U0.W0. — Initiates a return to reference.

M09 — Turns off the coolant.

N190

M05 —Turns off the spindle.

N200

M30 — Signals the end of program.

Taper Turning

- **Linear interpolation** on a lathe is used to **turn tapers**. It is similar in use to linear interpolation to cut angles when milling. In Figure 5, the part pictured is a taper to be bored

- The part is also a steel casting, requiring ***that the taper be rough*** and then finish machined. (A short program to perform these operations is shown in Figure 7)

- **Cutter offset calculations** that are necessary with taper turning are similar to those used when calculating angle cuts for milling. Figure 6 depicts the ***relationship of the lathe tool nose to the tapered part surfaces***

- Two coordinate locations require **cutter offsets**. Both locations present the identical situation so that ***calculating one offset will automatically yield the other***

Taper Turning

- In this case, the Y axis in the formula is the X axis on the lathe, and the X axis in the formula is the Z axis on the lathe

- The **offset** is calculated as follows, where **CR** is the **tool nose radius:**

$$X = TAN\left(\frac{\theta}{2}\right) \times CR \Rightarrow X = TAN\,40 \times 0.32 \Rightarrow X = 0.2685\,or\,0.027$$

Taper Turning

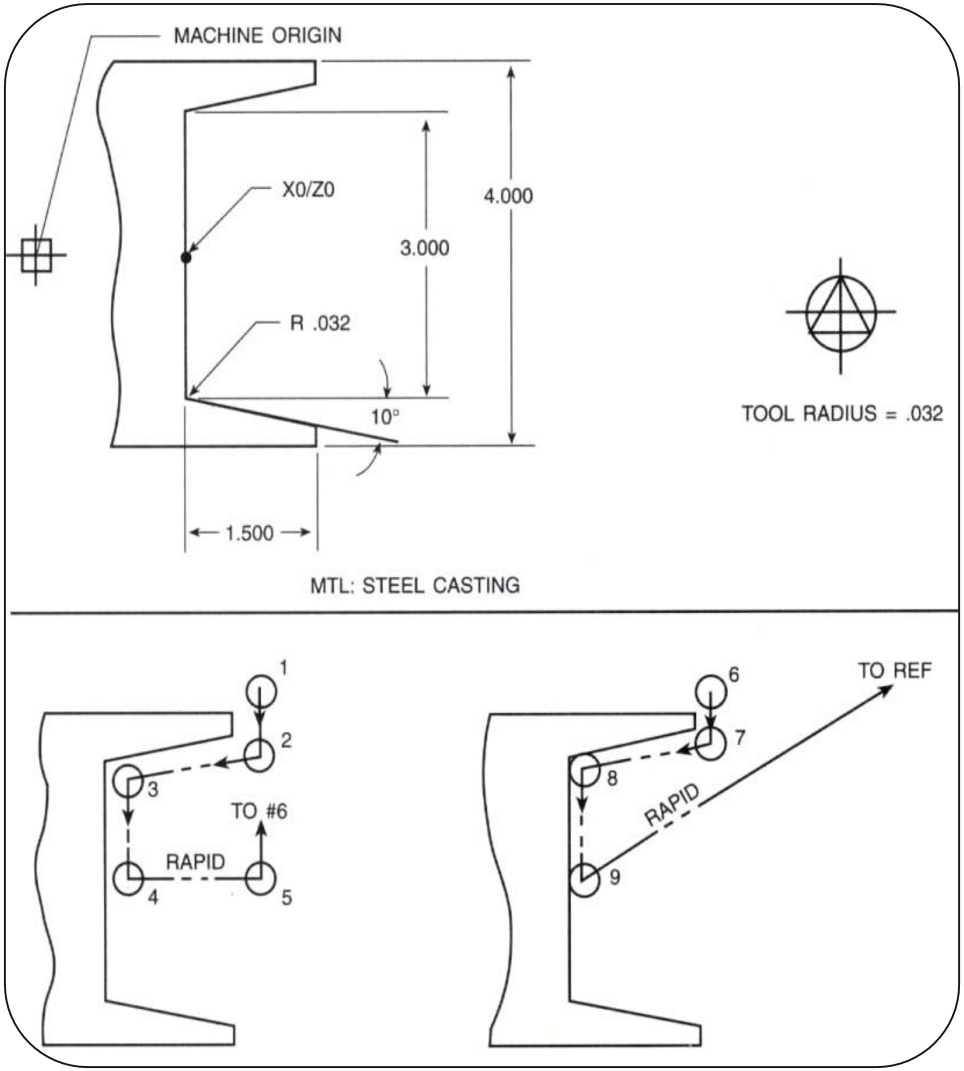
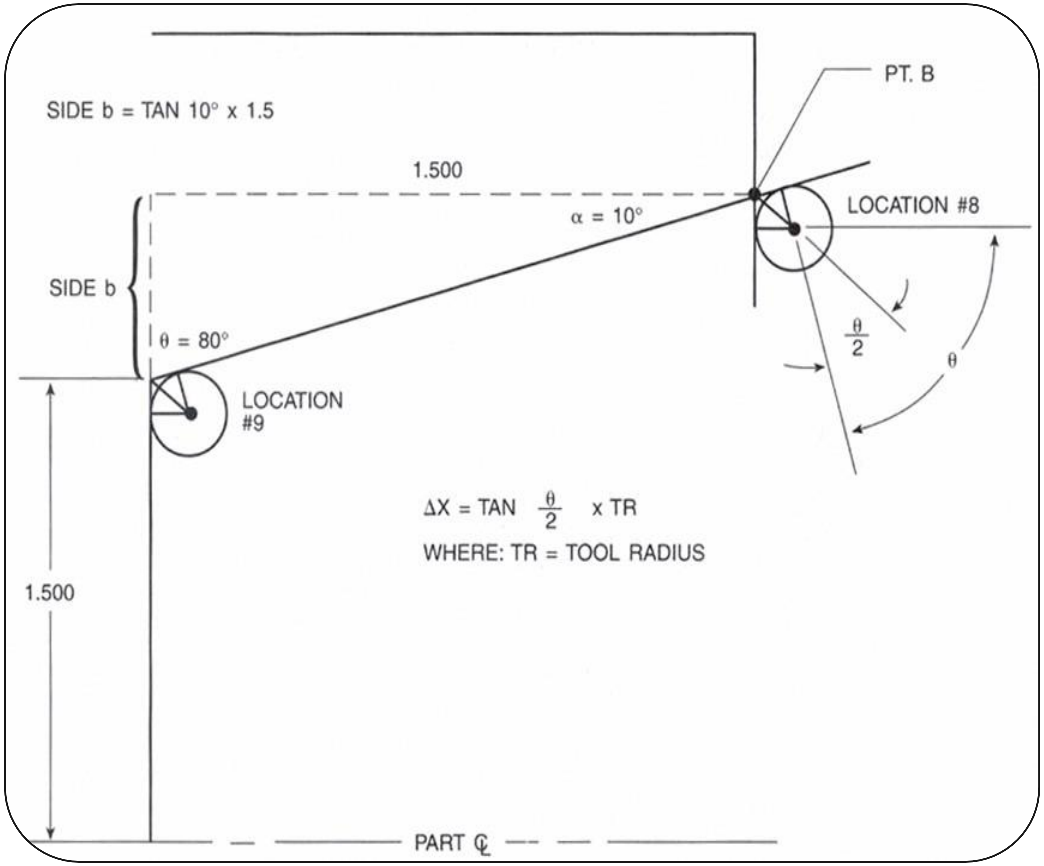


Figure 5: Taper turning

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

- In Figure 5, the part pictured is a **taper to be bored**

Taper Turning



- Figure 6 depicts the **relationship of the lathe tool nose to the tapered part surfaces**

Figure 6: Determining cutter offsets

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

Taper Turning

- Before the **cutter offset** can be used, however, it is necessary to **calculate the location of point B**, Figure 6

- By **solving the indicated triangle for side b** and **adding that length to the known radius of the taper** (1.5 inches), **the radius dimension** from the part center line to **point B can be determined**

$$\frac{b}{1.5} = \mathrm{TAN}10 \Rightarrow b = \mathrm{TAN}10 \times 1.5 \Rightarrow b = 0.26445 \; or \; 0.264$$

- The value of .264 added to the 1.5 radius gives a distance of 1.764 from the part centerline to point B

- The **cutter offset** can be subtracted from the 1.764 distance to find the dimension from the part centerline to cutter location #7

Taper Turning

- This distance is 1.737. The X coordinate for this location, however, will be diameter programmed

- The 1.737 must now be **doubled to arrive at the X coordinate** to be programmed, or 3.474.

- The calculated **tool offset can also be subtracted from the 1.5 known radius** to arrive at the 1.473 dimension from the part centerline to tool location **#8**

- **Doubling** this distance gives 2.946, the X-axis coordinate for location **#8**

- The **offset** for the Z axis in both these cases is simply the ***radius of the tool nose***

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

Taper Turning

Program Explanation

N010

G00 — Selects rapid traverse.

G99 — Specifies inches per revolution feedrate.

M08 —Turns the coolant on.

N020

T0101 — Select the tool and the offset. M42 — Selects high gear range.

N030

S800 — Sets the spindle speed to 800 rpm. M03 — Turns on the spindle.

N040

X4.1 Z1.51 — Position the tool to location # 1, Figure 5.

N050

G01 — Selects linear interpolation. The tool will feed in a straight line between the next coordinate programmed and the current tool location.

X3.454 — Feeds the tool from location #1 to location #2. This coordinate was determined by adding approximately the desired amount of finished stock to the cutter coordinate of location #8, calculated previously.

F.007 — Sets the feedrate.

N060

X2.974 Z.042 — Coordinates to feed the tool from location #2 to location #3. The X coordinate was determined by subtracting .020 from the calculated finished location coordinate. Although this coordinate will not leave exactly .010 inch of stock per side to be removed during finishing, the amount left will be close to that.

N070

X0. — Feeds the tool from location #3 to location #4.

Taper Turning

N080

G00 — Selects rapid traverse.

Z1.542 — Sends the tool at rapid to location #5. This is an intermediate location used before sending the tool to location #6. If the tool were moved from location #4 to location #6 directly, the corner of the part would be cut off. Laying a straightedge between location #4 and location #6 will demonstrate the point.

N090

X4.1 Z1.532 — Feeds the tool from location #5 to location #6 at rapid (G00 is active).

N100

G01 — Selects linear interpolation.

X3.474 — Feeds the tool from location #6 to location #7. This is the coordinate location calculated earlier.

F.003 — Sets the finish pass feedrate to .003 inch per revolution.

N110

X2.946 Z.032 — Coordinates of location #8.

N120

X0 — Feeds the tool from location #8 to location #9.

N130

G00 — Specifies rapid traverse. **G28U0.W0**. — Initiates a return to reference. **M09** — Turns off the coolant.

N140

M05 —Turns off the spindle.

N150

M30 — Ends the program.

Taper Turning

```
%
O1407
(* **********)
(* X0 = CENTERLINE OF SPINDLE)
(* Z0 = PART SHOULDER)
(* **********)
N010 G00 G99 M08            (SAFETY LINE, COOLNT ON)
N020 T0101 M42              (TURRET POS, HIGH RANGE)
N030 S800 M03               (SPINDLE ON)
N040 X4.1 Z1.51             (POSITION TO #1)
N050 G01 X3.454 F.007       (FEED TO #2)
N06 0X2.974 Z.042           (FEED TO #3)
N070 X0.                    (FEED TO #4)
N080 G00 Z1.542             (RAPID TO #5)
N090 X4.1 Z1.532            (RAPID TO #6)
N100 G01 X3.474 F.003       (FEED TO #7)
N110 X2.946 Z.032           (FEED TO #8)
N120X 0.                    (FEED TO #9)
N130 G00 U0. W0. M09        (RETURN TO REF & COOLNT OFF)
N140 M05                    (SPINDLE OFF)
N150 M30                    (END PRGM)
%
```

Figure 7: Program to turn part in Figure 5

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

Circular Interpolation

- **Circular interpolation** on a lathe does not differ significantly from circular interpolation when milling. There are two ways that an arc center can be programmed using CNC turning machines.

- The **centerpoint** can be programmed using **I** and **K**, or the center may be specified on some machinery as a **radius value**.

- Some machining centers may have an ***arc centerpoint specified by the radius method*** also.

- When **I** and **K** are used, **I** is programmed as the **X-axis coordinate** of the arc centerpoint, and **K** is programmed as the **Z-axis coordinate**. The **format** is:

N . .. G02/G03 X. . . . Z.. . . I... . K. ...

- Where
  - **G02** is **clockwise circular interpolation** and **G03** is **counterclockwise circular interpolation**,
  - **X** is the X-axis endpoint of the arc;
  - **Z** is the Z-axis endpoint of the arc;
  - **I** is the X-axis coordinate of the arc centerpoint; and
  - **K** is the Z-axis coordinate of the arc centerpoint.

Circular Interpolation

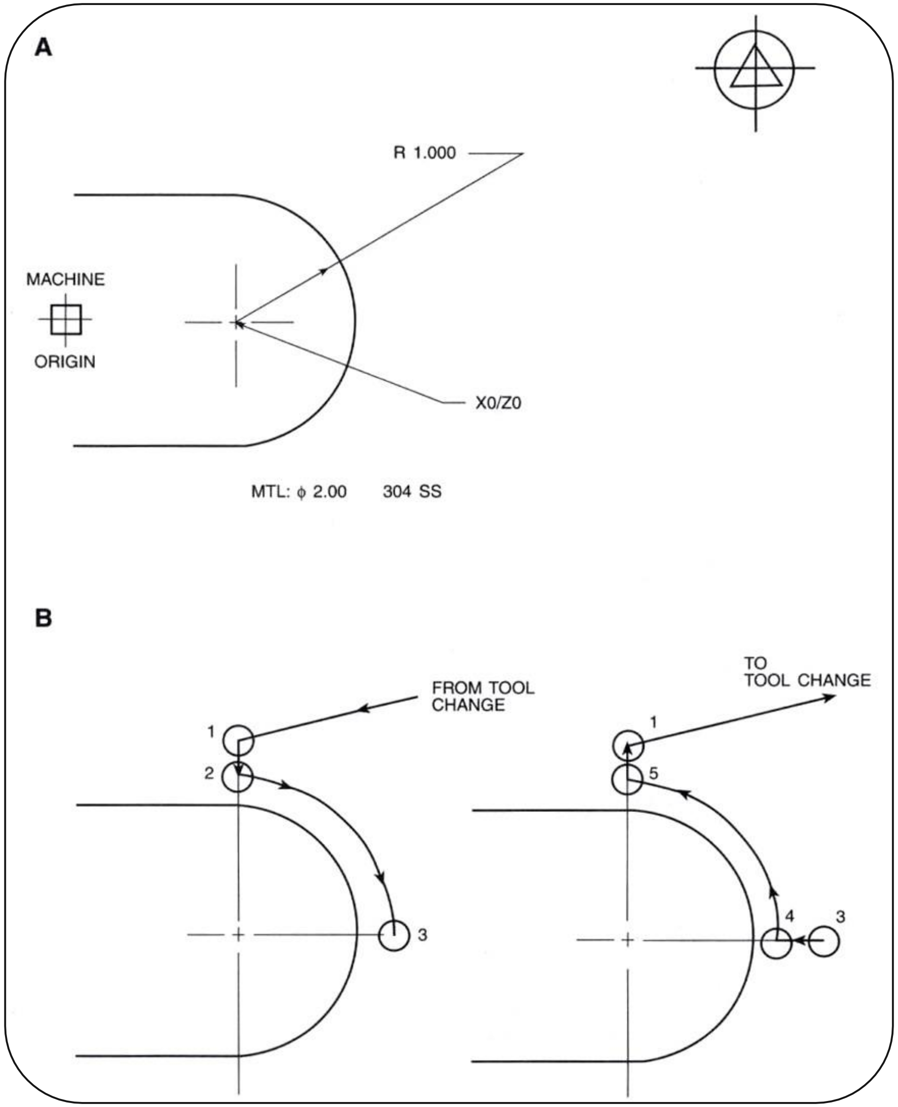
- When the center is specified using a radius, the R address is used. **R** is programmed as an ***incremental value from the current tool position***. The format is:

N . . . G02/G03 XZ. . .. R. . ..

- Two programs are presented here for turning a spherical end on a 2.000-inch-diameter piece of 304 stainless steel (see Figure 8)

- Figure 14-9(A) is a program to turn the end using **I** and **K**; Figure 14-9(B) is identical except that **R** is used instead

Circular Interpolation



- Figure 8 shows **turning a spherical end** on a 2.000-inch-diameter piece of 304 stainless steel

Figure 8: Turning a spherical end *(Seams W., "Computer Numerical Control, Concepts & Programming")*

Circular Interpolation

```
%
O1409
(* **********)
(* INCH VERSION OF PROGRAM)
(*X0/Z0 = CENTERLINE OF PART RADIUS)
(* **********)
N010 G00 G99 M08              (SAFETY LINE)
N020 T0101 M42                (TURRET POS, HIGH RANGE)
N030 S150 M03                 (SPINDLE ON)
N040 X2.1 Z0.                 (POSITION TO #1)
N050 G01 X2.084 F.003         (FEED TO #2)
N060 G02 X0. Z1.042 I0. K0.   (CW ARC TO #3)
N070 G01 Z1.032               (FEED TO #4)
N080 G03 X2.062 Z0. I0. K0.   (CCW ARC TO #1)
N090 G00 X2.084 M09           (RAPID TO #1, COOLNT OFF)
N100 G28 U0. W0. M05          (RETURN TO HOME)
N110 M30
%
```

```
%
O1409
(* **********)
(*METRIC VERSION OF PROGRAM)
(*X0/Z0 = CENTERLINE OF PART RADIUS)
(* **********)
N010 G00 G99 M08              (SAFETY LINE)
N020 T0101 M42                (TURRET POS, HIGH RANGE)
N030 S150 M03                 (SPINDLE ON)
N040 X2.1 Z0. M03             (POSITION TO #1)
N050 G01 X2.084 F.003         (FEED TO #2)
N060 G02 X0. Z1.042 R1.042    (CW ARC TO #3)
N070 G01 Z1.032               (FEED TO #4)
N080 G03 X2.062 Z0 R1.032     (CCW ARC TO #1)
N090 G00 2.084 M09            (RAPID TO #1, COOLNT OFF)
N100 G28 U0.W0. M05           (RETURN TO HOME)
N110 M30
%
```

Figure 9: An inch and a metric version program to turn part in Figure 8

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

Circular Interpolation

Program Explanation

N010

Safety line, returns tool to reference.

N020

T0101 — Selects tool #1, offset #1. M42 — Selects high gear range.

N030

S150 — Sets the spindle speed to 150 rpm. M03 —Turns on the spindle.

N040

X2.1 Z0.— Positions the tool to location #1, Figure 8.

N050

G01 — Selects feed r;

X 2.084 — Feeds the tool from location #1 to location #2.

F.003 — Assigns the feedrate.

N060

G02 — Selects clockwise circular interpolation.

X0.Z1.042 — Arc endpoint coordinates, location #3.

10.K0. — Centerpoints of the arc, Figure 9, top.

H1.042 — Radius value, Figure 9, bottom. The 1.042 value incremental distance from the arc start point (location #2) to the arc center

N070

G01 — Selects feedrate movement.

Z1.032 — Feeds the tool from location #3 to location #4.

Circular Interpolation

N080

G03 — Selects counterclockwise circular interpolation.

X2.062 Z0. — Endpoint coordinates of the arc.

10.K0. — Centerpoints of the arc, Figure 9, top,

R 1.032 —Radius of the arc, Figure 9, bottom.

N090

G00 — Selects rapid traverse.

X2.084 — Rapids the cutter from location #5 to location # 1.

M09 — Turns off the coolant.

N100

G28U0.W0. — Returns the tool to the reference point.

M05 — Turns off the spindle.

N110

M30 — Signals end of program.

Drilling on NC Lathe

- **Drilling** on NC lathes is accomplished in a ***similar manner*** **to turning and boring**
- The ***tool is sent to a desired start position*** and the coordinates are given to move along the ***proper path***
- When **drilling**, the tool point is programmed since there is **no tool radius involved**
- **Canned cycles** like those used for drilling on NC mills will be discussed in a later section
- To drill a $^3/_8$-diameter hole 1.500 inches deep in part Figure 1, a centerdrill and a $^3/_8$ drill can be added to the program in Figure 3
- This has been done in Figure 10

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

Drilling

```
O1410
(* **********)
(* X0 = CENTERLINE OF SPINDLE)
(* Z0 = PART SHOULDER )
(* **********)
N010 G00 G99 M08              (SAFETY LINE, COOLANT ON)
N020 T0101 M42                (TURRET POS., HIGH RANGE)
N030 S1200 M03                (SPINDLE ON, 1200 RPM)
N040 X2.6 Z2.042              (RAPID TO POSITION #1)
N050 G01 X0. F.007            (FEED TO #2)
N060 Z2.032                   (FEED TO #3)
N070 X2.314 F.003             (FEED TO #4)
N080 Z.042 F.007              (FEED TO #5)
N090 X2.6                     (FEED TO #6)
N100 G00 X2.32 Z2.132         (RAPID TO #4)
N110 G01 X2.084               (FEED TO #7)
N120 Z.042 F.003              (FEED TO #8)
N130 X2.6                     (FEED TO #9)
N140 G00 X2.084 Z2.132        (RAPID TO #7)
N150 G01 X2.062               (FEED TO #9)
N160 Z.032 F.003              (FEED TO #10)
N170 X2.55                    (FEED TO #11)
N180 G00 U0. W0. M09          (RAPID TO HOME/COOLANT OFF)
N190 M01                      (OPSTOP)
(* **********)
(* C'DRILL)
(* **********)
N200 M08                      (COOLNT ON)
N210 T0202 M42                (TURRET POS & HIGH RANGE)
N220 S1800 M03                (SPNDL ON, 1800 RPM)
N230 G00 X0. Z2.1             (POSITION TO START)
N240 G01 Z-1.85 F.003         (FEED TO DEPTH)
N250 G00 Z2.1                 (RAPID TO START POS.)
N260 G28 U0. W0. M09          (RETURN TO REF, COOLNT OFF)
N270 M01                      (OPSTOP)
```

```
(* ***********)
(* DRILL)
(* ***********)
N280 M08                      (COOLNT ON)
N290 T0303 M42                (TURRET POS & HIGH RANGE)
N300 S1600 M03                (SPINDLE ON, 1600 RPM)
N310 G00 X0. Z2.1             (RAPID TO START POS.)
N320 G01 Z1.625 F.003         (FEED TO 1ST PECKING DEPTH)
N330 G00 Z2.5                 (RAPID OUT OF PART)
N340 Z1.63                    (RAPID TO START OF PECK)
N350 G01 Z1.375               (FEED TO 2ND PECKING DEPTH)
N360 G00 Z2.5                 (RAPID OUT OF PART)
N370 Z1.38                    (RAPID TO START OF PECK)
N380 G01 Z1.                  (FEED TO 3RD PECKING DEPTH)
N390 G00 Z2.5                 (RAPID OUT OF PART)
N400 Z1.005                   (RAPID TO START OF PECK)
N410 G01 Z.625                (FEED TO 4TH PECKING DEPTH)
N420 G00 Z2.5                 (RAPID OUT OF PART)
N430 Z.63                     (RAPID TO START OF PECK)
N440 G01 Z.387                (FEED TO FINISH DEPTH)
N450 G00 Z.1                  (RAPID TO START POSITION)
N460 G28 U0. W0. M09          (RETURN TO REF, COOLNT OFF)
N470 M05                      (SPINDLE OFF)
N480 M30                      (END PRGM)
%
```

Figure 10: Program to machine part in Figure 1

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

Drilling

N010—N180

are identical to Figure 3.

N190

Optional slop code. This code aids the operator during setup. It the optional stop switch is turned on at the console, the program will stop at this line. The operator can then inspect the workpiece during setup. It is common practice to include an **M01** at the end of each tool.

N200—N220

Selects the tool, offset, gear range. Turns on the spindle and coolant.

N230

G00 — Rapid traverse mode.

X0.Z2.1 — Rapids the centerdrill to the start position, .100 away from the workpiece face.

N240

G01 — Feedrate mode.

Z—1.85 — Depth of centerdrilling (.150 deep).

F.003 — Sets feedrate at .003 ipr.

N250

G00 — Rapid traverse mode.

Z2.1 — Returns tool to the start position.

N260

Returns tool to the reference point and cancels the tool offset.

N270

M01 — Optional stop code.

Drilling

N280—N300

Selects tool, offset, gear range. Turns on spindle and coolant.

N310

Rapids tool tip to the start point.

N320

G01 — Feedrate mode.

Z1.625 — Depth of first drill peck.

F.003 — Sets the feedrate to .003 ipr.

N330

G00 — Rapid traverse mode.

Z2.5 — Sends the tool tip .500 away from the part face. The .500 distance gives the coolant sufficient area to enter the section of hole just drilled to lubricate the drill point on the next drill peck.

N340

Z1.63 — Sends the tool tip to the start of the next peck, .005 from the end point of the previous drill peck.

N350

G01 — Feedrate mode.

Z 1.375 — End point to the second drill peck.

N360

Rapids tool .500 out of part.

N370

Rapids tool tip to start of third peck.

Drilling

N380—N440

The pecking cycle is repeated until final hole depth is achieved.

N450

Tool rapids out of part to original start position.

N460

Returns to reference line.

N470

Spindle off.

N480

END of program.

Threading

- The geometrical shape of the **screw thread** is based on the diameter (d) of the thread and the pitch (p) : the distance axially on the component, from one point or valley on the profile to the corresponding next point along the thread

- This can also be seen as a **triangle** being unwound from the component, where the long base is the same as the circumference of the workpiece and the height is the pitch.

- The **angle of this triangle is called the helix angle** of the screw thread

- The **hypotenuse of the triangle forms the helix** that winds round the workpiece and defines the thread. The diameter in combination with the pitch will, therefore, indicate the definition of the thread

Threading

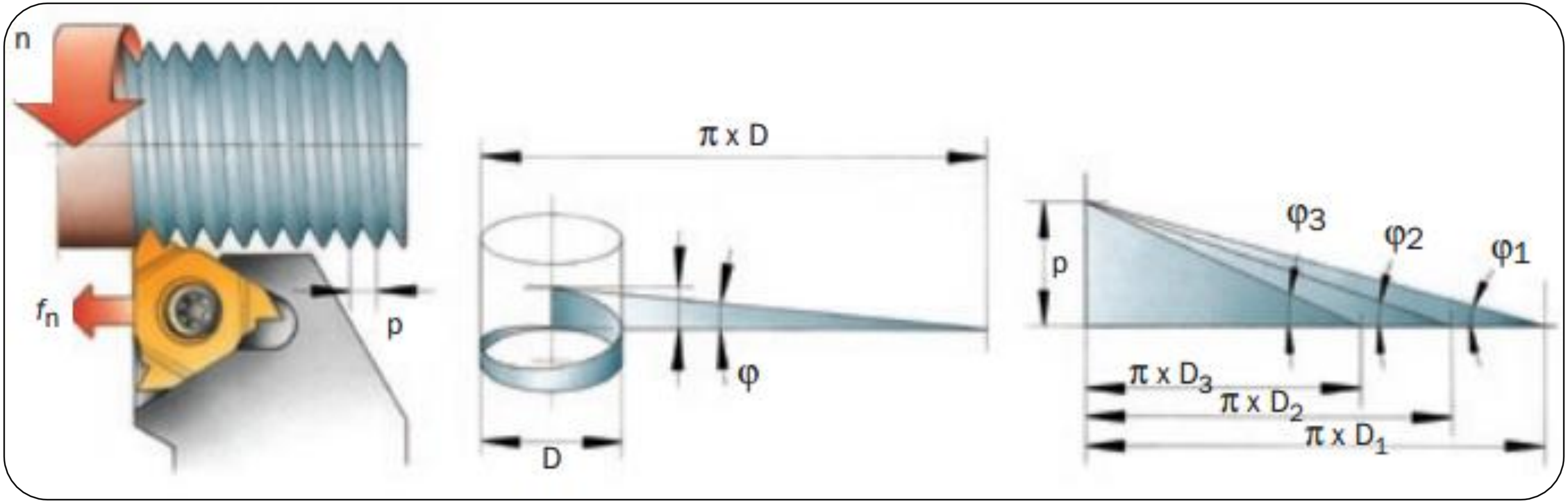


Figure11: Geometrical Shape of Threading

(Courtesy of SANDVIK Coromant)

Threading

- When **threading** on CNC lathes, one of three threading cycles is used: ***single pass threading*** (**G33**), ***multiple pass threading*** (**G92**), or ***multiple pass threading*** (**G76**

- When a **G33** is issued, the tool travels **the length of the thread and stops**

- The tool then has to be ***retracted from the thread***, returned to the starting point, and the whole procedure **repeated**

- When a **G92** command is issued, ***the tool moves to a programmed X coordinate***, feeds across the length of the thread to the programmed Z coordinate, and returns to the start point

- This process is **automatically repeate**d with the X axis moving to a new programmed X coordinate until the final X coordinate has been executed

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

Threading

- When a **G76** is issued, the machine ***makes a threading pass***, then automatically ***retracts the tool to the X-axis reference position*** and ***returns it to the Z-axis start position***

- Then, it **automatically repeats the procedure** until the final depth of the thread is achieved

- **Three types of threads** can be cut using a CNC lathe: ***constant lead, increasing lead, and decreasing lead***

- The **lead of a thread is the distance that the thread advances in one revolution**

- Some CNC lathes are capable of **cutting only constant lead threads**, depending on the thread-cutting options selected when the machine is purchased

- Threads of increasing and decreasing lead are specialized applications and will not be dealt with in this text

Threading

- When cutting threads, **the relationship between spindle speed and tool feedrate is very important**

- When a **G code** is used for **thread cutting**, the **feedrate override controls** on the MCU console, which allow the operator to ***adjust the feedrate during machining***, will not function

- When beginning a **threading pass**, a certain distance (A in Figure 12) must be allowed ahead of the part face ***to give the lathe carriage time to accelerate to the proper feedrate***

Failure to allow this distance will result in **improper leads** on the first several threads

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

Threading

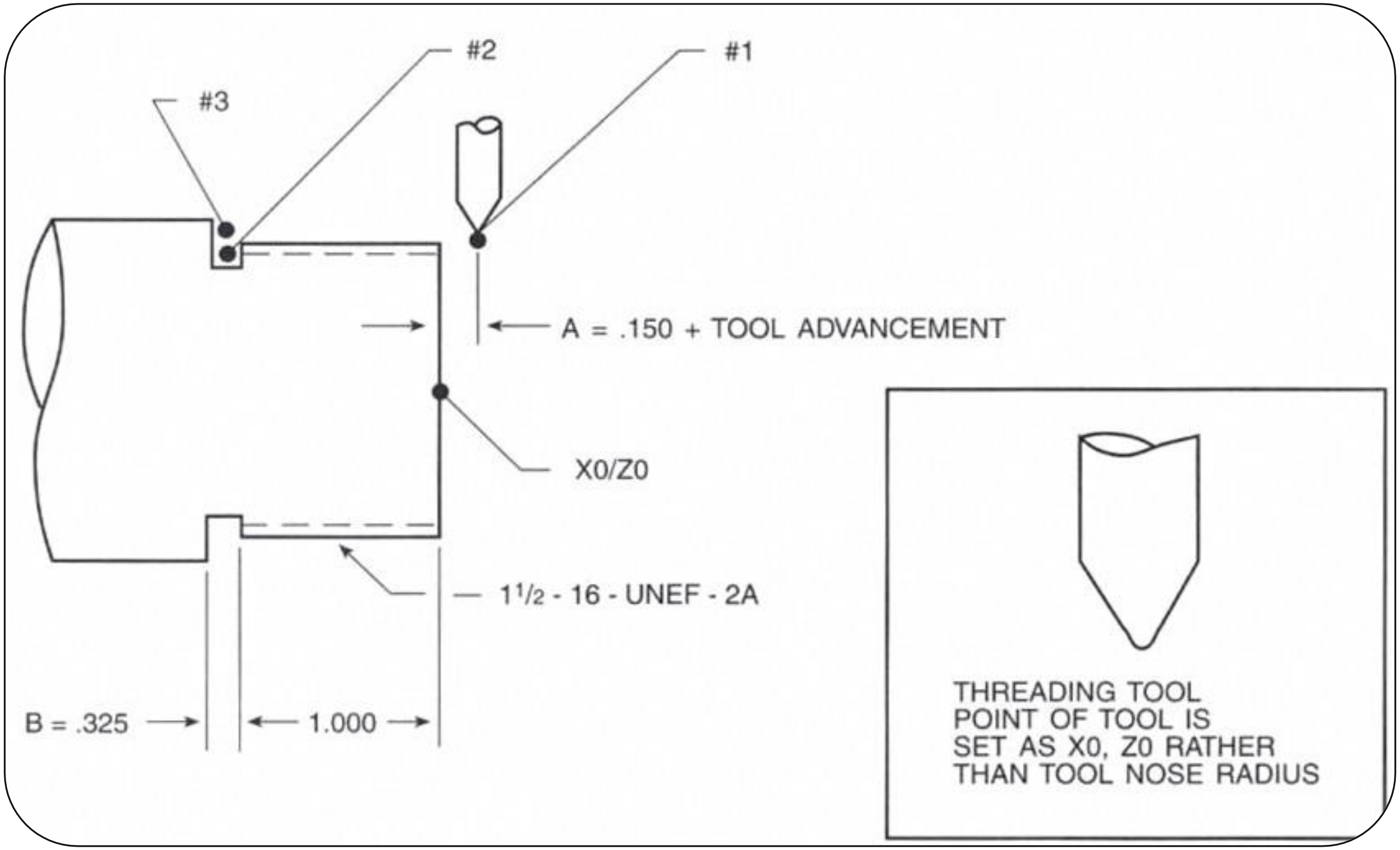


Figure 12: Part to be threaded

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

Threading

- Starting distance **A** varies from machine to machine. Charts giving the distance for a **particular thread on a particular machine** will be found in the programming manual. If a chart is not available, the following formula can be used:

A = (RPM x LEAD x .006) + Z

- Where **Z** is the **amount of tool advancement** in the Z axis. ***Tool advancement*** occurs, prior to the start of a threading cut, along two axes, as illustrated in Figure 13. Advancement along the **Z** axis is calculated by the formula:

Z = X (TAN 30)

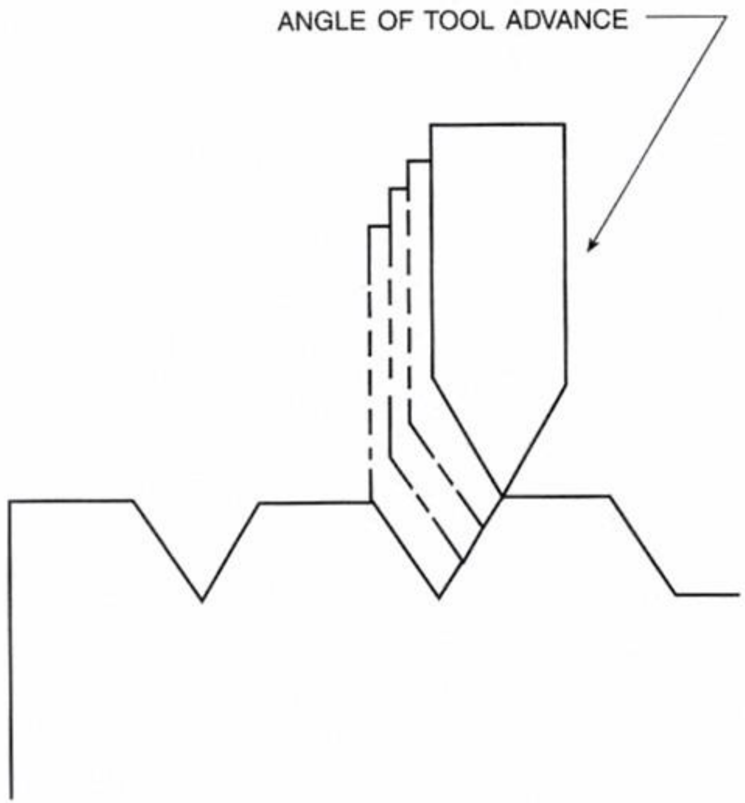
- Some programmers prefer to ***feed the tool in at a 29-degree angle instead of 30***. In this case, the formula would be:

Z = X (TAN 29)

- The **stopping distance is similar to the starting distance**. This distance is shown in Figure 12 as dimension B.The **minimum stopping distance** can be calculated by the following **formula** if a chart is not available:

B = rpm x LEADx .013

Threading



$$A = \left(RPM \times LEAD \times 0.006\right) + Z$$

$$Z = X\left(TAN\,30\right)$$

$$Z = X\left(TAN\,29\right)$$

$$B = rpm \times LEAD \times 0.013$$

Figure 13: Tool advancement

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

Threading

- Three threading programs have been written for the part shown in Figure 10

- The program in Figures 15 and 16 cut the thread using multiple-pass threading. The format for single-pass threading is:

n . . . G33 ... Z .... F ....
N ... G33 .... F .... (absolute positioning) ,or
N ... G33 U .... W .... (incremental positioning)

- On **FANUC**-style **lathe controllers**, **G90** and **G91** are not used to switch between absolute and incremental positioning

- Instead, a ***secondary set of axes are used to specify incremental movement***

- The **U axis** specifies **incremental motion along the lathe's X axis**. The **W axis** specifies **incremental motion along the lathe's Z axis**

Threading

```
%
O1413
(* ***********)
(* X0 = SPINDLE CENTERLINE)
(* Z0 = PART FACE)
(* ***********)
N0101 G00 G99 M08
N0202 T0101 M42
N030 S400 M03
N040 X1.47 Z.015              (POSITION TO #1)
N050 G91 G33 W-1.15 F.0625    (1ST THREAD PASS)
N060 G00 U.015                (RETRACT XAXIS)
N070 W1.168                   (RETURN ZAXIS TO START)
N080 U-.032 W-.018            (ADVANCE TOOL)
N090 G33 W-1.168 F.0625       (2ND THD PASS)
N100 G00 U.032                (RETRACT XAXIS)
N110 W1.186                   (RETURN ZAXIS TO START)
N120 U.032 W-.018             (ADVANCE TOOL)
N130 G33 W-1.186 F.0625       (3RD THREAD PASS)
N140 G00U .032 M09            (RETRACT XAXIS)
N150 G28 U0. W0. M05          (RETURN AXES TO HOME)
N160 M30
%
```

Figure 14: Thread program using G33 thread cycle

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

- The program in Figure 14 cuts the thread using **single-pass threading**

Threading

```
%
O1414
(* **********)
(* X0 = SPINDLE CENTERLINE)
(* Z0 = PART FACE)
(* **********)
N010 G00 G99 M08
N020 T0101 M42
N030 S700 M03
N040 G00 X1.6 Z.15          (THD. START POINT)
N050 G92 X1.58 Z-1.15       (1ST PASS)
N060 X1.57                  (2ND PASS)
N070 X1.55                  (3RD PASS)
N080 X1.53                  (4TH PASS)
N090 X1.51                  (5TH PASS)
N100 X1.49                  (6TH PASS)
N110 X1.47                  (7TH PASS)
N120 X1.46                  (8TH PASS)
N130 X1.455                 (9TH PASS)
N140 X1.45                  (10TH PASS)
N150 X1.445                 (11TH PASS)
N160 X1.443                 (12TH PASS)
N170 X1.44                  (13TH PASS)
N180 X1.438                 (4TH PASS)
N190 X1.437                 (15TH PASS)
N200 X1.436                 (16TH PASS)
N210 G28 U0. W0. M09
N220 M05
N230 M30
%
```

- The program in Figures 15 and 16 cut the thread using multiple-pass threading

Figure 15: Thread program using G92 thread cycle

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

Threading

```
%
O1415
(* **********)
(* X0 = CENTERLINE OF SPINDLE)
(* Z0 = PART FACE)
(* **********)
N010 G00 G99 M08
N020 T0101 M42
N030 S400 M03
N040 X1.6 Z.15                                   (THREAD START POINT)
N050 G76 X1.436 Z1. I0. K.032 F.0625 D.015 A60   (THREADING CYCLE)
N060 G00 G2 8U0. W0. M09
N070 M05
N080 M30
%
```

Figure 16: Thread program using G76 thread cycle

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

Threading

- Where **G33** is the thread-cutting G code, **Z** is the length of the threading cut and **F** is the lead of the thread. (Some lathe controllers use K to specify the lead of the thread)

- The format for **G92** multipass threading is:

```
N…G92 X…Z…F…
N…X…
N…X…
.
.
.
N…X…
```

- Where:

  G92 = multipass threading code

  X = X coordinate of the first threading pass

  Z = Z coordinate of the threading end point

  F = the feedrate (lead) of the thread

  X = depth of second pass

  X = depth of third pass and so on until

  X = depth of final pass

Threading

- Usually, the lead can be given to only **four decimal places** so that some round-off error will occur. This is so slight that it will **affect only threads several feet long**. Some machines have the capacity to accept thread leads to five or six decimal places

- The format for **G76** multiple pass threading is:

N…G76 X…Z…I…K…D…F…A…

- Where:

  G76 = multipass threading G code

  X = minor diameter of the thread

  Z = length of thread

  I = difference in thread radius from one end of the thread to the other. This value is used for cutting tapered threads. For straight threads, a value of zero is entered.

  K = height of the thread (a radius value, given from the crest of the thread to the root)

  D = depth of cut for the first pass

  F = lead of the thread

  A = angle of the tool tip. (For Unified, American National, and IFI metric threads, the angle is 60 degrees.)

Threading

Program Explanation

N010

Safety line, returns tool to reference.

N020

M06T0101 —Selects tool #1, offset #1.

N030

S400 — Sets the spindle speed to 400 rpm.

M03 —Turns on the spindle.

N040

X1.47 Z. 15 —Coordinates of location #1, Figure 12. The X coordinate is diameter programmed and positions the tool to the depth of the first pass. The Z coordinate is the starting distance. Subsequent passes will add to the starting distance the amount of Z-axis tool advancement.

N050

G91 — Selects incremental positioning.

G33 — Initiates single-pass threading.

W1.15 — Feeds the tool from location #1 to location #2, Figure 12.

F.0625 — Lead of the thread.

N060

G00 — Selects rapid traverse.

U.015 — Incremental coordinate to rapid the tool from location #2 to location #3.

N070

W1.168 — Incremental distance to rapid the tool back to the starting point This coordinate also compensates for the additional starting distance required by the tool advancement for the next pass.

Threading

N080

U-.032 — Incremental coordinate to advance the tool for the next cut. Two .015-inch roughing cuts are being made. This coordinate advances the X axis the .015 inch the tool was retracted at the end of the first pass, plus the .015 inch desired for the second.

W-.018 — Calculated Z-axis tool advancement to cause the tool to advance on a 30-degree angle.

N090

G33 — Initiates the threading cycle.

W-1.168 — Feeds the tool from the start point (location #1] to the end of the thread point (location #2).

F.0625 — Lead of the thread.

N100

G00 — Selects rapid traverse.

U.032 — Retracts the X axis from the thread.

N110

W1.168 — Returns the tool to the starting point of the thread.

N120

U.032 W-.018 — Advances the tool to final thread depth.

N130

G33 — Initiates thread cutting.

W-1.168 — Feeds the tool from #1 to *#2.*

F.0625 — Lead of the thread.

Threading

N140

G00 — Selects rapid traverse.

U.032 — Retracts the tool from the thread.

M09 — Turns off the coolant.

N150

G90 — Selects absolute positioning.

G28U0.W0. — Returns the tool to the reference point.

X6 Z6 — Intermediate point coordinates.

M05 — Turns off the spindle.

N160

M30 — Signals end of program.

Threading

N010

Safety line, returns to reference.

N020

T0101 — Selects tool and offset. **M42** — Selects high gear range.

N030

S700 M03 — Turns the spindle on at 700 rpm,

N040

X1 .6 2.15 — Start position of the thread.

N050

G92 — Initiates threading cycle.

X1.58 — X coordinate of first threading pass.

Z-1.15 — Z coordinate of the ending point.

F.0625 — The thread lead.

N060—N200

X coordinates of the **succeeding thread passes**. N200 is the last pass. Note that the passes gradually remove less and less stock per pass to eliminate tearing of the thread.

N210—N220

Returns the tool to reference. Turns off coolant and spindle.

N230

END of program.

Threading

N010

Safety line.

N020

T0101 — Selects tool #1, offset #1.

N030

S400 — Sets the spindle speed.

M03 — Turns on the spindle.

N040

Z1.5 — Positions the 2 axis at the start of the thread.

G76 — Initiates multipass threading.

X1.436 — Minor diameter of the thread.

Z1 — Length of the thread.

10 — Difference in radius of the thread from the starting point to the finish point.

K.032 — Height of the thread measured from the crest to the root. . D.015 — Specifies a .015-inch first pass.

F.0625 — Lead of the thread.

A60 — Specifies a 60-degree thread.

N060

G00 — Selects rapid traverse.

G28U0.W0. — Initiates a return to reference.

M09 — Turns off the coolant.

Threading

N070

M05 — Turns off the spindle.

N080

M30 — Signals the end of program.

A Complete Lathe Example

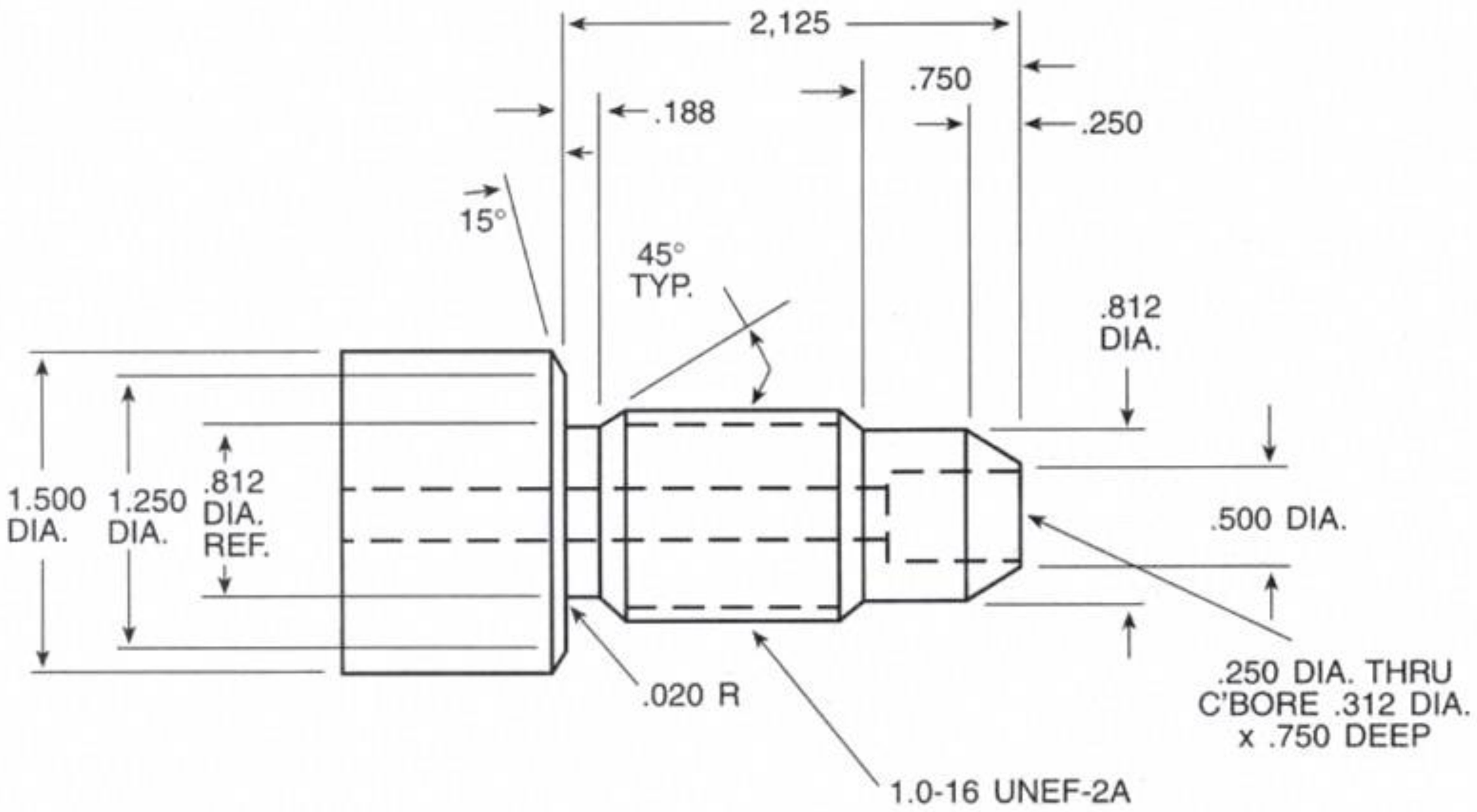


Figure 17: Part drawing

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

A Complete Lathe Example

- Up to this point, small lathe programming routines have been presented
- These routines illustrate various lathe operations which usually are parts of a single lathe program. Figure 17 is a part for which a program has been written
- The program is contained in Figure 18. A brief program explanation follows. There are several codes used in this program that should be noted

G98—used to select inch per revolution feedrates.

G97—used to select direct rpm programming.

M24—used when threading to cause the tool to pull straight out of the part.

- The **default condition** for a thread cycle is for the tool to pull out at a **60-to 45-degree angle**

A Complete Lathe Example

```
%
01417
(* **********)
(* LATHE PROGRAMMING EXAMPLE)
(* X0 = CENTERLINE OF PART)
(* Z0 = FACE OF PART)
(* **********)
(*.031R X 80 DEG. TURNING TOOL)
(* **********)
N1 G97
N2 G99
N3 M08
N4 G00 T0101
N5 S2133 M03
(ROUGH FACE PART - LEAVE .005 STK.)
N6 X1. Z.031
N7 G01 X0. F.007
N8 G00 Z.1
( ROUGH TURN 1.0 DIA. IN 2 PASSES - LEAVE .005 STK./SIDE)
N9 X1.172
N10 G01 Z-2.089 F.0070
N11 X1.672
N12 G00 Z.1
N13 X1.072
N14 G01 Z-2.089 F.007
N15 X1.2594
N16 X1.672 Z-2.1443
N17 G00 Z.136
N18 G28 U0. W0.
N19 M01
(* **********)
(* .007R X 35 DEG. TURNING TOOL)
(* **********)
N20 G99
N21 M08
N22 G00 T0202
```

```
N23 S2133 M03
(ROUGH THREAD RELIEF AREA)
N24 X1.074 Z-1.823
N25 G01 X.836 Z-1.942 F.003
N26 Z-2.105
N27 G02 X.852 Z-2.113 I.852 K-2.105
N28 G01 X1.084
( FINISH O.D.)
N29 G00 Z.007
N30 G01 X0. F.003
N31 X.489
N32 G03 X.5178 Z-.001 I.4889 K-.01
N33 G01 X.8208 Z-.2439
N34 G03 X.826 Z-.2529 I.792 K-.2529
N35 G01 Z-.7471
N36 X1.004 Z-.8361
N37 G03 X1.014 Z-.8481 I.98 K-.8481
N38 G01 Z-1.8389
N39 G03 X1.004 Z-1.8509 I.98 K-1.8389
N40 G01 X.826 Z-1.9399
N41 Z-2.105
N42 G02 X.852 Z-2.118 I.852 K-2.105
N43 G01 X1.2518
N44 X1.614 Z-2.1665
N45 G00 Z.1070
N46 G28 U0. W0.
N47 M01
(* **********)
(* THREADING TOOL)
(* THREAD O.D. 1-16-2A)
(* **********)
N48 G99
N49 M08
N50 G00 T0303
```

Figure 18: The program for Figure 16

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

A Complete Lathe Example

```
N51 S900 M03
N52 X-.5 Z.6 M74
N53 G92 X.99 Z-2.1 F.0625
N54 X.98
N55 X.9718
N56 X.9654
N57 X.96
N58 X.9552
N59 X.951
N60 X.947
N61 X.9434
N62 X.94
N63 X.9368
N64 X.9336
N65 X.9308
N66 X.9278
N67 X.9252
N68 X.9234
N69 X.9234 Z-2.1
N70 G28 U0. W0.
N71M01
(* **********)
(* NO. 4 C'DRILL)
(* C'DRILL TO .260 DIA.)
(* **********)
N72 G99
N73 M08
N74 G00 T0404
N75 S3000 M03
N76 X0. Z.1
N77 G01 Z-.278 F.003
N78 G00 Z.1
N79 G28 U0. W0.
```

```
N80 M01
(* **********)
(* 1/4 DRILL)
(* DRILL .250 DIA. THRU)
(* **********)
N81 G99
N82 M08
N83 G00 T0505
N84 S2000 M03
N85 X0. Z.1
N86 G01 Z-.3 F.003
N87 G00 Z.5
N88 Z-.295
N89 G01 Z-.6 F.003
N90 G00 Z.5
N91 Z1-.595
N92 G01 Z-.9 F.003
N93 G00Z.5
N94 Z-.895
N95 G01 Z-1.2 F.003
N96 G00 Z.5
N97 Z-1.195
N98 G01 Z-1.5 F.003
N99 G00 Z.5
N100 Z-1.495
N101 G01 Z-1.8 F.003
N102 G00 Z.5
N103 Z-1.795
N104 G01 Z-2.1 F.003
N105 G00 Z.5
N106 Z-2.095
N107 G01 Z-2.4 F.003
```

```
N108 G00 Z.5
N109 Z-2.395
N110 G01 Z-2.7 F.003
N111 G00 Z.5
N112 Z-2.695
N113 G01 Z-3. F.003
N114 G00 Z.5
N115 Z-2.995
N116 G01 Z-3.25 F.003
N117 G00 Z.1
N118 G28 U0. W0.
N119 M01
(* **********)
(* .005R BORING BAR)
(* **********)
N120 G99
N121 M08
N122 G00 T0606
N123 S3500 M03
(ROUGH C'BORE - LEAVE .005 STK/SIDE)
N124 X.292 Z.035
N125 G01 Z-.74 F.002
N126 X.152
N127 G00 Z.04
(FINISH C'BORE - DEBURR EDGE WITH .01R)
N128 X.332
N129 G01 Z.0105 F.002
N130 G02 X.292 Z-.0095 I.332 K-.0095
N131 G01 Z-.74
N132 X.132
N133 G00 Z.11
N134 G28 U0. W0. M09
N135 M05
N136 M30
%
```

A Complete Lathe Example

Program Explanation

First Tool:

NI-N5

Selects first tool. Tums on spindle and coolant.

N6-N8

Part is rough faced with .005 stock left for finishing.

N9-N11

First roughing pass on o.d.

N12-N17

Second roughing pass on o.d. The 15-degree angle is also rough turned at this time.

N18

Tool is returned to reference point. Tool offset cancelled.

Second Tool:

N20-N23

Selects second tool. Turns on spindle and coolant.

N24-N28

Thread relief area is rough turned .. 005 stock is left for finishing.

N29-N31

Face of part is finished

N32

Deburring radius is turned at the intersection of the first angle and the face of the part

A Complete Lathe Example

N33

First angle is finish turned.

N34

Deburring radius is turned at the intersection of the first angle and the .812 diameter.

N35

The .812 diameter is finish turned.

N36

The front thread chamfer is finish turned.

N37

A radius is turned at the intersection of the thread chamfer and major diameter.

N38

The major diameter of the thread is turned.

N39

A radius is turned at the intersection of the back thread chamfer and major diameter.

N40

The back thread chamfer is turned.

N41

The .812 diameter thread relief is turned.

A Complete Lathe Example

N42

The .020 radius is turned.

N43

The 2.125 dimension is faced.

N44

The 15-degree angle is finish turned.

N45-N47

The tool is returned to reference. The offset is cancelled.

Third Tool:

N48-N51

Tool and offset selected, spindle and coolant turned on.

N52

The tool is sent to the start position for threading.

The **M74** turns off the thread chamfering at the end of a thread pass.

N53

G92 multi-pass thread cycle initiated.

N54-N68

Succeeding X values for the **G92** cycle. Each X value is used on a separate thread pass.

N69

Last threading pass that is a repeat pass. The Z coordinate is optional.

A Complete Lathe Example

N70-N71

Returns the tool to reference. Offset is cancelled.

Fourth Tool:

N72-N75

Tool, offset, spindle speed selected.

N76-N78

Drill sent to start point, fed to depth, and rapids back to start position.

N79-N80

Returns to reference.

Fifth Tool:

N81-N84

Tool, offset, spindle speed selected.

N85-N116

Peck drilling of '/4-inch through hole. Each peck is .300 deep. At end of peck the tool is sent at rapid z.500 to clear out chips and allow coolant into the hole. The tool sequence repeats until fined depth is achieved in N116.

N117

Tool is returned to the starting position.

N118-N119

Returns to reference.

Sixth Tool:

N12D-N123

Tool, offset, spindle speed selected.

A Complete Lathe Example

N124-N127

The c'bore is rough bored and .005 stock is left for finishing.

N128-N130

A deburring radius is turned at the intersection of the c'bore and the part face.

N131-N133

The c'bore is finish bored and tool retracted from part.

N134

Return to reference line, coolant

N135

Spindle off.

N136

End of program.

Canned Cycles

- Most modern CNC lathe controllers ***contain a number of built-in canned cycles***. The threading cycles **G33**, **G92**, and **G76** are standard from controller to controller

- Other canned cycles are **options** offered by the controller manufacturer. These cycles are often **unique** to a given controller manufacturer (sometimes unique to a given model of controller) and therefore ***not transportable from controller to controller***

- With the current CNC lathe investment strategies by small and midsized companies, canned cycles will become as **standardized** as mill cycles at some future point

- It is not possible to cover the number of **cycle variations** in a text of this size. The student should be aware, however, that these cycles exist

Documentation on the use of these cycles will be contained ***in the programming and operational manuals for a given machine***

Canned Cycles

- How much a company relies on ***canned cycles*** for lathe programming ***depends on their use or non-use of computer-aided programming***

- Where computer-aided or graphics programming is utilized, there is **little need for canned cycles** aside from the standard lathe threading cycles

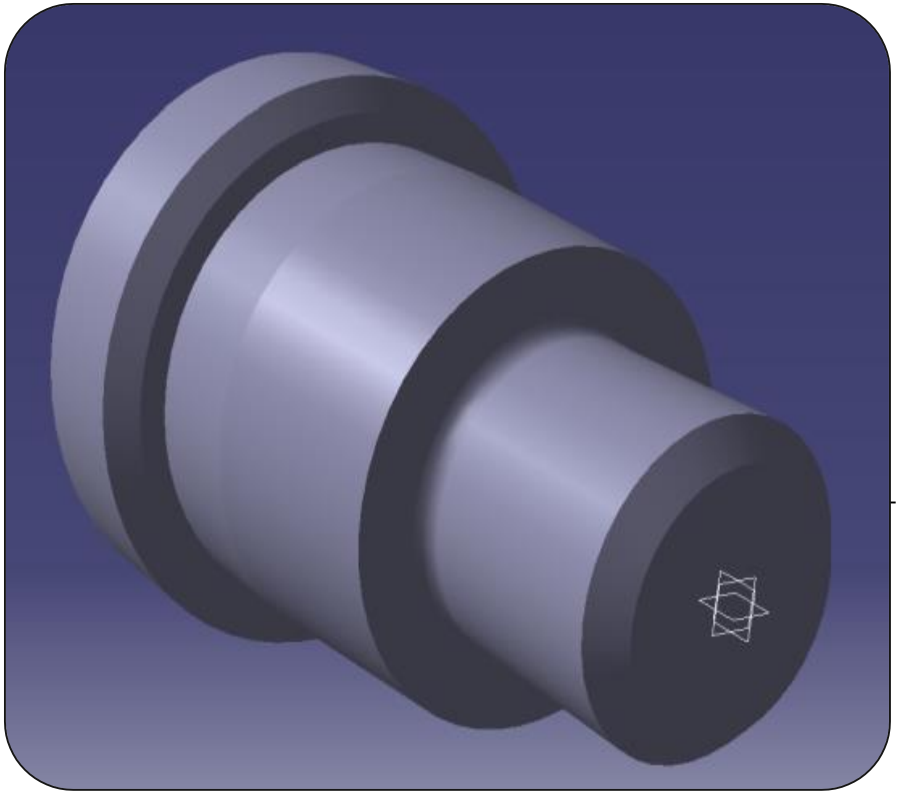
- Where MDI programming is used, **canned cycles can save many hours of programming time**

- The cycles used in these situations usually include: ***rough turning*** and ***boring cycle***, ***rough facing cycle***, ***finish turning*** and ***boring cycle***, ***finish facing cycle***, ***peck drilling cycle***, ***step drilling cycle***, ***chamfering cycle***, and ***growing cycle***

- One **caution** should be noted by the programmer: **Canned cycles** valid for one controller can cause a ***crash situation if run on an incompatible controller*** if the controller does not stop and put out an **alarm message** when the canned cycle is encountered

A Complete Lathe Example 2

- The part in Figure 19 will be constructed at a CNC Lathe:



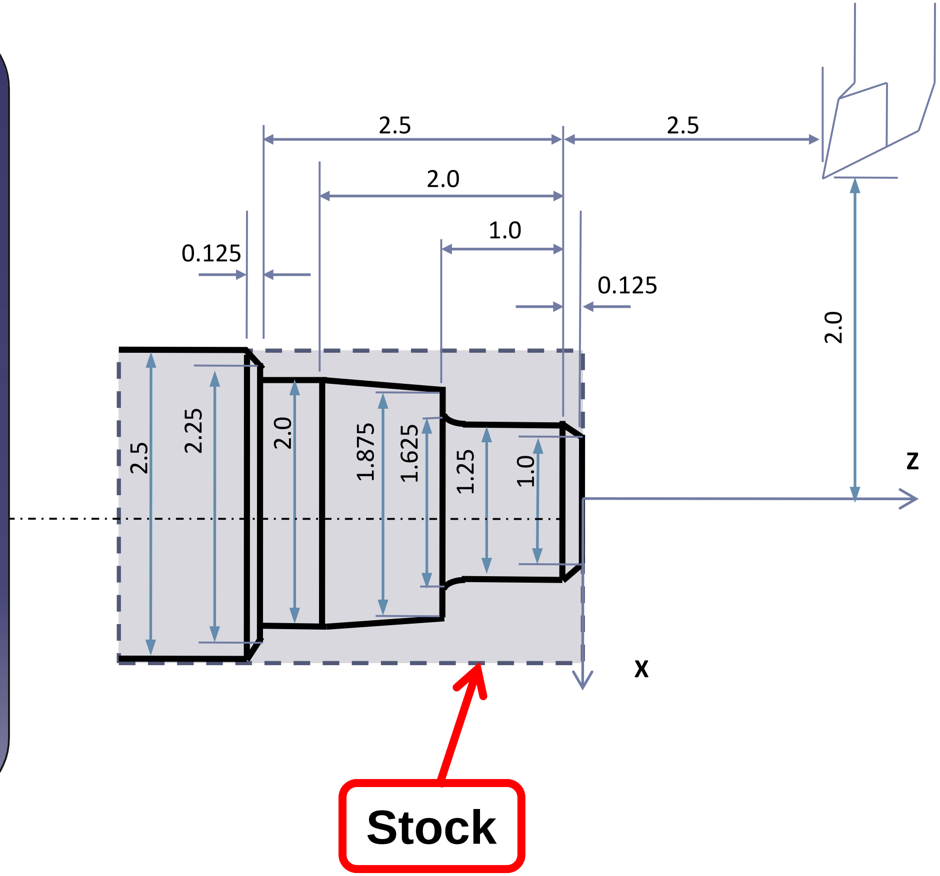
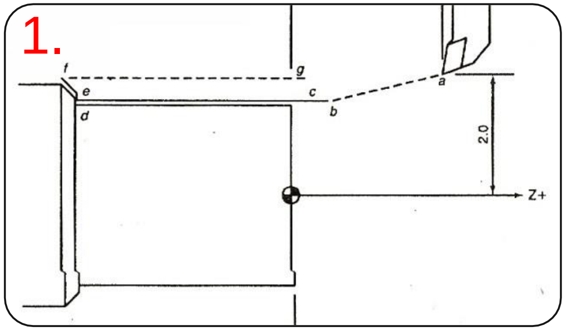


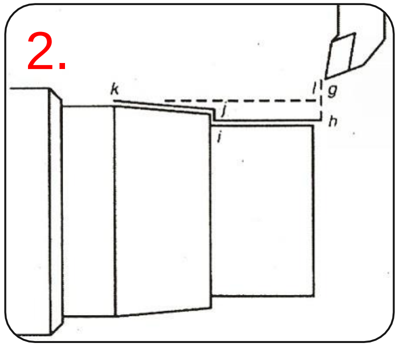
Figure 19:Part for turning example

Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ. Σαλωνίτης

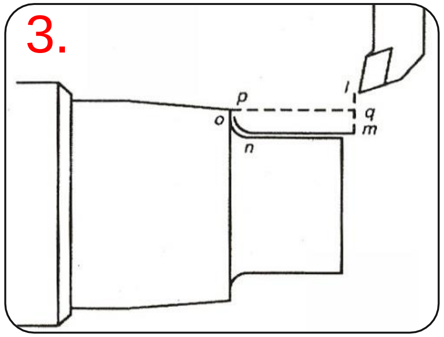
Material Removal Stages



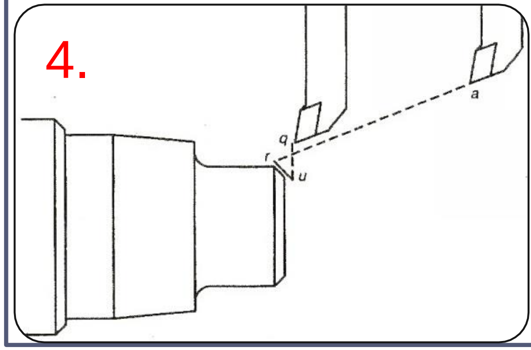
Turning of the first “step”



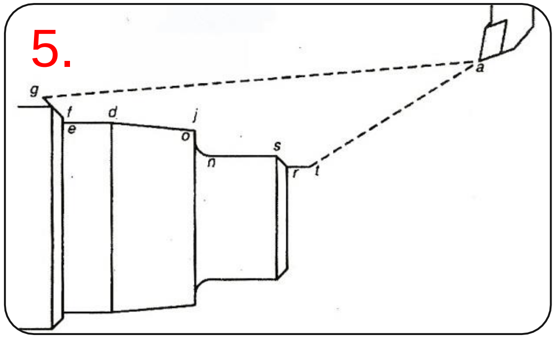
Turning of the second “step”



Turning of the third “step” and chamfer



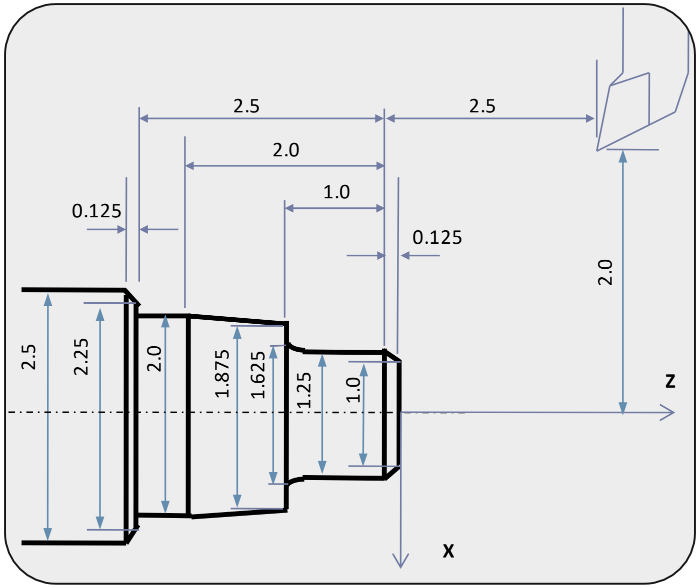
Angular Turning

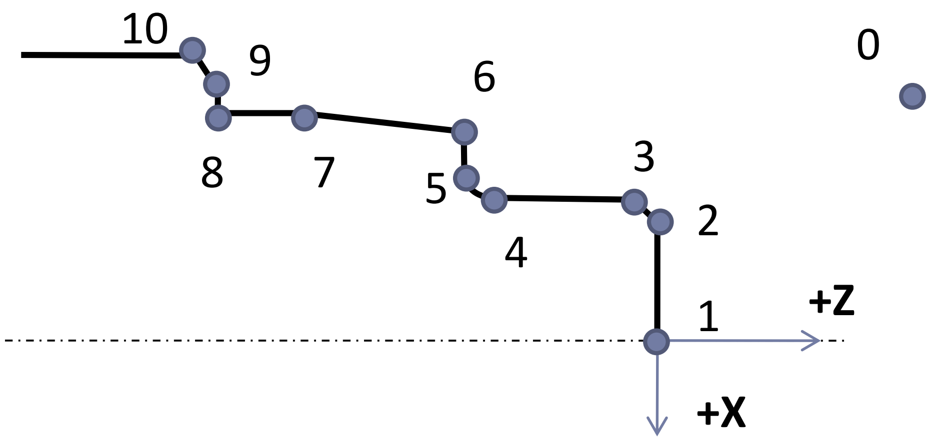


Finishing

Figure 20:Material removal stages for part in figure 19

Coordinates of Points





Point	X	Z
0	-2.0	2.5
1	0.0	0.0
2	-0.5	0.0
3	-0.625	-0.125
4	-0.625	-1.0
5	-0.825	-1.125
6	-0.9375	-1.125
7	-1.0	-2.125
8	-1.0	-2.625
9	-1.125	-2.625
10	-1.250	-2.75

Program

Program Explanation

%

Program Start

001

Program number

N010 G20

Inch coordinates

N020 G50 X-2.0 Z2.5 S1000

G50: reference point selection (absolute coordinate system)

S1000: maximum spindle speed 1000 rpm

N030 T0101 M41

T0101: tool 01 – activate offset 01

M41: low spindle speed

N040 G96 S100 M03

G96 S100: constant peripheral speed, 100 sf/min

M03: clockwise spindle turn

Program

N050 G00 X-2.0 Z.1 M08

G00: Rapid movement a-b

M08: Coolant start

N060 G01 Z.01F.120

Straight cut b-c

F120: Feedrate (in/rpm)

N070 Z2.49 F0.120

Straight cut c-d

N080 X-2.27

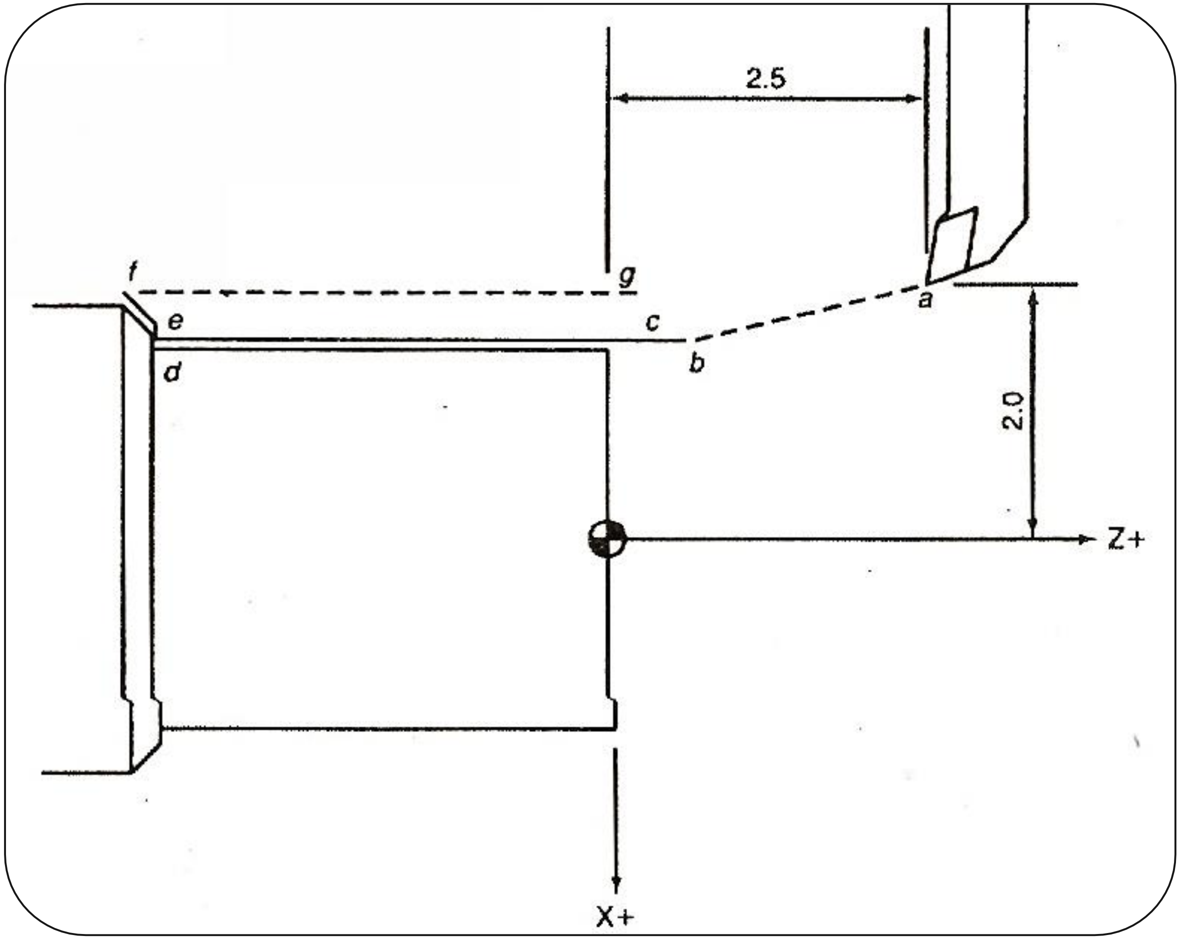
Straight cut d-e

N090 X-2.52 Z-2.625

Angle cut e-f

N100 G00 Z.01

Rapid movement f-g



Program

N110 X-1.645

Rapid movement g-h

N120 G01 Z.99

Straight cut h-i

N130 X-1.895

Straight cut i-j

N140 X-2.02 Z-2.0

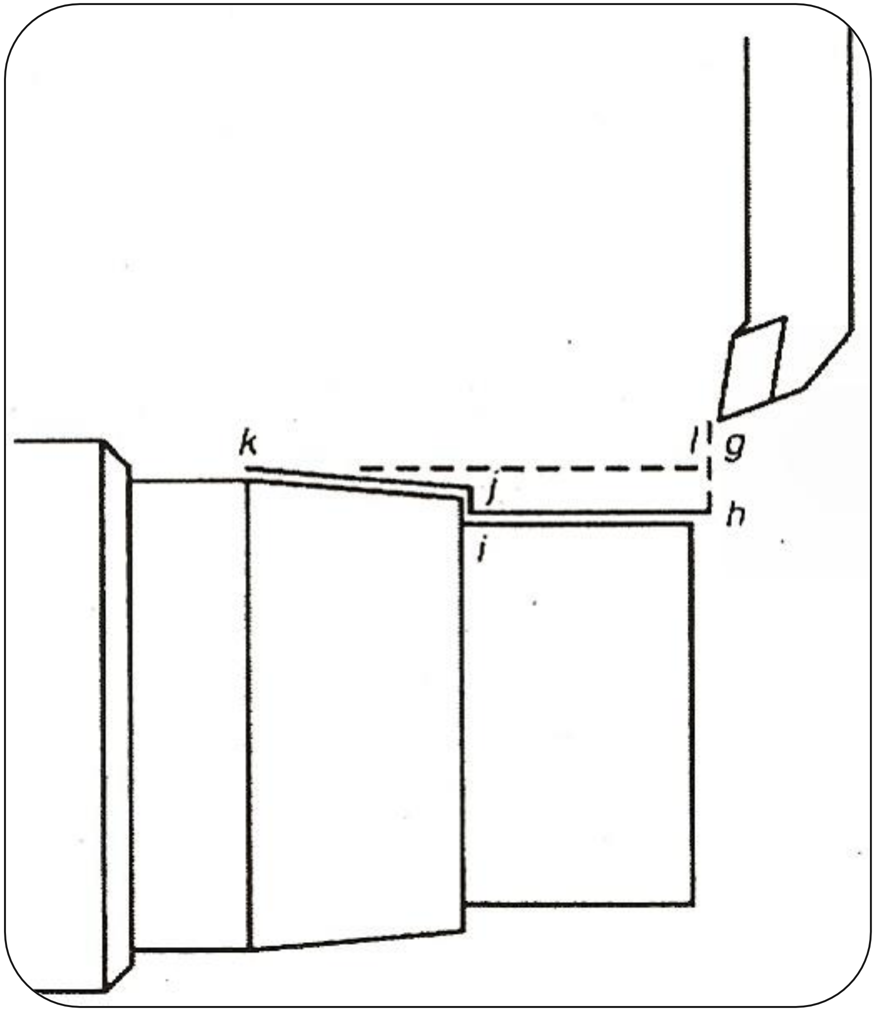
Straight cut j-k (taper)

N150 G00 Z.01

Rapid movement k-l

N160 X-1.27

Rapid movement l-m



Program

N170 G01 Z.79

Straight cut m-n

N180 G03 X-1.645 Z.99 R.2

G03: Circular interpolation CW h-i

N190 G01 X-1.895

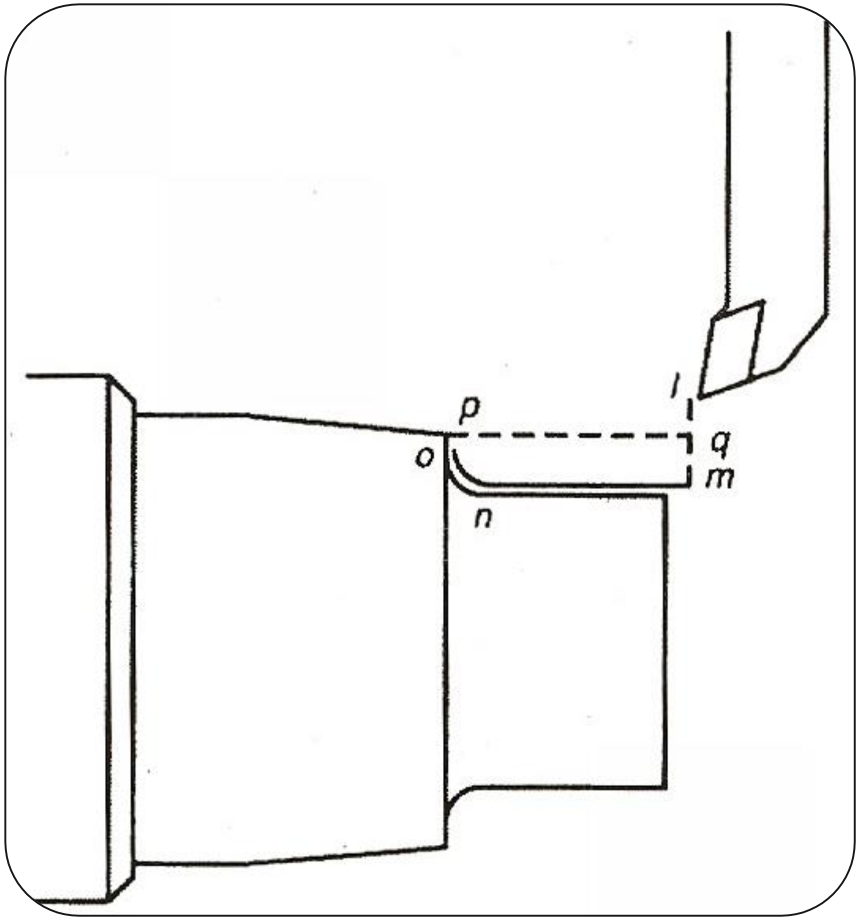
Straight cut o-p

N200 G00 Z.01

Rapid movement p-q

N210 X-1.02

Rapid movement q-u



Program

N220 G01 X-1.27 Z-.135

Angle cut m-r

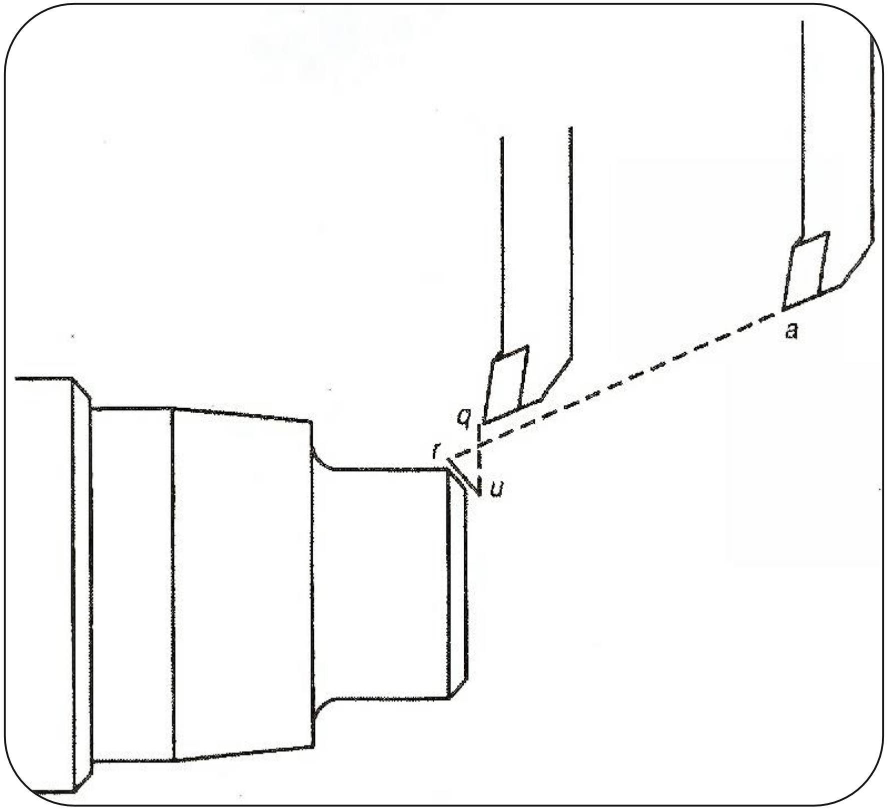
N230 G00 X-4.0 Z2.5 M05

Rapid movement r-a

M05: spindle stop

N240 T0100

Tool 01 – activate offset 00



Program

N250 G50 X-2.0 Z2.5 S2000

G50: Reference point selection(absolute coordinate system)

S2000: max spindle speed 2000 rpm

N260 T0202 M42

Tool 02 – activate offset 02

M42: max speed area

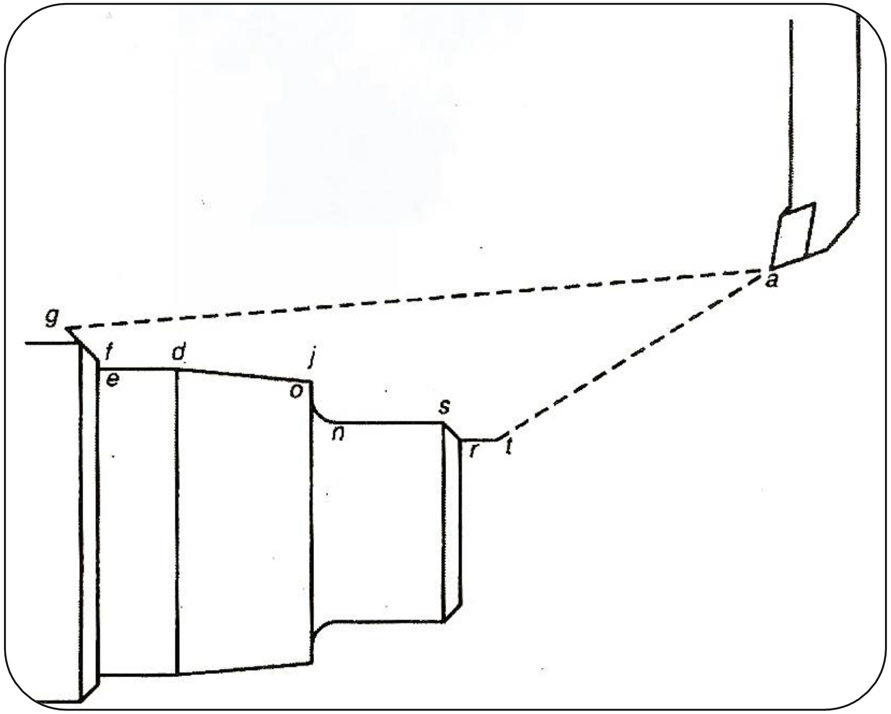
N270 G96 S150 M03

G96 S150: constant peripheral velocity,150sf/min

M03: clockwise spindle rotation

N280 G00 X-1.0 Z.2

Rapid movement a-t



Program

N290 G01 Z.1 F.040

Straight cut t-r

N300 X-1.25 Z-.125 F.006

Angle cut r-s

N310 Z-.8

Straight cut s-n

N320 G03 X-1.625 Z-1.0 R.2

Circular interpolationCW n-o

N330 G01 X-1.875

Straight cut o-j

N340 X2.0 Z-2.0

Straight cut j-d (taper)

N350 Z-2.5

Straight cut d-e

N360 X-2.25

Straight cut e-f

N370 X-2.5 Z-2.75 M09

Angle cut f-g.Coolant stop

N380 G00 X4.0 Z2.5 M05

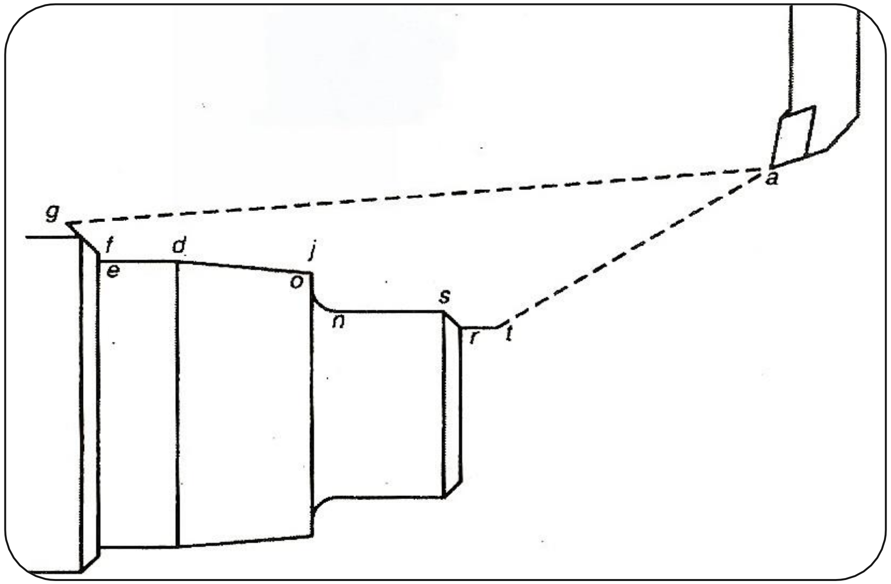
Rapid movement g-a. Spindle stop.

N390 T0200

Tool 02. activate offset 02

N400 M30

End of program



Summary 1/2

- In **diameter programming**, the X-axis coordinates are one-half the actual tool movement.
- In **radius programming**, the X-axis coordinates and the tool movement are the same.
- **G01**, ***linear interpolation***, is used for feedrate moves
- Coordinates for **taper turning** must be ***calculated using trigonometry*** (or other math methods), just as when milling angles
- **G02** and **G03** are used for **circular interpolation**
- **I** and **K** are the addresses used to program the **center points of an arc**

Summary 2/2

- The **R address** is used in place of **I** and **K** to program an arc using the arc radius ***instead of the arc centerpoints***
- **Single-pass threading** cycles produce one threading cut. The cycle must be reinitiated for each threading pass
- **Multi-pass threading** can produce an entire finished thread without additional programming
- When **threading**, the Z-axis tool advance must be calculated from the X-axis depth of cut by the formula **Z = X TAN(30)**
- Minimum ***starting and stopping distances must be calculated*** for use in a threading program

Vocabulary Introduced in this chapter

- Constant lead thread
- Decreasing lead thread
- Diameter programming
- Increasing lead thread
- Intermediate point
- Radius programming
- Reference point

References

1. Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006, Springer-Verlag
2. http://www.coromant.sandvik.com/
3. Kalpakjian S., «Manufacturing Engineering and Technology», 2nd Edition, 1992, Addison-Wesley Publishing company
4. Kalpakjian,Schmid,<<Manufacturing Processes for Engineering Materials>>, 5th ed. 2008
5. Mattson M., “CNC Programming, Principles and Applications”, Delmar, 2002
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

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 – II. Τεχνολογία κατεργασιών κοπής των μετάλλων», 1998, Εκδόσεις Ζήτη
12. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ. Σαλωνίτης