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 14:
The Numerical Control Lathe

Table of Contents

Chapter 14: The Numerical Control Lathe……………………………….4

14.1 Conventional & Slant Bed Arrangement.……………..……………………………...…….………..8

14.2 Axis Movement on a CNC Lathe…..……………………….………………………….17

14.3 Tool-holding Methods used on CNC Lathes......……………………………………….………21

14.4 Tool Offset Number……………………..…………………….…………………..…26

14.5 Other Parts of Modern CNC Lathes…………………………………………...………37

14.6 How Feedrates are Specified on CNC Lathes….………………………………………..42

14.7 Machine Origin and Work Coordinate System……..…………………….………..……..49

Objectives

- Describe the **difference** between a ***conventional lathe bed arrangement*** and a ***slant bed arrangement***, listing the advantages of the slant bed for NC

- Explain **axis movement** on a CNC lathe

- Describe the **method of tool-holding** used on CNC turning machines

- Explain what a **tool offset** number is

Objectives

- Describe two methods of **tool selection** used on CNC turning machines
- Describe how **spindle speed** is designated on gear head and variable speed lathes
- Explain how **feedrates** are specified on CNC turning equipment
- Define Tool Nose Radius compensation **(TNR)**

Introduction

- Up to this point, the programming features of CNC mills have been discussed, but **numerical control** is used for **turning equipment** as well

- For the turning programs discussed in Chapter 14, **word address format** will be used

- The **coding** will be a version used with **FANUC lathe controllers**, designed to be generic and so to illustrate the basic programming steps involved

- A numerical control lab in a school ***will have equipment that differs*** in one way or another from that presented here

- Students are advised to **familiarize themselves** with the codes used for the machines they will be using

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

Main Parts of a CNC Lathe

- The Main parts of a CNC lathe are depicted below

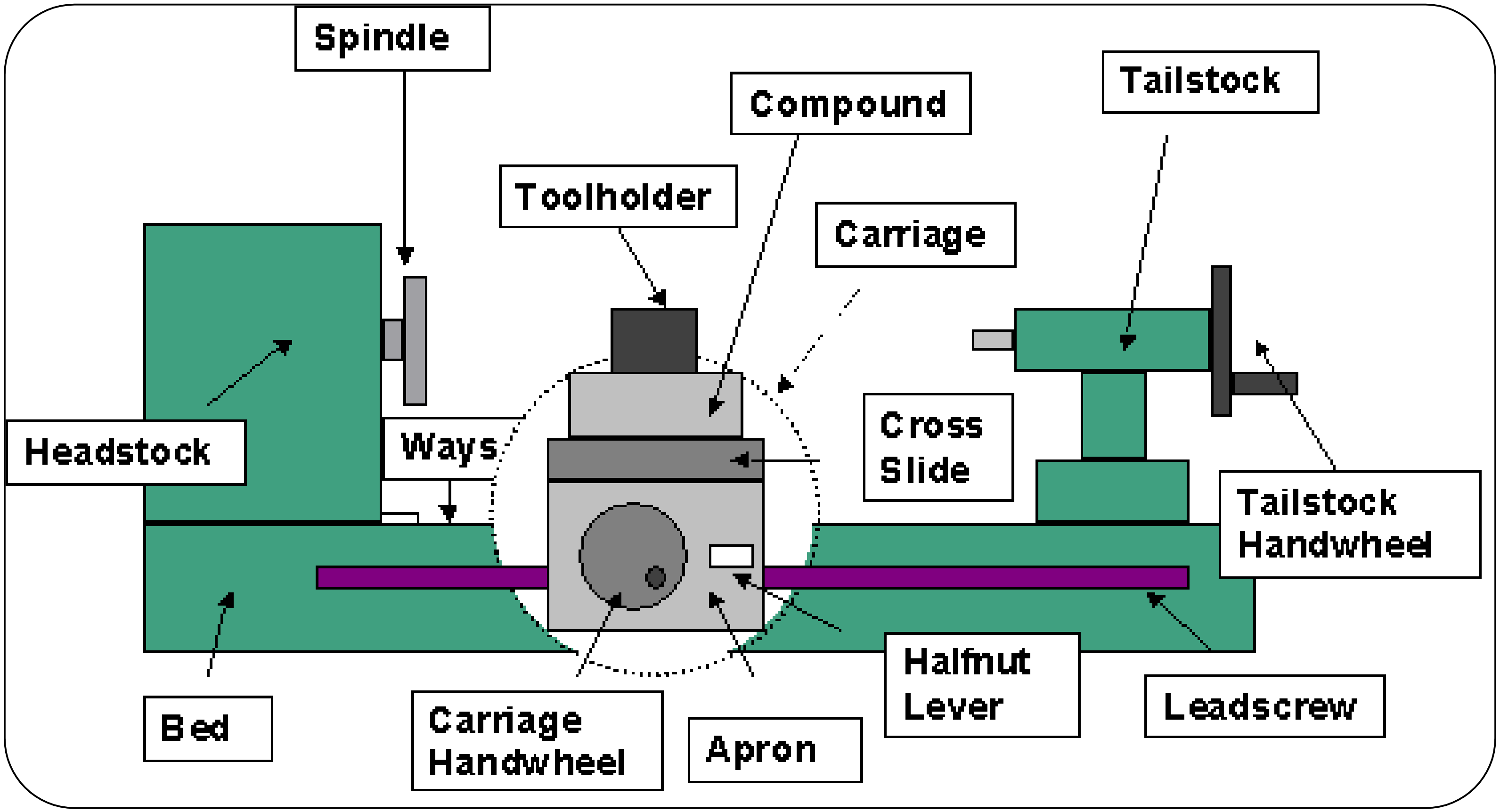


Figure 1:Main parts of a CNC lathe

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

Lathe Bed Design

- Older NC lathes, and those that have been converted to numerical control with retrofit units, look like traditional engine lathes

- **The lathe carriage rests on the ways**

- The ways are in the **same plane** and are **parallel** to the floor, as illustrated in Figure 2

- This arrangement allows the machinist to reach all the controls readily. Since the CNC lathe performs its operations **automatically**, this type of arrangement is not necessary

Lathe Bed Design

- In fact, it is quite awkward since the operator will be busy with other responsibilities while the program is running and will ***not necessarily be there to brush the chips off the ways***

- In a **conventional lathe bed** arrangement, the chips have nowhere to fall except on the ways

- To overcome this problem, many CNC lathes make use of the **slant bed design** illustrated in Figure 3

Lathe Bed Design

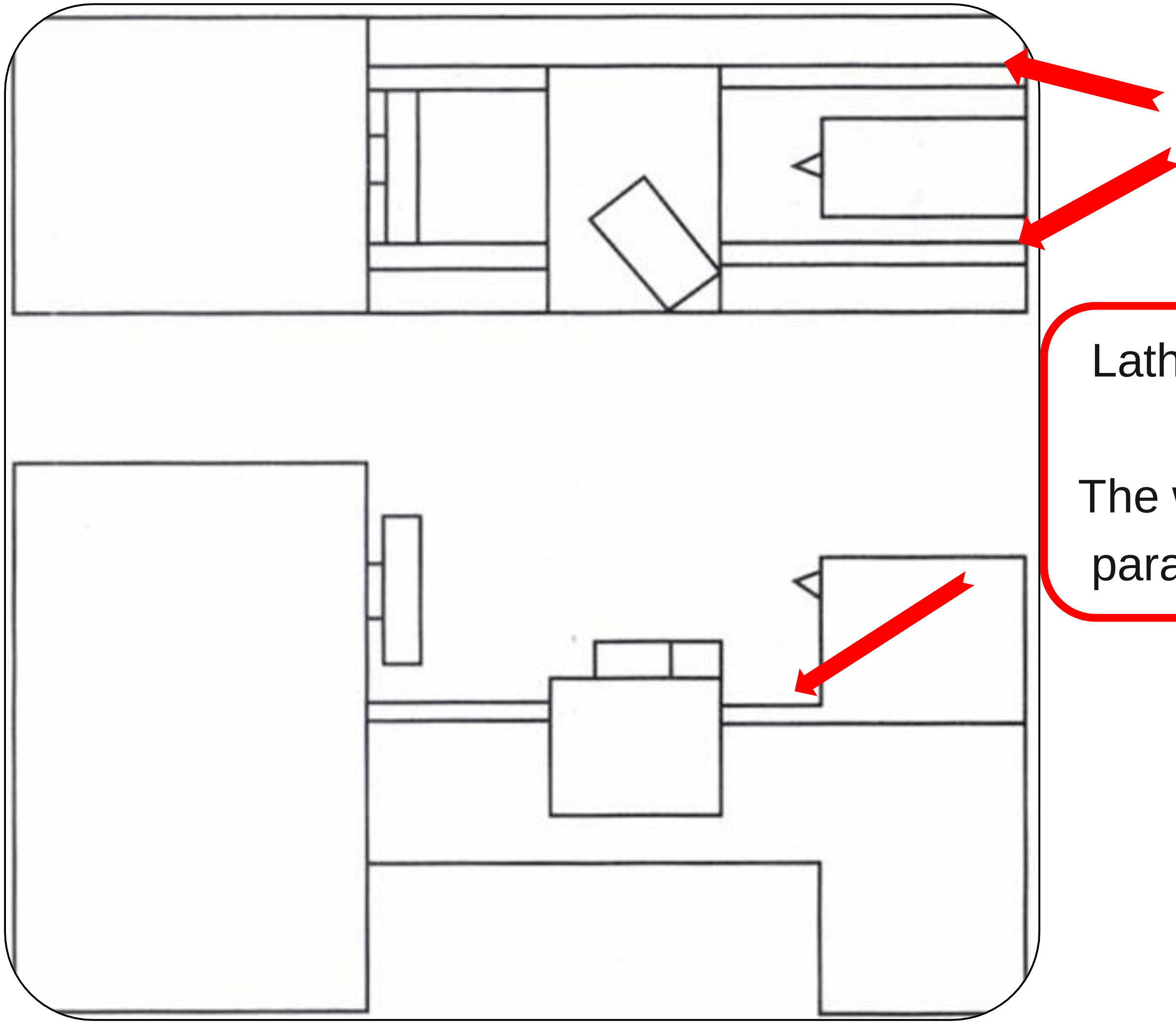
- On many NC lathes, the **turret tool post** is mounted on the opposite side of the saddle, compared to a conventional lathe, to take advantage of the **slant bed design**

- The **slant bed** allows the chips to fall into the **chip pan** rather than on tools or bedways

- Despite its odd appearance, the slant bed NC lathe **functions just like a conventional lathe**
  - Figures 5,6 and 7 show **modern CNC turning machines**
  - Notice the **slant bed arrangement**

Lathe Bed Design

- The **ways** on a conventional lathe:



Lathe carriage rests on the ways

The ways are in the same plane and parallel to the floor

Figure 2: Bed arrangement on a conventional lathe

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

Lathe Bed Design

- The **slant bed arrangement** allows the chips to fall into the chip pan rather than on tools or bedways

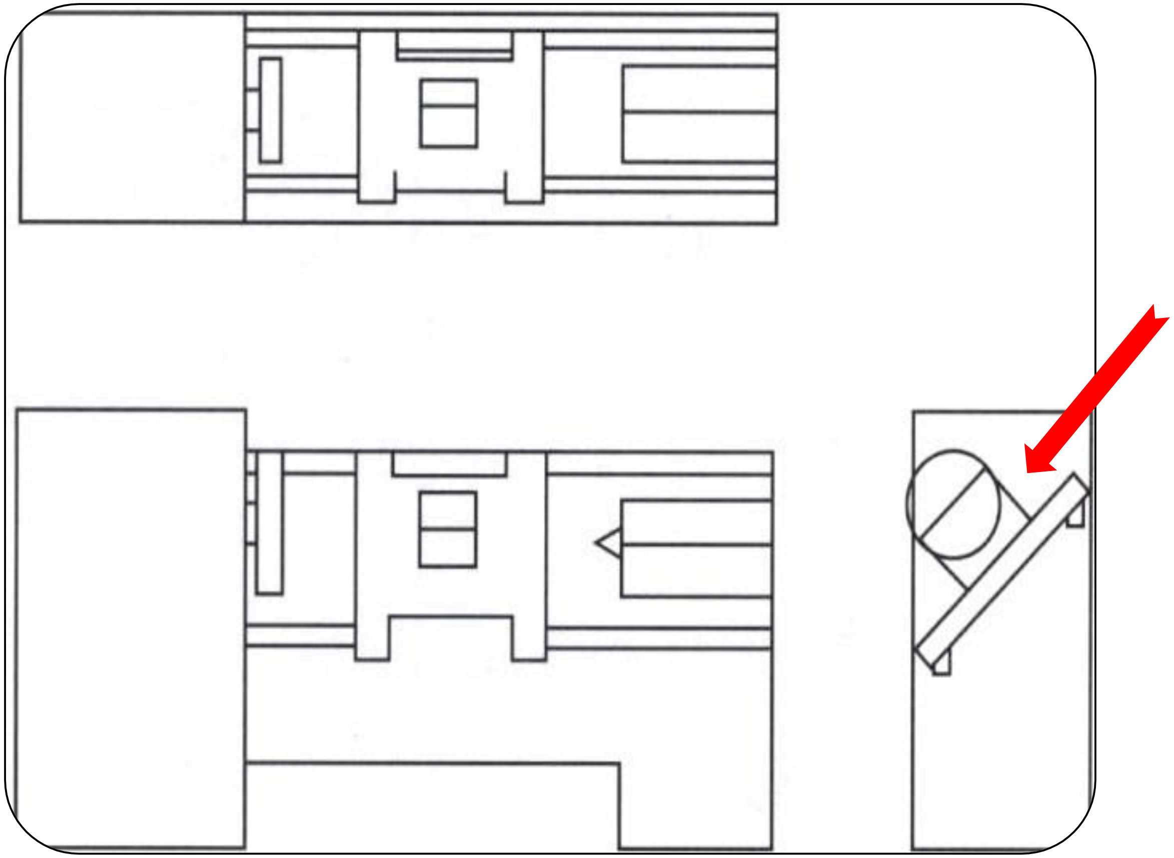


Figure 3: Slant bed for NC or CNC lathe *(Adapted from Seams W., “Computer Numerical Control, Concepts & Programming”)*

Lathe Bed Design

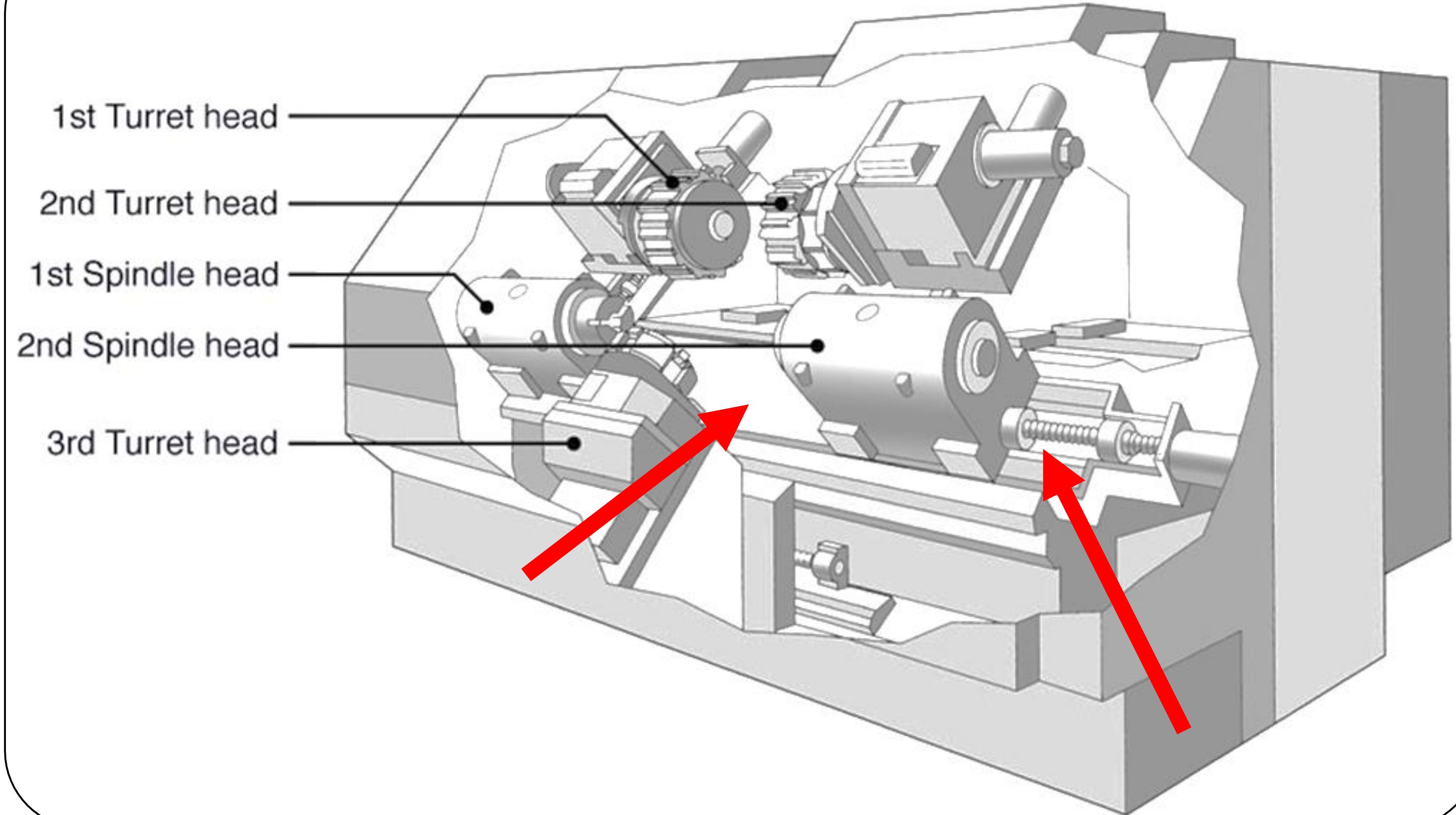


Figure 4:Schematic illustration of a computer numerical-controlled turning center. Note the slant bed arrangement (Manufacturing, Engineering & Technology, Fifth Edition, by S.Kalpakjian and Steven R. Schmid.)

Lathe Bed Design

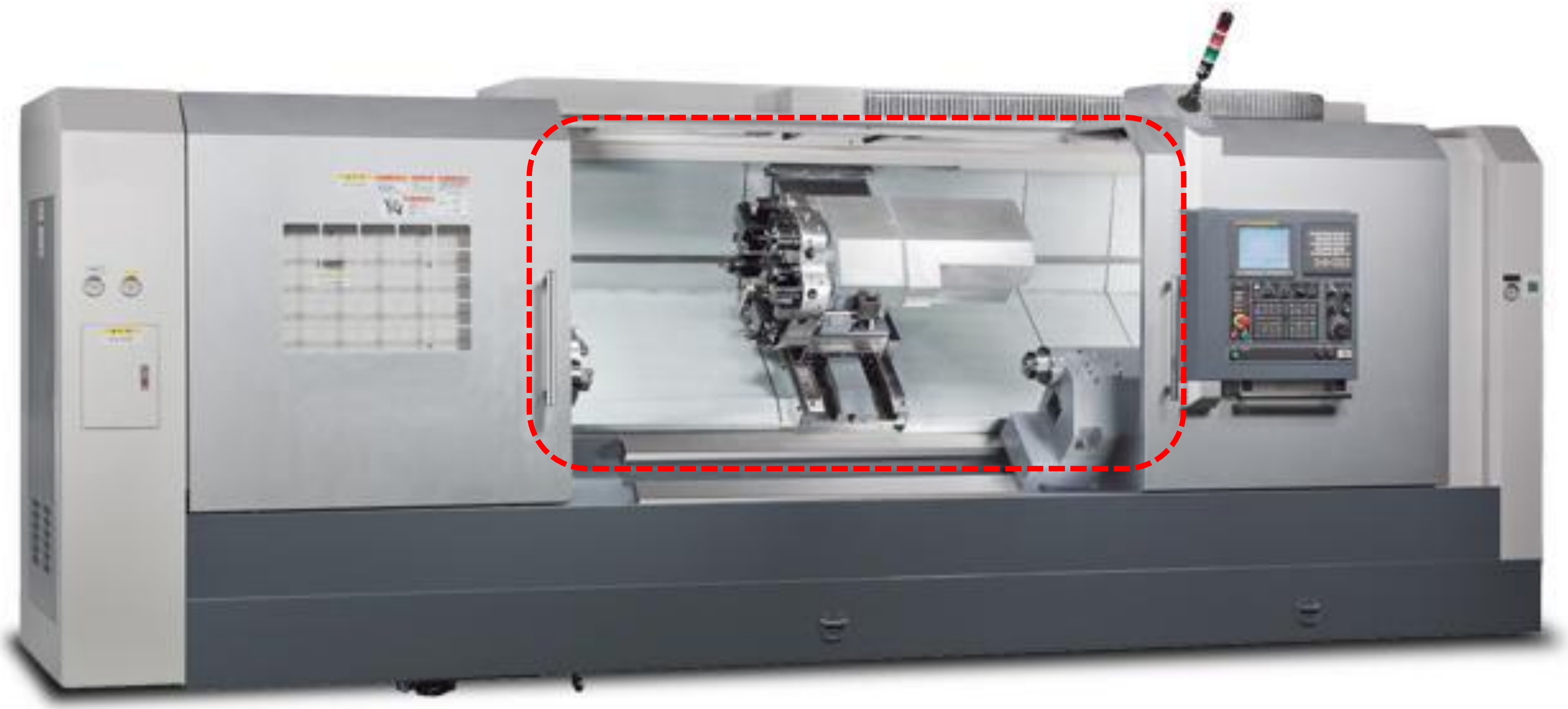


Figure 5:Slant Bed CNC Lathe

(Photo courtesy of www.helmancnc.com)

Lathe Bed Design

Chuck Size 381 mm Bar Capacity 117 mm Swing Diameter Over Bed 875 mm	Chuck Size 254 mm Bar Capacity 76 mm Swing Diameter Over Bed 673 mm

Figure 6: Modern CNC turning centers

(Photo courtesy of HYUNDAI KIA,SKT 460 CNC Turning Centre and MAZAK, Quick Turn Nexus 250MSY)

Lathe Bed Design

Figure 7: A five-axis CNC turning center

(Photo courtesy of Mazak, Integrex 100-IIIST CNC Turning Center)

Axis Movement

- The **axis movement** of a basic CNC lathe is diagrammed in Figure 8. Some turning machines, such as that shown in Figure 7, are **five-axis machines**

- In this chapter, only the basic two-axis machine is programmed. The programming concepts learned on a two-axis machine are the **foundation necessary** to program more complex machinery

NOTE

The ***basic lathe has only two axes, X and Z***

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

Axis Movement

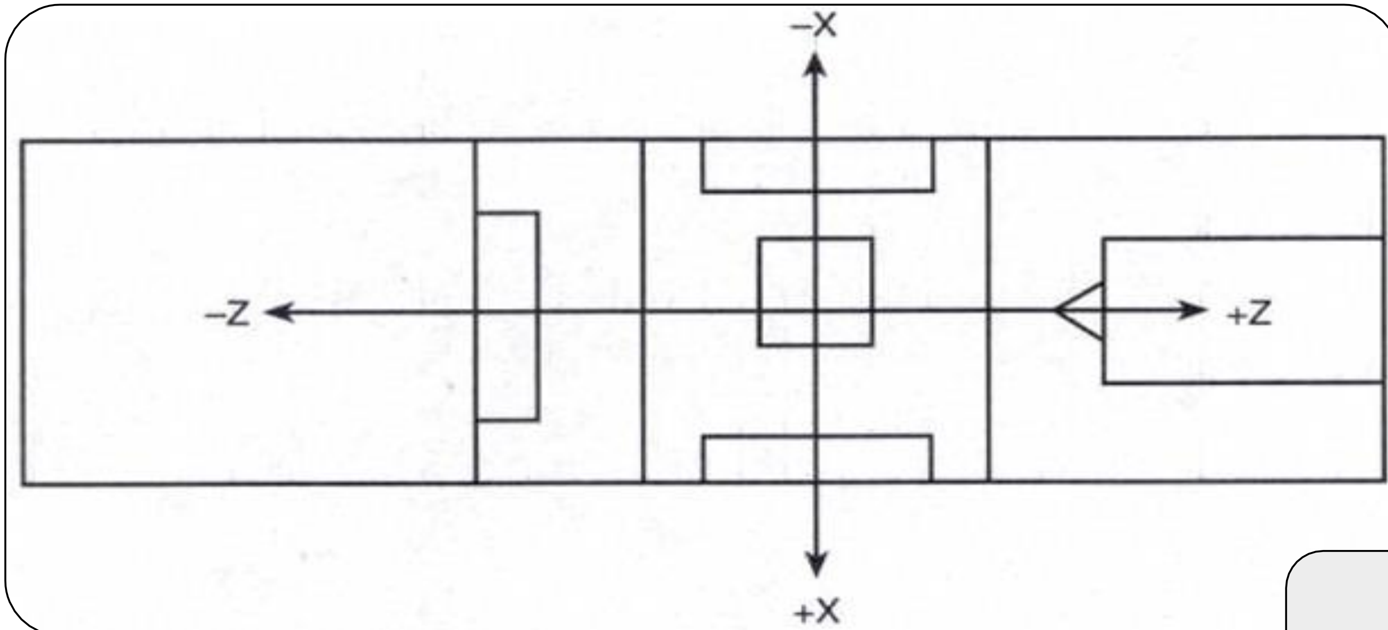
- Since the **Z** axis is always parallel to the spindle, **longitudinal** (carriage) travel is designated **Z**

- The cross slide movement is **designated X**, since it is the primary axis perpendicular to **Z**. If it were possible to move the carriage up and down, that axis would be **Y**. There is, however, a potential **problem** with this arrangement

- There appear to be **two Z axes**: ***the carriage movement and the tailstock movement***

Axis Movement

- Basic lathe has only two axes , X and Z



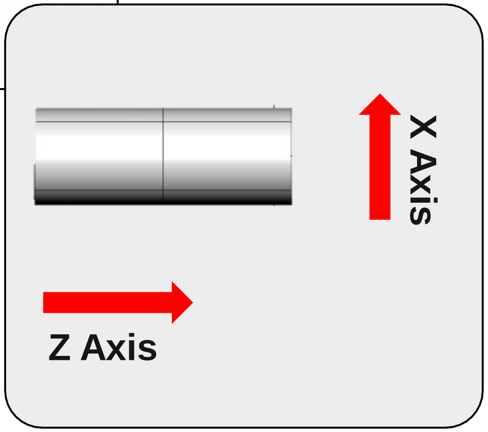


Figure 8: Lathe axis movement

(Adapted from Seams W., “Computer Numerical Control, Concepts & Programming”)

Axis Movement

- To eliminate this problem, the ***tailstock is usually called the W axis*** on lathes with **programmable tailstocks**

- **Programmable tailstocks**, which are **rear turret assemblies** on CNC equipment, are the third and sometimes fourth axes on more complex equipment

- The turning center in Figure 7 has two programmable **saddles**

- In such cases, the axes of the second saddle are usually designated **W** and **U**, with **W being saddle travel and U being cross slide travel**

- There are some imported lathes on which the X-axis direction is **reversed**. The programmer must **determine** if such a situation exists **before** writing the lathe program

Toolholders and Tool Changing

- Either a **rigid toolholder** or a **tool turret** is used to ***hold the tools*** on an NC lathe

- Figure 6 shows a CNC chucker employing a **rigid toolholder**. The turning center in Figure 7 employs a tool turret in which the **various tools** needed for lathe operations are placed in **tool-holders**

- When a **tool change is necessary**, the appropriate ***turret is indexed to the next tool*** needed

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

Toolholders and Tool Changing

- Simple lathes use six-sided turrets; larger turning machines use eight-, ten-, and twelve-sided turrets

- With the development of **robotics**, new tool changing and work handling schemes are appearing. Figure 15 shows a **robot arm used for handling workpieces**, and Figure 16 illustrates the robot in operation

NOTE

To teach the basics of CNC programming, this text **will focus on non-robotic tool change**

Toolholders and Tool Changing

Figure 9:CNC lathe tool turret

(Photo courtesy of www.helmancnc.com)

- **Tool capacity** for CNC lathes **varies** depending on the machine
- Small CNC lathes normally comes with a tool turret capacity of 4 or 8 tools stations

Toolholders and Tool Changing

- **Toolholders** used on NC turning machines are of very **rigid design**

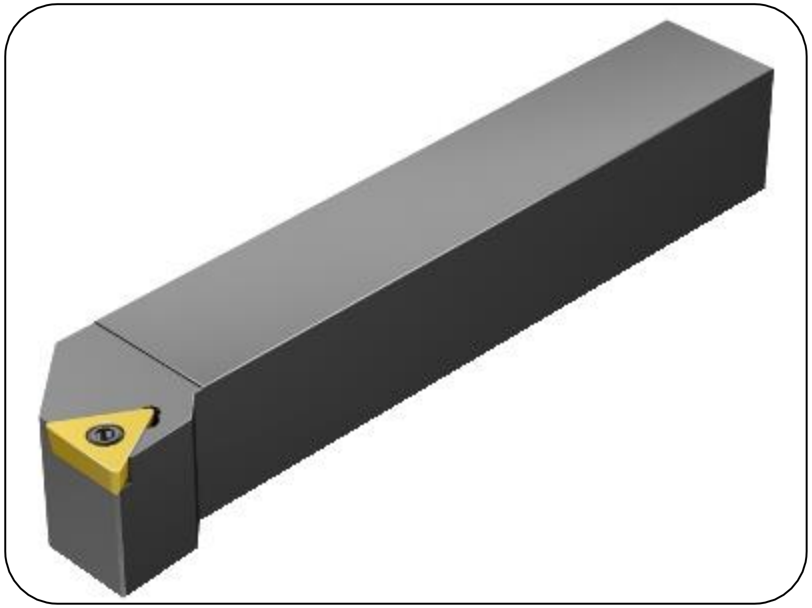
- The tools used for turning are of the **carbide insert type**, made to much more exacting **tolerances** than conventional lathe insert tooling

- A tool change command in a turning program either ***changes the turret position*** or ***causes an automatic tool change***, depending on the type of machine used

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

Toolholders and Tool Changing

- **Carbide**, results in **maximum strength** when looking at **hardness** and **wear resistance**



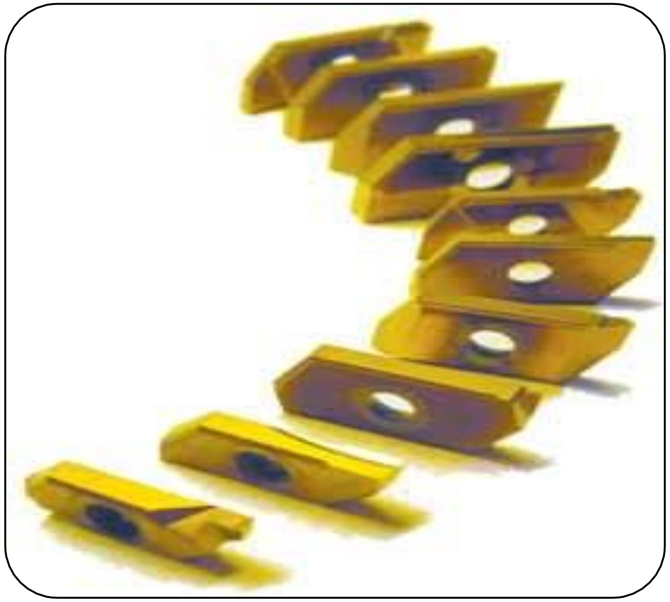


Figure 10:Carbide insert type turning tool and carbide inserts

(Photo courtesy of (a) Sandvik coromant, and (b) WhizCut inserts)

Toolholders and Tool Changing

Automatic Tool Change

- In a CNC turning program for a machine with a **rigid toolholder**, **M06** is used to initiate an ***automatic tool change***. The **T** address is used (as it is in milling programs) to specify the desired tool. The T address also **calls up the tool offsets**. The format for automatic tool change is:

M06Tn1 n2

  - Where **M06** initiates the tool change,
  - **T** is the tool address,
  - **n1** is the tool number, and
  - **n2** is the tool offset number

- Turret Position **T** is used in a similar manner with ***turret tool selection***. The format is:

Tn1 n2

  - Where the **first number** is the turret position and
  - the **second** is the tool offset number

Toolholders and Tool Changing

- Since one tool may be used in **several positions**, a turret position is used rather than a tool number. ***The turret position corresponds to the turret station number***

- **T01** will index the tool in **station one** into position

- Some NC lathes can utilize more than one tool on a single station. It is possible, therefore, for **T0101** to refer to one tool and **T0111** to refer to another. This is referred to as ***piggybacking a tool station***

Toolholders and Tool Changing

- One other point should be kept in mind when changing tools**: the carriage (or tailstock) does not necessarily move to a tool change location**

- It is often necessary, therefore, first to move the carriage or tailstock turret **out of the way** before making a tool change

It may also be necessary to program a dwell **(G04)** to **halt the program**, giving the tool time to index to ***position safely***

Toolholders and Tool Changing

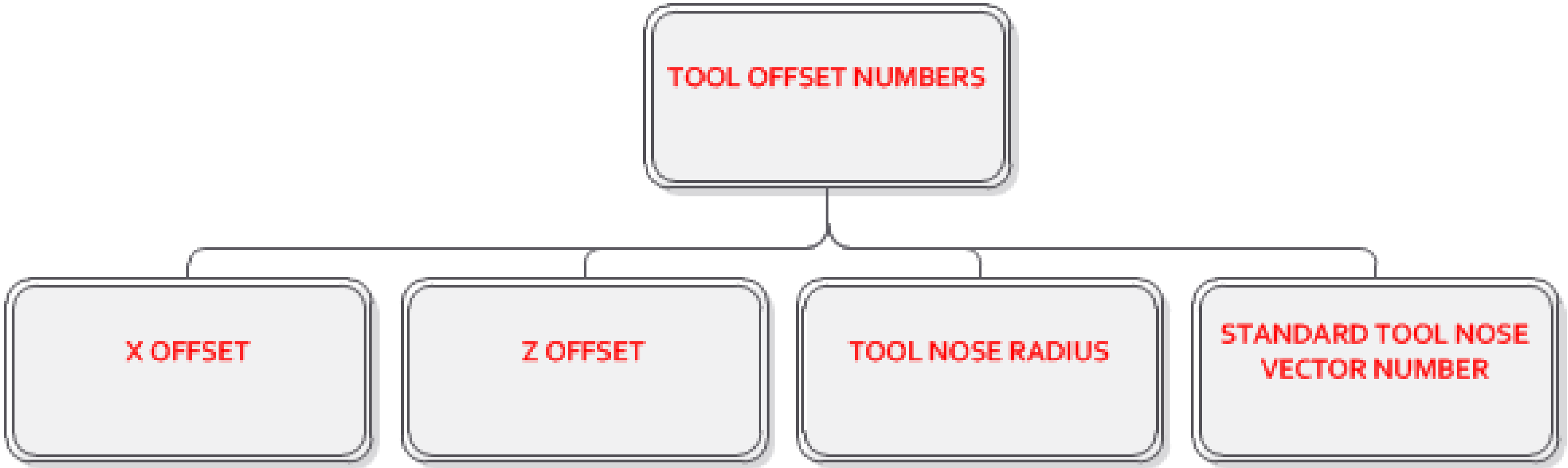


Figure 11:Tool offset diagram

Tool Nose Radius Compensation

Tool Nose Radius & Standard Tool Nose Vector Numbers

- The **tool nose radius** and **tool nose vector** numbers are optional. They are entered if using **cutter comp**
- Cutter diameter **compensation** is called ***tool nose radius compensation (TNR comp)*** on turning machines
- The **tool radius** tells the MCU the **amount of compensation** that is to be used. With NC machining centers this value was entered in a **comp register**

Tool Nose Radius Compensation

- **TNR comp** is utilized just as **cutter comp** was in Chapter 10. It can be used to **program** the part line or **fine tune** the tool path to **compensate** for tool wear

- The **major difference** is that lathe tools are not completely **circular** as is a milling cutter

- To aid in proper **compensation** of the tool path and correctly identify alarm conditions, a **tool nose vector number** is entered in the register

Tool Nose Radius Compensation

- **Tool nose vector numbers** tell the MCU the **orientation** of the tool nose
- Figure 12 shows the various directions in which a tool may be oriented
- These directions are referred to as ***vectors***
- Each vector has a number associated with it that is **used to describe the tool orientation** to the MCU

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

Tool Nose Radius Compensation

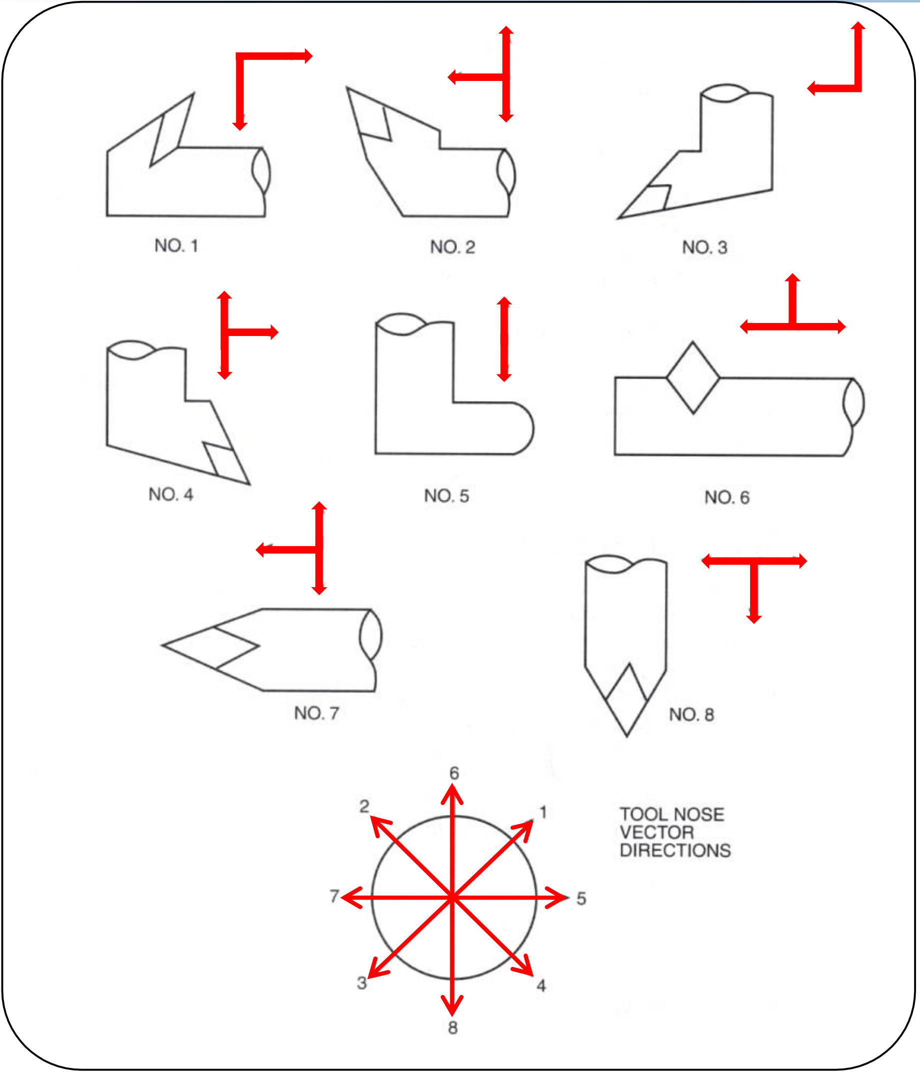


Figure 12: Tool nose vectors

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

- The **various directions** in which a tool may be **oriented**

Tool Edge versus Centerline Programming

- The **tool nose** may be programmed in one of two ways when **TNR comp** is not active:

    1. by the **tool edge** or
    2. by the **tool nose radius centerline**

- **Tool edge programming** is adequate for **simple straight line cuts** where the part surfaces intersect each other at right angles

Problems are encountered, when ***angles and especially arcs*** are programmed this way

- Figure 17(a) illustrates this point

Tool Edge Programming

- If the **tool edge** is programmed, the **I** and **K center-points** of the illustrated arc must be **shifted**. This results in a ***tool path that does not follow the desired arc*** exactly

- The amount of error that is induced **depends on the size of the cutter** and the **radius of the arc**

In any case, **tool edge programming should not be used when encountering arcs and angles**

Tool Centerline Programming

- **Tool centerline programming** is **identical** to the centerline programming done when **milling**

- Figure 17(b) demonstrates how the cutter **centerlines** and part surface centerlines **coincide *when the center of the tool nose radius is programmed***

Other Parts of Modern CNC lathes

Quicksetters

- A fairly recent development has been the use of ***quicksetters***—**arms with tool sensors** on them

- During job setup, the arm is lowered into position, the operator **jogs a tool** to the presetting position, and **touches it off on the sensor**

- The **quicksetter** automatically ***sets the values of the work coordinate and the tool offset registers***

Other Parts of Modern CNC lathes

- ***Quicksetters*** can also measure the size of tools **before** cutting starts, and
- ***Check for tool damage or breakage*** during the machining operation

Figure 13: CNC lathe tool setter (Photo courtesy Renishaw probe systems)

Other Parts of Modern CNC lathes

Continuous supply feedstock systems

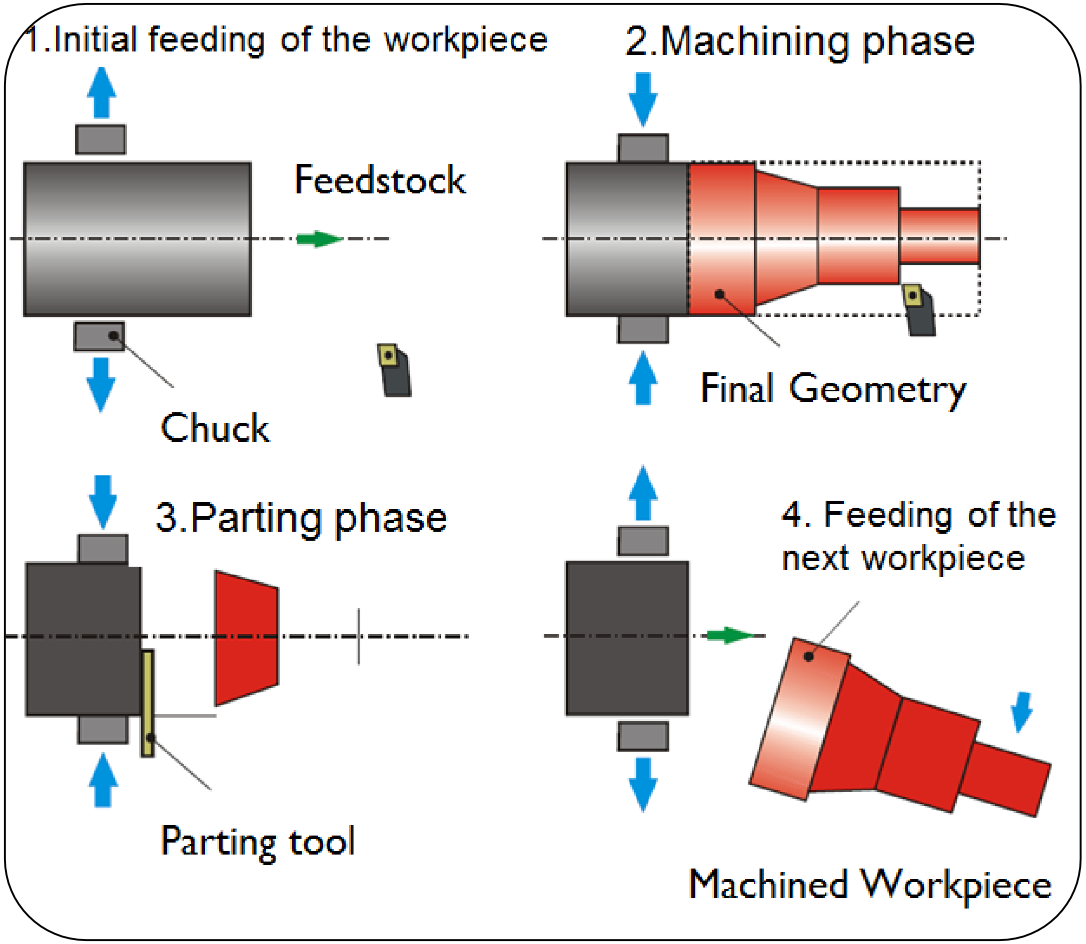
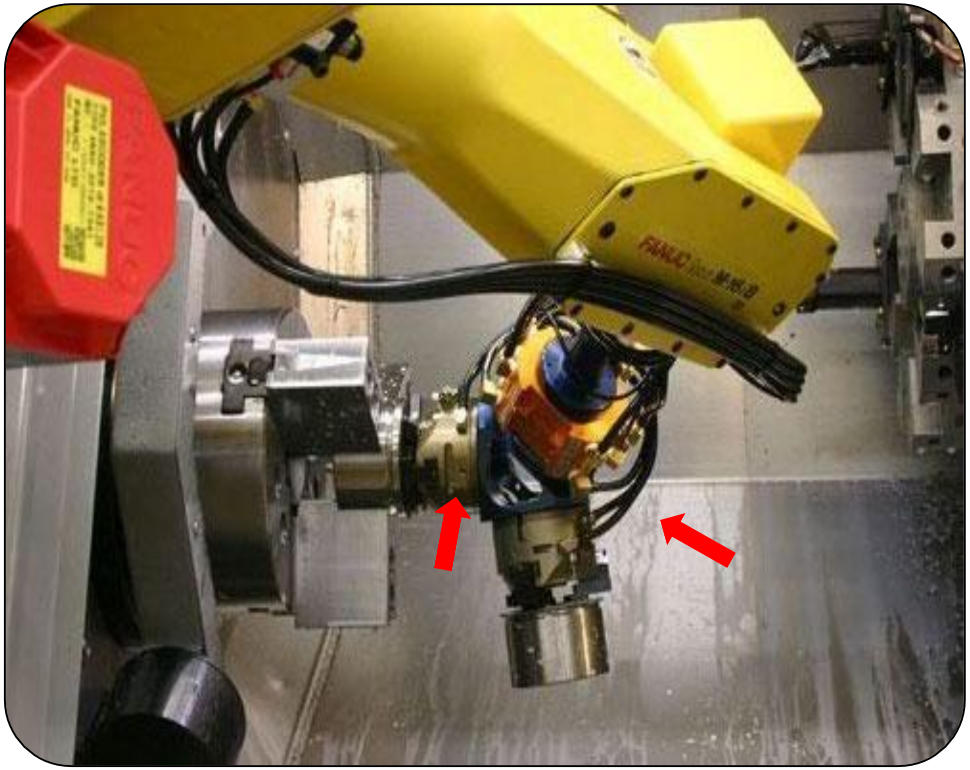


Figure 14:Continuous supply feedstock system

Other Parts of Modern CNC lathes

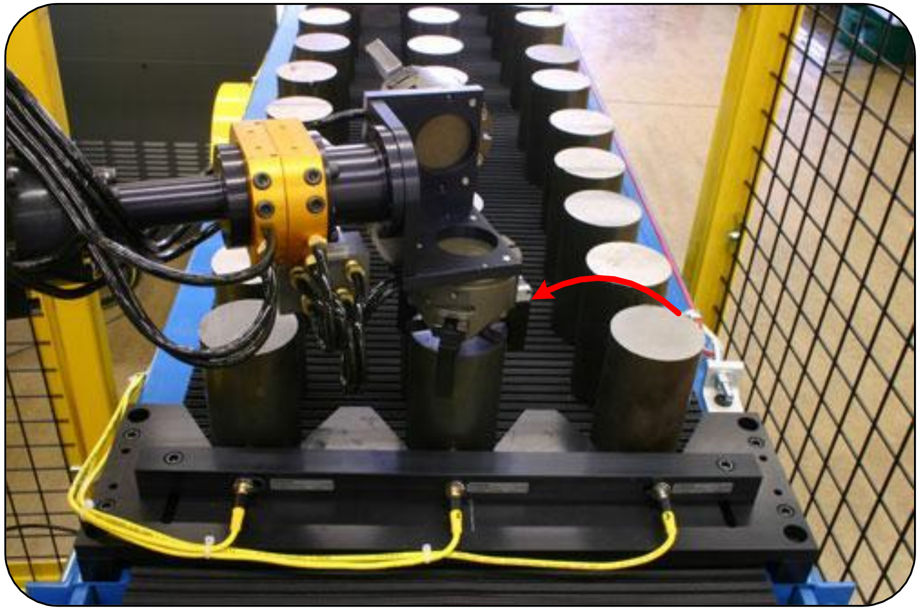
Robot arms used for handling workpieces



- Fanuc M16iB robot arm used for **handling workpieces**
- The second gripper head allows the **exchange of the previous workpiece**

Figure 15: A robot arm used for part load and unload
(Photo courtesy of Wanner Engineering, Nakamura-Tome WT-150 lathe, using a Fanuc M16iB robot arm)

Parts of CNC lathes



A **bar feeder** provides the blank ⇨ The **robot arm** delivers it to the machine and also **removes** the **previous** workpiece

Figure 16: A robot arm in action

(Photo courtesy Centurion Systems, manufacturer of gate motors and access automation products using Arcmate 120iB from FANUC Robotics)

Spindle Speeds

- **Spindle speed** is specified using an **S address**, just as in milling. On turning machines with a gear head design, the spindle speed is changed by shifting gears in the headstock
- On **gear head machinery**, there are usually **two or more** gear ranges
- An **M** function is used to select the gear range in which the desired speed is located
- **M40** through **M46** generally serve this purpose
- For **gear head** examples in this text:
  - **M40** will be used for low range
  - **M41** for mid range, and
  - **M42** for high range

Spindle Speeds

- The following chart shows a sample of **speed ranges for gear head machines**. This chart is not for a particular machine but is **representative** of the type of **spindle speed spread** found on a machine

- Some CNC turning machines use a **variable speed drive** with which an **infinite number of speeds** are available between the highest and lowest speeds

- In these cases, the speed is selected using the **S address** as it is in milling

Spindle Speeds

LOW RANGE

10	15	20	25
30	40	50	65
75	90	110	125

MEDIUM RANGE

55	70	95	120
140	155	175	200
235	260	290	300

HIGH RANGE

285	335	380	450
530	660	900	1200
1800	2100	2500	3000

Chart 1: A sample of **speed ranges** for gear head machines

Spindle Speeds

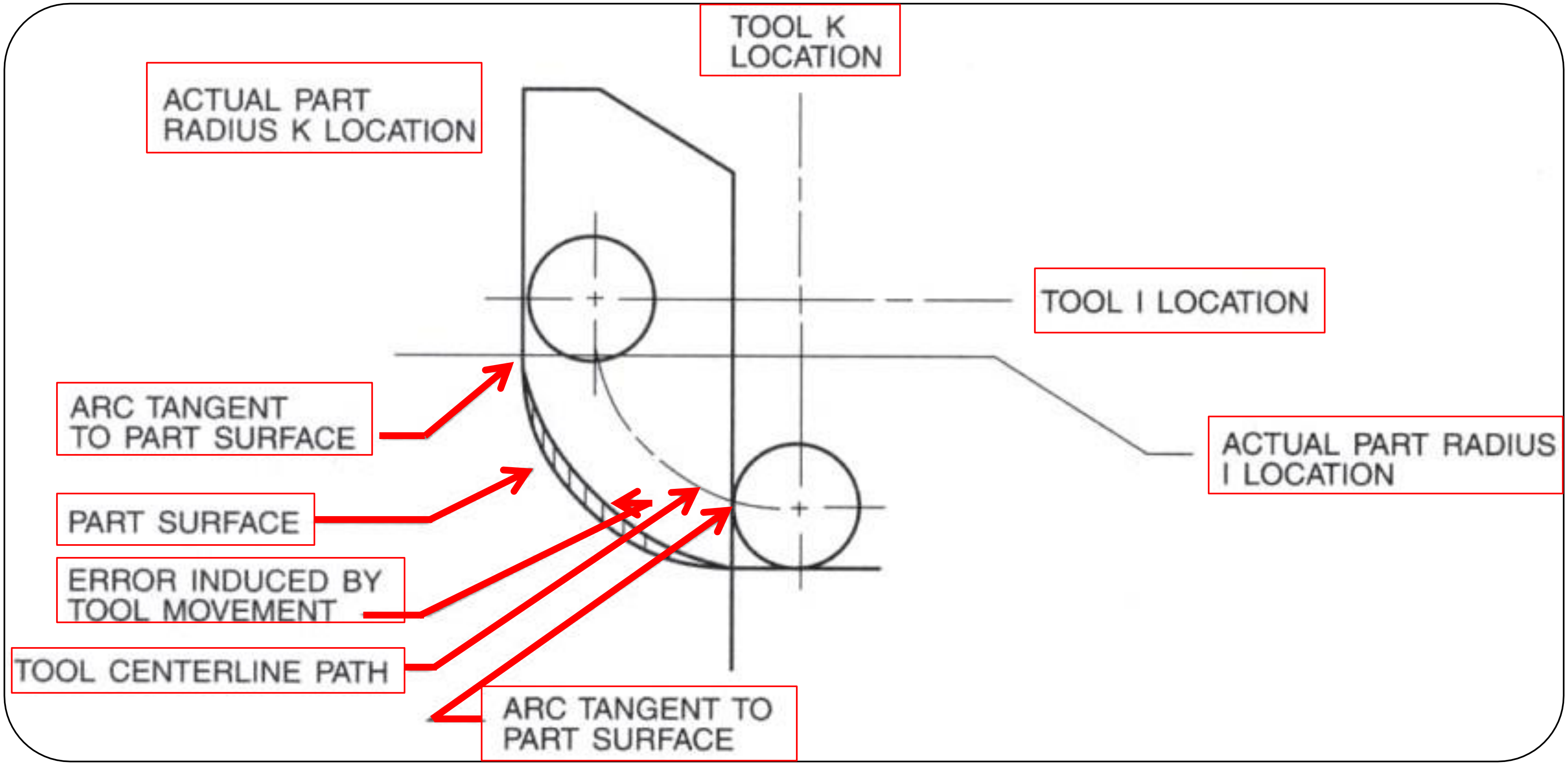


Figure 17(a): Error induced by programming tool edge

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

Spindle Speeds

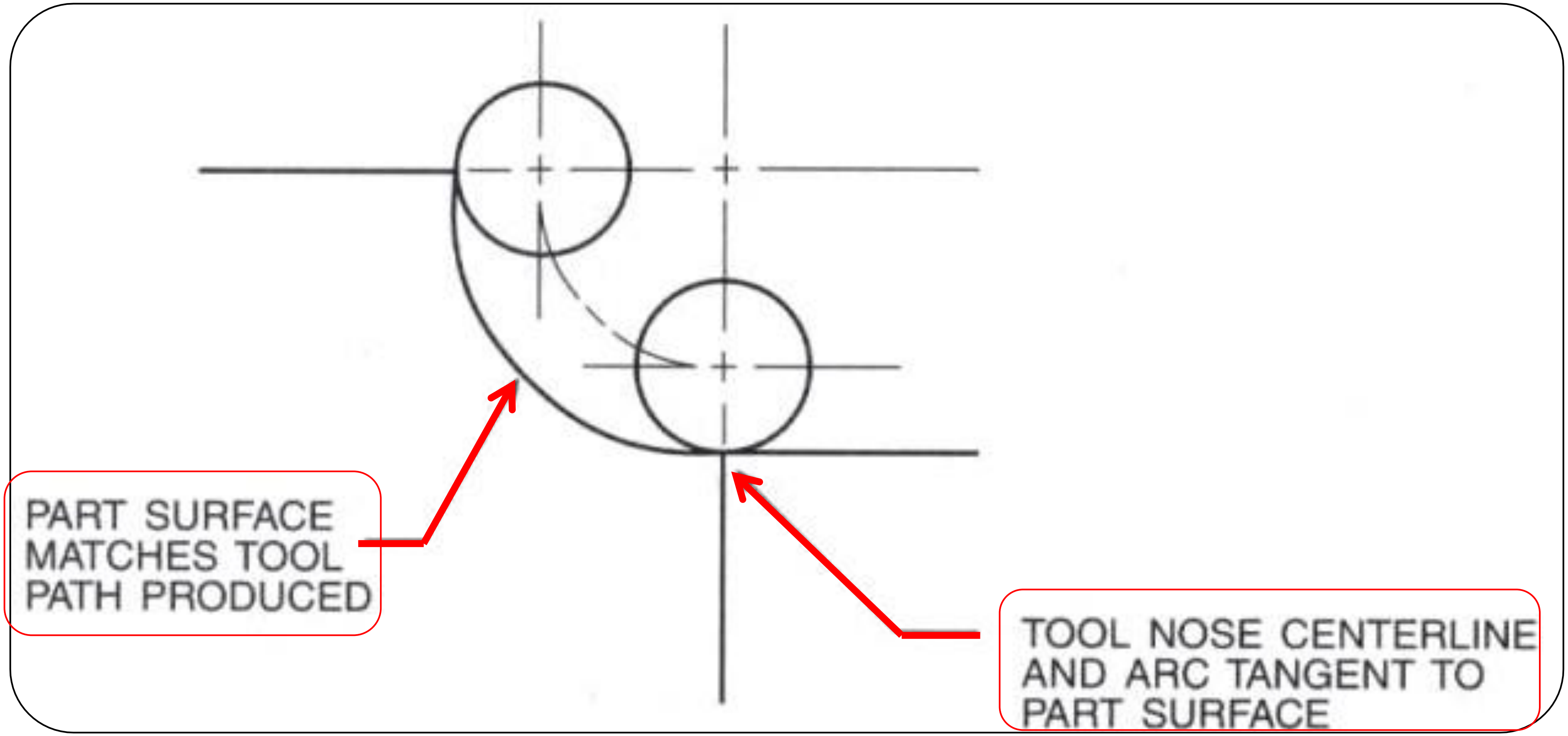


Figure 17(b):Tool nose centreline programming

(Adapted from Seams W., “Computer Numerical Control, Concepts & Programming”)

Feed-rates

- With a CNC lathe, assigning **feedrates** is quite simple

- A **G98** or **G94** code (depending on the controller) tells the MCU that the following feedrate is in inches per minute

  - For example, **G98 F7** specifies a **feedrate of 7 inches per minute**. A **G99** or **G95** in a turning program specifies a feedrate in inches per revolution

  - For example, G99 F.015 specifies a feedrate of .015 inch per revolution

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

Machine Origin and Work Coordinate Systems

- An NC lathe generally has a **fixed zero position** assumed by the executive program upon power-up. This position is known as the ***home zero*** or ***machine origin***

- The physical location of this position **varies** from controller to controller and machine model to machine model

- It is usually one of two locations: **X0 = centerline of the spindle**, **Z0 = the chuck mounting surface of the spindle**, or **X0** = **extreme X + location**, **Z0 = extreme Z + location**

- It is usually necessary to ***establish a zero point*** on the part different from the machine origin location. This position is called the ***work coordinate system or part zero***

Machine Origin and Work Coordinate Systems

- There are two methods used to accomplish this:
- The first method involves the use of an axis preset command—**G50**. The **G50 transfers** the zero point from the **home zero to the coordinates specified** with the command
- The **format** for a **G50** command is:

G50 Xxx.xxxx Zzz.zzzz

Where:

G50 = the axis preset command

xx.xxxx = the X-axis distance to the part zero

zz.zzzz = the Z-axis distance to the part zero

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

Machine Origin and Work Coordinate Systems

- A **G50** command is issued at the **start of each tool**. Since the programmer will not know the axis preset distances in advance, **zeros** should be used or some other prearranged value in the **G50** line

NOTE

The actual values will be ***determined by the setup person*** and edited in the control when the job is set up

- The second method uses registers called ***work coordinates***. These are registers in the MCU that tell the MCU the **distance** from **home zero to the part zero**

Machine Origin and Work Coordinate Systems

- If a machine has more than one available work coordinate, **multiple zero points** may be used for complex programming

- Another advantage to multiple work coordinates is the ***ability to have more than one program loaded*** in the MCU, each with its **own work coordinate**

- This is a decided advantage when running several repeating jobs through a turning center

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

Machine Origin and Work Coordinate Systems

- Each **work coordinate** is called by a **G code**. If a program were to use four work coordinates, they would be selected by the codes **G54**, **G55**, **G56**, and **G57**

- The first work coordinate (**G54** in this case) is the default work coordinate

- This work coordinate is **automatically activated** upon power-up

Machine Origin and Work Coordinate Systems

- If using only the default work coordinate, the **G code** may be omitted

- The work coordinate values are **entered by the setup person** when the job is prepared

The programmer must **instruct** the setup personnel the position on the part of the ***part zero location***

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

Summary 1/3

- CNC turning machines often use a **slant bed arrangement** to protect the machine ways from chips. Although different in appearance, the ***functioning of a slant bed and conventional bed machine is identical***

- There are ***two basic axes, X and Z***, on a CNC lathe. If the lathe has additional axes, they are generally designated **U** and **W**

- **TNR** stands for **tool nose radius** compensation. **TNR** is the **equivalent** in CNC turning to cutter diameter **compensation** in milling

- A **tool turret** or a rigid toolholder is used to **hold the tools** on an NC lathe

Summary 2/3

- **Tool offset** are entered into the MCU **prior** to running the program to **compensate** for minor setup adjustments

- A standard **tool nose vector number** is used to ***identify the orientation*** of a particular tool when using **TNR**

- A **tool change command** in turning programs will either **change the turret position** or cause an automatic tool change, depending on the type of machine used

- The tool change format for turret changing is: **T n1 n2**

  Where **T** is the tool change command, **n1** is the turret position and **n2** is the tool offset

Summary

- The format for automatic tool change is : **M06 T n1 n2**

  Where **M06** initiates the tool change, **T** is the tool address, **n1** is the tool number, and **n2** is the tool **offset** number

- **Spindle speeds** are specified directly using the **S address**. On gear head machines, it is necessary ***to specify the gear range*** when selecting a range **outside** the active one.

- **Feedrates** on CNC lathes can be specified either in **inches per minute** (using **G94** or **G98**), or in **inches per revolution** (using **G95** or **G99**).

- To set a part at **X0/Z0** point, it is necessary to **transfer the machine origin** to the workpiece using a G code

Vocabulary Introduced in this chapter

- Centerline programming
- Lathe bed
- Quicksetter
- Slant bed
- Tool edge programming
- Tool nose radius
- Tool nose vector number
- Tool offset numbers
- Tool turret
- Turret position

References

1. Chryssolouris G., «Manufacturing Systems: Theory and Practice», 2nd Edition, 2006, Springer-Verlag
2. http://www.centsys.co.za/
3. http://www.dptechnology.com/
4. http://www.sandvik.coromant.com/
5. http://www.hyundai-wiamachine.com/
6. http://www.renishaw.com/
7. Kalpakjian S., «Manufacturing Engineering and Technology», 2nd Edition, 1992, Addison-Wesley Publishing company
8. Kalpakjian,Schmid,<<Manufacturing Processes for Engineering Materials>>, 5th ed. 2008
9. Mattson M., “CNC Programming, Principles and Applications”, Delmar, 2002
10. Moriwaki T., “Multi-Functional Machine Tool”, CIRP Annals Manufacturing Technology, Vol. 57/2, 2008, pp. 736-749

References

11. Seams W., “Computer Numerical Control, Concepts & Programming”, 4th Edition, Delmar, 2002
12. www.mazak.com
13. www.whizcut.com/carbide.html
14. Γ. Χρυσολούρης, «Συστήματα Παραγωγής Θεωρία και Πράξη» Μέρος I και II, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2001,
15. Γ. Χρυσολούρης, Δ. Μούρτζης, Κ. Τσίρμπας, Σ. Καραγιάννης, “Ορθογωνική Κοπή”, Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2000
16. 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)
17. Δ. Μούρτζης, “Αριθμητικός Έλεγχος Εργαλειομηχανών” Εκπαιδευτικές Σημειώσεις, Πανεπιστήμιο Πατρών, 2011 (3η Έκδοση)
18. Πετρόπουλου Π.Γ., «Μηχανουργική Τεχνολογία – II. Τεχνολογία κατεργασιών κοπής των μετάλλων», 1998, Εκδόσεις Ζήτη
19. Σύγχρονες μέθοδοι κατεργασίας υλικών και προγραμματισμός με Ηλεκτρονικό Υπολογιστή (Η/Υ) ,Δ. Μούρτζης ,Κ. Σαλωνίτης