Machining

Level-III

Based on October 2023, curriculum version 2



Module Title: - Perform Advanced Lathe CNC Operations

Module Code: INDMAC3 M02 0322

TTLM Code: IND MAC3 TTLM02 1023-v1

Nominal Duration: 120Hour

Prepared by: Ministry of Labor and Skill

October, 2023

Addis Ababa, Ethiopia

Acknowledgment

The Ministry of Labor and skill wishes to thank and appreciation to MoLS leaders and experts, Regional Labor and skill/training Bureaus leader, experts, TVT College Deans, Instructors and industry experts who contribute their time and professional experience to the development of this Training Module.

Acronym

CNC	**Computer numerical control**
EOB	End of block
F	Feed
N	Sequence numbers.
OD	Outside Diameter
MDI	Manual Data Input
PD	Pitch diameter
RPM	Revolutions per minute
S	Spindle speed
T	TOOL
TTLM	Teaching, Training and Learning Material.
TVT	Technical and Vocational Training.

Table of Contents

Acknowledgment i
Acronym ii
Table of Contents iii
Introduction to the Module v
Module units vi
Unit one:- The Fundamentals of lath CNC 1
1.1 Initial information of CNC programing 2
1.2 Sequence of CNC lathe operation 8
1.3 CNC Lathe Machine Components, accessory and measuring instruments 10
1.4 Cutting tools and parameters. 33
1.5 CNC Lathe Process/job/adjustment sheets. 39
1.6 work shop safety 41
Self-Check 1 43
Operation sheet-1 45
LAP Test 1. 46
Unit TWO: program CNC lathe machining process 47
2.1 CNC lathe coordinate systems. 47
2.2 CNC lathe operations program 52
2.2.1 Part Program 52
2.3 Simulate and edit CNC lathe Program. 55
2.4 Save CNC lathe Program. 58
2.5 CNC lathe program downloaded to the machine. 60
Self-Check 2 62
Operation sheet-2 65
LAP Test 2 67
Unit Three: CNC Lathe operations 68
3.1 Mounting work-piece. 69
3.2 CNC lathe tool path dry run. 71
3.3 Advance CNC lathe operations. 73
3.4 Corrective measures/adjustments with standard. 83
Self-Check 3 86
Operation sheet 3 88

LAP Test 3 91
Unit Four: Quality assurance of produce parts 92
4.1 Inspect components. 93
4.2 Deviations handling procedures and standard 94
4.3 Comparing product quality of the CNC production with conventional production. 97
Self – Check 4 99
Book of reference 100

Introduction to the Module

Computer Numerical Control (CNC) is the process of having a computer control the operation of a machine. This would include the table motions, the spindle speed, and various other machining functions. Having the computer control the machine is accomplished by a program that is written using NC code. These coded instructions are carefully arranged by using letters (A, B, C, Z), numbers (1,2,3...9) and symbols (+, -, /...etc.) to create a program. These instructions can range from a few lines for a simple part to many thousands of lines for a complex part. If the cumulative instructions (program) are written correctly it will provide specific guidance on exactly what functions you want the machine to perform, resulting in the production of a quality part. On the other hand, a poorly written program can have disastrous results... scraping parts, destroying tools, and possibly damaging the machine. In extreme cases the operator's own personal safety may be in jeopardy. One of the most common types of machining processes of modern time is Lathe CNC Operations. CNC lathe machining operation involves the use of various type of lathe cutting tools to perform such operations like facing, turning, chamfering, grooving, knurling, threading, drilling, boring, reaming.
In machining filed; the Perform Advanced Lathe CNC Operations module cover skills, knowledge and attitudes required in carrying out Perform Advanced Lathe CNC Operations manufacturing of different types of Turning, boring, facing, grooving, and Treading of different profile of metal work.
This module is designed to meet the industry requirement under the machining occupational
standard, particularly for the unit of competency: Performing Advanced Lathe CNC Operations.

Module units

This module covers the units:

- Fundamentals of CNC lathe
- CNC Lathe machining process
- Lathe operations
- CNC Lathe Product Quality

Learning objectives of the Module

At the end of this session, the students will able to:

- Determine job requirements
- Prepare CNC lathe machining process / Write program
- Perform appropriate Lathe operations
- Comply with Quality assurance

Module Learning Instructions:

For effective use this modules trainees are expected to follow the following module instruction:

1. Read the information written in each unit
2. Accomplish the Self-checks at the end of each unit
3. Perform Operation Sheets which were provided at the end of units
4. Do the “LAP test” given at the end of each unit and
5. Read the identified reference book for Examples and exercise

Unit one:- The Fundamentals of lath CNC

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Initial information of the Geometry
- CNC Lathe operation sequence
- CNC Lathe Machine Components
- Cutting tools and parameters.
- Process/job/adjustment sheets.

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Identify and interpret the required drawings
- Understanding sequence of operation.
- Identify CNC Lathe Machine Components
- Selecting proper cutting tools and cutting parameters for the required operation
- Work adjustment and selecting raw material filed sheet

1.1 Initial information of CNC programing

Development of any CNC program should always begin with a very carefully planned process. Such process typically starts with an engineering *drawing* (also called a *blueprint* or a *technical print*) of the required part, released for production. Before any machining process can be completed, several steps have to be considered and carefully evaluated. Placing a greater effort into program planning will yield better program and better machined part.

The main purpose of most engineering drawings is to define the part shape, individual dimensions, and relationships between part features. Some drawings may also include data about the initial blank material (stock), such as type, size, and shape. In CNC programming, good familiarity with various materials is important. For programming purposes, materials used to machine a part are evaluated by their ***size, type, shape, condition, hardness***, etc. Easily inspect and select the rough size of the work stock blank. (smid peter)

PARTS LIST			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	body	
2	1	plate	
3	1	handle	
4	1	hallow shaft	

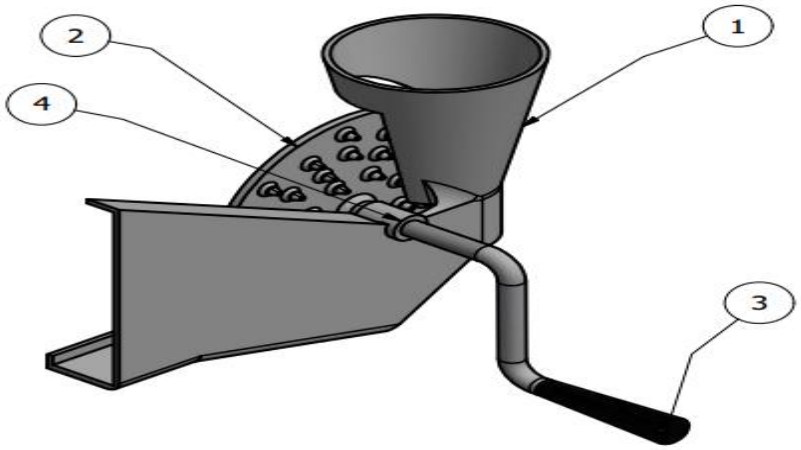


Figure 1.1 assembly drawing

Standardization: - is the process of formulating and applying rules for an orderly approach to a specific activity for the benefit and with the cooperation of all concerned, and in particular for the promotion of optimum overall economy taking due account of functional conditions and safety requirements."

Working drawings

Part drawing and material data are the primary sources of information about a specific part to be machined. They define the starting point of program planning. The objective of such a plan is to collect all available data and use all initial information for one purpose - to establish grounds for the most efficient method of machining, along with all related considerations.

- mainly ***part accuracy, productivity, safety and convenience***.

Is classified in to two:

- Detail drawing
- Assembly drawing

1.1.3 Detail drawing: Detail drawing is multi view presentation of a single part with dimensions and notes. And it conveys the information and instructions for the manufacturing the part. *Fig. 1.2*

1.1.4 Assembly drawing: is a drawing of various parts of a machine or structure assembled in their relative working position. Fig 1.1. Assembly drawings show how different parts go together, identify those parts by number, and have a parts list, often referred to as a bill of materials.

Drawing and material data provide much of initial information, but they are not the only source. A great part of what is needed to develop a part program is not found in the drawing directly, but in other documentation. For example, a process sheet (routing sheet) provides many engineering requirements *not covered* in the drawing, such as

- *pre*- and *post*- machining operations,
- grinding allowances,
- assembly features,
- requirements for hardening,

- next machine setup, and many others.

Collecting relevant information from all available sources provides enough groundwork to start planning a CNC program development.

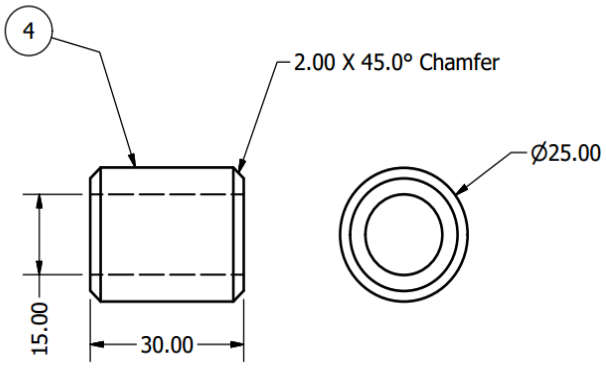


Figure 1.2 detail drawing

Part`s information

Shape:

- orthographic drawing
- Pictorial drawing

Size: - dimensions and tolerances

Specification: -

- part number, name, numbers required
- Types of material used
- General notes
- Heat treatment
- Surface finish
- General tolerance

1.4.5 Tolerances and fits

Among the important things to consider to define or express a part, we have to have enough knowledge about: -

- Surface finish
- Tolerance

- Size
- Geometry

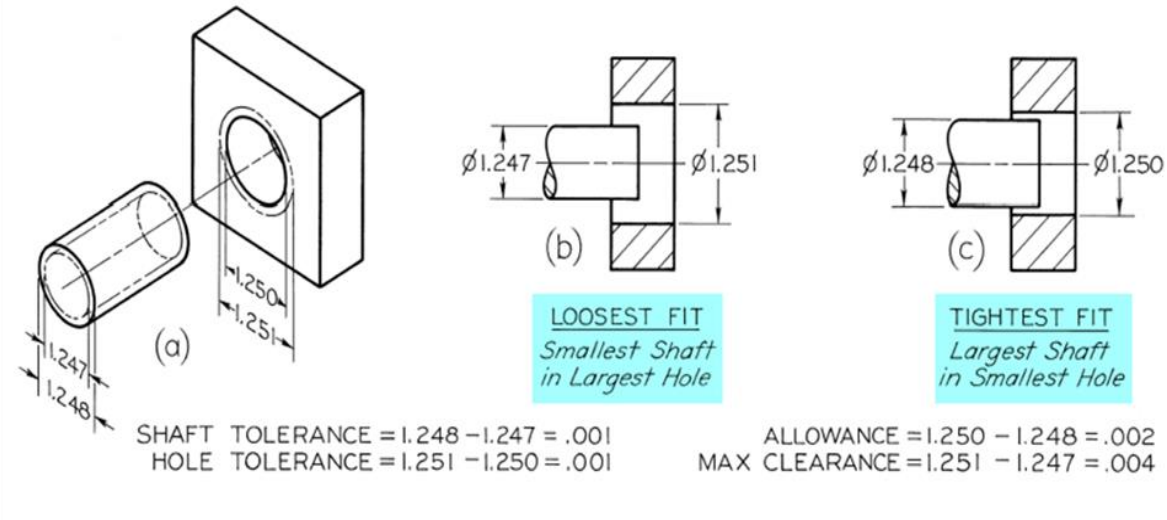


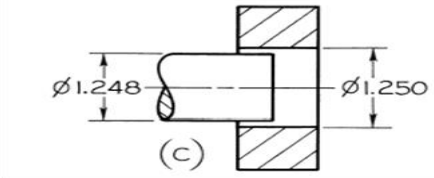
Figure 1:3 Types of fit

1.4.6 Fits between Mating Parts

Fit is the general term used to signify the range of tightness or looseness that may result from the application of a specific combination of allowances and tolerances in mating parts.

There are four types of fits between parts

A. **Clearance Fit:** an internal member fits in an external member (as a shaft in a hole) and always leaves a space or clearance between the parts.



Minimum air space is 0.002". this is the allowance and is always positive in clearance fit.

B. **Interference Fit:** The internal member is larger than the external member such that there is always an actual interference of material. The smallest shaft is 1.2513" and the largest hole is 1.2506", so that there is an actual interference of metal amounting to at least 0.0007". Under maximum material conditions the

interference would be 0.0019". This interference is the allowance, and in an interference fit it is always negative.

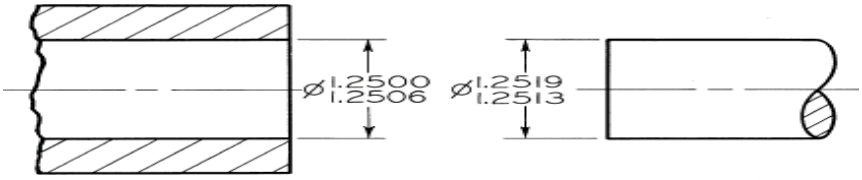


Figure 1:4. interference fits

C. **Transition Fit**: may result in either a clearance or interference condition. In the figure below, the smallest shaft 1.2503" will fit in the largest hole 1.2506", with 0.003" to spare. But the largest shaft, 1.2509" will have to be forced into the smallest hole, 1.2500" with an interference of metal of 0.009".

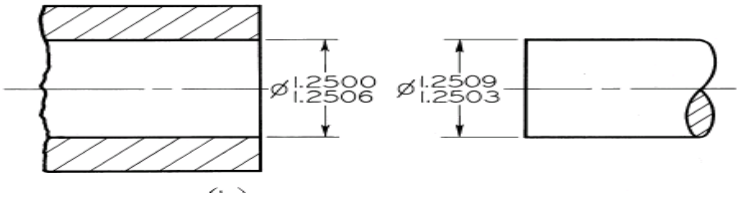


Figure .1;5 Transition fits

D. **Line Fit**: the limits of size are so specified that a clearance or surface contact may result when mating parts are assembled.

Specifications of Tolerances

1.1.5 CNC Machine Features

No amount of initial information is much useful if the selected CNC machine is not *suitable* for a particular job. During program planning, CNC programmer concentrates on a *particular machine tool with* a *particular CNC system*. These two major parts of a CNC machine are always connected and they must always be considered in any single CNC machine definition. It is just not enough to select a special fixture or a special setup - the CNC machine itself has to be suitable to handle any required setup.



Figure 1.6 CNC lathe machine

1.1.6 Advantages of CNC Machine

- Increased Program storage capability at the machine tool
- Program editing at the machine tool
- Control systems upgrades possible
- Option -resident CAM system at machine tool
- Tool path verification
- Quality control through process control
- Accommodates simple to complex parts geometry
- Improved part aesthetics
- Increased productivity

1.1.7 Limitation of CNC Machine

- High capital cost (Machine tools cost $30,000 - $1,500,000)
- Retraining and recruitment of staff (the requirement of skilled part programmer).
- High maintenance requirements
- Not cost-effective for low-level production on simple parts i.e. as geometric complexity or volume increases CNC becomes more economical.
- Maintenance personnel must have both mechanical and electronics expertise

1.1.8 Machine Type and Size

Possibly the two most important steps in program planning relate to the *type* and *size* of CNC machine, particularly its *work space* or *work area*. Other features, equally

important, are machine power rating, spindle speed and feed rate range, number of tool stations, tool changing system, available accessories, etc. Typically, small CNC machines have higher spindle speeds and lower power rating, large machines have lower spindle speeds available, but their power rating is higher.

1.1.9 Part Complexity

When all initial information, such engineering drawing, material stock, available tools and CNC equipment are evaluated, the complexity of programming task becomes much clearer. How difficult is to program a given part manually? What are the machine capabilities? What are the costs? Many questions have to be answered before starting actual program development.

1.2 Sequence of CNC lathe operation

Machining sequence defines the order of machining operations. Technical skills and machine shop experience do help in program planning, but a good quantity of common sense approach is equally important. Machining sequences must have logical order - for example, ***drilling must be programmed before tapping, roughing operations before finishing, first operation before second,*** **etc**. Within this logical order, further specification of the order of individual tool motions is required for a particular tool. For example, in turning, *a face cut may be programmed on the part first, then roughing all material on diameters will take place*. Another method is to program a roughing pass for the first diameter, then face and continue with the remainder of the diameter roughing afterwards. In drilling, a spot drill before drilling may be useful for some applications, but in another program a center drill may be a better choice. There are no fixed rules, no absolutes, that determine which method is better - each CNC programming assignment has to be considered individually, based on the basic criteria ***of safety, quality, and efficiency. (simin Peter)***

Machining sequence must follow logical order

1.2.1 Tooling Selection

Selecting tool holders and cutting tools is another important step in planning a CNC program. Category of tooling, their selection and usage, includes a lot more than just cutting tools and tool holders - it covers an extensive line of accessories, involving numerous ***vises, fixtures, chucks,***
sub-tables, steady-rests, tailstock, indexing tables, clamps, collets and many other holding and auxiliary devices. Cutting tools require special attention, due to the very large variety available and their direct effect in machining.

Cutting tool used for the job is usually the most important selection. It should be selected by two main criteria:

- **Efficiency of usage**
- **Safety in operation**

1.2.2 Part Setup

Another decision in program planning relates to the actual part setup - *how to mount* the raw or pre machined material, what *supporting* tools and other devices should be used, how many *operations* are required to complete as many machining sequences as possible, where to select a *program zero,* etc. Setup is necessary and it should be done efficiently.

1.2.3 Selection of cutting parameters

The next stage of CNC program planning involves selection of ***spindle speeds, cutting feed rates, depth of cut, coolant application, etc***. All of the already considered factors will have their influence. For example, the available range of spindle speeds is fixed for any CNC machine, size of the cutter and material type will influence speeds and feeds, power rating of the machine tool will help determine what amount of material can be removed safely, etc. Other factors that influence program design include tool extensions, setup rigidity, cutting tool material and its condition. Not to be overlooked is the proper selection of cutting fluids and lubricants - they, too, are important for overall part quality. (Smid Peter)

1.3 CNC Lathe Machine Components, accessory and measuring instruments

1.3.1 Parts of a CNC Lathe

The Parts of the lathe machine are as follows.

1. Lead Screw
2. Bed
3. Guideways
4. Prime mover
5. Transmission system
6. Feed Gearbox
7. Headstock
8. Spindle
9. Chuck
10. Carriage
11. Tailstock
12. Tool post
13. Slide ways with saddle
14. Thread Chasing Dial
15. Split nut
16. Compound rest
17. Quill
18. pan
19. legs
20. apron

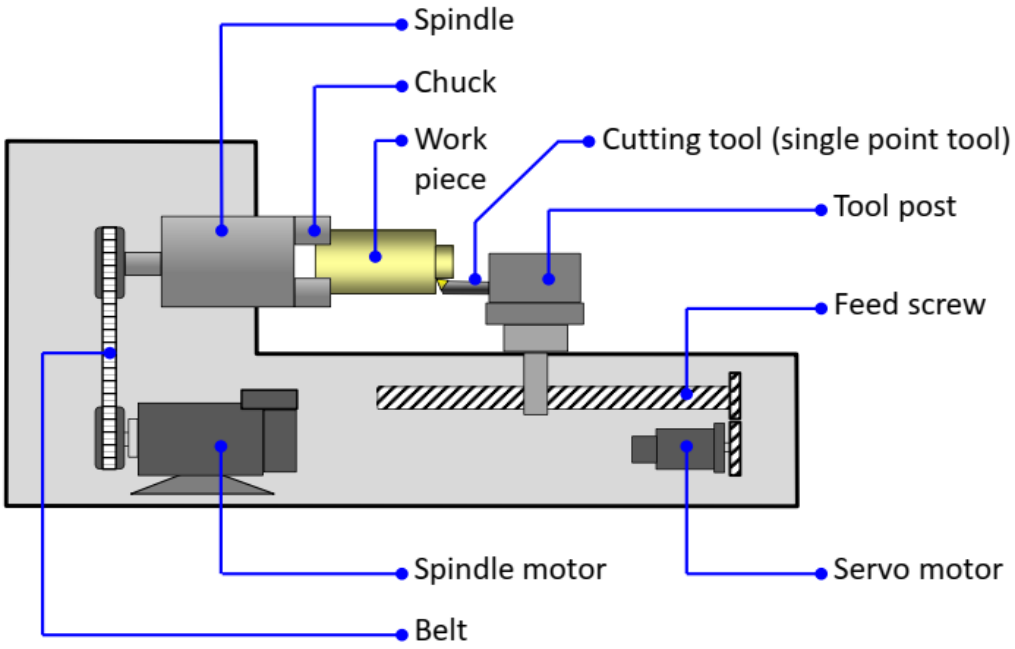


Figure 1:7 CNC lathe parts

a. Lathe Machine Parts and Functions:

The detailed explanation of all the parts is as follows.

b. Leadscrew:

- The purpose of the leadscrew is to convert rotational movement into linear movement.
- The material of the lead screw must be selected such that it should have high wear resistance
- Due to this, the best preferable material for the manufacturing of lead screw is the brass.
- Because of the sagging problem, the maximum length of the brass lead screw will be 2 meters only.
- The type of threads used on the lead screw is square thread because it gives the highest efficiency of power transmission. For example, Buttress Thread.

c. Bed of Lathe machine:

- It should have high damping qualities i.e. it can absorb the vibrations induces during machining.
- It should act as a house for keeping some of the parts in a machine tool like Prime mover of a machine, gearbox, coolant tank, etc.

Material:
Grey C.I: It is used for manufacturing of bed of the machine because it has high damping qualities and rigid qualities.

Shape:

The shape of the bed is selected such that the reduction in the rigidity of machine tool should be minimum and it should provide the maximum amount of the space.

Guideways:

- The guideways are guiding the moving parts on the surface of a machine tool.
- The material used for the manufacturing of guideways should possess High wear resistance, high damping qualities, and low ductility.

Prime Mover:

- The very first Prime mover used in machining is IC engine but because of noise pollution and result of the reciprocating parts after the invention of electrical Motors, the electrical Motors are used as a prime mover in the machine tools.
- Whenever the power supply is given to the induction motor, it always starts rotating at only one particular rpm i.e. designed rpm.

Transmission System:

- The most commonly used mechanical transmission system is the belt drive with stepped pulley because during machining operation any sudden increase In forces will cost to split the belt but no failure is taking place.

Head Stock:

Head stock is the widely used component on lathe machine. It is generally used to load the work piece into the chuck and perform various operations.

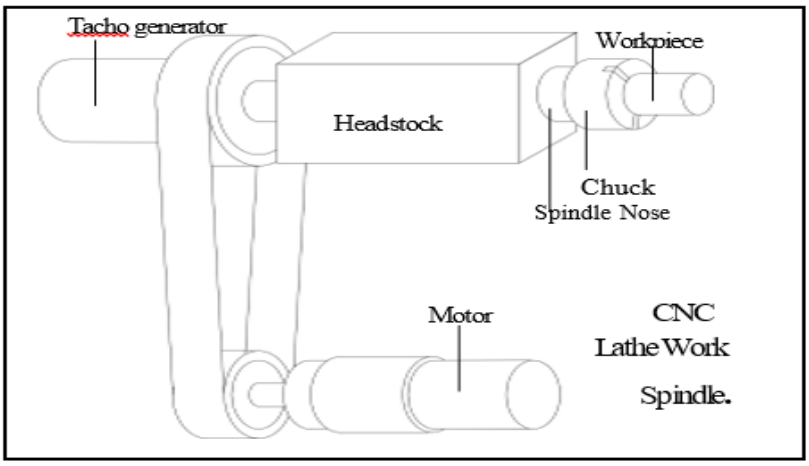


Figure 1:8 Headstock.

Spindle:

- It is the hollow shaft used for transmitting power from the transmission system to the work piece.
- It also supports the longer length of the work piece during machining.

Chuck:

It is a work holding device used for holding the work piece at one end.

Chuck Is classified into two types:

a. **Three jaw chuck: -** It is used to hold circular or cylindrical shape jobs.

a. **Four jaw chuck: -** It is used to hold non-circular jobs.

Feed Gear Box: - At various speeds, it is used for transmitting power from spindle to the lead screw.

Thread Chasing Dial (TCD):

To avoid multiples, start of the thread with multiple cuts, Thread chasing Dial is used in thread cutting.

Split nut: - For converting rotational movement into linear movement, split nut with the lead screw will be used.

Carriage: - The carriage is the device used for carrying a moving part in a lathe machine globally.

Slide ways with Saddle: - It is used for moving the tool locally. (google .com)

1.3.2 Accessories of CNC lathe machine

1. **Lathe Accessories**

A number of lathe accessories extend the usefulness of a standard lathe. These include:(lathe attachment pdf)

1. Centers
2. Chuck
3. Catch plate or carriage
4. Face plate
5. Angle plate
6. Mandrel

2. **Lathe Attachments**

Following are the 5 different types of lathe attachments:

1. Taper Turning attachments
2. Grinding attachments
3. Gear cutting attachments
4. Milling attachment
5. Thread chasing dials

3. Centers

The most common methods of holding the workpiece in a lathe are between the two centers i.e. live center and dead center. The different types of centers are used for different types of workpieces and also for specific purposes. The important types of centers used as shown in the figure below.

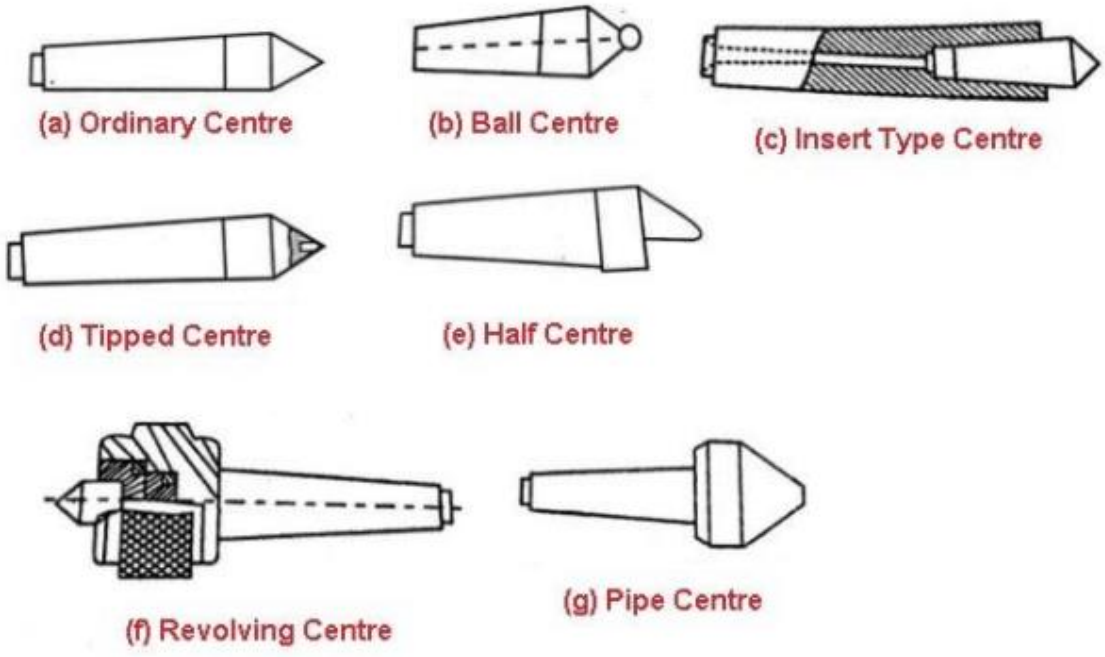


Figure. 1. 9 lathe centers

a. Ordinary Centre

This type of center is used for general purposes and for small operations.

b. Tipped Centre

This is very important of all other centers and here the tip is made of hard material.

These hard materials will resist deflection, wear, and tear.

c. Ball Centre

This type of centre is used to minimize wear and strain while taper turning by set over method.

d. Insert Type Centre

This type of centre is used for the economic purpose and here we can replace only the high-speed steel "insert", instead of replacing the whole center.

e. Half Centre

This is very similar to ordinary centre expect that little, less than half of the center has been grounded away.

f. Pipe Centre

The pipe centers are used for supporting the open end of pipes. They are commonly used in production works.

g. Frictionless Centre

This is used in the tailstock, it has the capacity of bearing high-applied force and this can rotate the center along with the job or workpiece. This can operate at very high speed.

The centers mentioned above will withstand the higher temperature during the

working and the centers h=generally have an included angle ranging from 60° to 75°.

4. **Chuck**

Chucks are generally used to support and hold the workpiece in order to carry out different operations in lathe. The different types of chucks are as follows

a. 3 jaw universal chuck
b. 4 jaw independent chuck
c. Combination chuck
d. Magnetic chuck
e. Collet chuck

Now let us discuss some of the important chucks briefly

A. **3 Jaw Universal *Chuck***

The 3 jaws, which are generally made of high quality steel, are arrogated at an

angle of 120° to each other. During the operation, the jaw teeth are made to mesh with scrawl spiral teeth (Bevel's teeth).

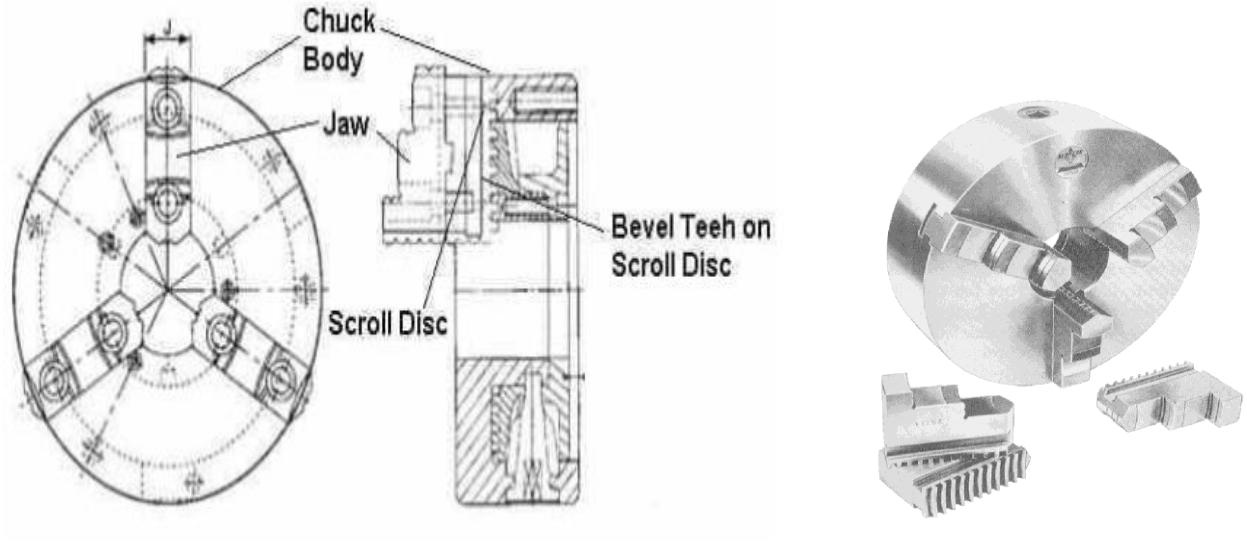


Figure 1.10. 3 jaw chuck

The meshing causes a moment of all 3 jaws either towards or away from the chuck center, depending upon the direction of rotation of bevel pinion. The pinion is performed by square end key "chuck key". 3 jaw universal chucks are used for workpieces of circular and hexagonal shapes.

B. 4 Jaw Independent Chuck

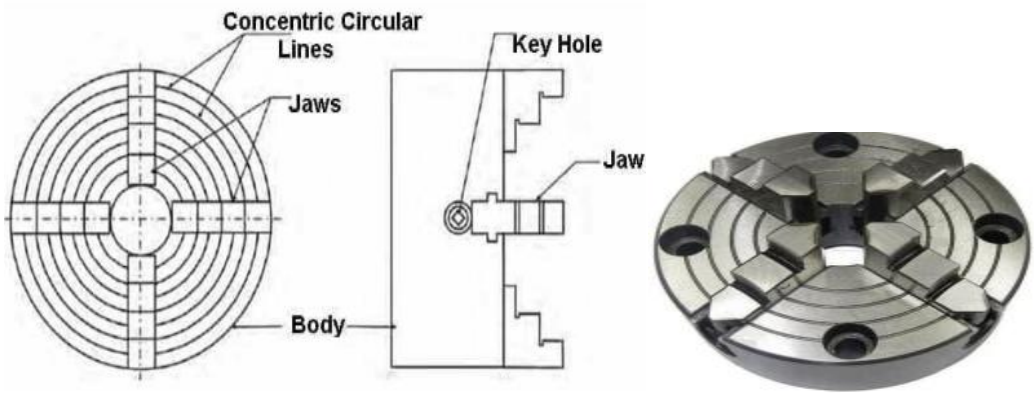


Figure 1.11 four Jaw lathe chuck

The arrangement of 4 jaw independent chuck, which has 4 jaws, located at an angle of 90° to each other. Here all the four jaws are operated independently with 4 screws in

place of scroll disk used in 3 jaw chuck. Hence it is called as 4 jaw independent chuck. This type of chuck is generally used to hold irregularly shaped workpieces.

5. **Combination Chuck**

The arrangement of combination chuck, which combines the characters of universal chuck and independent chuck. This type of chuck can be used either as self-centring chuck or an independent chuck

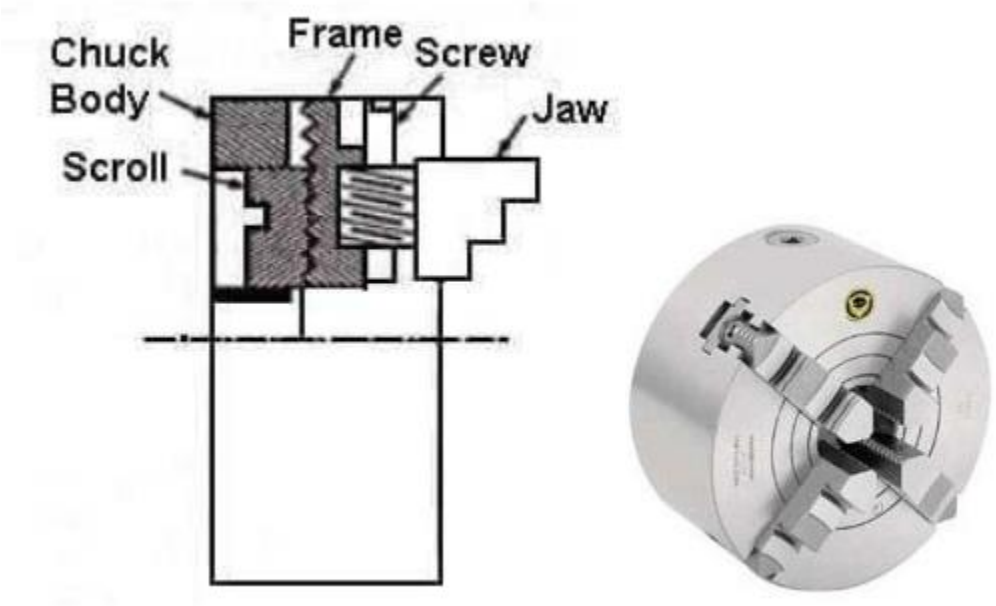


Figure1.12. Combination chuck

It has both the independent screws and the scroll disk. In this chuck, the jaws can be either moved separately by operating the screws or simultaneously by operating scroll disk. It can be used for regular as well as irregularly shaped workpieces.

6. **Magnetic Chuck**

The chuck is used for holding a very thin workpiece made of magnetic material which cannot be held in an ordinarily chuck. It is also used where any distortion of the workpiece due to the pressure of the jaws is undesirable.

The holding power of the chuck is obtained by the magnetic flux radiating either from the electromagnets or from the permanent magnets introduced within the chuck.

In the ON position, the flux passes through the workpiece and grips it. In OFF position

the magnets are set aside bringing them in contact with high permeable " keepers" which short-circuit the flux and prevent them from passing through the workpiece.



Figure. 1.13. Magnetic chuck

7. **Catch Plates or Carriers**

   Carriers and catch plates are used to drive a workpiece when it is held between two centers. Carriers are driving dogs attached to the end of the workpiece by a setscrew. Catch plates are bolted to the nose of the head stock spindle.

   A projecting pin from the catch plate or carrier fits into the slot provided in wither (anyone) of them. The projecting pin of a single pin catch plate drives the straight end or tail of the carrier attached to the workpiece.

   Two pins of a double pin catch plate engage with the double tail or double slotted carrier and provide uniform drive. The bent tail type is used with a faceplate or slotted

   catch plate.

8. **Face Plate**

   It is a circular plate threaded at its center with plain and T-slots which are machined rapidly. It is fitted to the lathe spindle with its central threaded portion. The job on the workpiece is held by the face plate using bolts and clamps in the slots.

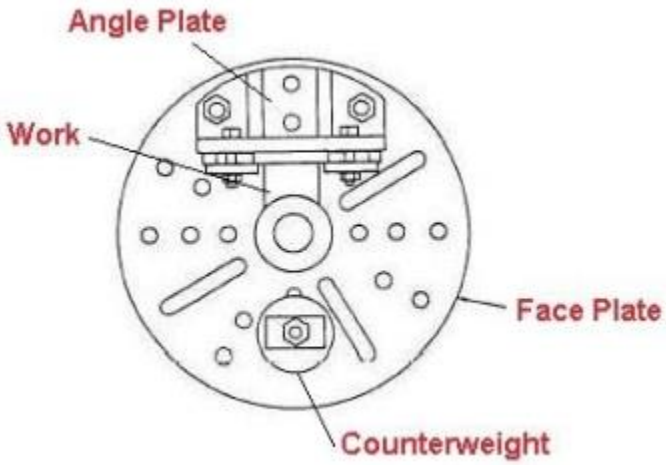


Figure .1. 14 Face plate

The faceplate is suitable to hold both regular and irregular shaped workpieces, which cannot be help conveniently by chucks or on centers.

9. **Angle Plate**

   It is a cast iron plate with two faces that make them at exactly the right angle to each other. Holes and slots are provided on both the faces so that it may be clamped on a faceplate and can hold workpieces on the other face by bolts and clamps.

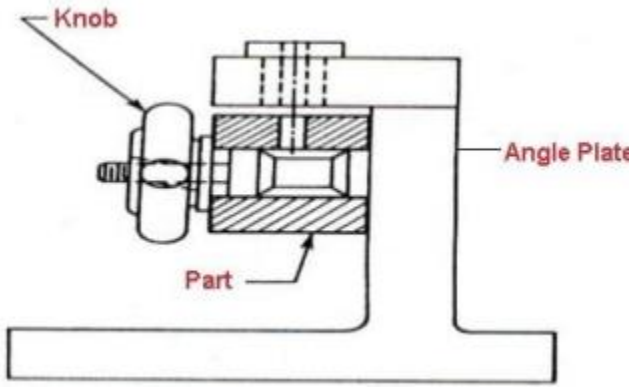


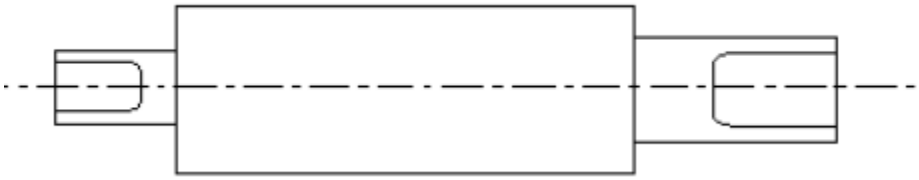
Figure. 1.15 angle plate

Angle plates are used with faceplate when the holding surface of the workpiece should be kept horizontal. For example, in machining, a flange of a pipe elbow this plate is used. When eccentric jobs are bolted to the faceplate, a balance weight or counterweight must be added.

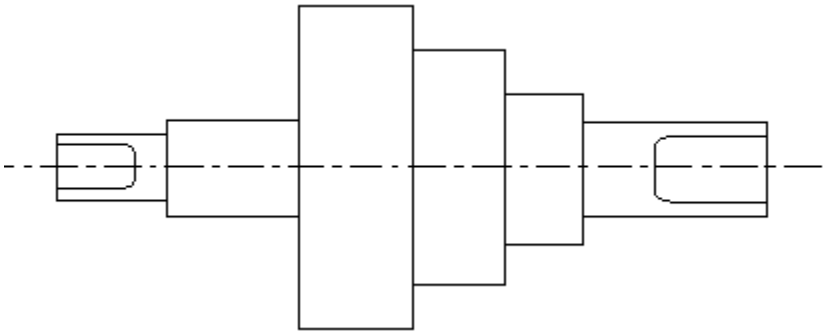
10. Mandrel

It is used for holding and rotating a hollow job. The hole in the job must be clean and accurate to ensure it running true. The mandrel is placed on centers and being rotated by driving plate and lathe dog and it drives the job by friction. Material of mandrel is Generally High carbon steel, may also be mild steel case hardened.

Plain mandrel: - Whenever the parts of the workpiece are plain in nature, this type of mandrel is used.



Step mandrel: - Whenever the inner portion of the workpiece is in the form of steps, in that case, to hold the workpiece rigidity, this type of mandrel is used.



Cone Mandrel: - Whenever the part of the workpiece is conical in shape, in that case this type of mandrel is used.

11. Steady and Follow Rests:

Rests are used to prevent bending of a slender job (having its length more than 12 times the diameter of the job approximately) due to its own weight and the vibrations during operations**.**

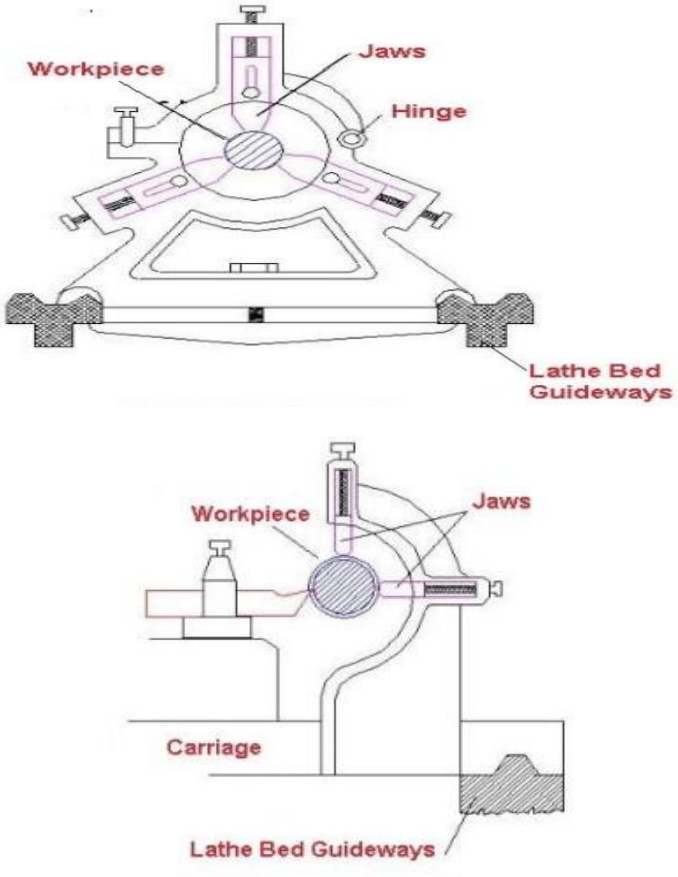


Figure.1.16. steady and follower rest

1.3.3 Measuring Instruments

We know that the primary objective of measurement in industrial inspection is to determine the quality of the component manufactured. Different quality requirements, such as permissible tolerance limits, form, surface finish, size, and flatness, have to be considered to check the conformity of the component to the quality specifications. In order to realize this, quantitative information of a physical object or process has to be acquired by comparison with a reference. The three basic elements of measurements (schematically shown in Fig. 1.15), which are of significance, are the following.

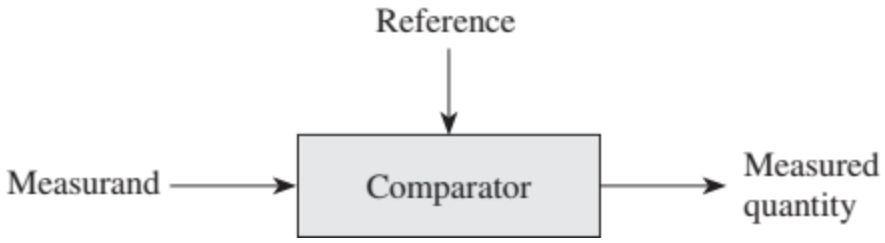


Figure.1.17 the three basic elements of measurement

1. *Measurand*, a physical quantity such as length, weight, and angle to be measured
2. *Comparator*, to compare the measurand (physical quantity) with a known standard (reference) for evaluation
3. *Reference*, the physical quantity or property to which quantitative comparisons are to be made, which is internationally accepted

All these three elements would be considered to explain the direct measurement using a calibrated fixed reference. In order to determine the length (a physical quantity called measurand) of the component, measurement is carried out by comparing it with a steel scale (a known standard).

Telescopic Gauge
Use :- Indirect measurement of Bore, Slot, Recesses, etc. Used in accordance with micrometer /Vernier.

Parts :-

1. Handle
2. Pair of Plunger
3. Lock

Construction:-

• It consists of handle, two telescopic rods & locking screw

• For taking measurement, telescopic rods are compressed against spring & inserted into the hole whose diameter to be measured.

• Then extended up to the walls of hole.

• Then they locked by locking screw & rods can be measured by micrometer or vernier calliper.

Telescopic Gauge

• Indirect measuring ***device***

• Used for measuring ***internal diameter of holes, slots and grooves etc***

• ***Consists of*** handle, two telescopic rods and locking screw

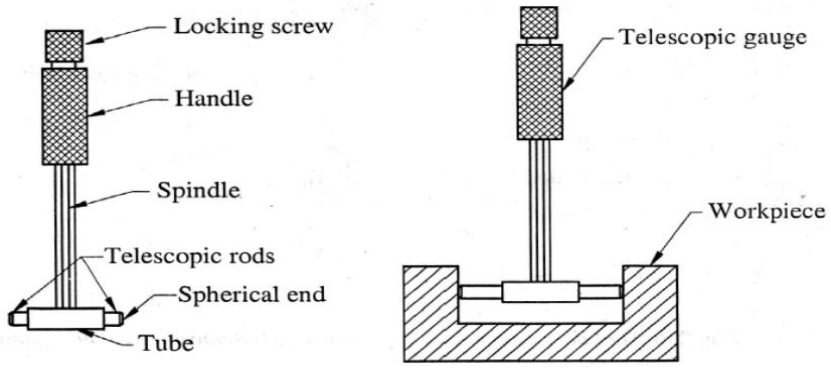


Figure 1.18 telescopic gauge

Precision Instruments

List of Precision Instruments

• Vernier Calliper

• Micrometer

• Vernier Height Gauge

• Dial Indicator

• Comparators

• Bore Dial Gauge

Vernier Calliper

Vernier Principle: When two scales (Main and Vernier scales) or divisions slightly different in size are used, the difference between them can be utilized to enhance the accuracy of measurement.

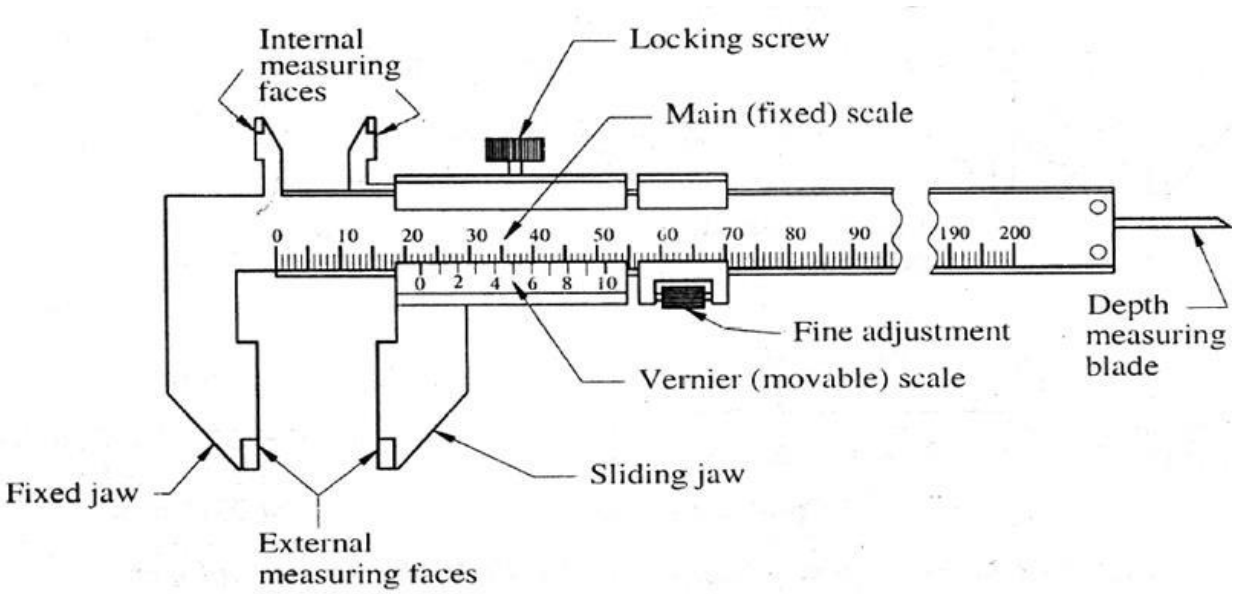


Figure.1.19 Vernier Calliper

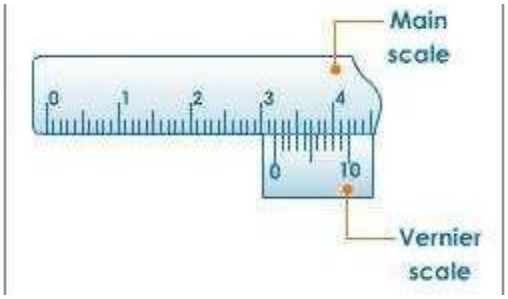
The Vernier Caliper is an instrument for measuring outside diameters, bores, lengths and depths on components.

Least Count of Vernier Calliper

• It is the capability of an instrument to measure minimum distance accurately.

• The least count is the difference between the value of main scale division and vernier scale division.

Least Count (L.C.) of Vernier Calliper

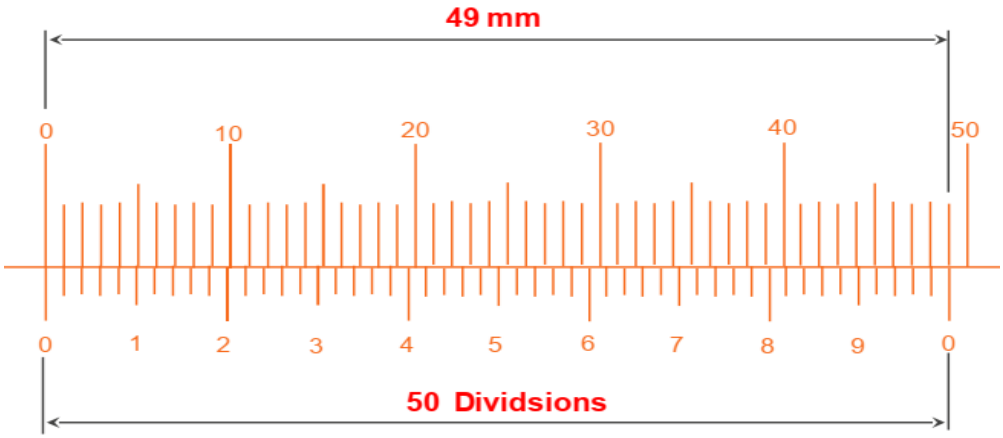


L.C. = Value of smallest division on Main Scale / no. of division on Vernier Scale

= 1 mm/10

= 0.1 mm = 0.01 cm

Vernier Calliper Least Count



Least Count (L.C.) = 1 M.S.R. – 1 V. S.R. = *1* - 49 = 1- 0.98 = 0.02 mm

Micrometers

• Useful device for magnifying small measurement.

• Micrometers works on the principle of screw and nut. The screw is attached to thimble.

• A screw is turned through nut by one revolution, its axial movement is equal to pitch of the thread of screw.

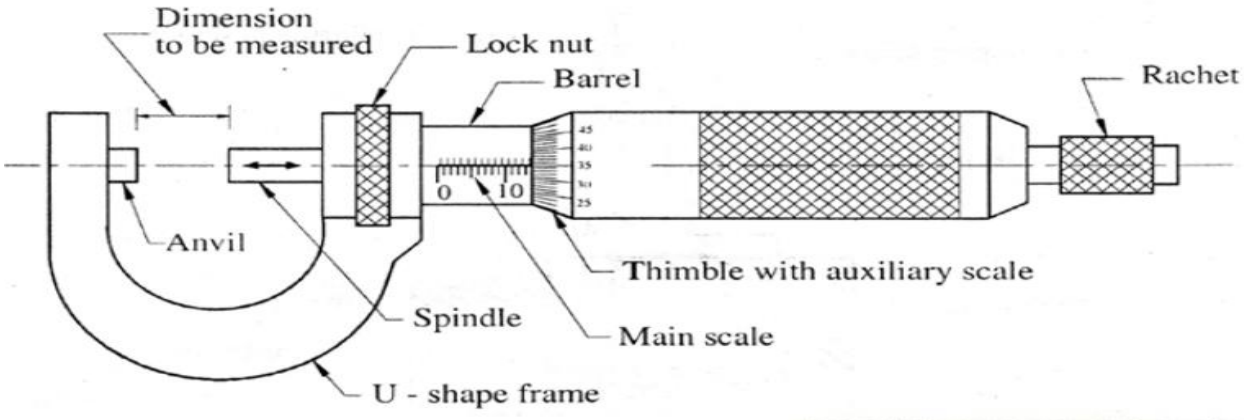


Figure. 1.20. micrometer

External Micrometer: - used for measuring external diameter or thickness.

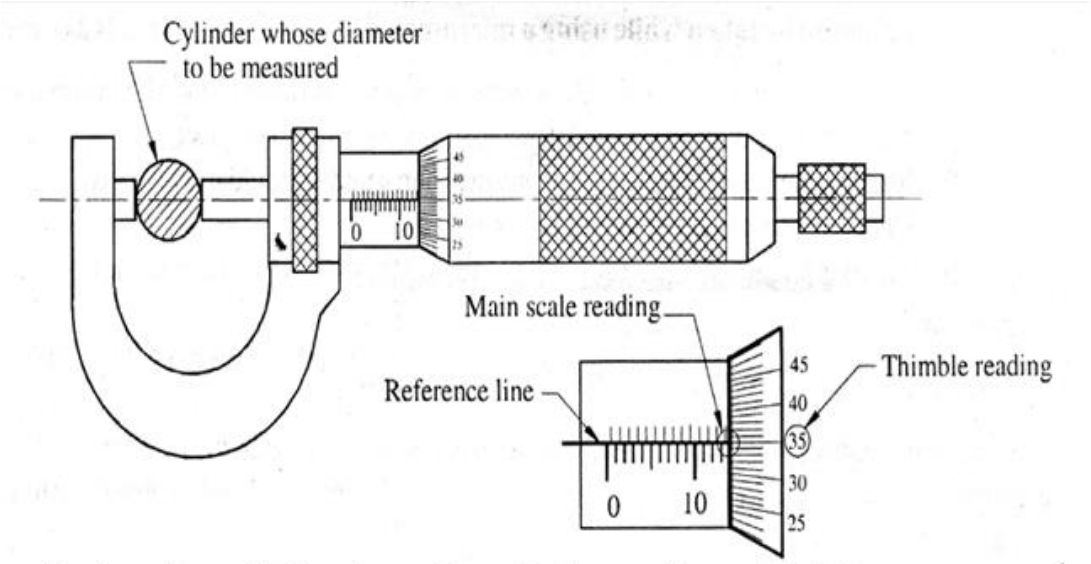
Least Count of a MICROMETER

Total divisions in the Circular Scale = 50 divisions

Linear movement in one rotation of Circular Scale (Pitch) = 0.5 mm

Least Count = 0.5/50 = 0.01 mm

Reading on a micrometer



Total reading = Main scale reading + (L.C. × reading on thimble)

$= 13.5 + 0.01 \times 35$

$$\left(\because \text{L.C.} = \frac{\text{Value of smallest division on main scale or pitch}}{\text{No. of divisions on thimble}} = \frac{0.5}{50} = 0.01 \right)$$

= 13.85 mm

Inside Micrometer

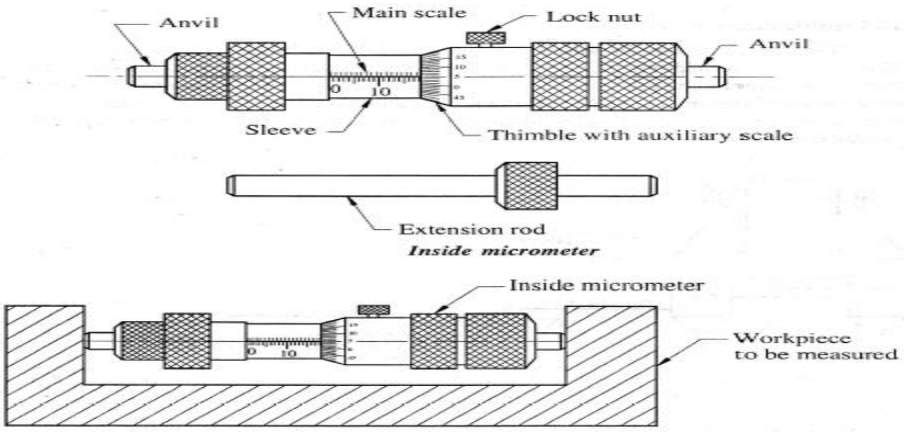


Figure.1.21 Inside Micrometer

Depth Micrometers: **-** Used for measuring depths of holes and shoulders accurately

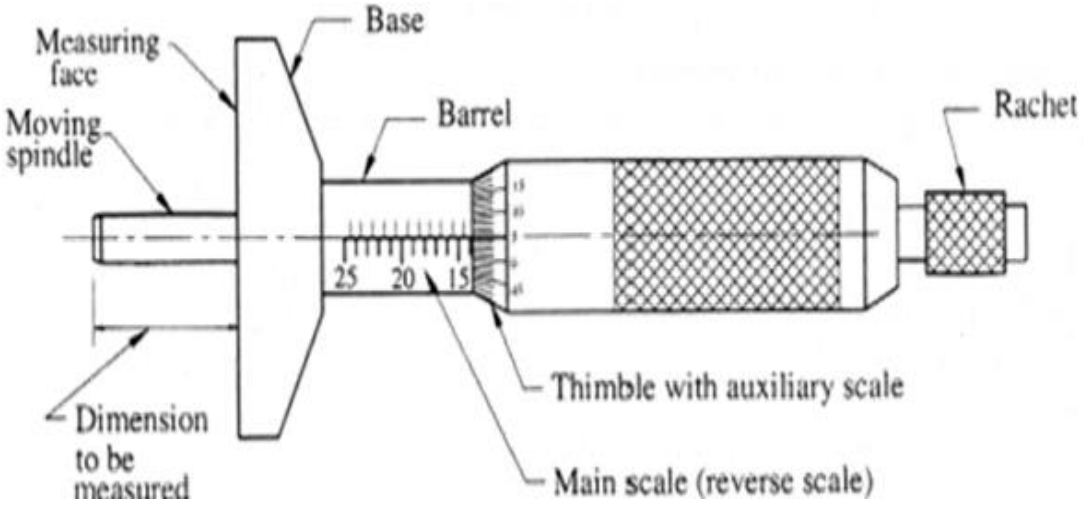


Figure. 1.22 Depth Micrometer

Linear Measurements – Gear Tooth Vernier

Similar to two Verniers used at 90°. Used for measuring chordal tooth thickness of gear teeth

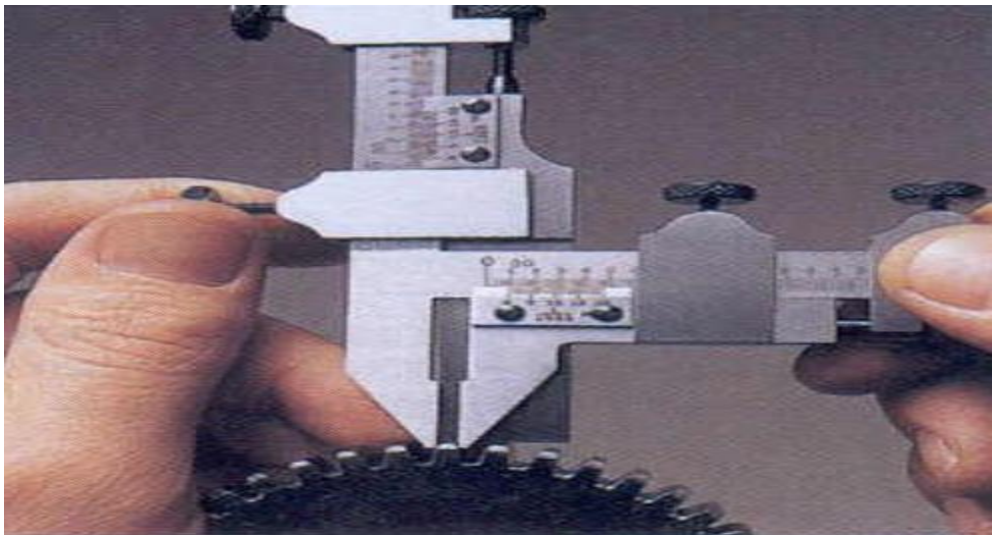


Figure.1.23 gear tooth Vernier

Vernier Height Gauge

• Used in connection with a surface table.

• Fitted with a carbide tips.

• Used for marking the lines on the workpiece.

• Also used for taking height

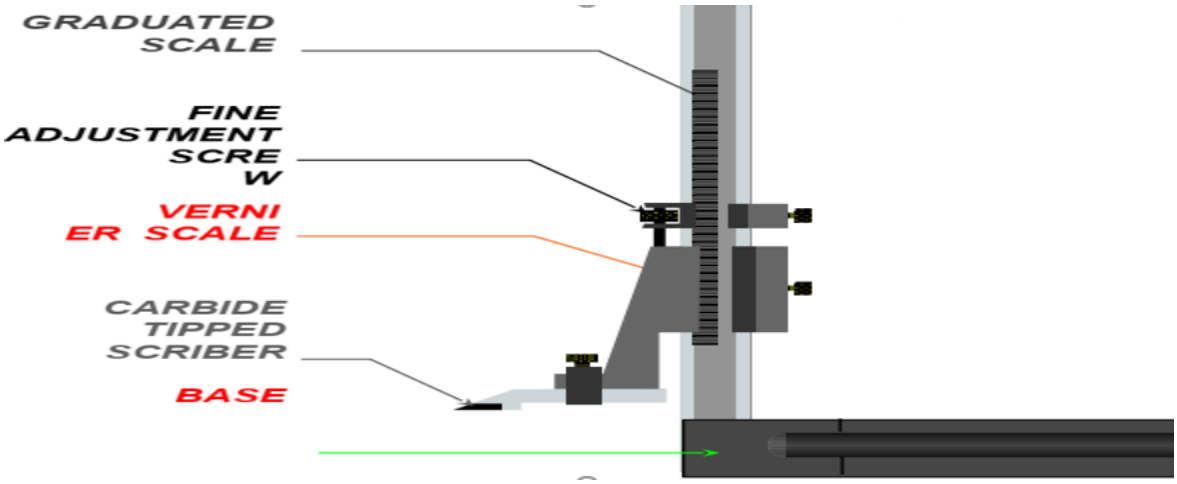


Figure 1.24. Vernier height gauge

Dial Indicator

Checking the alignment of lath centers



Figure. 1.25 Dial Indicator with Stand

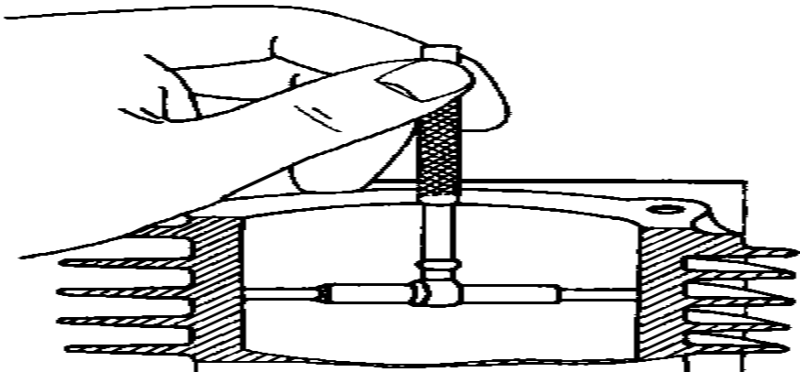
Dial Bore Gauge

A Dial Bore Gauge is used to measure cylinder taper and out-of-round as well as main bearing (block housing) bore for taper and out-of-round.



Figure 1.26. Dial Bore Gauge

Telescopic Gauges



SLIP GAUGE/GAUGE BLOCKS – Slip Gauges are known as Gauge Blocks. They are Precise Measuring Instruments. These Slip Gauges are universally accepted end Standard of length.



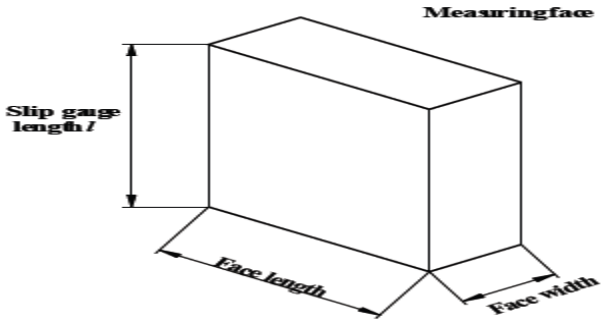


Figure 1.27-gauge block

Slip Gauges are blocks of steel that have been hardened and stabilized by heat treatment.

• They are ground and lapped to size to very high standards of accuracy and surface finish.

• A Gauge Block (also known Johansson Gauge, Slip Gauge, or Jo Block) is a precision length measuring standard.

Classification

(International Standard)

AA SLIP GAUGES– Master slip gauges

– Accurate to plus or minus two microns per meter

• **A SLIP GAUGES**– Reference purpose

– Type A is guaranteed accurate up to plus or minus four microns per meter

• **B SLIP GAUGES**– Working slip gauges

– Type 'B' for plus or minus eight microns per meter

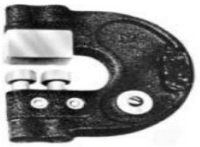
Gauges –Master Setting Rings- Master Setting Rings are made to nominal size and are used to set the measuring instrument or comparator. Master Setting Plugs are also similar in application.

Limit gauges

• the "go" side of the limit gauges should enter the hole (or cavity) or just pass over the shaft under the weight of the gauge without using any force.

• the "not go" side of the gauge must not enter or pass.


Cylindrical Pin Gauge


Adjustable Snap Gauge


Plain Ring Gauge


Thread Plug Gauge


Cylindrical Plug Gauge


Progressive Cylindrical Plug Gauge


Adjustable Thread Ring Gauge

Figure. 1.28 limit gauges

Gauges – Screw Pitch Gauges

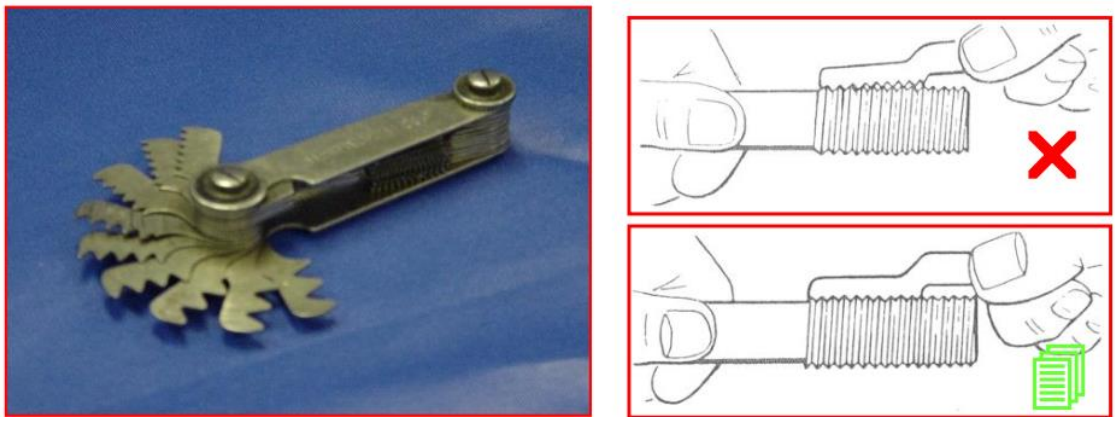


Figure. 1.29 Screw Pitch Gauges

A screw pitch gauge has a set of teeth which are of a known thread form. The gauge is used to determine the pitch of a thread on a bolt or screw of any diameter.

Note: Both Pitch gauges and thread, should be clean and free of burrs. It is useful to hold the screw up to a light to see small gaps. Use the full length of the gauge for greater accuracy.

Gauges – Radius Gauges

A radius gauge has a profile of a known radius. The gauge is used to determine the radius of any feature.



Figure. 1.30 radius gauge

1.4 Cutting tools and parameters.

1.4.1 Cutting tools

Cutting tools are made from hardened steel or carbide. Carbide will hold its edge longer than hardened steel, but is brittle and breaks easier. The tools are often ground to various shapes by the lathe operator to suit a particular cutting requirement. The clearances ground behind cutting edges are adjusted for the type of material the tool will cut and the direction the tool will be fed along the work piece.

Not enough clearance behind the cutting edge will cause the tool to rub. Too much clearance will produce a ridged or wavy finish because of the small length of tool edge in contact with the work piece.

Standard tool types are: turning tools, side tools, parting tools, and boring tools.

Side Tools

Side tools are used to face-off the ends of shoulders or to make facing cuts in the surface of a work piece held in a chuck. They may also be used as turning tools.

Right side (or right-hand) tools feed from right to left and are used to reduce the work piece to a desired diameter. The shape of the cutting edge and the clearance (behind the point between the end of the tool and the work piece) determine the surface finish of the work piece. Rough cuts are made in small increments until the tool is within approximately 0.010 inch (0.25mm) of the desired diameter. Final cuts are made at slow feed rates with a very shallow depth of cut.

Left side (or left-hand) tools feed from left to right. Side tools cut very flat surfaces and can be used to produce a part with an exact thickness.



Left-hand



Right-hand



Parting Tools

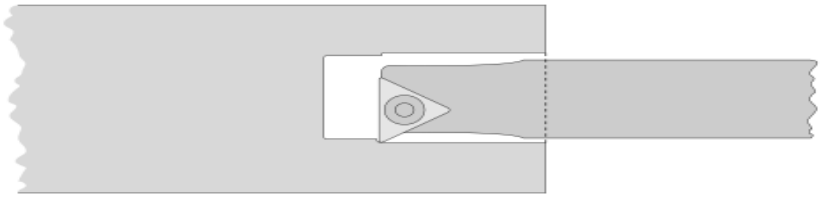
A parting tool has a dual-edged, dovetail-shaped cutting edge which is used to cut off workpieces. The parting tool is plunged into the work piece and the cross slide is moved across the lathe bed until the workpiece is severed.



Boring Tools

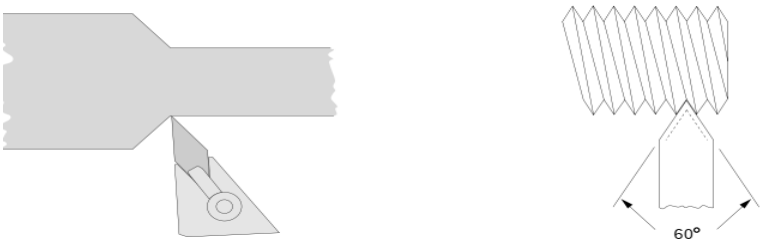
Boring tools are used to enlarge or modify a drilled or cored hole in a workpiece. Clearance
must be maintained behind the cutting point of the tool. A slow feed rate and frequent tool
withdrawals are required because the turnings cannot freely escape from the hole.
A boring tool should not be driven deeply into the hole; damage to the chuck may occur.
Depth of cut is increased by moving the tool and cross slide in the X direction to enlarge

the diameter of the hole. Boring tools are often mounted to the tool post with greater than normal overhang. Depth of cut and feed rates must be reduced to avoid chatter.



Profiling Tools

Profiling tools are quite popular in CNC applications because they can cut on both sides and in both directions.



Threading Tools

Most threads are cut with a 60o threading tool. For fine threads, the tool can be ground to a sharp point. For coarse threads, you may wish to radius the point. A threading tool is used for external threading operations. An inside threading tool is used for internal threading, like on a nut

Cutting Tool holders

The cutting tool of a lathe is clamped into a *tool holder*, which stays with the cutting tool when it is removed from the machine. Ideally, there is a separate tool holder for each cutting tool. Tool holders include *standard tool holders, carbide insert holders* and *boring bars.*

Tooling for modern lathes consists almost exclusively of carbide inserts and their holders Carbide inserts offer significant advantages in terms of accuracy, cutting efficiency, ease of use and cost.

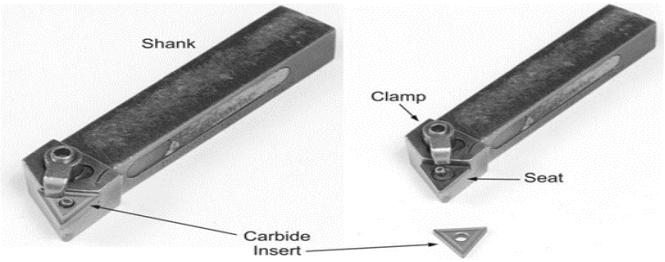


Figure. 1.31. Carbide insert holder

1.4.1 Calculating CNC lathe Machining Parameters

i. Cutting speed

- Rate at which point on work circumference travels past cutting tool
- Always expressed in feet per minute (ft./min) or meters per minute (m/min)
- Important to use correct speed for material
- Too high: cutting-tool breaks down rapidly
- Too low: time lost, low production rat
- Cutting Speed (Surface Speed):

$$Cs = \frac{\pi DN}{1000}$$

Cs= cutting speed

Where D= is diameter of a job in mm

N= is spindle or job speed in rpm

ii. Spindle Speed

- Given in revolutions per minute
- Cutting speed of metal and diameter of work must be known
- Proper spindle speed set by dividing cutting speed (m/min) by circumference work in mm

Spindle Speed (RPM) = (Cutting Speed x 1000) / (π x Diameter of workpiece)

Factors affecting cutting speed

- Material to be cut
- Material of the cutting tool
- Grinding set up of the cutting tool

- Type of cut to be taken
- Rigidity of the set up
- Condition and capacity of the machine
- Type of coolant to be used

Table 1.1 Cutting Speed (Cs) In Meters Per Minute

Materials	**Roughing cut**	**Finishing**
Cast iron	18	24
Mild steel	27	36
Tool steel	15	22
Brass	45	60
Bronze	27	30

Example: Calculate the CS required to turn a 50 mm diameter work piece of machine steel (rev/min= 255).

(V) = $\frac{\prod DN}{1000}$ m/min. = $\frac{3.14 \times 50 \times 255}{1000}$ = 40m/min

iii. **Lathe Feed**

- It is the amount of tool advancement per revolution of job parallel to the surface being machined.
- It is given in mm per revolution of the job.

The rate at which the tool is fed depends upon various factors such as

- Finish required,
- Depth of cut and

Feed per Revolution:

Feed per Revolution (IPR or mm/rev) = Feed Rate (IPR or mm/rev) / Number of Cutting Edges

iv. **The rigidity of the machine**.

A high rate of feed will get the job done in less time but will give a rough finish and a slower rate will give a better finish but will take a longer time. Normally feed varies from 0.1 to 1.5mm. .003- to .005-in (0.07- to 0.012-mm)

Table 2.2 High-speed steel cutting tool feed rate per revolution of spindle

Material	Rough Cuts		Finish Cuts	
	in.	mm	in.	mm
Mach steel	.010–.020	0.25–0.5	.003–.010	0.07–0.25
Tool steel	.010–.020	0.25–0.5	003–.010	0.07–0.25
Cast iron	.015–.025	0.4–0.65	.005–.012	0.13–0.3
Bronze	. 015–.025	0.4–0.65	.010 0.07	0.07– 0.25
Aluminum	.015–.030	0.4–0.75	.005– .010	0.13–0.25

v. Depth of Cut

It is the advancement of the tool in the job in a direction perpendicular to the surface being machined. Or the penetration of tool in to the work.

It is equal to half of the the difference between the diameters befor and after machining

Depth of chip taken by cutting tool and one-half total amount removed from workpiece in one cut

Only one roughing and one finishing cut Roughing cut should be deep as possible to reduce diameter to within .030 to .040 inch or .75 to 1 mm of size required.

Factors Determining Depth of Rough-Turning Cut

- ✓ Available Stock
- ✓ Work piece Material
- ✓ Condition of machine
- ✓ Type and shape of cutting tool used
- ✓ Rigidity of work piece, machine, and cutting tool
- ✓ Rate of feed

Depth of cut , (t) = $\frac{D-d}{2}$

Where, D = is diameter of the workpiece

d = is diameter of the work required

EXAMPLE: - Determine the DoC for D=50 and d=30

Depth of cut , (t) = $\frac{D-d}{2}$ in mm = $\frac{50-30}{2}$ = 10mm

vi. Calculating Machining Time

Factors such as spindle speed, feed and depth of cut must be considered

Machining time = $\frac{\text{Noof passes Xlength of work piece}}{\text{f x N}}$

Example: - A brass pin is of 500 mm length and 40mm diameter. Find the turning time to reduce the pin to 38.8mm in one pass, when cutting speed is 60 m/min and feed is 0.8 ^{mm}/min.

Solution: N = $\frac{\pi cs\ D}{1000}$ = $\frac{3.14\ x\ 40\ x\ 60}{1000} = 478\ RPM$

Machining time = $\frac{Noof\ passes\ Xlength\ of\ work\ piece}{f\ x\ N}$ = $\frac{1\ x\ 500}{0.8\ x\ 478} = 1.31min$ =

1.5 CNC Lathe Process/job/adjustment sheets.

A process sheet consists of manufacturing instructions for a specific batch, lot, or run. It describes the operating parameters and settings for the equipment and facilities used and associated tooling or supplies.

It contains part information, routing information, and operation detail information.

A process sheet is a set of instructions that can be followed to achieve the desired goal. It is a manual that involves detailed procedures of tasks.

Importance of process sheet

It outlines the detailed mechanisms and describes every event in a step-by-step manner. That is where the importance of the manufacturing process sheet lies.

Firms with infrequent productions, a complex production pattern, or elaborate and irregular steps must follow a process chart efficiently.

It is developed way before a firm starts operating. That helps in having control over operations from day one.

The components outlined in a process sheet

A process sheet for machining comprises two units,

A. A route sheet

A route sheet is like a map that mentions the path and how the production and management activities proceeded.

B. An operation sheet

An operation sheet mentions the step-by-step programs to follow, outlines all the materials required for each task, and maintains a log book of items that have arrived or are yet to come.

It is an irreplaceable tool. It keeps a tab on the entire work process it has been designed.

It keeps a count on the amount of raw material required, total processing time taken, the number of workers, specifications of the machines functioning, and so on for every separate component involved in the production.

1.5.1 Structure of process sheet for manufacturing

- Outlining the sequence of work
- Figuring out the assemblies and sub-assemblies involved and drawing a map to figure out where each one fits
- Determining the number of units to be produced
- Machines, tools, and details of other instruments are required, along with further information like their efficiency, capacity, and run times.
- The sequence of these operations is drawn in detail on a map that develops a clear conception of how the raw material is converted into the finished product.
- A certain quality is required throughout the process; hence quality controlling instructions are also important.
- Instructions during packing and handling the products or movement during the processing are also mentioned.

Process sheets are an integral part of the manufacturing process. They can be used as a planning tool to plan wisely and efficiently, help with quality control, or be used simultaneously for various purposes by different departments.

1.6 work shop safety

Safety is a very important issue in a machining environment. You hear stories all the time about
the dangers of operating machinery. People getting their hair caught in rotating machines, fingers
getting cut off by students and teachers not paying attention to their work; some are life threatening injuries, others, at the least are permanently damaging. We require that you read and
understand all of the following safety rules, that you pass, *with a score of 100%*, a written test on
these rules, and sign the test stating you will follow these guidelines every time you operate the
machine. Safety is not to be taken lightly.
We have included a Safety Checklist that should be copied and posted next to the Turning Center. It must be completed by the machine operators every time the machine is run. In the each activity you will be instructed to complete this checklist.(CNC Technology: 9802 proLIGHT Lathe)

The following is the list of the Safety Rules for operating the Turning Center. Read through them until you are familiar with them. When everyone has finished, your instructor will give you the test. You must pass with a perfect score.

1.6.1 Safety Checklist

I. Before you enter the work area:

- Put on safety glasses.
- Tie back loose hair and clothing.
- Remove jewelry including rings, bracelets and wristwatches.

II. Before turning a part:

- Make sure you have the correct tool for the job.
- Secure the tool properly.
- Make sure all tool positions have been properly initialized.
- Verify the NC program on the computer before machining.
- Remove all loose parts and pieces from the machine.
- Remove adjusting keys and wrenches from the machine.
- Close the safety shield.
- Only operate the machine after being properly trained in its use.

III. **Perform a dry run:**

- Set the spindle speed to lowest manual speed setting.
- Make certain there is no workpiece in place.
- Run the NC program to make sure all the moves make sense before running the program with a workpiece in place.
- After completing the dry run, properly secure the workpiece to the machine.
- Keep fluids away from all electrical connections, electronic or electrical devices, the computer and nearby electrical outlets.

IV. While machining a part:

- Do not touch moving or rotating parts.
- Press the Emergency Stop button before opening the safety shield.
- Only open the safety shield after the chuck has stopped rotating.
- Press the Emergency Stop button whenever changing tools or mounting or removing a workpiece.
- Pull the Emergency Stop button out only after closing the safety shield.
- Keep all unauthorized persons away from the work area.

Self-Check 1

Instructions: answer all questions given below accordingly

- Given 25 minutes,
- Enumerate the following. Use a separate sheet of paper for answering.
- For Short answers, write the answer for the questions on the provided space.

Part I. Choose the Correct answer from the given alternatives

1. Reading and understanding the information given by the drawing about the component to be manufactured is called?

   a) Sequence of operation
   b) Drawing of parts
   c) Drawing interpretation
   d) All

2. A cast iron is of 600 mm length and 45mm diameter. What is the turning time to reduce the work to 35 mm in one pass, when cutting speed is 65 m/min and feed is 0.75 ^{mm}/min

   a) 17 min
   b) 15min
   c) 12.3 min
   d) 11.3min

3. Which one of the following is the advantages of drawing interpretation?

   a) Easily inspect and select the rough size of the work stock blank.
   b) Recognized the surface finish expected.
   c) For plan the machining sequence operation technically.

4. A cutting tool material must possess the following characteristics except

   a) Hardens
   b) Wear resistance
   c) Wear resistance
   d) Toughnes

Part II. Say True if the statement is true or else say False

1. In the case of interference Fit the external member is larger than the internal member.

2. Cutting capacity of the cutter and surface quality of the work-piece depend mainly on the rigidity of Lathe machine.
3. Multi view representation of a single part with dimensions and notes is called assembly drawing
4. Mandrel is used for holding and rotating a hollow job.
5. The methods used for mounting and holding the work in the lathe depend upon the shape and length of the work.

Part III: - give clear answer to the given questions bellow

1. Write the criteria's used to select cutting tools
2. List down the types of CNC lathe accessories
3. Write down the very applicable measuring tools in CNC lathe operations
4. List out the safety precautions practicable in CNC lathe machine operation.

Operation sheet-1

Operation title: -Sequence of operation to produce component to the given specifications

Instruction: - Put these steps in the correct sequence

Purpose: - To interpreter drawing and understanding sequence of operation.

Required tools and equipment's: a clear understanding about sequence of operations

Precaution: - Follow each and every necessary steps as much as possible, if not you are doing wrong Machin setup for wrong out put

a) Determine stock and tool.

b) Enter NC code on coding sheet.

c) Determine feed and depth of cut.

d) Sketch the part.

e) Determine cutting sequence.

f) Draw the part to scale.

Quality criteria: the trainee should follow arrange according to logical CNC lathe operation sequence

LAP Test 1.

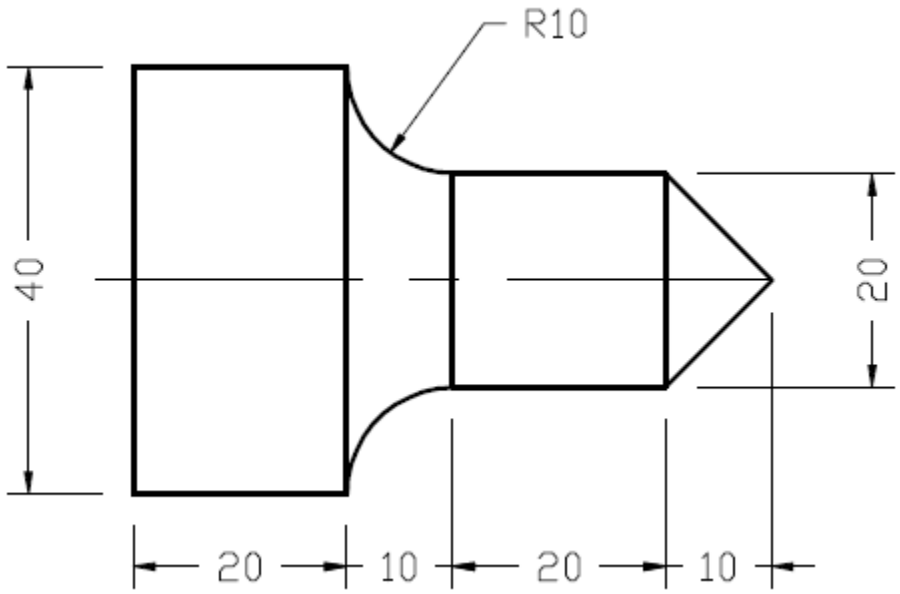
Practical Demonstration

Name: _____ Date: _____TVT Technical Layouts

Time started: _____ Time finished: _____

Instruction I: Given necessary templates, tools and materials you are required to perform the following tasks within 10 hours

Task 1: - Reading the given drawing and writing the sequences of operation to produce the part.



Task 2: - calculate the cutting parameters of the above workpiece based on the given dimension

Unit TWO: program CNC lathe machining process

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- CNC lathe coordinate systems.
- CNC lathe operation Programing
- CNC lathe Program Simulate and edit
- CNC lathe program save and document
- Program Upload to the CNC lathe machine.

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Calculate coordinate system
- Program the operation
- Simulate and edit the program
- Save and document the program
- Upload the program to the machine

2.1 CNC lathe coordinate systems.

2.1.1 Co-ordinate Dimensioning - Absolute

Data in an absolute dimension system always refers to a fixed reference datum point, as shown in the example below. The dimension lines run parallel to the co-ordinate axes and always start at the reference point. Absolute dimensions are also referred to as reference dimensions. When plotting absolute co-ordinates, this point has the function of the co-ordinate zero point.

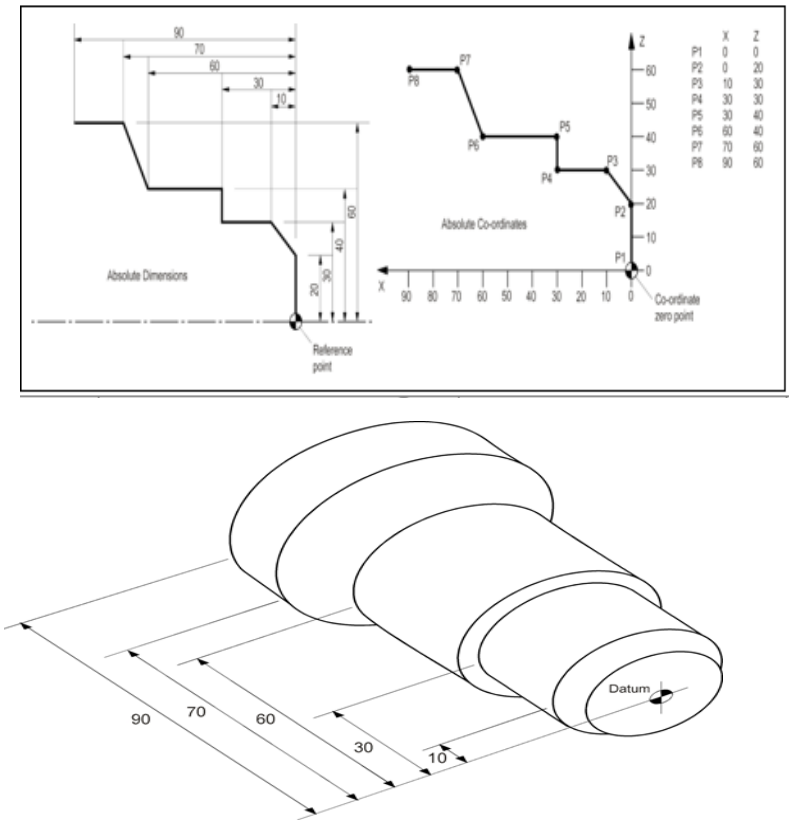


Figure 2.1Absolut dimensioning example

2.1.2 Co-ordinate Dimensioning – Incremental

When using an incremental dimension system, every measurement refers to a previously dimensioned position. An incremental dimension is the distance between adjacent points. These distances are converted into incremental co-ordinates by accepting the last dimension point as the co-ordinate origin for the new point. Incremental dimensions are also referred to as relative dimensions or chain dimensions.

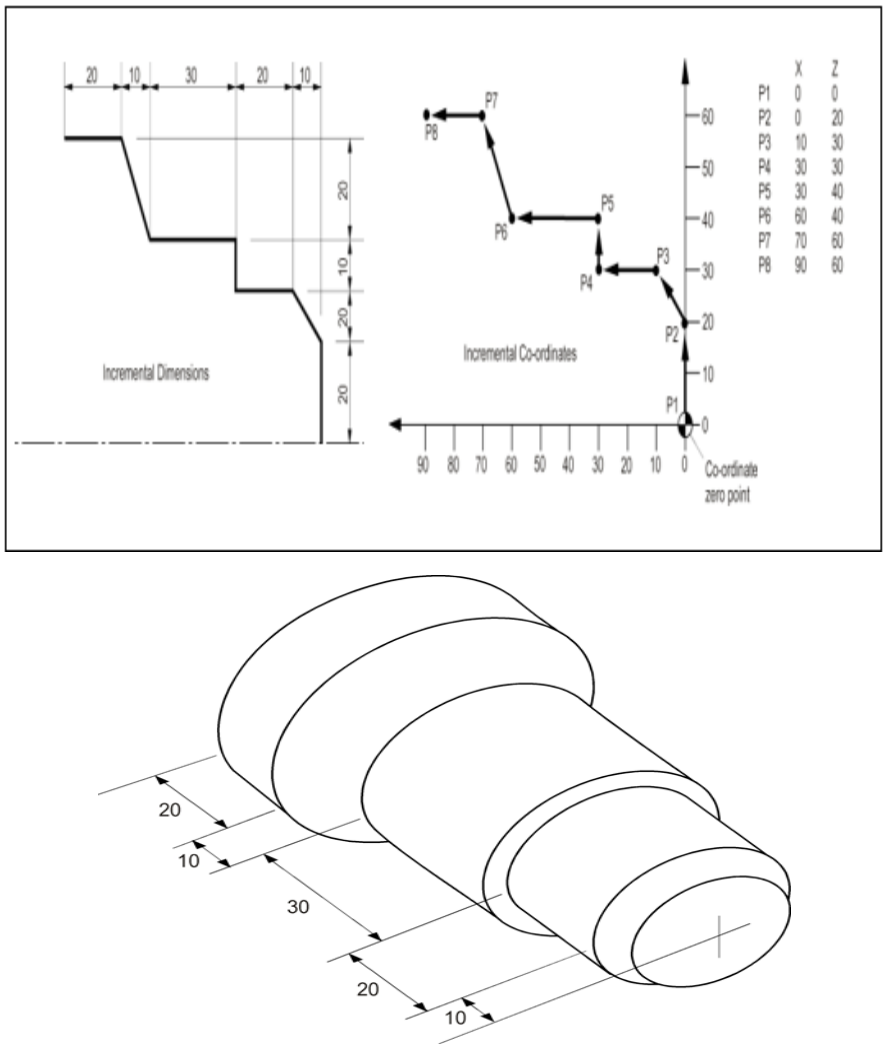


Figure 2.2 Incremental dimensioning example

2.1.4 Interpolation

Interpolation refers to an operation in which the machine tool moves along the workpiece parts. There are five methods of interpolation: linear, circular, helical, parabolic, and cubic. Most CNC machine can provide linear interpolation and circular interpolation. The other three methods of interpolation (helical, parabolic, and cubic interpolation) are usually used to manufacture the complex shapes, such as aerospace parts. In this manual, linear and circular interpolation are introduced.

a) **Linear Interpolation**

There are two kinds of linear interpolation:

1) Tool movement along a straight line

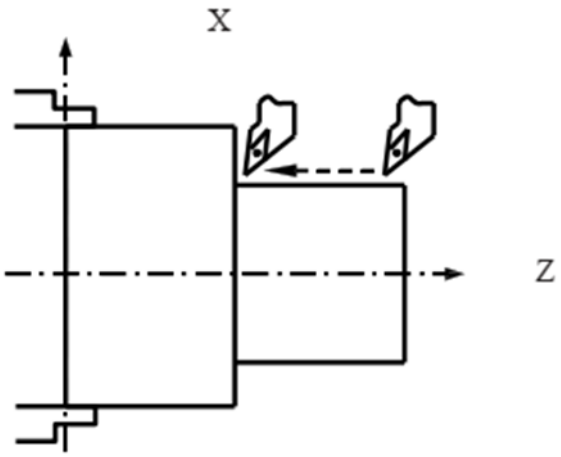


Figure 2.3 Linear Interpolation

2**) Tool movement along the taper line**

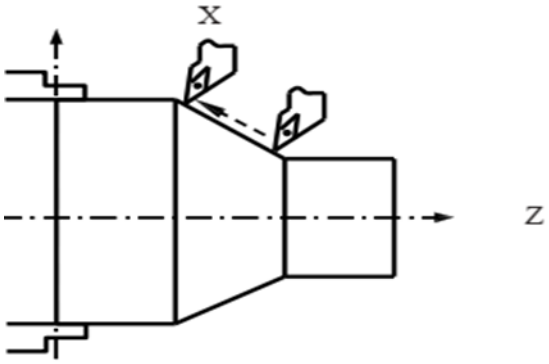


Figure 2.4 Linear Interpolation

b) Circular Interpolation

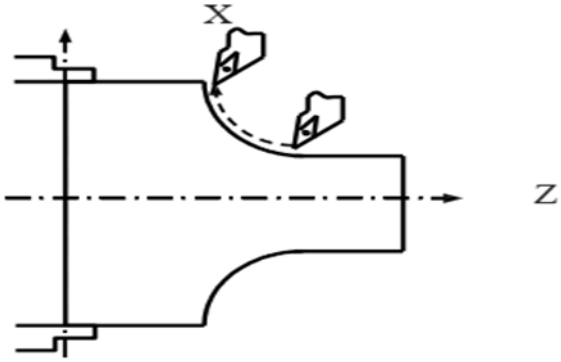
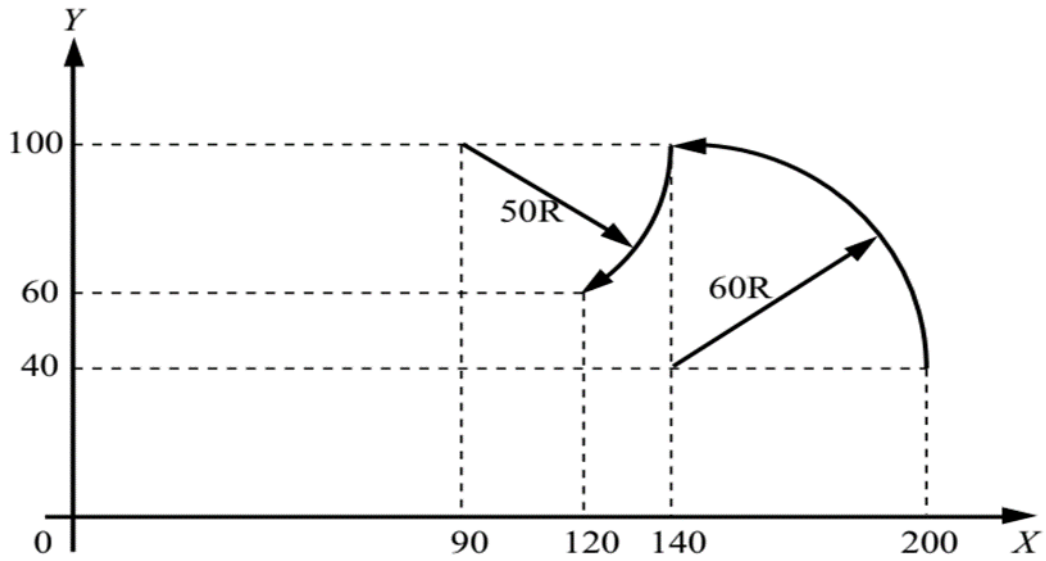


Figure 2.5 Circular Interpolation



i) Absolute programming mode

G92 X200 Y40 Z0 ;
G90 G03 X140 Y100 I-60 F300;
G02 X120 Y60 I-50 ;
or
G92 X200 Y40 Z0 ;
G90 G03 X140 Y100 R60 F300;
G02 X120 Y60 R50 ;

ii) Incremental programming mode

G91 G03 X-60 Y60 I-60 F300;
G02 X-20 Y-40 I-50 ;
or
G91 G03 X-60 Y60 R60 F300;
G02 X-20 Y-40 R50 ;

In order to define the relative motion, reference directions are required to be defined.

a) Machine Origin

- It is a fixed point set by the machine tool builder (cannot be changed).
- Any tool movement is measured from this point.
- The controller always remembers tool distance from the machine origin.

b) Program Origin (home position of the tool)

A point from where the tool starts for its motion while executing a program and returns back at the end of the cycle.

Can be any point within the workspace of the tool which is sufficiently away from the part.

c) Part Origin

It can be set at any point inside the machine's electronic grid system

2.1.6 Setting up of Origin

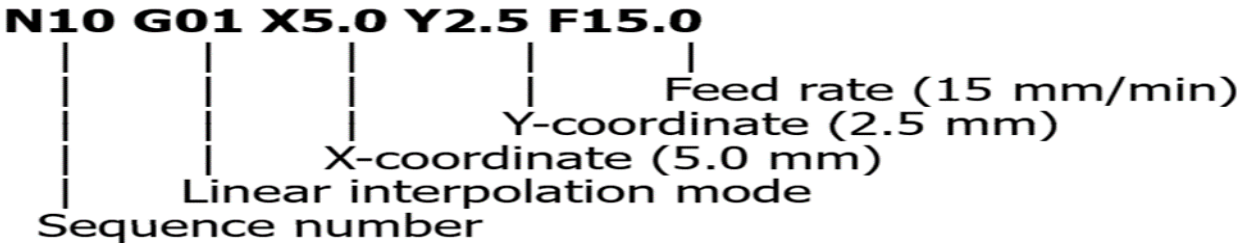
Origin can set by selecting three reference planes X, Y and Z.

Planes can be set by touching tool on the surfaces of the work piece and setting that surfaces as X=x, Y=y and Z=z

2.2 CNC lathe operations program

2.2.1 Part Program

A part program is a series of coded instructions required to produce a part. It controls the movement of the machine tool and the on/off control of auxiliary functions such as spindle rotation and coolant. The coded instructions are composed of letters, numbers and symbols and are arranged in a format of functional blocks as in the following example.



2.2.2 NC words

S**equence** n**umber** (N-**words***):* identify the block

P**reparatory** *w***ork** *(*G-**words***):* prepare the controller for instructions that are to follow

*c***oordinates** *(x-***,** *y-***, and** *z-* **words***):* give coordinate positions of the tool**.**

F**eed** r**ate** (F-**word***):* specifies the feed rate in a machining operation (in./min)

*C***utting** *S***peed** *(*S-**Word***):* **specifies the cutting speed of the process (rev/min)**

Tool Selection (T-Word): specifies the tool to be used in the operation. Only be needed for machines with a tool turret or automatic tool changer

Miscellaneous Function (M-Word): specifies certain miscellaneous or auxiliary functions which may be available on the machine tool. The programmer and the operator must use a coding system to represent information, which the controller can interpret and execute.

A) Preparatory codes:

Used to prepare the control system to be ready for implementing the information that follows in the next block of instructions.

The programmer and the operator must use a coding system to represent information, which the controller can interpret and execute. designated in a program by the word address “G” followed by two digits are also called G-codes and they specify the control mode of the operation.

Some of the G- codes are given below.

G CODE LIST

Code	Description
G00	Rapid Move
G01	Linear Feed Move
G02	Clockwise Arc Feed Move
G03	Counter Clockwise Arc Feed Move
G04	Dwell
G09	Exact stop
G10	Fixture and Tool Offset Setting
G18	ZX Plane Select
G20	Inch
G21	Millimeter
G28	Zero Return
G30	2nd, 3rd, 4th Zero Return
G31	Probe function
G32	Threading
G40	Tool Radius Compensation Cancel
G41	Tool Radius Compensation Left
G42	Tool Radius Compensation Right
G50	Max Spindle Speed
G52	Local Coordinate System Shift
G53	Machine Coordinate System
G54	Fixture Offset
G54.1	Additional Fixture Offsets
G55	Fixture Offset
G56	Fixture Offset
G57	Fixture Offset
G58	Fixture Offset

G59	Fixture Offset
G61	Exact Stop Mode
G64	Cutting Mode (Constant Velocity)
G76	Thread Cutting Cycle
G80	Canned Cycle Cancel
G83	Face Drilling

Code	Description
G84	Face Tapping
G85	Face Boring
G87	Side Drilling
G89	Side Boring
G90.1	Arc Center Absolute Mode
G91.	Arc Center Incremental Mode
G93	Inverse Time Feed
G96	Constant Surface Speed
G97	Constant Speed
G98	Feed Per Minute
G98.1	Initial Point Return
G99	Feed Per Revolution
G99.1	R Point Return

Miscellaneous codes (functions)

use the address letter “M” followed by two digits. often referred to as machine functions or M-functions perform a group of instructions such as coolant on/off, spindle on/off, tool change, program stop or program end.

Some of the M codes are given below.

M CODE LIST

Code	Description
M00	Mandatory Program Stop
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward/Clockwise
M04	Spindle Reverse/Counterclockwise
M05	Spindle Stop
M07	Mist Coolant On
M08	Flood Coolant On

M09	All Coolant Off
M19	Spindle Orient
M30	Program End and Rewind
M48	Enable Feed/Speed Overrides
M49	Disable Feed/Speed Overrides
M98	Sub program Call
M99	Return From Sub program / Rewind

2.3 Simulate and edit CNC lathe Program.

2.3.1 CNC Lathe Simulator is a software simulator of a numerical control lathe is an educational methodological development intended for basic familiarization of novice machine building specialists with the principles of programming parts turning operations using standard G-code (Fanuc System A).

Graphical Simulation.

A graphical simulation package emulates the CNC lathe by using computer graphics to plot out any tool movement and cutting of the workpiece on the VDU screen. Any errors in the program will be observed and highlighted prior to the program being entered into the machine. The programmer can alter or 'Edit' their program as the simulation takes place.

The basis of the three-dimensional simulation model is a lathe machine with a classical arrangement of units, equipped with a CNC system, an eight-position turret, a three-jaw chuck, a tailstock, a coolant supply system and other machinery. Material processing is performed on two

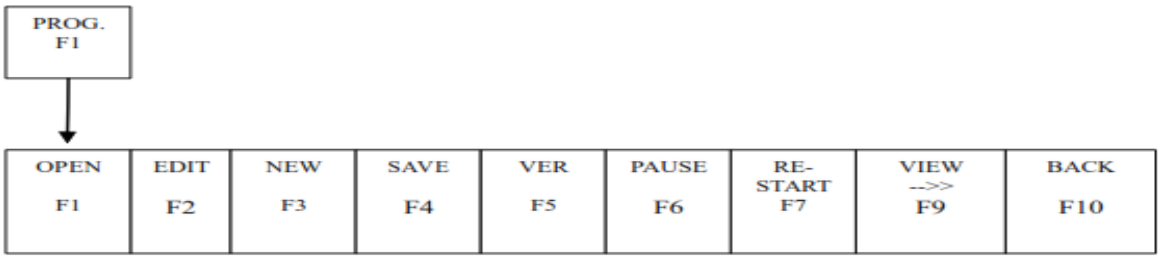
2.3.2Addres

Let’s discuss the various “address” that composes each word prior to looking into the primary unit of machining program “word”. Address has the specific name and meaning as shown below.

Table 2.1 address name and meaning

Address	Applications
N	Block line identification
G	Preparatory-purpose command
X,Z	Point designation to which the axis moves
U,W	
F	Feed rate/speed designation of the axis
S	
T	Tool Designation
M	Miscellaneous function

2.3.3 PROG Submenu



OPEN picks which program is to be run or edited.

EDIT allows the opened program to be edited.

NEW creates a new program while SAVE stores an edited program in memory.

VER verifies whether the opened program has correct syntax and tool path.

PAUSE temporarily halts a running program immediately.

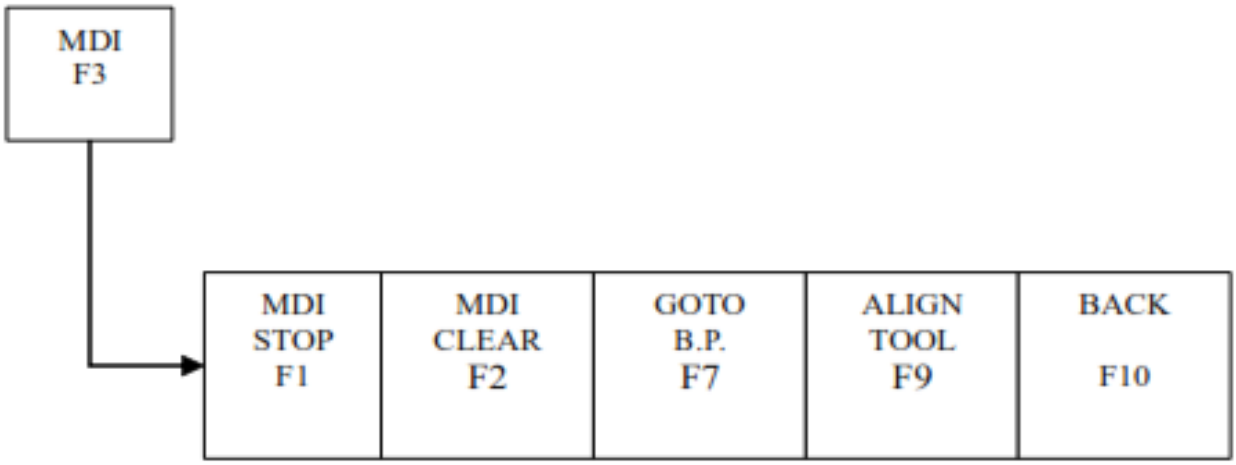
RESTART resumes program running.

VIEW selects one of several ways to view the screen display.

BACK goes back to the main menu.

2.3.4 MDI Menu

The MDI submenu has five soft keys



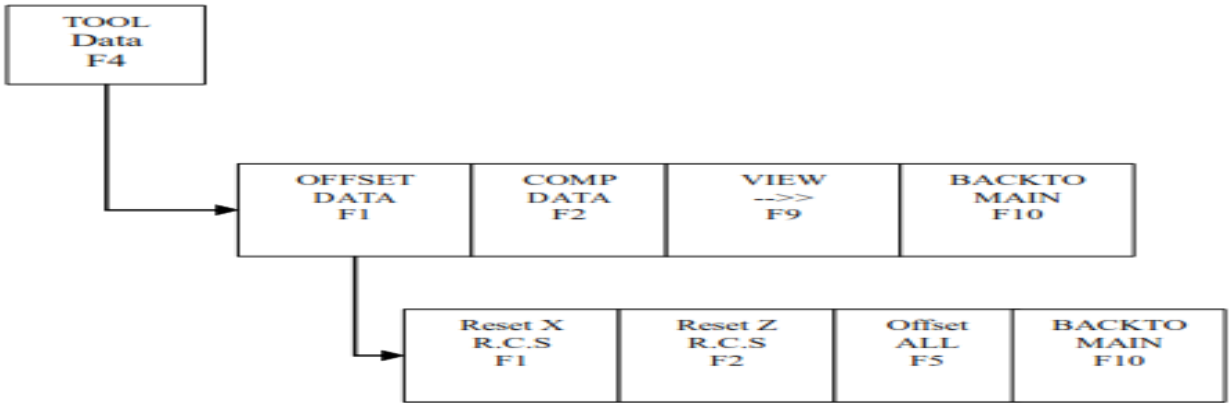
MDI STOP stops the executing of the MDI command.

MDI CLEAR clears all the dimensional data manually input.

GOTO BP let the tool go back to the breakpoint after an interactive intervention during automatic machining.

ALIGN TOOL allows operator to re-align the tool's dimension. BACK goes back to the main menu.

2.3.5 TOOL Submenu



OFFSET allows the user to measure and set the tool's offset data.

COMP DATA let the operator type the tool's compensation data.

VIEW selects one of several ways to view the screen display.

BACK goes back to the main menu.

Manual Data Input Keyboard

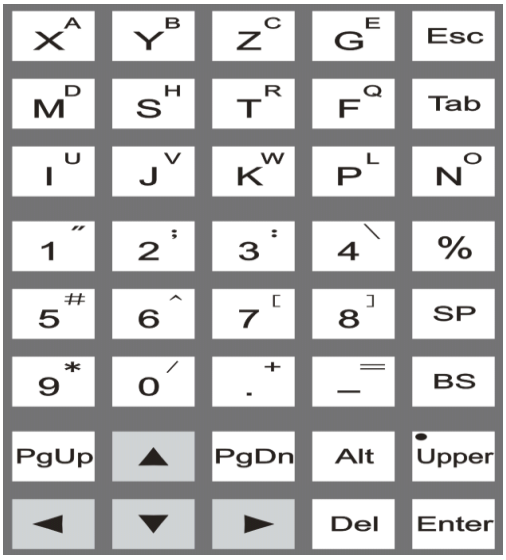


Figure 2:6 Manual Data Input keyboard on the machine control panel

2.4 Save CNC lathe Program.

2.4. 1Programming modes

A. **Absolute mode of programming (G90**)

All measurements are made from the part origin established by the programmer and set up by the operator. The machine control uses the part origin as the reference point in order to position the tool during program execution.

B. Incremental mode of programming (G91)

Tool movement is measured from the last tool position. The programmed movement is based on the change in position b/n two successive points.

2.4.2 Tool selection

Accomplished using 'T' function followed by a 4- digit number first two digits are used to call the particular tool and **last** two digits are used to represent tool offset in the program (used to correct the values entered in the coordinate system preset block). Done quickly on the machine without actually changing the values in the program.

2.4.3 Feed rate control

Cutting operations may be programmed using two basic feed rate modes: Feed rate per spindle revolution and Feed rate per time

G94 =Designate the feed rate per minute(mm/min)

G95= Designate the feed rate per the rotation of principal spindle(mm/rev.)

2.4.4 Program File Operation

Program can be stored in CF, floppy Disk and hard disk. From Main menu, press F1, the program file management menu is displayed.



Functions:

1) F1[OPEN]: Open a program file saved in CF, DNC or floppy disk.
2) F2[EDIT] F3[NEW]: Create a new file.
3) F2[EDIT]: Edit the current program file.
4) F4[SAVE]: Save the current program file. If you specify another name as, this function is “Save as”.

Opening a Program

A program should be loaded to memory before editing, verifying or automatic machining. The following steps are to open a program:

1) From file operation menu, press F1[OPEN]. Screen looks like:

filename	size	date
O001	1K	2006-12-13
O3453	1K	2006-12-18
O6666	1K	2006-12-13
O9990	1K	2006-12-13
O9998	1K	2006-12-13
O9999	1K	2006-12-13
OEXAM	1K	2006-12-13
OEXAM.TXT	1K	2006-12-13
OEXAM1	1K	2006-12-13
OJIUBEI1	1K	2006-12-13
OT4	1K	2006-12-13
OT5	1K	2006-12-13
OXH	1K	2006-12-13

Press “Enter” to open the file.

Figure 2.7 The sequence to open a program

2.4.5 Creating a new Program File

Create a new file at a specified directory on a specified disk. The name of the new file should be different from the existing files.

The following steps are to create a new program file:

1) From file operation menu, press F2[Edit], then F3[New].
2) Type a file name for the new file.
3) Confirm the name by "Enter"
4) Edit the new file.

2.5 CNC lathe program downloaded to the machine.

2.5.1 Standard Operating Procedure

A Standard Operating Procedure (SOP) is a standardized process that outlines a set of detailed instructions to help workers perform complex tasks properly and safely. The main objective of standard operating procedures is to develop an effective quality system and comply with industry-specific regulations and standards. Failure to follow SOPs can cause significant errors in production and operations, increasing unwanted work variations.

Standard Operating Procedure (SOP) for Lathe

- All stock must be properly secured in the lathe chuck or mounted prior to the machining process taking place. Use the correct sized clamp or vise for the stock being machined.
- Turn the chuck or faceplate by hand to ensure there is no binding or danger of the work striking any part of the lathe.
- Check to ensure the cutting tool will not run into the chuck or lathe dog. If possible, feed away from the chuck or dogs.
- Before starting the lathe, ensure the spindle work has the cup center imbedded; tail, stock and tool rests are securely clamped; and there is proper clearance for the rotating stock.
- Prior to starting the lathe, ensure that small diameter stock does not project too far from the chuck without support from the tail stock center.
- Select turning speed carefully. Large diameter stock must be turned at a very low speed.

- Always use the lowest speed to rough out the stock prior to final machining.
- The correct speed and feed for the specific material and cutting tool must be used.
- Stop the machine before making adjustments or measurements.
- Do not remove metal or wood chips from the table or stock by hand. Use a brush or other tool to properly remove chips or shavings from the table or stock.
- Never attempt to run the chuck on or off the spindle head by engaging the power.

Self-Check 2

Instructions: Answer all the questions listed below. Illustrations may be necessary to aid some explanations/answers. Write your answers in the sheet provided.

Part I. Choose the correct answer to the following questions

1. After editing a part program, the program remains in the computer:

A) Forever

B) Until the program is deleted

C) Until the computer is turned off, unless you save the program

2. When entering a part program, the address characters must be typed as:

a. Upper-case letters

b. Lower-case letter

c. Upper- and/or lower-case letters

3. Before adding a comment to a line of NC Code, type a:

A) Space

B) Semicolon

C) Command

4. What is the meaning of CNC code G21 in lathe operation?

a. Measuring unit mm

b. Measuring unit inch

c. Rapid movement of the tool

5. What is the name of CNC code Mo2?

a. Program start

b. Coolant on

c. Program end

6. Which of the following coordinate systems used by CNC programing?

a. Absolute

b. Incremental

c. Cartesian

d. All

Part II:- Match the NC words in the first column to their correct meaning in the second column

Column A

d. Move the tool to 2.5 on the X axis (absolute)
e. Move the tool to .25 on the Z axis (absolute)
f. Move the tool rapidly to the indicated position
g. Make the center point of the arc .15 on the X axis
h. Use clockwise circular interpolation
i. End of program
j. Turn off the spindle motor
k. Make the center point of the arc .5 on the Y axis
l. Use incremental coordinates
m. Use absolute coordinates
n. Move the tool at a feed rate of one inch per minute
o. This is the block marked number twelve
p. Use tool #1

Column B

M02
G90
T1
J.5
G00
X2.5
N12
G02
F1
Z.25
I.15
M05
G91

Part III. Answer the following questions clearly

1. What is a coordinate system means ?
2. What is the theory of René Descartes' coordinate system?
3. What is the point where the axes meet called, and what is the value on each axis? at this point?
4. Write the steps to create manual a new program file in CNC lathe?

Operation sheet-2

Operation title: Writing CNC lathe machine program for the given drawing.

Purpose: Understanding the give drawing and producing part convert into machine language or CNC program.

Instruction: Carefully read the drawing and write the program with correct sequence.

Required tools and equipment's:

- CNC lathe machine
- PPE and materials
- Personal computer
- CAD/CAM software
- CNC lathe accessories

Precautions: For every adjustment use the required tools and alignment parallel to each other.

Procedure: -

1. read the given drawing of the part.

2. Draw the part to scale on graph paper.

3. Determine the stock size and select the tool.

4. Determine the feed rate and depth of cut.

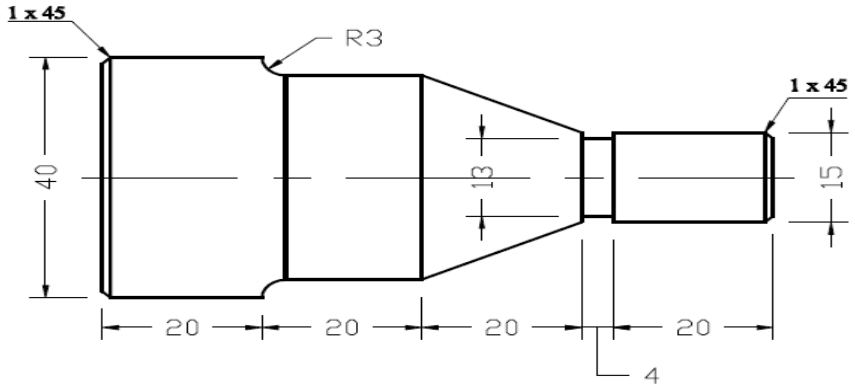
5. Determine how to cut the part.

6. Enter the code on a coding sheet.

7. load the program on the machine

8. run the program

9. follow safe work procedure



Part Programing

O0001
%1234
G21 G90 G18
G00X100Z200
T0101
M03 S800 F100
G00 X47 Z2
G71 U0.5 R0.25 P1 Q13 X0.2 Z0.5
N1 G01 X13 Z0
N3 G01 X15 Z-1
N5 G01 X15 Z-24
N7 G01 X34 Z-44
N9 G01 X34 Z-61
N11 G02 X40 Z-64 R3
N13 G01 X40 Z-84
G00 X100
G00 Z300
M05
T0303
M03 S300 F50
G00 X18 Z-24
G75 X13 Q0.25 R0.25 F5
G00 X100
G00 Z300
M05
M30

Quality criteria: The final product is inspected according to the given dimension with in a tolerance dilation of +-0.05

LAP Test 2

Practical Demonstration

Name: _____ Date: _____TVT Technical Layouts

Time started: _____ Time finished: _____

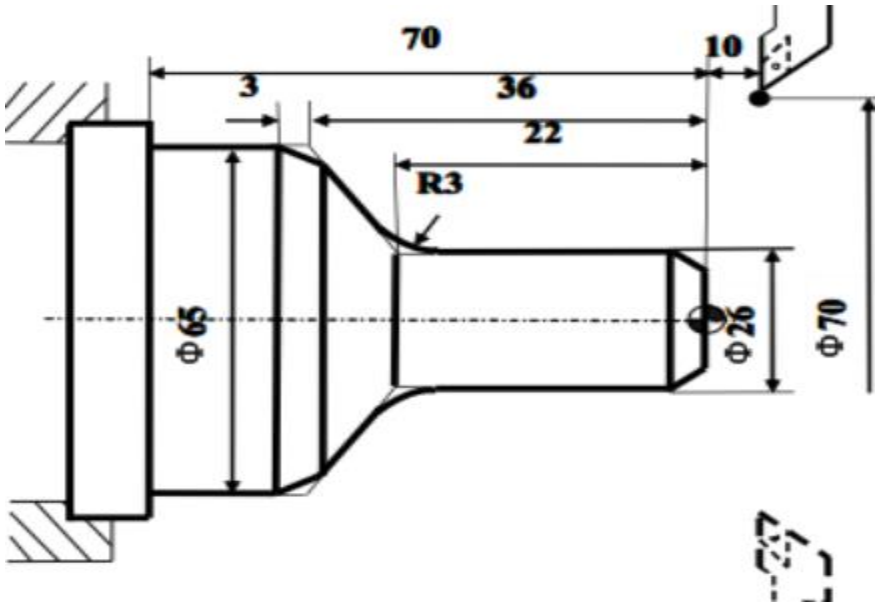
Instructions: Given necessary job instruction and specification for the project to be performed, working plan (sketch), workshop, cutting tools with complete standard accessories, Tools, PPE and materials you are required to perform the following tasks within 4hour per person.

Task 1: - Carefully reading the drawing and write the program with operation sequence.

Use the chamfering and rounding command (G01):

Task 2: load the program to the machine

Task 2: - Dray Run the program



Task 3: load the work piece

Task 4: run the program

Unit Three: CNC Lathe operations

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Mounting work-piece.
- CNC lathe tool path dry run.
- Advance CNC lathe operations.
- Corrective measures/adjustments with standard.

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Safety Procedures in Performing CNC lathe Operations
- Performing dry run tool path for editing program.
- Understanding different types of CNC lathe operation
- Check Corrective measures/adjustments for produced part
- Mount and setting the work-piece for required operation

3.1 Mounting work-piece.

3.1.1 Automatic Chuck Systems

When it is necessary to frequently change the chuck jaws, to accommodate different sized workpieces, or when additional machining operations require a different method of holding the workpiece, a quick jaw-changing system can be used. Systems such as these can reduce the time taken to change jaws from around 30 minutes to 1-2 minutes, or less. Although initially expensive, the system can pay for itself in a very short space of time. There is less machine down time, so the productivity of the machine can improve quite dramatically.

CNC lathes can also be equipped with fully automatic jaw, or chuck, changing systems. Some automated systems change on jaw at a time, whilst other systems may be capable of changing all jaws at once.

CNC machine tool has coordinate systems, called Machine coordinate system and Workpiece coordinate system. The machine coordinate system is used to instate the positions specific to the machine tool.

The “Machine position” counter shown on the monitor screen of CNC unit displays the coordinates in the machine coordinate system.

The position at which the “machine position” counter displays zero is called machine coordinate origin.

The workpiece coordinate system is commonly used when writing the machining program. The position (X, Z) = (0, 0) in the machining program is called workpiece coordinate origin.

The “Workpiece coordinate position” counter shown on the monitor screen of CNC unit displays the coordinates in the workpiece coordinate system.

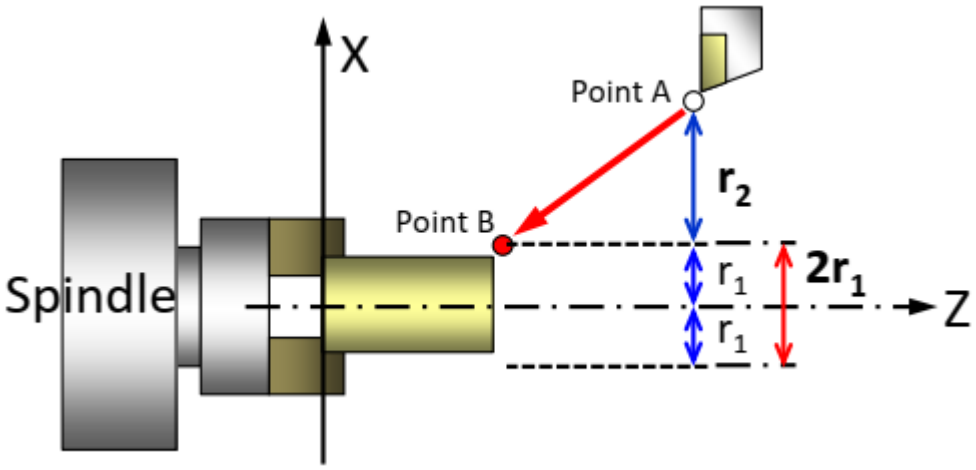


Figure 3.1 w/p and m/c origin

Reference position is a specific point defined by a coordinates in the machine coordinate system, and is utilized as the point at which the machine is told as the start point of motion.To move the tool to reference position, we use G28 (Reference position return).

By the axis letter address in absolute mode (X, Z) included in G28 command, you can specify the intermediate position (intermediate point) to which the axis will first move prior to moving to the reference position. This intermediate point is used to avoid unintended interference between the tool and the workpiece.

For example, the command shown below tells the machine to first move the tool from point A to (X, Z) = (150.0, -200.0) then to move the reference position.

3.1.1 Reference position and return: G28

In case you would not like to specify the intermediate point, use zero in incremental positioning (U0, W0) in G28 to directly move the tool to the reference position. In this case, it is generally preferred to move the axis one by one to avoid unintended interference between the tool and the workpiece.

Following show, the example when moving the X axis and Z axis independently to reference position. G28 U0

G28 W0;

3.2 CNC lathe tool path dry run.

This method of program proving is done on the machine tool but the workpiece or billet to be cut is not installed. As the tool or cutter moves in air the operator looks to see that the cutting tool will not collide with proposed clamping arrangements or other projections within the set-up. Federate override facilities can be used to slow down the speed at which the program runs.

Dry run in CNC is a program testing process. In this process all axis movement will be tested according to program. During this process spindle or chuck will not rotate. The programmer will manually test the program/code. This massage will appear when “DRY RUN M41 SPINDLE ON” switch will ON.

3.2.1 Toolpaths: -The most basic toolpath for lathe machine is 2D. As its name suggests, 2D toolpaths only ever move in two dimensions, or two of the machine’s axes, while cutting. Generally, these are the x and z axes.

G00 – position of the tool and not concerned with the path that is followed in arriving at the final destination.

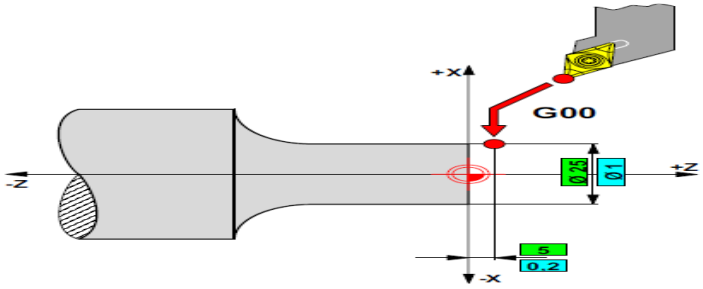


Figure 3.2 G00 tool path

G01 – Tool is required to move in a straight line connecting current position and final position. Used for tool movement without any machining- point to point control. (linear interpolation)

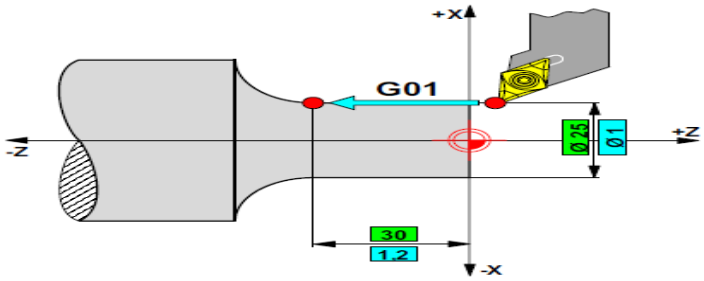


Figure 3.3 G01 tool path

G02 – Circular interpolation (clockwise)

- This command is anything with circular interpolation

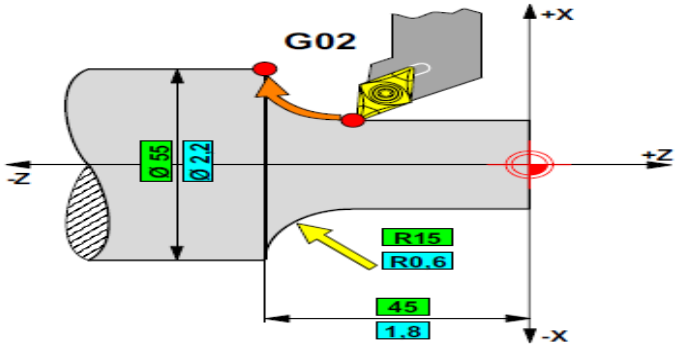


Figure 3.4 G02 tool path

G03– Circular interpolation (Counter clockwise) This command is anything with circular interpolation

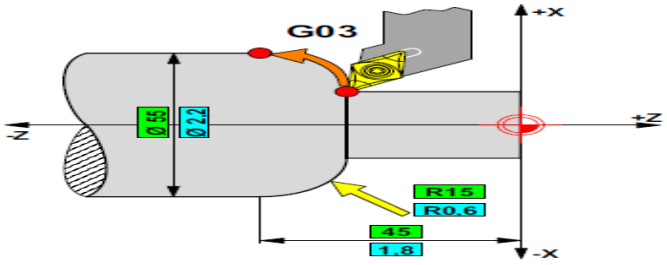


Figure 3.5 G01 tool path

3.3 Advance CNC lathe operations.

3.3.1 Automatic Parallel And Taper Turning

Automatic (AUTO) operation is used to machine parts under program control. In an NC machine tool, most production machining is done as an AUTO operation. In Automatic mode, you can execute part programs automatically. If you are a manual lathe operator, it is the AUTO operations that will be most unfamiliar to you.

AUTO operation requires a program to execute. Several sample programs are already included in the controller's memory. If you are a beginner, execute one of the sample programs in AUTO mode.

Facing is the processes of reducing the Length of work piece.

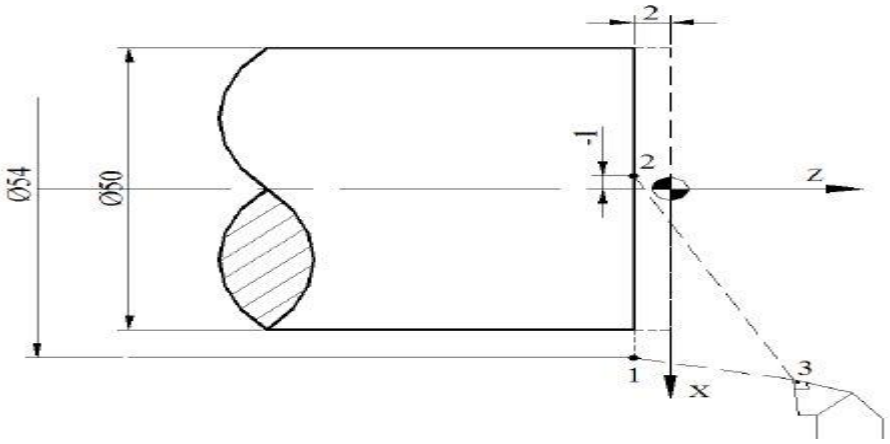


Figure 3.6 Facing operation

O0001(CNC Facing Program)

N10 T03 M06

N20 G96 S150 G95 F0.1 D2500 (D = Maximum spindle speed)

N30 G00 X54 Z-2 M03 M08

N40 G01 X-1

N50 G00 X200 Z200 M09

N60 M30

Turning is the processes of reducing the diameter of work piece.

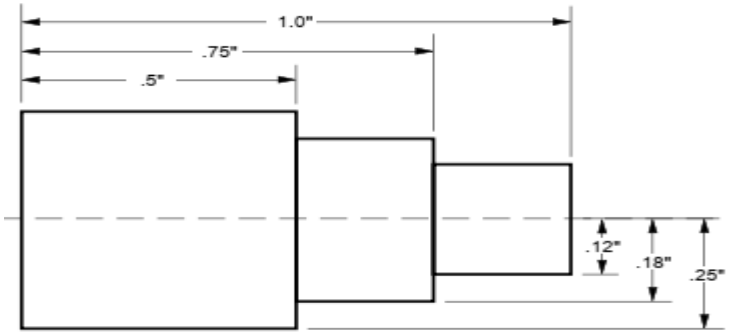


Figure3.7 turning operation

G90 G95;
M03 S90;
T1D1;
G00 X355 Z2;
CYCLE95("TURN"2.00000, 0.00000, 1.00000, 0.00000 0.30000, 0.30000, 0.30000, 9, , ,1.00000);
G00 X360;
G00 Z2.0;
M30;

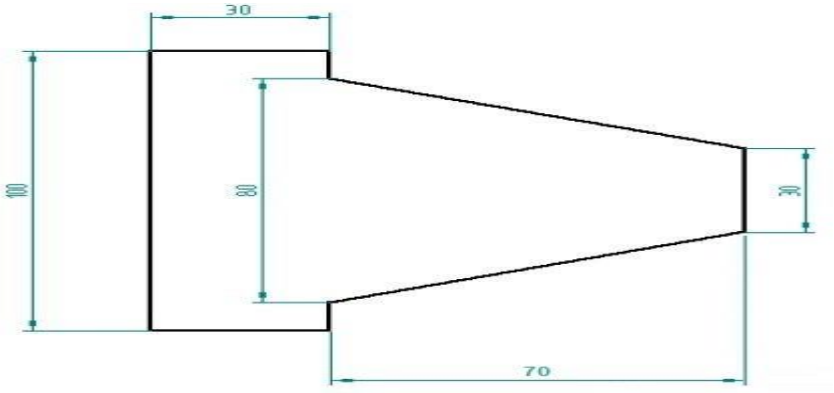


Fig 3.8 Taper turning

Program-Taper turning cnc code

N05 G94 M06 T0101

N10 G00 X0 Z0 M04 S600

N15 G01 X30 F30

N20 G01 X80 Z-70

N25 G01 X100

N30 G01 Z-30

N35 G00 X120

N40 G00 Z10

N45 M05

N50 M30

3.3.2 Internal and external turning

A program file must be “loaded” into the lathe’s memory before it can be “executed” (i.e., cut a part or do some other operations automatically). The program file itself is a series of program lines called blocks that direct the lathe through a series of various operations automatically. The programmer who writes the program file sets these operations.Only one program at a time can be “loaded” for execution by the lathe. Usually many programs that have been written are stored in the lathe’s computer. “Loading” is the process of picking one of these programs to be executed.

i. G04 – Dwell(drilling)

This command is used when drilling. when the drilling tool dips at the end of the drilling depth tool will stops in a matter of seconds depending on the program.

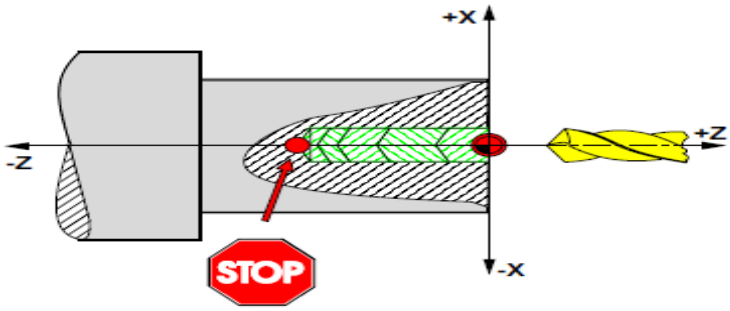
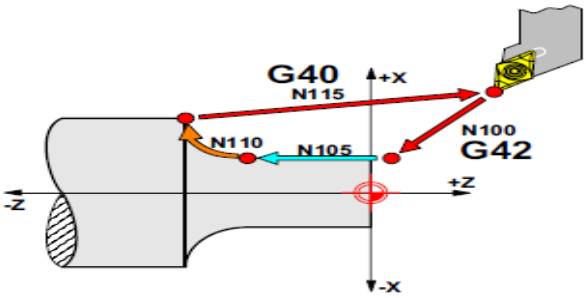


Figure 3.9 Dirlling

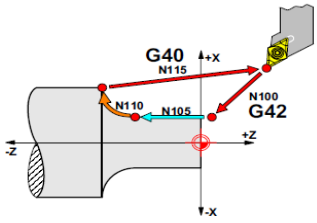
ii. G40 – Cancel Radius compensation

This command is used cancel the radius Compensation of a tool used**.**




```
N100 G42 G00 X.. Z..
N105 G01 Z-.. F..
N110 G02 X.. Z-.. R..
N115 G40 G00 X.. Z..
```

Figure 3.7 Cancel Radius compensation



iii. G41 – Cutter radius compensation left

This command is used when your tool Is viewed at the left side of the contour

```
N100 G41 G00 X.. Z..
N105 G01 Z-.. F..
N110 G03 X.. Z-.. R..
N115 G00 Z..
N120 G40 G00 X.. Z..
```

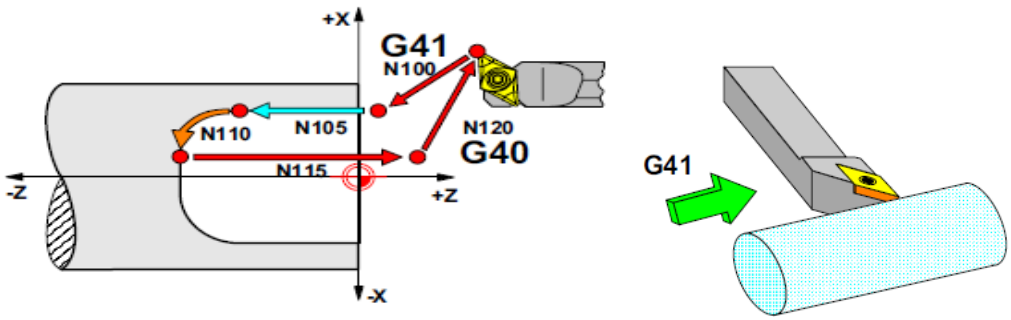
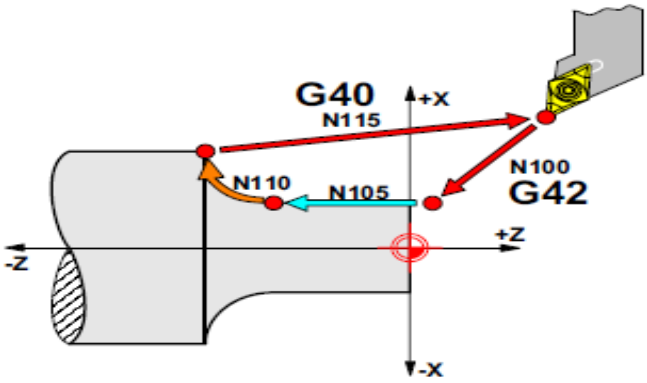


Figure 3.10 Cutter radius compensation left

iv. G42 – Cutter radius compensation right

This command is used when your tool Is viewed at the right side of the contour




```
N100 G42 G00 X.. Z..
N105 G01 Z-.. F..
N110 G02 X.. Z-.. R..
N115 G40 G00 X.. Z..
```

Figure 3.11 G42 – Cutter radius compensation right

v. G72 – Finishing cycle

This cycle will start its cycle from the very first path until the end path of the machining operation

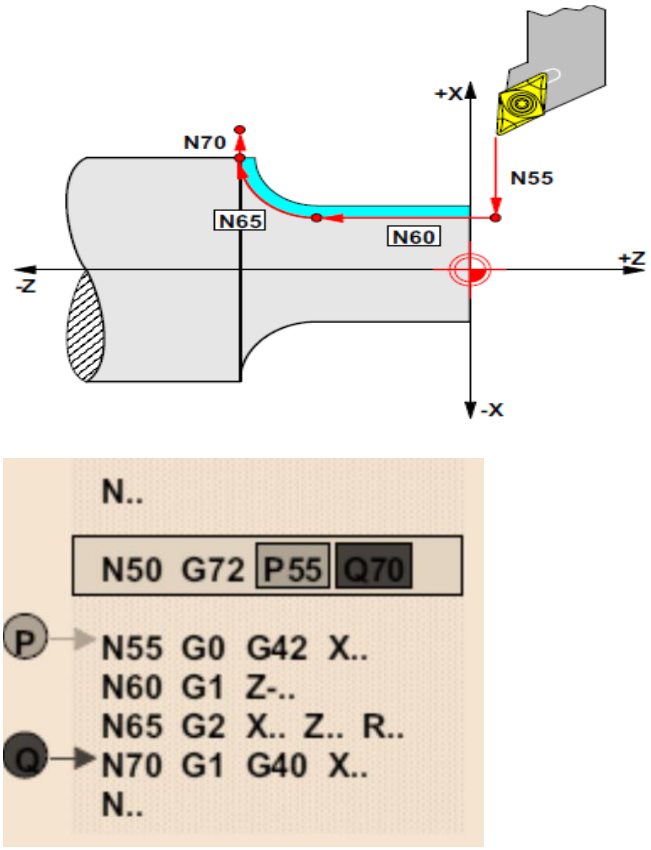


Fig 3.12 G72 – Finishing cycle

vi. Automatic G01 corner rounding

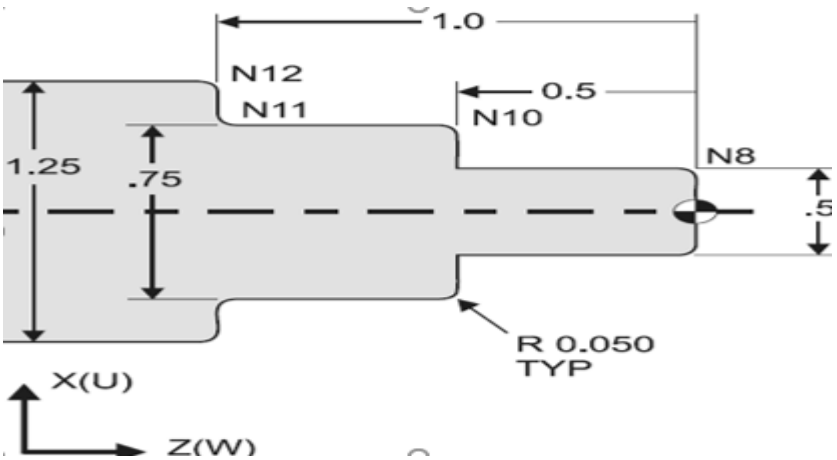


Figure 3.13 **Automatic G01 corner rounding**

O00042 (Linear G01 with Radius using R) N1 G53 G00 X0. Z0. T0

N2 T101 (O.D. TOOL x .031 TNR) N3 G50 S3000

N4 G97 S3000 M03

N5 G54 G00 X0.3 Z0.1 M08 N6 G96 S390

N7 G42 G01 Z0. F0.01 (Tool nose compensation right) N8 G01 X0.5 R-0.05

N9 G01 Z-0.5

N10 G01 X0.75 R-0.05

N11 G01 Z-1.0 R0.05 (R is to a positive X axis) N12 G01 X1.25 R-0.05 (R- is to a negative Z axis) N13 G01 Z-1.5

N14 G40 G00 U0.01 Z0.1 M09 (cancel tool nose comp) N15 G53 G00 X0. Z0. T0

N16 M30

vii. G75-grooving operation

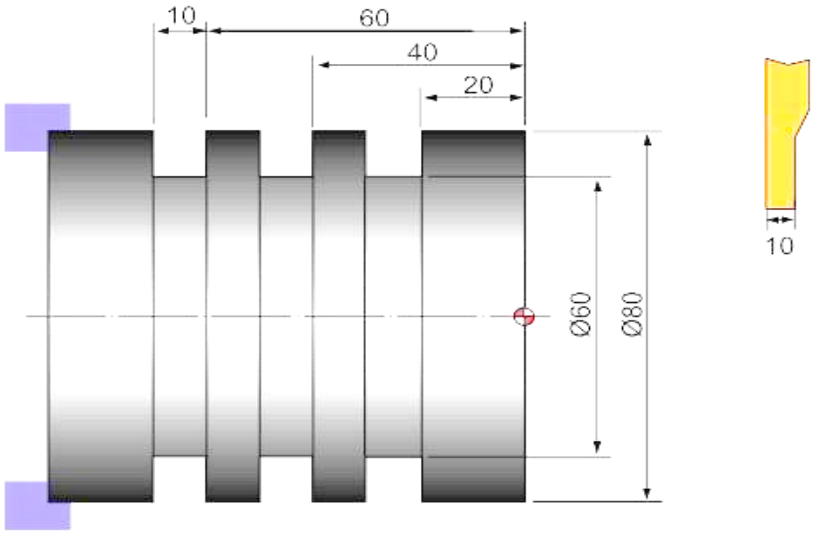


Figure 3.14 G75-grooving operation

N10 G50 S500 T0100

N20 G97 S400 M03

N30 G00 X90.0 Z1.0 T0101

N40 X82.0 Z-60.0

N50 G75 R1.0

N60 G75 X60.0 Z-20.0 P3000 Q20000 F0.1

N70 G00 X90.0

N80 X200.0 Z200.0 T0100

N90 M3

3.3.3 Thread cutting

A. G76 Thread Cycle Example

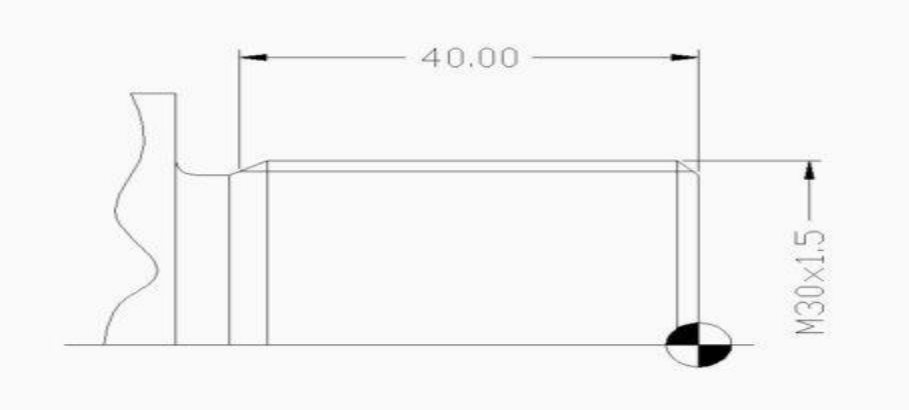


Figure 3.15 Thread cycle

N17 T101

N18 G54

N19 G97 S800 M3

N20 G0 X32 Z6 M8

N21 G76 P010060 Q100 R0.02

N22 G76 X28.161 Z-50 P919 Q250 F1.5

N23 G0 X150 Z100

C. G92 Taper Threading Cycle

Other parameter of Fanuc threading cycle G76 are explained here. The R parameter in second block of G76 is the tapered value. Note that R is given as Radius value. How to calculate R parameter for Tapered Threading on Fanuc with G76 Threading Cycle.

R = (Start Diameter – End Diameter) / 2

G92 threading cycle is used for threading on a CNC lathe machine. The added benefit of G92 thread cycle is that G92 G-code gives CNC machinist full control over the depth of every cut while threading. For full description of G92 threading cycle read CNC Fanuc G92 Threading Cycle. The below is a CNC programming example which show you how taper threads can be machined with G92 thread cycle. Taper Threading with G92 Threading Cycle.

CNC Programming Example G92 Taper Threading Cycle

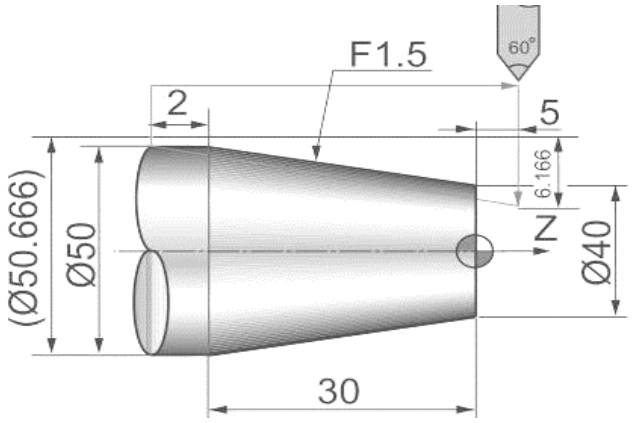


Figure 3.16 G92 Taper threading cycle

N10 T1

N20 G97 S1000 M03

N30 GOO X70 Z5 M08

N40 G92 X49.4 Z—32 R—6.166 F1.5

N50 X49

N60 X48.7

N70 X48.5

N80 X48.3

N90 X48.2

N100 G00 X100 Z100

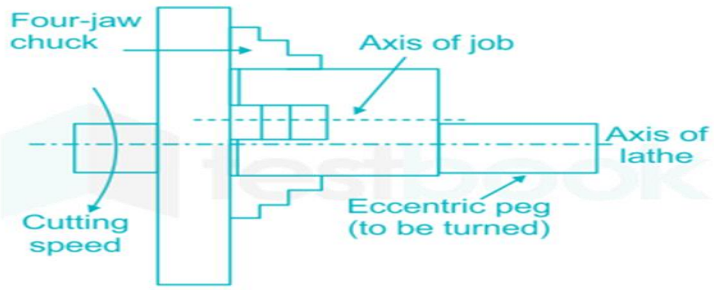
N110 M30

3.3.4 Eccentric Turning

Eccentric turning, also known as offset turning, is a type of lathe operation. It is typically used to machine parts with features that are not concentric to the spindle axis, such as the eccentric drive of a steam engine and a crankshaft.

The main difference between turning and eccentric turning is the center of rotation of the workpiece. In the turning process, The axis of rotation of the chuck and the geometric axis of the workpiece and in alignment. While in eccentric turning, the axis of rotation and the geometric axis of the workpiece is not aligned. Most CNC turning jobs are concentric and use a 3 jaw chuck to hold the workpiece. The 3-jaw chuck moves all jaws in unison when the chuck is adjusted. While in eccentric turning, a 4 jaw chuck is usually used or the piece is turned between centers. The 4 jaw chuck independently moves the jaws which allow parts to be moved off-center.

Eccentric parts refer to the workpiece whose external axis and internal axis are not in the same straight line but are parallel to each other. This kind of workpiece can convert straight-line motion into rotary motion, so it is widely used in the field of mechanical transmission. Eccentric workpieces can be divided into eccentric shafts and eccentric sleeves according to the types of the parallel axis.



Eccentric turning

This program will produce an eccentric circle with a radius of 0.5 inches and an eccentricity of 0.25 inches. The cutting tool will travel at a feed rate of 100 inches per minute.

G00 X0.0 Y0.0 Z0.0

G01 X0.5 Y0.0 F100.0

G01 X0.5 Y0.25 F100.0

G01 X0.25 Y0.5 F100.0

G01 X0.0 Y0.5 F100.0

G01 X0.0 Y0.25 F100.0

G01 X0.25 Y0.0 F100.0

G00 Z1.0

M30



3.4 Corrective measures/adjustments with standard.

Why the Alignment Is Important?

Lathe turret alignment is a crucial step. If not properly aligned this may lead to a machine crash eventually. The alignment will determine how square the drills run in the boring bar pockets. It must be made sure that they run alongside the Z-axis and precisely cut on the centre.

3.4.1 Replacement of cutting tools

T - Tooling Arrangement.

This refers to the number of the tool required to be used for a particular operation. Each tool is given a number and placed in a waiting area before being called upon. The tools must be pre-set in the machine prior to storage. Any data that is fed into a CNC computer must conform to the manufacturer's classification or the computer will not understand it.

A typical example would be:

N4 G2 X4/3 Z4/3 etc.

To explain this take X4/3:

The first digit (in this case 4) refers to the maximum number of digits that can be allowed before the decimal point.

The second digit (in this case 3) refers to the maximum number of digits that can be allowed after the decimal point.

For example, X 1689.913.

If there is only one figure after the address character for example N4

This means that the largest number of digits before a decimal point and none after it.

It is very important that any data entered into the machines computer is done so in a way that the computer understands.

Two methods are described below:

Fixed Block- Each block of information must be entered into the computer even though some of the words have not changed from the previous block.

Variable Block- The data can be entered in any order without having to repeat unchanged data.

Zero Suppression. It may be that the data classification of a block is 3/2. As we know, this refers to the maximum and minimum number of digits before and after the decimal point. But what if the programmer has to program a figure of 16.1 into this block?

It can be seen that there are only two digits prior to the decimal point and only one after it.

Some systems will require the programmer to input leading and trailing zeros. For example, Ø16.1Ø Other systems will just require the original digits to be fed into the computer.

The leading zero and the trailing zero are not required. We call this omitting of the zeros, *Zero* Suppression.

All CNC part programs should be carefully checked before they are set in motion on a machine tool.

Methods of Checking and Verifying:

Visual Inspection.

The actual program is run, with no workpiece present in the chuck. Through purely visual inspection, all the programmed movements in all axes are checked, together with any tool offset and cutter compensation features. This method represents the least form of verification and should not be relied on entirely.

3.4.2 Adjustment of tool offset

Tool Setting: -No matter what configuration or type of tool used, it must be positioned at the correct height, in the tool holder and tool post, for efficient cutting. Ideally, the tool should be positioned as far back into the tool post as possible. Too great an overhang can cause vibration and chatter, leading to poor surface finishes.

The choice of feed rate and depth of cut determines the size of the chip cross section.

The following formulae are used to determine the size of the chip cross section and the amount of material removed during a machining operation:

Chip cross section=Feed rate per revolution x Depth of cut.

Stock removal volume=Chip cross section x Cutting speed.

3.4.3 Adjustment of cutting speed and federate

The lathe cutting parameters determined by the operator are the feed rate, depth of cut and spindle speed. All three must coordinated to take into account a number of factors:

1) **To achieve short cycle times.**

Increasing the federate, depth of cut or spindle speed parameters will remove more material chips per unit time, reducing the time required to complete the CNC program. This must be balanced against accompanying high tool wear which will counter any time saved, due to more frequent tool tip and tool holder changes.

2) **To achieve lower machined part costs.**

Any increase in cutting parameters leading to a reduction in cycle times will also reduce overall labor and machining costs. Tool costs will rise due to greater wear. A balance must be chosen so that the wear-related tool costs are not excessive, possibly through the selection and use of an appropriate coolant.

3) **To achieve high machined workpiece quality.**

The choices for flexibility of cutting parameters become much more limited, due to the specialized settings for obtaining high quality surface finishes and dimensional accuracy. The choice of parameters will depend on the type of tool tip being used, the load capabilities of the machine and the vibration characteristics of the tool, workpiece and machine.

The selection of the correct spindle speed, feed rate, cutting depth and cutting tool for each machining operation is important since incorrect settings can lead to chattering, poor finishes and tool breakage. There are three basic types of machining operation:

Self-Check 3

Instructions

Given minutes 30 min provide what is asked.

- Enumerate the following. Use a separate sheet of paper for answering.
- For Short answers, write the answer for the questions on the provided space.

Part II. Choose the correct answer

1. Which of the following operation is performing on lathe, except?
   A. Turning operation C. Rack gear
   B. Thread operation D. grooving operation

2 CNC machine tool has coordinate systems, called
   A. Machine coordinate system C. Workpiece coordinate system
   B. Tool coordinate system D. A & C correct

3 Which of CNC machine Operators Must Know About Sizing Adjustments
   A. Explain, demonstrate and have newcomers practice making sizing adjustment at the machine.
   B. Show how to determine whether a sizing adjustment is required and, if so, its value
   C. Explain, demonstrate and have newcomers practice taking measurements
   D. All

4. Tool is required to move in a straight line connecting current position and final position using the code known as.
   A. G- 00 C. G-03
   B. G-01 D. G-02

Part II. True/false

1. Closing movements between the slide-ways and fixed parts (tail stock) can result in hand crushing
2. The most basic toolpath for lathe machine is 3D
3. in incremental positioning (U0, W0) in G28 to directly move the tool to the reference position
4. The position at which the “machine position” counter displays zero is called machine coordinate origin.
5. Dry run in CNC is a program testing process

Part III. Give brief and short answer to the given questions

N5 G90 G20

N10 M06 T3

N15 M03 S1250

N20 G00 X1 Y1

N25 Z0.1

N30 G01 Z-0.125 F5

N35 X3 Y2 F10

N40 G00 Z1

N45 X0 Y0

N50 M05

N55 M30

Use the above program to answer the following questions

1. Did your program run through verification without any problems?

2. If not, what were the problems? How did you fix them?

3. Do you get any error messages while the program is running? How would you correct them if you did?

Operation sheet 3

Operation title: - Thread cutting operation

Instruction: -From the following drawing calculate and write CNC lathe program and produce the required part all the dimensions are in mm

Purpose: To produce the given thread with the given program

Tools and material requirement:

- Turning cutter
- grooving cutter
- Threading cutter

Part Name: Piston

Material: Aluminum 6061

Tool: HSS Turning Tool

Speed: 1000 RPM

Feed Rate: 0.005 IPR

Procedures

- Cutting the required length of W/P using power hack-saw
- Adjust the work on cuck
- Turn the Required diameter on lathe machine
- Set up or align the center of work with thread cutting tool
- Check pitch of thread using pitch gage.
- Cut the thread to the required pitch

Operations:

1. Center the workpiece in the chuck.
2. Face the end of the workpiece.
3. Turn the outside diameter of the workpiece.
4. Turn the groove for the piston ring.
5. Cut off the piston.

Instructions:

1. Mount the workpiece in the chuck and center it using the tailstock.

2. Set the tool to the desired height and angle.
3. Start the machine and engage the feed.
4. Face the end of the workpiece to the desired length.
5. Turn the outside diameter of the workpiece to the desired size.
6. Turn the groove for the piston ring to the desired depth and width.
7. Cut off the piston using a parting tool.

Precautions:

- Use Flood coolant throughout the machining process.
- Check the dimensions of the part frequently to ensure accuracy.
- Be careful when cutting the piston ring groove to avoid damaging the tool.

Part Programing

O0002

%1234

G21 G90 G18

G00X100Z200

T0101

M03 S800 F100

G00 X42 Z5

G71 U0.5 R0.25 P1 Q17 X0.2 Z0.5

N1 G01 X13 Z0

N3 G01 X16 Z-1.5

N5 G01 X16 Z-20

N7 G01 X22 Z-20

N9 G01 X24 Z-21

N11 G01 X24 Z-30

N13 G02 X24 Z-45 R12

N15 G01 X33 Z-60

N17 G01 X33 Z-75

G00 X100

G00 Z300

M05

T0303

M03 S300 F50

G00 X17 Z-20

G75 X12 Q0.25 R0.25 F5

G00 X17 Z-18

G75 X12 Q0.25 R0.25 F5

G00 X100

G00 Z300

M05

T0404

M03 S300

G00X17Z5

G00X15.5

Z5

G32Z-14.5F2

X14.26

G00X17

G32Z-14.5F2

Z5

G00X17

X15.1

Z5

G32Z-14.5F2

X14.26

G00X17

G32Z-14.5F2

Z5

G00X17

X14.7

Z5

G32Z-14.5F2

X14.26

G00X17

G32Z-14.5F2

Z5

G00X100

X14.4

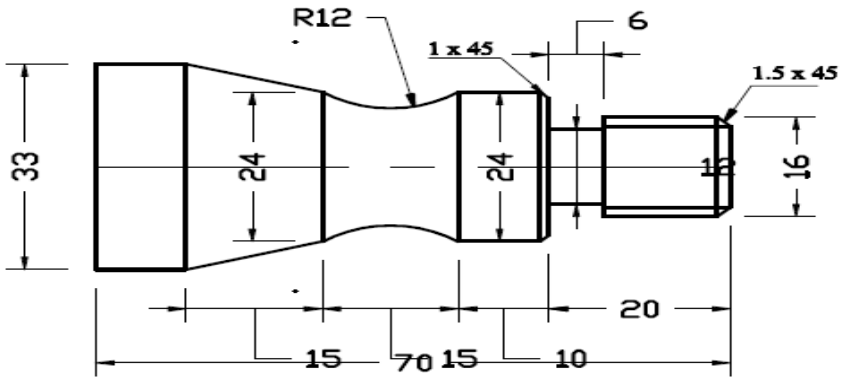
Z300

G32Z-14.5F2

M05

G00X17

M30



LAP Test 3

Practical Demonstration

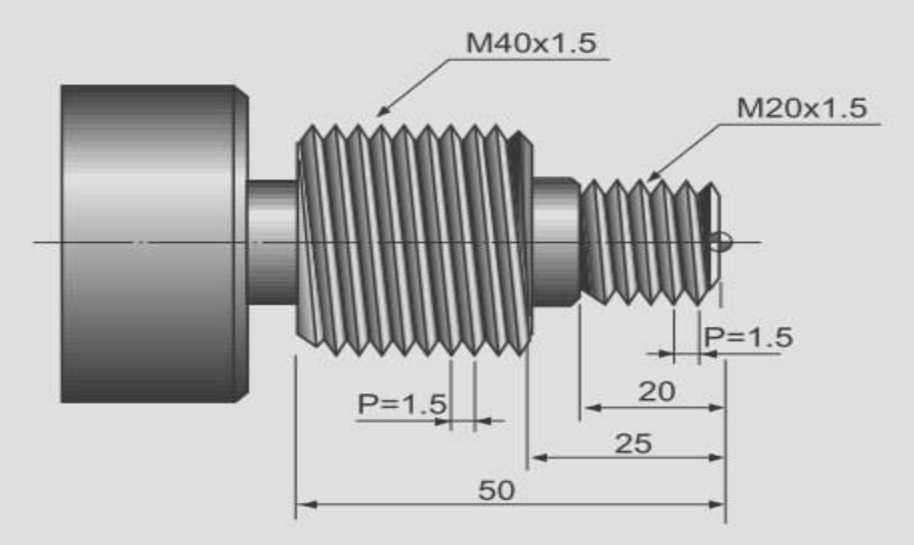
Name: _____ Date: _____TVT Technical Layouts

Time started: _____ Time finished: _____

Instruction I: Given necessary templates, tools and materials you are required to perform the following tasks within 4 hours.

Task 1: - Write a program to Cute ISO standard Thread using the following drawing

Task 2 Calculating the cutting depth of thread shown below



Unit Four: Quality assurance of produce parts

This learning unit is developed to provide the trainees the necessary information regarding the following content coverage and topics:

- Inspect components.
- Handling deviations.
- Comparing product quality of the CNC production with conventional production.

This unit will also assist you to attain the learning outcomes stated in the cover page. Specifically, upon completion of this learning guide, you will be able to:

- Inspect components.
- Handle deviations
- Compare Product quality

4.1 Inspect components.

Ensuring product quality is crucial in CNC lathe operations to meet customer requirements and maintain high standards. Here are some key aspects of CNC lathe product quality assurance:

Machine Calibration and Maintenance: Regular calibration and maintenance of the CNC lathe are essential to ensure its accuracy and performance. This includes checking and adjusting the machine's axes, spindle runout, tool offsets, and other critical parameters. Adhering to the manufacturer's maintenance schedule and guidelines is important for optimal machine performance.

Tooling Selection and Inspection: Selecting appropriate cutting tools for the machining operations is vital. High-quality tooling, such as inserts, holders, and drills, should be used to maintain dimensional accuracy, surface finish, and tool life. Inspect tools regularly for wear, damage, or excessive runout, and replace them when necessary.

Workpiece Inspection: Conduct regular inspections of machined workpieces to ensure they meet the required specifications. This may involve dimensional measurements, surface roughness analysis, hardness testing, or other specific quality checks based on the product requirements. Use calibrated measuring instruments and follow inspection procedures to ensure accurate and consistent results.

Programming and Simulation: CNC lathe programming plays a significant role in product quality. Use reliable CAM software to generate precise toolpaths and optimize cutting parameters. Perform simulations or virtual machining to verify the toolpaths, detect collisions, and ensure the desired results before running the program on the actual machine.

Process Monitoring: Implement process monitoring techniques to detect deviations or abnormalities during machining. This can involve using sensors, gauges, or monitoring

software to measure variables such as cutting forces, tool wear, temperature, or vibration. Real-time monitoring allows for timely intervention to prevent quality issues and maintain process stability.

Quality Control Documentation: Maintain thorough documentation of quality control processes and inspection results. This includes recording machine settings, tooling data, inspection reports, and any corrective actions taken. Documentation facilitates traceability, analysis of trends, and identification of areas for improvement.

Continuous Improvement: Regularly review and analyze quality data to identify opportunities for improvement. This may involve analyzing scrap or rework rates, customer feedback, and non-conformance reports. Implement corrective actions, conduct root cause analysis, and establish preventive measures to enhance product quality and overall process efficiency.

Training and Skill Development: Provide training and skill development programs for machine operators, programmers, and quality control personnel. Enhancing their knowledge of CNC lathe operations, programming techniques, measurement tools, and quality assurance methods contributes to consistent product quality.

4.2 Deviations handling procedures and standard

Handling deviations in CNC lathe operations is important to address any issues that may arise during machining and ensure that product quality is maintained. Here is a procedure for handling deviations in CNC lathe operations.

Identify the Deviation: When a deviation is detected, such as dimensional inaccuracy, surface finish problems, or other quality issues, it is essential to identify the specific nature

and extent of the deviation. This can involve inspecting the workpiece, reviewing measurement data, and comparing the results against the desired specifications.

Stop the Operation: If a significant deviation is observed, it is advisable to stop the CNC lathe operation to prevent further machining on potentially defective workpieces. This helps avoid the production of non-conforming parts and minimizes waste.

Analyze the Root Cause: Determine the root cause of the deviation. This can involve examining various factors, including machine settings, tooling conditions, programming errors, material issues, or environmental factors. Thoroughly analyze the situation to understand the underlying cause of the deviation.

Make Immediate Corrections: If the root cause is identified and can be immediately corrected, take the necessary corrective actions. This may involve adjusting machine parameters, replacing worn or damaged tools, reprogramming the CNC lathe, or addressing any other issues that contributed to the deviation.

Document the Deviation: Record all details related to the deviation, including the nature of the issue, the corrective actions taken, and any other relevant information. Proper documentation helps in tracking the occurrence of deviations, analyzing trends, and implementing preventive measures in the future.

Perform Validation and Verification: After implementing the corrective actions, validate and verify the effectiveness of the adjustments made. This can involve rechecking the workpiece dimensions, conducting additional inspections, or performing test runs to ensure that the deviation has been adequately addressed.

Implement Preventive Measures: Once the immediate deviation has been resolved, focus on implementing preventive measures to minimize the likelihood of similar deviations occurring in the future. This may involve reviewing and improving machining processes,

enhancing operator training, updating programming guidelines, or investing in better tooling or equipment.

Continuous Improvement: Continuously monitor the CNC lathe operation and collect data to identify any recurring deviations or trends. Regularly analyze the data and seek opportunities for continuous improvement in machining processes, quality control procedures, or other relevant aspects.

International Tolerance Grade Number and Applications (Processes)

How precise are common CNC machining processes? Different IT grade numbers represent different precision and suitable varying processing techniques.

IT Grade 1-4: very precise, usually used for Gage making and precision work, precision measuring tools

IT Grade 5-16: suitable for cutting operations like turning, milling, boring, grinding and sawing.

IT Grade 12-16: suitable for manufacturing operations like pressing, rolling & other forming operations.

Application,Process	Tolerance (μm)	IT Grade
Slip blocks, reference gages	1-2	1
High quality gages, plus gages	2-3	2
Good quality gages, gap gages	3-5	3
Fits produced by lapping	4-10	4
Ball bearings, Diomand or fine boring, fine grinding	5-12	5
Grinding, fine honing	6-20	6
High quality turning, broaching	12-35	7
Center lathe turning and boring, reaming	14-50	8
Horizontal or vertical boring machine	30-80	9
Milling, slotting, planing, metal rolling or extrusion	50-100	10
Drilling, rough turning and boring, precision tubing	70-140	11
Light press work, tube drawing	120-240	12
Press work, tube rolling	150-500	13
Die casting or molding, rubber moulding	250-1000	14
Stamping	400-1400	15
Sand casting, flame cutting	500-2000	16

4.3 Comparing product quality of the CNC production with conventional production.

When comparing the product quality of CNC production (Computer Numerical Control) with conventional production methods, several factors come into play. Here are some key points to consider:

Accuracy and Precision: CNC production typically offers higher levels of accuracy and precision compared to conventional production methods. CNC machines can consistently reproduce complex geometries with tight tolerances, resulting in more precise and consistent parts. Conventional methods, such as manual machining or fabrication, rely on the skill and experience of the operator, which may introduce variability and limitations in achieving the desired level of precision.

Reproducibility: CNC production excels in reproducing identical parts or components with high repeatability. Once a CNC program is developed and optimized, it can be reused to produce multiple parts with consistent quality. In conventional production, reproducing identical parts may require manual setup and adjustments, which can introduce variations and increase the potential for errors.

Flexibility and Customization: CNC production offers greater flexibility and customization capabilities compared to conventional methods.

Complex Geometries: CNC production is particularly advantageous when dealing with complex geometries and intricate designs.

Consistency and Quality Control: CNC production allows for greater control over the manufacturing process, leading to consistent and reliable quality. CNC machines can be programmed to monitor and adjust parameters such as cutting speeds, feeds, and tool changes automatically. This reduces human error and ensures that parts are machined consistently

Efficiency and Productivity: CNC production is often more efficient and productive compared to conventional methods. CNC machines can operate continuously, 24/7, with minimal downtime and faster cycle times. They can also perform multiple operations in a single setup, reducing the need for manual intervention and setup changes. Conventional methods may require manual tool changes, repositioning, or additional setups, which can slow down the production process and reduce overall efficiency.

Self – Check 4

Instructions:

- Given 30 minutes, provide to complete the question .
- Enumerate the following. Use a separate sheet of paper for answering.
- For Short answers, write the answer of the questions on the provided space.

Part I. Choose correct answer of the following questions

1. Which of the following operation is performing on lathe, except

A Turning operation C. Rack gear

B. Thread operation D. grooving operation

2.CNC machine tool has coordinate systems, called

A. Machine coordinate system C. Workpiece coordinate system

B. Tool coordinate system D. A & C correct

3.Which of CNC machine Operators Must Know About Sizing Adjustments

A. Explain, demonstrate and have newcomers practice making sizing adjustment at the machine.
B. Show how to determine whether a sizing adjustment is required and, if so, its value
C. Explain, demonstrate and have newcomers practice taking measurements

4. Tool is required to move in a straight line connecting current position and final position using the code known as.

A. G- 00 C. G-03

B. G-01 D. G-02

Part II: - say True if the statement is true or else False

1. Insuring product quality is a critical in CNC operation
2. Machine calibration and maintenance is not factor of quality.
3. Tool selection can affect product quality

Book of reference

1. Smid, Peter -(2007) CNC Programming Handbook - A Comprehensive Guide to Practical CNC Programming-Industrial Press
2. proLIGHT February, 1998) corresponds to the Turning software program.
3. 2014 -Fundamentals of CNC Machining Desk Copy,
4. Theory and Design of CNC Systems

Developer's Profile

No	Name	Qualification	Field of Study	Organization/ Institution	Mobile number	E-mail
1)	Abebe Hailiye	(MSc)	Manufacturing engineering	M/G Mulugeta Buli Polytechnic College	0913521369	Aabebe52@gmaile.com
2)	Tsegaye Mulat	(MSc)	Manufacturing technology	A.A tegbaried poly technic college	910411511	Ttshgm97@gmail.com
3)	Boka Terefe	(MSc)	Manufacturing Technology	Ambo poly technic college	917038870	bokaterefe@gmaile.com
4)	Mesay Teshome	BSC	Manufacturing	Hawassa poly technic college	0905 456888	mesayy2018@gmail.com
5)	Mesret Woldemariam	MSC	Manufacturing technology	Wolkite poly technic college	912117416	bmesi2007@gmail.com
6)	Mohammed Bedri	MSC	Manufacturing Technology	Robe poly technic college	913897297	
7)	Mohammed Abdela	MSC	Manufacturing Technology	Ethio italy poly technic college	973794977	mehamed abdela 49@gmail.com