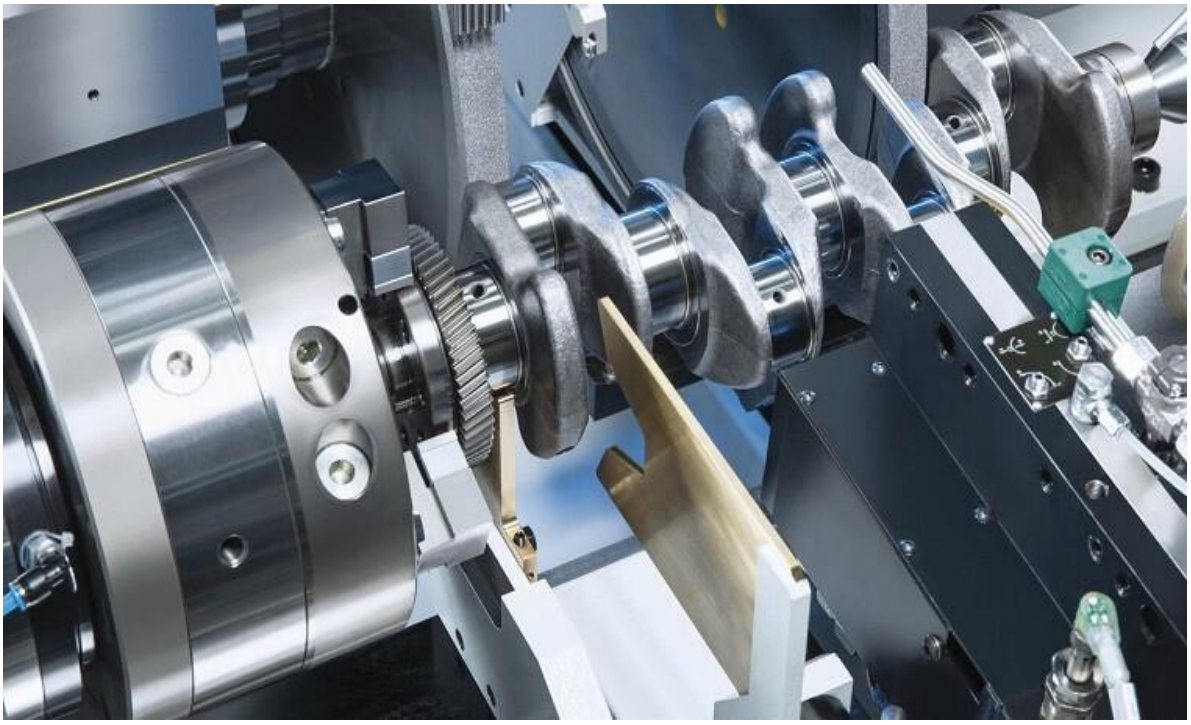
MACHINING

Level-III

Based on October 2023, Curriculum Version II

Module Title: -Perform Advanced Lathe Operations



Module code: IND MAC3 07 1221

Nominal duration: 100Hour

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.

BSW	British Standard with Worth Thread
‘ei’	lower Deviation basic size shaft
‘EI’	lower Deviation minimum limit of size hole
‘es’	Upper Deviation basic size. Shaft
‘ES’	Upper Deviation maximum limit of size hole
‘IT’	Fundamental Upper and Lower Deviation
ISO	International standard organization
NC	National Course
NF	National Fine
NS	National Special
NPT	National Pipe

Introduction to the Module

In machining filed; Lathe is one of the most versatile and widely used machine tools all over the world. It is commonly known as the mother of all other machine tool. The main function of a lathe is to remove metal from a job to give it the required shape and size. The job is securely and rigidly held in the chuck or in between centers on the lathe machine and then turn it against a single point cutting tool which will remove metal from the job in the form of chips. Lathes are manufactured in a variety of types and sizes, from very small bench lathes used for precision work to huge lathes used for turning large steel shafts. But the principle of operation and function of all types of lathes are same.

This modules cover the unite

- Fundamental concept of lathe
- Set- up the machine & work-piece
- Advanced lathe machine operations
- product quality assurance

Learning objectives of the modules

- understand Fundamental concept of lathe
- Set- up the machine & work-piece
- Perform Advanced lathe machine operations
- Inspect product quality assurance

Modules learning instruction

For effective use these 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” giver at the end of each unit and
5. Read the identified reference book for Examples and exercise

Conten

Acknowledgment ..... **ii**

Acronym. iii

Introduction to the Module iv

Unit one: Fundamental concept of lathe **1**

1.2. Basic Information of Lathe 2

1,2, Sequence of Lathe Operation 10

1.3. Lathe Cutting Tools 12

1.4. Work Shope Safety 16

Self-Check 1.1 21

Operation sheet 1.1. 23

Unit Two. Set- Up the Machine & Work-Piece **24**

2.1. Mounting And Centering Work-Piece Instruments/Equipment 25

Self-Check -2.1 37

Operation Sheet 2.1 38

Lap Test –2: Practical Demonstration 40

Unit Three. Advanced Lathe Machine Operations **41**

3.1. Cutting Parameter 41

3.2. Lathe Accessories 43

3.3. Advanced Lathe Operations 51

Self-Check -3.1 61

Operation Sheet 3.1 63

Operation Sheet 3,2 66

Operation Sheet 3.3 69

Operation Sheet 3.4 71

Operation Sheet 3. 5 73

Operation Sheet 3.6 76

Operation Sheet 3.7 79

Operation Sheet 3. 8 81
LAP Test: Practical Demonstration..... 83

Unit Four: Product Quality Assurance 84
4.1. Measuring Tools and Equipment..... 85
4.2. Checking Techniques Of Work Pieces 94
4.3. Handling Deviation 98
Self-Check -4.1..... 101
Operation Sheet 4.1. 102
Operation title: Polishing on a lathe operation of shaft 102
List Of Reference Materials..... 104

Unit one: Fundamental concept of lathe

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

- Basic information of lathe
- Sequence of lathe operation
- Lathe cutting tools
- Work Shope safety

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:

- Understand Basic information of lathe
- Determine Sequence of lathe operation
- Select Lathe cutting tools
- Identify Shope safety

1.2. Basic Information of Lathe

The lathe is one of the oldest and most important tools in human history. Its origins can be traced back to ancient civilizations, where it played a crucial role in shaping various materials, particularly wood and metal.

The earliest lathes were simple machines consisting of a rotating spindle and a tool holder. They were operated manually, with the user applying pressure to the material against the turning tool. Ancient Egyptian and Mesopotamian civilizations are known to have used lathes as early as 1300 BCE.

Over time, lathes evolved and became more sophisticated. During the Middle Ages, pedal-powered lathes appeared, allowing the operator to use both hands to manipulate the material. This innovation significantly increased efficiency and precision in turning operations.

During the Industrial Revolution in the 18th century, the lathe underwent a major transformation. The introduction of steam power and the development of complex gearing systems enabled lathes to be driven by external power sources, such as steam engines. This led to the creation of larger and more powerful lathes capable of handling heavy-duty work.

The 19th and 20th centuries witnessed significant advancements in lathe technology. The introduction of electric motors and the development of more precise controls allowed for greater precision and automation in the turning process. Additionally, the invention of specialized lathes, such as the turret lathe and the CNC (Computer Numerical Control) lathe, revolutionized industrial manufacturing by enabling mass production and complex machining operations.

Today, lathes are widely used in various industries, including metalworking, woodworking, and the production of precision components. Modern lathes feature advanced technologies like computer control systems, automatic tool changers, and multi-axis capabilities, greatly enhancing their versatility and efficiency.

The historical background of the lathe showcases its enduring significance as a fundamental tool in manufacturing and craftsmanship, contributing to the development and progress of human civilization.

A lathe is a machine tool used in machining operations to perform various cutting and shaping operations on a workpiece. It is a versatile and widely used tool in manufacturing and metalworking industries. **Here are some basic pieces of information about lathes:**

Purpose: Lathes are primarily used for turning operations, where the workpiece rotates while a cutting tool is used to remove material and shape it into the desired form. Turning operations include cylindrical turning, facing, taper turning, threading, and grooving.

Components: A lathe consists of several main components, including the bed, headstock, tailstock, carriage, cross-slide, tool post, and chuck. The bed provides a rigid base for supporting the other components. The headstock houses the main spindle that holds and rotates the workpiece. The tailstock can be positioned along the bed and is used for supporting long workpieces. The carriage moves along the bed and holds the cutting tool. The cross-slide moves perpendicular to the axis of the lathe and is used for precise positioning of the cutting tool. The tool post holds various cutting tools.

Types of Lathes: There are various types of lathes, including engine lathes, bench lathes, turret lathes, automatic lathes, and CNC lathes. Engine lathes are the most common type and are manually operated. Bench lathes are smaller and designed for use on a workbench. Turret lathes have multiple tool stations and can perform a variety of operations without manual tool changes. Automatic lathes are capable of performing multiple operations automatically. CNC (Computer Numerical Control) lathes are controlled by computer programs and can perform complex operations with high precision.

Workpiece Size: Lathes are available in various sizes to accommodate different workpiece sizes. The size of a lathe is typically specified by its swing capacity, which is the maximum diameter of the workpiece that can be accommodated over the bed. The distance between the headstock and tailstock, known as the “between centers” distance, determines the maximum length of the workpiece that can be turned.

Safety Considerations: When operating a lathe, it is important to follow safety guidelines. Operators should wear appropriate personal protective equipment, such as

safety glasses or goggles. They should also be familiar with the lathe's controls, emergency stop features, and safety interlocks. Proper tooling and workpiece clamping techniques should be used to prevent accidents and ensure stability during machining operations.

These are some basic aspects of lathes. However, lathes can be quite complex machines with a wide range of features and capabilities depending on their type and configuration.

Basic Information of Lathe

Drawings Interpretation

Drawing interpretation means reading and understanding the information given by the drawing about the component to be manufactured. The information includes the shape, surface condition, dimension, number of parts (work piece) and the type of material to be used. It provides the following advantages.

- Easily inspect and select the rough size of the work stock black.
- Recognized the surface finish expected.
- For plan the machining sequence operation technically.

Views and projection

projection is a shadow and view are an observation of an object in a different angle that is nothing but while drawing an object on a paper view is a shadow of an object in different angles.

- A view of an object is known technically as a projection.
- A projection is an object which is three dimensional objects on a two-dimensional surface or plane.
- A projection is a view conceived to be drawn or projected on to a plane, known as the plane of projection.
- Multi-view drawings are convectional projections of a three-dimensional object on a two-dimensional plane.
- Multi-view or orthographic projection is a system of views of an object formed by projectors from the object perpendicular to the desired plane of projection.

- Perpendicular lines or projectors are drawn from all points on the edges or contours of the object to the plane of projection.

There are three planes perpendicular to each other, which are the basis for multi-view projection. These are:

- Horizontal projection plane.
- Frontal projection plane.
- Profile projection plane.

The position of the plane is illustrated on the figure below

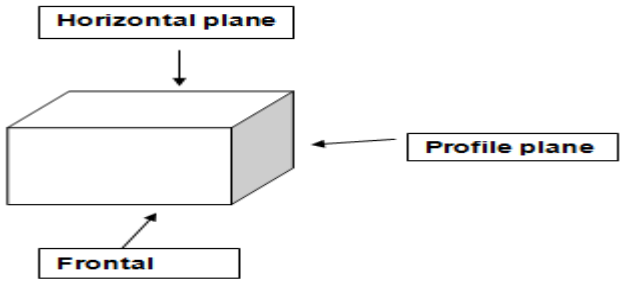


Figure.1.1. Planes of projection

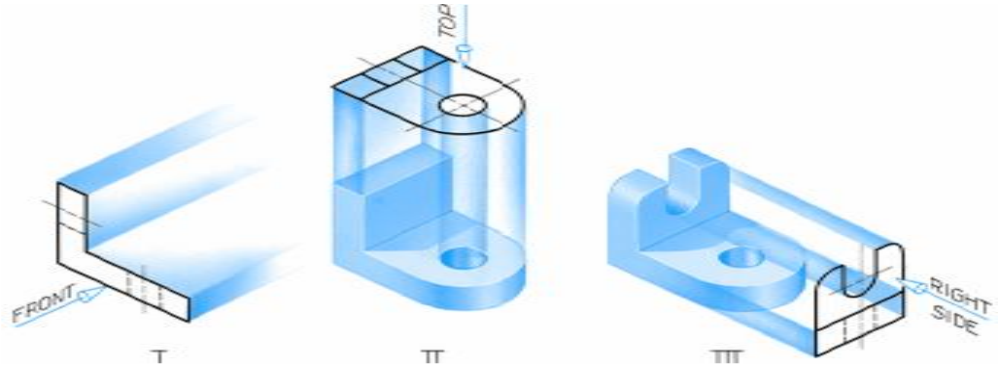


Figure.1.2. Planes of projection

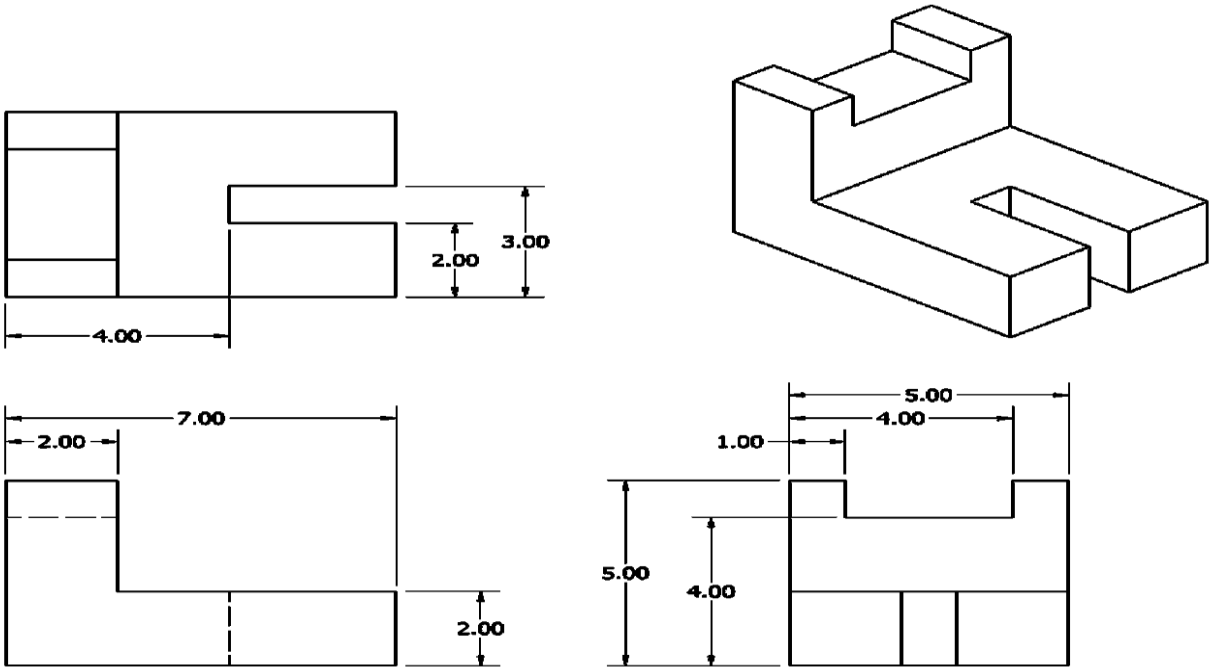


Figure.1.3. views using 3rd angle projection

Drawing symbols

The drawings for a component are composed largely of symbols and conventions representing its shape and material. Symbols are the shorthand of drawing. They graphically portray the characteristics of a component with a minimal amount of drawing.

Engineering Drawing Symbols

Basic types of symbols used in engineering drawings are countersink, counterbore, spot face, depth, radius, and diameter. Here are more commonly used engineering drawing symbols and design elements as below. You can also check out the GD&T symbols and terms on our site.

SYMBOL FOR :	ANSI Y14.5
STRAIGHTNESS	—
FLATNESS	▱
CIRCULARITY †	○
CYLINDRICITY †	⌭
PROFILE OF A LINE	⌒
PROFILE OF A SURFACE	⌓
ALL AROUND–PROFILE	⌓
ANGULARITY	∠
PERPENDICULARITY	⊥
PARALLELISM	//
POSITION	⌖
CONCENTRICITY/COAXIALITY †	◎
SYMMETRY ‡	NONE
CIRCULAR RUNOUT	↗
TOTAL RUNOUT	⌰
AT MAXIMUM MATERIAL CONDITION	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ
REGARDLESS OF FEATURE SIZE	Ⓢ
PROJECTED TOLERANCE ZONE	Ⓟ

* MAY BE FILLED IN

SYMBOL FOR :	ANSI Y14.5
DIAMETER	⌀
BASIC DIMENSION	50
REFERENCE DIMENSION	(50)
DATUM FEATURE	-A-
DATUM TARGET	Ø A1
TARGET POINT	×
DIMENSION ORIGIN	⌖
CONICAL TAPER	⋈
SLOPE	⟋
COUNTERBORE/SPOTFACE	⌴
COUNTERSINK	⌵
DEPTH/DEEP	↧
SQUARE (SHAPE)	□
DIMENSION NOT TO SCALE	15
NUMBER OF TIMES/PLACES	8X
ARC LENGTH	105
RADIUS	R
SPHERICAL RADIUS	SR
SPHERICAL DIAMETER	SØ

† Not recommended, use runout
‡ Not used

Figure 1.4. tolerancing symbol

Dimensions and features

Generally speaking, the dimensions shown in a detailed drawing are the dimensions that will be needed to produce the object. Assembly drawings may contain no dimensions, or only those dimensions critical to the assembly or operation of the item. Care should also be taken not to 'over-dimension' the object. If a dimension is shown twice on a drawing, confusion arises in the factory or on site as to which dimension to use, especially if a change is made to one of the dimensions.

As with any other part of a drawing, dimensions must be placed on the drawing in accordance with the current drawing standards AS100.

The basic rules of dimensioning are:

Dimensions for the size of a component's features should be placed on the drawing once only.

Dimensions should be read from the bottom and right side of the drawing sheet.

Placing the dimension on the centerline, on the outline or inside the outline of the object should be avoided. Dimensions should be placed away from the object be using extension

Dimensions are placed above the dimension

lines from the points to be dimensioned. Features of Size should be located by basic dimensions. The size should be specified with coordinate dimensions, and the tolerance for location should be specified with Tolerance of position.

Below is an example of how to dimension features of size. In this case, we have an assembly that includes buttons and a cover. The buttons and the holes in the cover are all dimensioned relative to the same datums. Those datums apply to the assembly itself, so anything within the assembly can be located to those datums.

The holes and the buttons are patterns, so in each case we can show the tolerance once and use the 4X to indicate that the dimension applies in all four places. For clarity, we add a note under each tolerance to state explicitly whether it applies to holes or buttons.

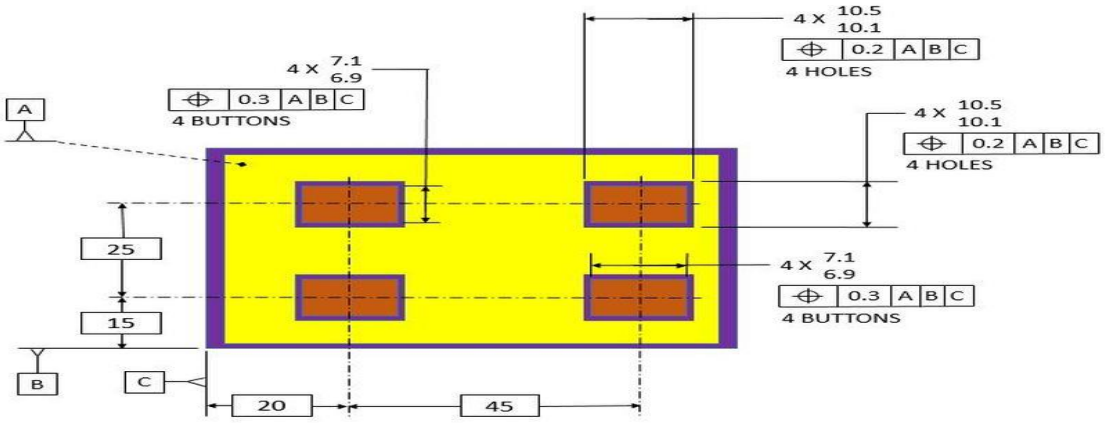


Figure 1.5. dimenssioning

Tolerances and fit

Machining tolerance is defined as the total amount a specific dimension is permitted to vary from the given value. This can be expressed in a few different ways: The upper and lower limits (e.g., 0.2500, 0.2498) The allowable amount above and below a defined dimension (e.g., 0.2499 ±0.0001)

Tolerances is the ability or willingness to tolerate the existence of opinions or behavior that one dislikes or disagrees with.

It is the capacity to endure continued subjection to something such as a drug or environmental conditions without adverse reaction.

Total amount by which a specific dimension is permitted to vary • Difference between the maximum and the minimum limits for the dimension. • For Example, a dimension given as 1.625 ± .002 means that the manufactured part may be 1.627" or 1.623", or anywhere between these limit dimensions.

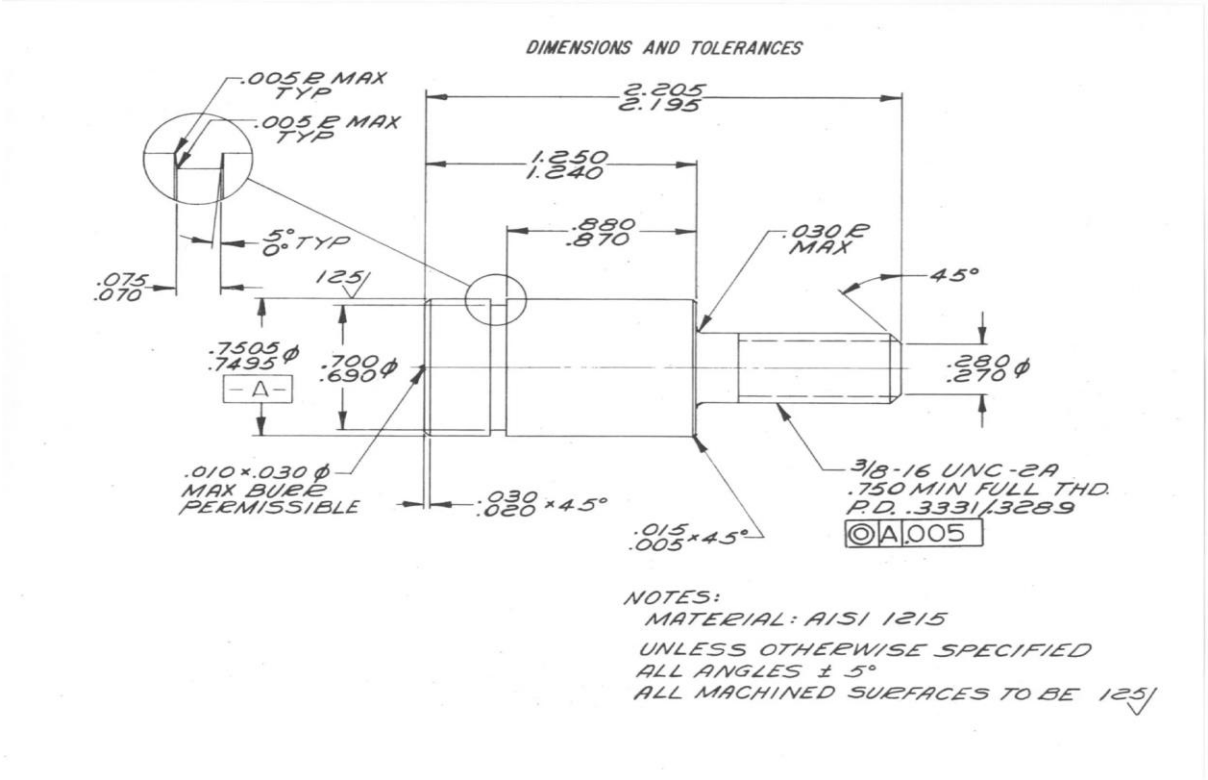


Figure 1.6. dimenssioning and tollerancing

Fit

It is a suitable quality, standard, or type to meet the required purpose.

in good health, especially because of regular physical exercise.

be of the right shape and size for.

the particular way in which something, especially a garment or component, fits.

Engineered products sometimes come as components that must slip or press against each other to deliver their functions. Therefore, a fit is used to describe these dimensional relationships between the components. It is used to determine whether

the components are loose or tight which aids in slipping or pressing property. Understanding what a fit comes with understanding certain terms, which are explained below.

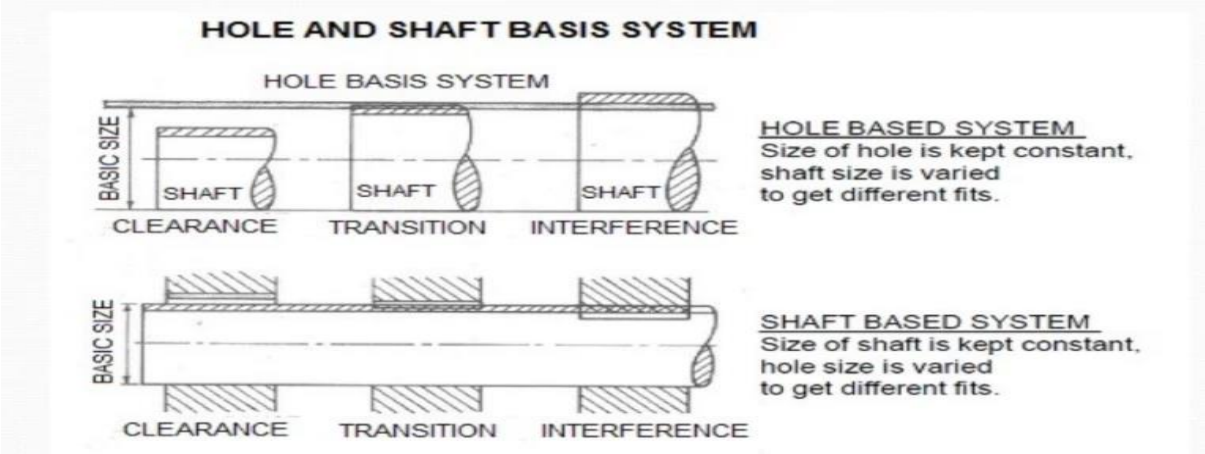


Figure 1.7. fitting

1,2, Sequence of Lathe Operation

The sequence of lathe operations may vary depending on the specific task and workpiece being machined. However, here is a general sequence of operations commonly followed when using a lathe:

Select the workpiece: Choose the appropriate workpiece material and dimensions according to the desired final product.

Mount the workpiece: Securely mount the workpiece in the lathe's chuck or between centers, ensuring it is properly aligned and centered.

Set the cutting tools: Select the appropriate cutting tools for the desired operation, such as turning, facing, threading, or grooving. Install and secure the cutting tool in the lathe's tool post.

Adjust the tool position: Position the cutting tool at the desired starting point on the workpiece. Adjust the tool height, angle, and position as necessary to achieve the desired cutting action.

Set the cutting parameters: Determine the appropriate cutting speed, feed rate, and depth of cut based on the workpiece material and the operation being performed. These parameters may vary depending on the specific lathe and cutting tool being used.

Start the lathe: Turn on the lathe and adjust the speed to the desired level. Ensure that the

lathe is rotating in the correct direction.

Perform the operation: Engage the cutting tool with the workpiece and gradually feed it into the material. Monitor the cutting process, making adjustments as needed to maintain proper cutting conditions.

Check the dimensions: Periodically measure the dimensions of the workpiece using appropriate measuring tools, such as calipers or micrometers. Ensure that the workpiece is being machined to the required specifications.

Repeat or change operations: Depending on the complexity of the part, you may need to repeat the same operation multiple times or change to different cutting tools to perform additional machining operations.

Finish the operation: Once the desired dimensions and surface finish are achieved, disengage the cutting tool from the workpiece and stop the lathe.

Inspect the final product: Carefully inspect the machined part to ensure it meets the required specifications and desired quality.

It's important to note that this is a general sequence of lathe operations and may vary depending on the specific machining task and the operator's experience. Safety precautions should always be followed, and the manufacturer's guidelines for the specific lathe model should be consulted.

1.3. Lathe Cutting Tools

Lathe cutting tools are used to remove material from a workpiece during turning operations. There are various types of lathes cutting tools designed for specific tasks and materials. Here are some common lathes cutting tools:

Turning Tool: Also known as a lathe bit or tool bit, a turning tool is the most commonly used cutting tool in a lathe. It has a cutting edge that is ground to a specific shape, such as a square, rectangular, or triangular profile. Turning tools come in various sizes and are used for general-purpose turning operations.

Facing Tool: A facing tool has a wide cutting edge and is used to create a flat surface at a right angle to the workpiece axis. It is commonly used to remove material from the end face of a workpiece or to create a smooth and flat surface.

Parting Tool: A parting tool is used to cut off a workpiece from the main stock. It has a narrow cutting edge and is designed to create a clean and precise cut. Parting tools are commonly used for separating finished parts or creating grooves.

Boring Tool: Boring tools are used to enlarge existing holes or create internal cylindrical features in a workpiece. They have a cutting edge on the tip and can be adjusted to achieve the desired diameter and depth.

Threading Tool: Threading tools are used to cut external or internal threads on a workpiece. They have a specific profile that matches the thread pitch and angle. Threading tools can be used for creating various thread types, such as V-threads or metric threads.

Grooving Tool: Grooving tools have a thin and wide cutting edge and are used to create grooves or recesses on a workpiece. They are commonly used for creating internal or external grooves, such as for O-rings or snap rings.

Knurling Tool: Knurling tools are used to create a knurled pattern on the surface of a workpiece. The knurled pattern provides a better grip or aesthetic appearance. Knurling tools have two cutting wheels that press into the workpiece surface to create the pattern.

Threading Insert: Threading inserts are replaceable cutting tools used in CNC lathes or indexable toolholders. They have a specific profile for cutting threads and can be easily replaced when worn out.

These are some of the common lathes cutting tools used in turning operations. The selection of the cutting tool depends on factors such as the type of operation, workpiece material, and desired outcome. It's important to choose the appropriate cutting tool and ensure it is properly sharpened and secured in the tool post for safe and effective machining.

Tool Grinding operations

Grinding wheels are expendable wheels composed of an abrasive compound. A Grinding wheel has multiple cutting edges made up of many hard a particle called abrasives. The abrasives are crushed to have sharp edges for cutting operations.



Figure 3.1. grinding wheels

The abrasive grains are properly mixed with is a suitable bond, which acts as a holder when the wheel is in use. The grinding wheel may be manufactured in one piece or segments of oppressive blocks build up into a solid wheel. These wheels are available in different shapes and mounted on different forms of Machines for a particular type of work. Grinding wheels are manufactured with higher geometrical accuracy resulting in a more uniform and consistent wheel.

Abrasive

Abrasive is a material that helps to provide a shiny look on a surface. Abrasives are of two types

1. Natural Abrasives (Diamond, Quartz, Sand)
2. Artificial Abrasive (Synthetic diamond, Tin Oxide, Aluminum oxide, Silicon Carbide)

Grinding wheel abrasives types





Diamond Grinding Wheels

In diamond wheels, industrial diamonds remain bonded to the edge. This is used to grind hard materials like concrete, gemstones & carbide tips. A slitting saw is designed for slicing gemstones like hard materials.



Figure 3.2. Diamond Grinding Wheels

silicon grinding

Fine grinding of silicon wafers refers to the grinding operations with #2000 mesh (36 μm grit size) or finer diamond wheels. The wafer surfaces to be fine-ground generally have no damage or very little damage and the surface roughness is less than 0.03 μm in Ra.

Aluminum oxide is generally used for grinding:

- Carbon steel
- Alloy steel
- High speed steel
- Wrought iron
- Hard bronzes

Silicon carbide is generally used for grinding:

- Grey iron
- Nonferrous metals
- Stone
- Marble

1.4. Work Shope Safety

1.4.1. general safety guidelines for lathe machine

Operating a lathe machine safely is of utmost importance to prevent accidents and ensure the well-being of operators. Here are some general safety guidelines to follow when working with a lathe machine:

Familiarize Yourself with the Machine: Before operating the lathe machine, thoroughly read and understand the manufacturer's operating manual and safety instructions. Familiarize yourself with the machine's controls, emergency stop features, and safety interlocks.

Wear Appropriate Personal Protective Equipment (PPE): Always wear the necessary PPE, such as safety glasses or goggles, a face shield, ear protection, and safety shoes. Depending on the specific task, additional PPE like gloves or a dust mask may be required.

Keep the Work Area Clean and Organized: Ensure that the work area is clean, well-lit, and free from clutter. Keep the floor clear of tools, debris, and obstructions that may cause slips, trips, or falls.

Securely Mount the Workpiece: Use proper clamping techniques to securely mount the workpiece in the lathe. Make sure it is properly centered and aligned to prevent it from becoming loose during machining.

Check for Damaged Tools: Inspect cutting tools for any signs of damage, such as chipped or dull edges. Replace damaged tools before using them to prevent accidents or poor machining results.

Set the Correct Speed and Feed Rates: Ensure that the lathe is set to the appropriate speed and feed rate for the specific material and operation being performed. Consult machining guides or refer to the lathe's manual for recommended cutting parameters.

Approach the Workpiece Cautiously: When starting the lathe, approach the rotating workpiece cautiously to avoid sudden contact. Keep hands, clothing, and jewelry away from the rotating parts at all times.

Avoid Loose Clothing and Jewelry: Wear fitted clothing and avoid loose sleeves, ties, or jewelry that could get caught in the rotating parts of the lathe.

Use Proper Cutting Techniques: Follow proper cutting techniques and avoid aggressive cuts or excessive force that could cause tool breakage or workpiece displacement. Take light passes and gradually increase cutting depth or feed rates as needed.

Regularly Inspect the Machine: Conduct regular inspections of the lathe machine to ensure all components, such as belts, pulleys, and guards, are in good working condition. Report any abnormalities or malfunctions to the appropriate personnel.

Seek Training and Supervision: Obtain proper training in lathe machine operation and seek guidance from experienced operators. If you are new to lathe machining, work under the supervision of an experienced mentor until you gain sufficient expertise.

Remember, these guidelines provide a general overview of lathe machine safety. Always adhere to the specific safety recommendations provided by the lathe manufacturer and your workplace.

1.4.2. lathe shop safety practices

When working in a lathe shop, it is crucial to follow safety practices to minimize the risk of accidents and ensure a safe working environment. Here are some important lathe shop safety practices to consider:

Personal Protective Equipment (PPE): Always wear the appropriate personal protective equipment, including safety glasses or goggles, a face shield, hearing protection, and safety shoes. Use gloves when handling sharp tools or materials, and wear a dust mask if necessary.

Training and Knowledge: Ensure that all personnel operating the lathe machines have received proper training on lathe safety and operation. Familiarize yourself with the specific lathe machines in the shop, including their controls, emergency stop features, and safety interlocks.

Good Housekeeping: Keep the lathe shop clean, organized, and free from clutter. Clear the work area of any unnecessary tools, debris, or obstructions that could cause slips, trips, or falls. Maintain proper lighting in the shop to ensure good visibility.

Machine Inspection and Maintenance: Regularly inspect the lathe machines for any signs of wear, damage, or malfunction. Ensure that all guards and safety devices are

in place and functioning correctly. Follow the manufacturer's recommended maintenance schedule and promptly report any issues to the appropriate personnel.

Proper Tool Handling: Handle cutting tools with care and use the appropriate tool holders or collets when mounting them on the lathe machines. Inspect tools for damage or wear before use, and replace or repair them as needed.

Safe Workpiece Handling: Securely mount workpieces in the lathe machines using appropriate clamping methods. Make sure the workpiece is centered and aligned properly to prevent it from becoming loose during machining.

Safe Start-Up and Shutdown Procedures: Follow the recommended start-up and shutdown procedures for the lathe machines. Ensure that the work area is clear of any obstructions or hazards before starting the machines, and never leave the machines unattended while they are running.

1.4.3. Safe Machining Practices:

- Use proper cutting techniques and avoid aggressive cuts or excessive force.
- Take light passes and gradually increase cutting depth or feed rates as needed.
- Keep hands, clothing, and jewelry clear of rotating parts at all times.
- Avoid reaching over or across the lathe's spinning components.

Emergency Preparedness: Know the location of emergency stop buttons, fire extinguishers, first aid kits, and emergency exits. Understand the emergency procedures for evacuating the lathe shop and responding to accidents or equipment malfunctions.

Continuous Monitoring and Communication: Regularly monitor the lathe machines during operation for any signs of abnormal behavior or unusual sounds. Encourage open communication among shop personnel to report any safety concerns or incidents.

Remember, these are general lathe shop safety practices. Always follow the specific safety guidelines provided by your workplace and prioritize safety at all times. Regularly review and reinforce safety practices to create a culture of safety in the lathe shop.

1.4.4. First Aid and Fire Extinguisher

A. First Aid:

Having basic knowledge of first aid is crucial in any workplace, including a lathe machine shop. Here are some essential first aid practices to keep in mind:

Assess the Situation: Evaluate the situation and ensure it is safe for you to approach and provide assistance. If there are any hazards present, such as moving machinery or electrical dangers, address those first or seek help from a qualified person.

Call for Help: If the injury is serious or life-threatening, immediately call for emergency medical assistance. Provide clear and concise information about the location, nature of the injury, and any other relevant details.

Control Bleeding: If there is bleeding, apply direct pressure using a clean cloth or sterile dressing. Elevate the injured area if possible. Maintain pressure until medical help arrives or the bleeding stops.

tabilize Fractures and Injuries: If a fracture or serious injury is suspected, avoid moving the injured person unless necessary to prevent further harm. Immobilize the injured area using splints or other available materials to minimize movement.

Provide Comfort and Reassurance: Stay with the injured person and provide comfort and reassurance while waiting for medical help to arrive. Keep them calm and monitor their condition.

B. Fire Extinguisher:

In a lathe machine shop, fire safety is crucial. Understanding how to use a fire extinguisher correctly is essential in the event of a fire. **Follow these general steps:**

Evaluate the Situation: Before attempting to use a fire extinguisher, assess the fire's size and ensure it is safe for you to extinguish it. If the fire is large, spreading rapidly, or you are unsure about using the extinguisher, evacuate the area immediately and call the fire department.

Remember the PASS Technique: The PASS technique is a common method for using a fire extinguisher effectively:

Pull the pin: Pull the pin located at the top of the extinguisher to break the seal.

Aim at the base: Aim the nozzle or hose at the base of the fire, not at the flames.

Squeeze the handle: Squeeze the handle to release the extinguishing agent.

Sweep from side to side: Sweep the extinguisher from side to side, covering the base of the fire with the extinguishing agent.

Maintain a Safe Distance: Stand at a safe distance from the fire and use the extinguisher within its effective range. Do not approach the fire too closely, as it may be unsafe or cause the fire to spread.

Evacuate if Necessary: If the fire becomes unmanageable, grows rapidly, or you are unable to extinguish it, evacuate the area immediately and sound the fire alarm if available. Alert others and follow the established evacuation procedures.

Call the Fire Department: Even if you have successfully extinguished a small fire, it is important to call the fire department to ensure the area is safe and to receive professional assistance.

Remember, it is crucial to have proper training in first aid and fire safety. Consult your workplace's specific safety guidelines, receive certified training if available, and always prioritize your personal safety when responding to emergencies.

Self-Check 1.1

Directions: I. Choose the best answer for the for the given alternative

1. which is the following being NOT lathe main components (1 point)

   A. headstock,
   B. tailstock,
   C. lead screw
   D. toolpost

2. _____is material that helps to provide a shiny look on a surface. (1 point)

   A. Abrasive
   B. ingot
   C. Tool steel
   D. silver

3. one of the followings are NOT Basic types of used in engineering drawings symbols are

   A. countersink,
   B. counterbore,
   C. diameter
   D. Datum

4. Silicon carbide is generally used for grinding:

   A. Grey iron
   B. Nonferrous metals
   C. Marble
   D. all

Direction 2, Matching Column A With Column B (1 Point for each question)

Column ``A``	**Column ``B``**
1. Evaluate the Situation	A. First Aid
2. Control Bleeding	B. Fire Extinguisher
3. Familiarize Yourself with the Machine	C. safety practices
4. Good Housekeeping	D. Sequence of lathe operation
5. Select the workpiece	E. general safety guidelines
6. Turning Tool	F. lathe cutting tool

Directions: III. Write if the statement is correct say True or if the statement wrong say False

1. Grinding wheels are expendable wheels composed of an abrasive compound .
2. A projection is a view conceived to be drawn or projected on to a plane, known as the plane of projection
3. Lathe cutting tools are used to remove material from a workpiece during turning operations
4. Operators should wear appropriate personal protective equipment, such as safety glasses or goggles is not mandatory

5. Boring tools are used to enlarge existing holes or create internal cylindrical features in a workpiece.
6. Machining tolerance is defined as the total amount a specific dimension is permitted to vary from the given value.

Operation sheet 1.1.

Operation title 1.1 procedures for setting single point cutting tools

Purpose: To practice and demonstrate the knowledge and skill required in selecting and setting cutting tools.

Instruction: Use the given figure below , the tools and equipment and then follow given steps. For this operation you have given 3Hour and you are expected to show the shape on the Lathe machine given computer.

Tools and requirement:

- Grinding machine
- Lathe machine
- HSS cutting tools

Quality Criteria: the cutting tools must be align with the lathe center

Precautions: check and save all activity

Procedures in doing the task

Steps 1- Apply safety precaution.

Steps 2- grinding HSS cutting tool.

Steps 3- Mount the cutting tool in the tool post.

Steps 4- Mount the leave center in the tail stock.

Steps 5- Adjust the tool bit touches the tip of leave center in the tail stock

Unit Two. Set- Up the Machine & Work-Piece

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

- Mounting and Centering work-piece and tools,
  - set up instruments/equipment
  - Material selection

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:

- Mount and Centering work-piece and tools,
  - set up instruments/equipment
  - Select material

2.1. Mounting And Centering Work-Piece Instruments/Equipment

2.1.1. mounting and centering work piece

All jobs have to be securely clamped in the chuck and centered before any of the above listed operations can be performed on a lathe. 3-jaws chuck is a self centering device and is used for clamping round bar set. A four jaws chuck is for clamping irregularly shaped jobs. In 4-jaws chuck each jaw moves in radially independent of other jaws. Centering means that the centre line of the work piece should nearly coincide with centre line of machine spindle. It is not enough to hold the job centrally in the chuck, the portion of work piece projecting out of chuck should also be centrally placed. Collect chuck, face plate set. Are some other holding devices for the work piece?

A. Mounting Work pieces in Chucks

When installing the chuck or any attachment that screws onto the lathe headstock spindle, the threads and bearing surfaces of both spindle and chuck must be cleaned and oiled. In cleaning the internal threads of the chuck, a spring thread cleaner is very useful Turn the spindle so that the key is facing up and lock the spindle in position. Make sure that the spindle and chuck taper are free of grit and chips. Place the chuck in position on the spindle. Engage the draw nut thread and tighten by applying four or five hammer blows on the spanner wrench engaged with the draw nut. Rotate the spindle 180°, engage the spanner wrench, and give four or five solid hammer blows to the spanner wrench handle. The work piece is now ready for mounting. Work automatically centers itself in the universal (3 jaw) scroll chuck, drill chuck, collets chucks, and step chuck, but must be manually centered in the independent (4 jaw) chuck. To center work in the independent chuck, line the four jaws up to the concentric rings on the face of the chuck, as close to the required diameter as possible.

Mount the work piece and tighten the jaws loosely onto the work piece (Figure7). Spin the work piece by hand and make approximate centering adjustments as needed, then firmly tighten the jaws.

For rough centering irregularly shaped work, first measure the outside diameter of the work piece, then open the four jaws of the chuck until the work piece slides in. Next tighten each opposing jaw a little at a time until the work piece is held firmly, but not too tightly. Hold a piece of chalk near the work piece and revolve the chuck slowly with your left hand. Where the chalk touches is considered the high side.

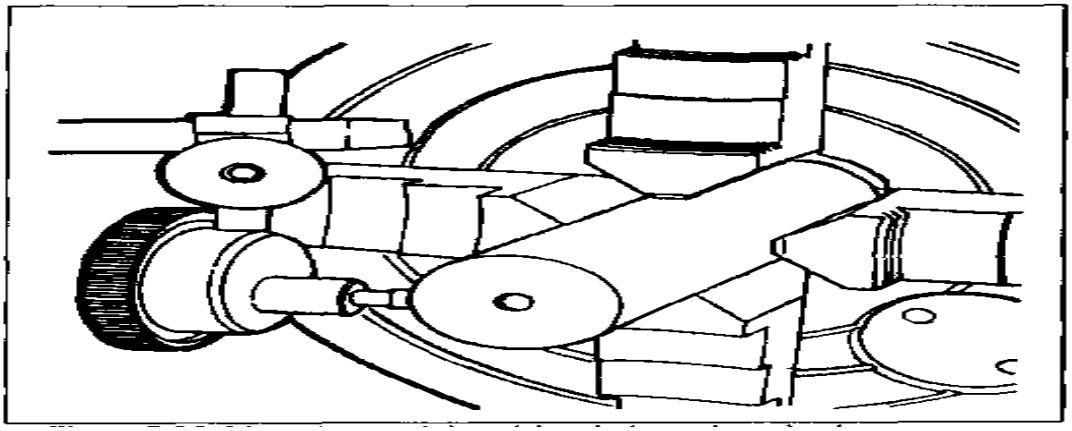


Figure 2.1. Mounting work in a 4-jaw independent chuck

To center a work piece having a smooth surface such as round stock, the best method is to use a dial test indicator. Place the point of the indicator against the outside or inside diameter of the work piece. Revolve the work piece slowly by hand and notice any deviations on the dial. This method will indicate any inaccuracy of the centering in thousandths of an inch.

If an irregularly shaped work piece is to be mounted in the independent chuck, then a straight, hardened steel bar can be used with a dial indicator to align the work piece. Experienced machinists fabricate several sizes of hardened steel bars, ground with a 60° point, that can be mounted into the drill chuck of the tailstock spindle and guided into the center- punched mark on the work piece. A dial indicator can then be used to finish aligning the work piece to within 0.001 inch. If a hardened steel bar is not readily available, a hardened center mounted in the tailstock spindle may be used to align the work while using a dial indicator on the chuck jaws. This method is one of several ways to align a work piece in an independent chuck. Ingenuity and experience will increase the awareness of the machine operator to find the best method to set up the work for machining.

B. Mounting Work to Faceplates

Mount faceplates in the same manner as chucks. Check the accuracy of the faceplate surface using a dial indicator, and true the-faceplate surface by taking a light cut if necessary. Do not use faceplates on different lathes, since this will cause excessive wear of the faceplate due to repeated truing cuts having to be taken. Mount the work piece using T-bolts and clamps of the correct sizes (Figure 8). Ensure all surfaces are wiped clean of burrs, chips, and dirt.

When a heavy piece of work is mounted off center, such as when using an angle plate, use a counterweight to offset the throw of the work and to minimize vibration and chatter. Use paper or brass shims between the work and the faceplate to protect the delicate surface of the faceplate. After mounting the work to an approximate center location, use a dial indicator to finish accurate alignment.

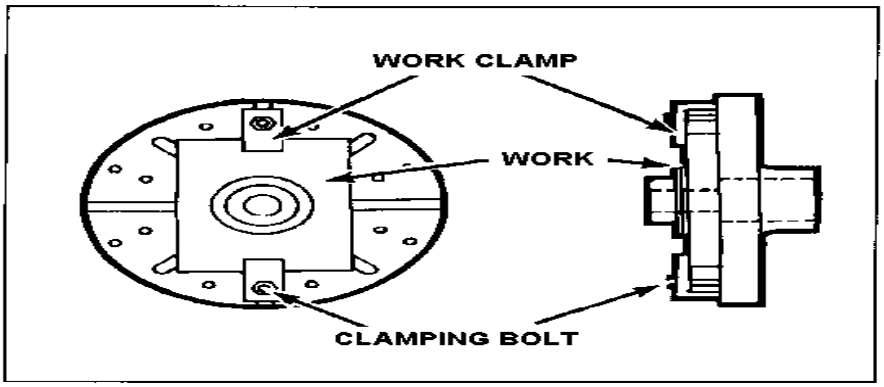


Figure 2.2. Mounting Work to Faceplates

C. Mounting Work Between Centers

Before mounting a work- piece between centers, the work piece ends must be center- drilled and countersunk. This can be done using a small twist drill followed by a 60° center countersink or, more commonly, using a countersink and drill (also commonly called a center drill). It is very important that the center holes are drilled and countersunk so that they will fit the lathe centers exactly. Incorrectly drilled holes will subject the lathe centers to unnecessary wear and the work piece will not run true because of poor bearing surfaces. A correctly drilled and countersunk hole has a uniform 60° taper and has clearance at the bottom for the point of the lathe center. Figure 9 illustrates correctly and incorrectly drilled center holes. The holes should have a polished appearance so as not to score the lathe centers. The actual drilling and countersinking of center holes can be done on a drilling machine or on the lathe itself. Before attempting to center drill using the lathe, the end of the work piece must be machined flat to keep the center drill from running off center.

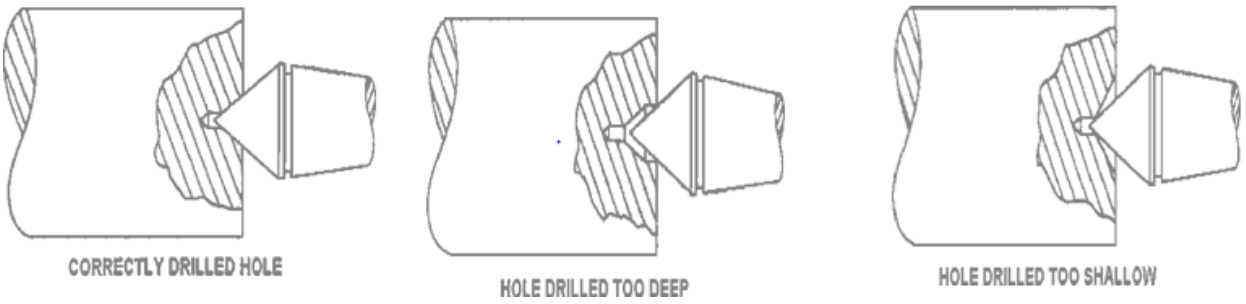


Figure 2.3. Correctly and incorrectly drilled hole

Mount the work in a universal or independent chuck and mount the center drill in the lathe tailstock (Figure 10). Refer to the section of this chapter on facing and drilling on the lathe, prior to doing this operation. Center drills come in various sizes for different diameters of work. Calculate the correct speed and hand feed into the work piece. Only drill into the work piece about 2/3 of the body diameter. High speeds and feed them into the work slowly to avoid workmanship depends as much on the condition of the lathe breaking off the drill point inside the work. If this happens, centers as on the proper drilling of the center holes. Before the work must be removed from the chuck and the point mounting lathe centers in the headstock or tailstock, extracted. This is a time-consuming job and could ruin the thoroughly clean the centers, the center sleeve, and the tapered work piece. Sockets in the headstock and tailstock spindles. Any dirt or chips on the centers or in their sockets will prevent the centers from seating properly and will cause the centers to run out of true.

Install the lathe center in the tailstock spindle with a light twisting motion to ensure a clean fit. Install the center sleeve into the headstock spindle and install the lathe center into the center sleeve with a light twisting motion.

To remove the center from the headstock spindle, hold the pointed end with a cloth or rag in one hand and give the center a sharp tap with a rod or knockout bar inserted through the hollow headstock spindle.

To remove the center from the tailstock, turn the tailstock hand wheel to draw the tailstock spindle into the tailstock. The center will contact the tailstock screw and will be bumped loose from its socket.

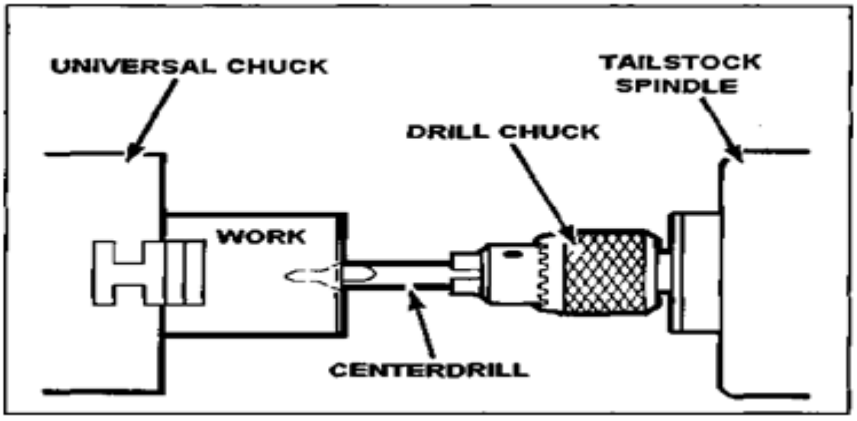


Figure 2.4. center drilling

D. Mounting Work on Mandrels

To machine a work piece of an odd shape, such as a wheel pulley, a tapered mandrel is used to hold and turn the work. The mandrel must be mounted between centers and a drive plate and lathe dog must be used. The centers must be aligned and the mandrel must be free of burrs. Mount the work piece onto a lubricated mandrel of the proper size by
using an arbor press. Ensure that the lathe dog is secured to the machined flat on the end of the mandrel and not on the smooth surface of the mandrel taper). If expansion
bushings are to be used with a mandrel, clean and care for the expansion bushings in the same manner as a normal mandrel. Always feed the tool bit in the direction of the large end of the mandrel, which is usually toward the headstock end, to avoid pulling the work out of the mandrel. If facing on a mandrel, avoid cutting into the mandrel with the tool bit.

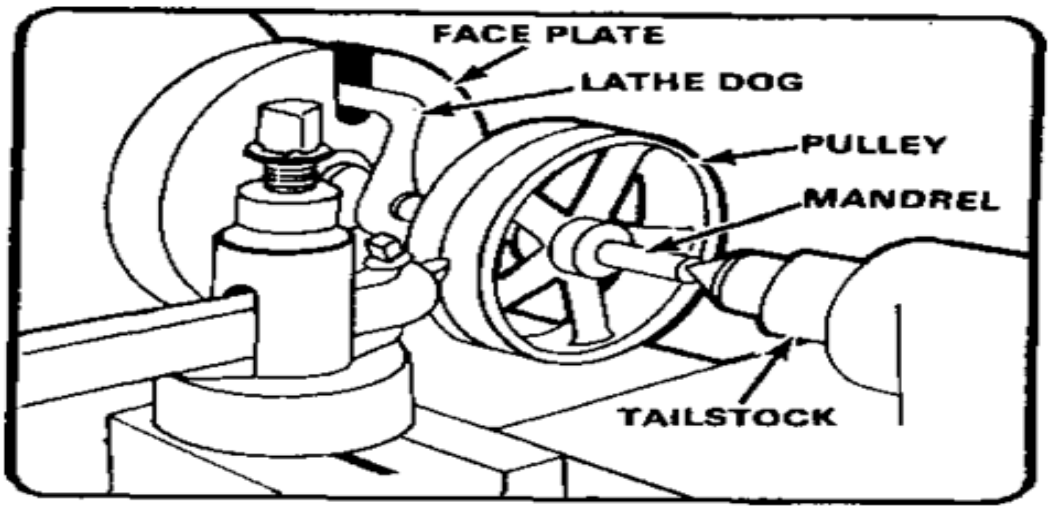


Figure 2.5. Pulley mounted on a mandrel

If the work piece is to be mounted between centers, check the alignment of the headstock center with the tailstock center (live or dead center) and make the necessary changes as needed. Ensure that the tool holder and the cutter bit are set at the proper height and angle. Check the work holding accessory to ensure that the work piece is held securely. Use the center rest or follower rest to support long work pieces.

The first step in preparing the centers is to see that they are accurately mounted in the headstock and tailstock spindles. The centers and the tapered holes in which they are fitted must be perfectly clean. Chips and dirt left on the contact surfaces will impair accuracy by preventing a perfect fit of the bearing surfaces. Be sure that there are no burrs in the spindle hole. If burrs are found, they must be removed by carefully scraping or reaming the surface with a Morse taper reamer. Burrs will produce the same inaccuracies as chips or dirt.

Center points must be accurately finished to an included angle of 60° Check the angle of the center with the use of a center gage. If the test shows that the point is not perfect, true the point in the lathe by taking a cut over the point with the compound rest set at 30°. The hardened tailstock center must be annealed before it can be machined or set up in the lathe and ground with the use of a tool post grinder.

Figure 2.6a. Centering Work in a 4-Jaw Chuck Figure 2.7.b. Centering Work in a 4-Jaw Using by surface gage Using by dial indicator

The surface gauge is usually used with a V Block and angle plate as well. To find the Centre place your round section steel object in a V block. The surface gauge is them moved across the surface of the steel, scribing a line. The steel is then rotated through 90 degrees and another line is scribed.

A surface gage is used for many purposes, but is most often used for layout work. The gage can be used to scribe layout lines at any given distance parallel to the work surface.

The rocker adjusting screw provides for finer adjustment of the spindle by pivoting the spindle rocker bracket. The scriber can be positioned at any height and in any desired direction on the spindle by adjusting the scriber. A surface plate and combination square are needed to set the surface gage to the correct dimension.

Hardened and ground base Ground “V” groove in base for mounting on cylindrical work Retractable pins provided in the base for work against edge of flat surface Double ended scriber with hardened points Fine adjustment of spindle may be made at any angle above or below the base The spindle may be adjusted to any position with respect to the base and tightened in place with the spindle nut.

SIZE

- Surface gage [9″]
- Steel ruler [6″ 12″ 24″] – 3 pic set.
- Weight: 1Kg. (Approx)

2.1.2. Setup instrument /equipment

Setting up a lathe requires several instruments and equipment to ensure proper operation and safety. Here are some essential tools and equipment you'll need for lathe setup:

Lathe Machine: This is the main equipment used for shaping and machining workpieces. The type and size of the lathe will depend on your specific needs and the size of the workpieces you plan to work on.

Lathe Tools: These include various cutting tools used for turning, facing, threading, and other operations on the lathe. Common lathe tools include turning tools, parting tools, boring bars, and threading tools.

Tool Post: The tool post holds the lathe tools and allows for quick and easy tool changes. It should be sturdy and adjustable to accommodate different tool sizes.

Chuck: The chuck is used to hold the workpiece securely while it is being machined. There are different types of chucks available, such as three-jaw chucks, four-jaw chucks, and collet chucks, depending on the specific application.

Centers: Centers are used to support the workpiece during turning operations. The headstock center is mounted in the lathe's spindle, while the tailstock center can be adjusted to align with the headstock center.

Tailstock: The tailstock is used to provide additional support to the workpiece and can be moved along the lathe bed. It is often used for drilling, reaming, and other operations.

Lathe Chuck Key: This is a specialized key used to tighten and loosen the chuck jaws when mounting or removing the workpiece.

Lathe Chuck Guard: A protective guard should be installed around the chuck to prevent accidents and shield the operator from flying debris.

Lathe Cutting Fluid: Cutting fluids such as coolant or lubricant are used to cool the workpiece and the cutting tool during machining, reducing heat and extending tool life.

Safety Equipment: Personal protective equipment (PPE) such as safety glasses, gloves, and ear protection should be worn to ensure operator safety.

Measuring Instruments: Various measuring tools like calipers, micrometers, dial indicators, and depth gauges are essential for accurate machining and dimensional control.

Lathe Stand or Bench: A sturdy and stable stand or bench is required to mount the lathe securely and provide a comfortable working height.

These are some of the common instruments and equipment used when setting up a lathe. The specific requirements may vary depending on the type and size of the lathe and the nature of the work you'll be doing. Always refer to the lathe manufacturer's instructions and safety guidelines for proper setup and operation.

instruments and equipment commonly used in lathe operations. Here are some examples:

Cutting Tools: Various types of cutting tools are used in lathe operations, including turning tools, boring tools, threading tools, parting tools, and grooving tools. These tools are made from high-speed steel (HSS), carbide, or other specialized materials and are designed to remove material from the workpiece during machining.

Tool Holders: Tool holders are used to securely hold the cutting tools in place on the lathe's tool post. They come in different configurations, such as quick-change tool holders or indexable tool holders, which allow for easy and efficient tool changes.

Chuck: A chuck is a device used to hold the workpiece securely in place on the lathe's spindle. Different types of chucks are available, such as three-jaw chucks, four-jaw chucks, collet chucks, and faceplates, depending on the workpiece requirements.

Collets: Collets are used for holding small-diameter workpieces in a lathe. They are typically used in precision applications where a high degree of concentricity is required.

Centers: Lathe centers are used to support the workpiece during turning operations. The **headstock center** is mounted in the spindle, while the tailstock center is positioned in the tailstock. Together, they provide stability and help align the workpiece.

Faceplate: A faceplate is a flat, circular plate that attaches to the lathe's spindle. It is used for holding large, irregularly shaped workpieces that cannot be held by a chuck.

Steady Rest: A steady rest is a device used to support long, slender workpieces to prevent bending or vibration during machining. It is typically used when turning between centers or when the workpiece extends beyond the chuck.

Live Center: A live center is a rotating center mounted in the tailstock. It has bearings that allow it to rotate with the workpiece, reducing friction and heat buildup.

Coolant System: Many lathes are equipped with a coolant system to lubricate the cutting tools and workpiece, remove chips and swarf, and dissipate heat generated during machining.

These are just some of the instruments and equipment commonly used in lathe operations. The specific tools and equipment used may vary depending on the type of lathe and the specific machining requirements.

Figure Magnetic Base Indicator Stands

2.1.3. Selection of material

Types of Metals

Ferrous and non-ferrous metals

What is the difference between ferrous and non -ferrous metal



what metals are non-ferrous or ferrous? Ferrous metal contains a significant quantity of iron, while non-ferrous metal does not contain iron. Both metal types have properties and attributes that make them desirable.

Manufacturers value ferrous metals for their durability and strength. Making them the ideal materials for use in the construction industry. Common ferrous metals include alloy steel, carbon steel, wrought iron, and cast iron. However, ferrous metals' carbon and iron content make them susceptible to rust on exposure to moisture. The only ferrous metal that resists rust is wrought iron, and this is due to its low carbon content.

On the other hand, non-ferrous metals are rust and corrosion-resistant. This makes them ideal for making outdoor signs, roofs, gutters, and liquid pipes. They are also highly malleable, making them suitable for creating ornaments and jewelry.

Additionally, non-ferrous metals have properties and attributes that make them highly demanded, from lightweight to zero-magnetic ability. Being non-magnetic makes them ideal for wiring and electronic applications. Common examples of non-ferrous metals include; aluminum, silver, zinc, copper, gold, titanium, lead, and nickel.

The two types of metal are; -

A. Ferrous Metals

Iron (Fe) – atomic number 26. Most widely used of all metals as base metal in steel and cast iron.

Pig iron - the intermediate product of smelting iron ore with a high-carbon fuel such as coke, usually with lime stone as a flux. Cast iron

Cast iron – is derived from pig iron

White cast iron is named after its white surface when fractured, due to its carbide impurities which allow cracks to pass straight through.

Grey cast iron is named after its grey fractured surface, which occurs because the graphitic flakes deflect a passing crack and initiate countless new cracks as the material breaks.

Wrought iron - iron alloy with a very low carbon content, in comparison to steel, and has fibrous inclusions (slag) tough, malleable, ductile and easily welded

Steel is an alloy that consists mostly of iron and has a carbon content between 0.2% and 2.1% by mass.

Carbon is the most common alloying material for iron, but various other alloying elements are used, such as manganese, chromium, vanadium, molybdenum, tungsten, etc.

Stainless steel (inox steel) is a steel alloy with a minimum of 10.5 or 11% chromium content by mass.It does not corrode, rust, or stain with water as ordinary steel does.

High speed steel is commonly used in tool bits and cutting tools. It can withstand higher temperatures without losing its hardness. This property allows HSS to cut faster than high carbon steel, hence the name high speed steel.

B. Non-Ferrous Metals

Copper Latin cuprum (Cu)– ranks next to iron in importance and wide range of application, good heat and electrical conductivity, resistance to corrosion

Alloys: brass, bronze, cupro- nickel (copper nickel) alloys Aluminum (BrE) or aluminum (AmE) – Al, atomic number 13. the third most abundant element (after oxygen and silicon), and the most abundant metal in the Earth's crust. Low density and ability to resist corrosion; good conductivity

Structural components made from aluminum and its alloys are vital to the aerospace industry and are important in other areas of transportation and structural materials

Zinc (Zn), atomic number 3. Corrosion resistant in air due to a thin oxide film forming on its surface. Used as a coating for protecting steel - galvanization is the process of applying a protective zinc coating to steel or iron, in order to prevent rusting.

Tin Latin stannum (Sn), - atomic number 50, white, lustrous, soft, malleable, ductile, resistant to corrosion. Used as coating for steel and sheet iron.

Lead Latin plumbum (Pb)- atomic number 82, metallic lead has a bluish-white color after being freshly cut, but it soon tarnishes to a dull grayish color when exposed to air. Has a shiny chrome-silver luster when it is melted into a liquid.

Property: soft, malleable, has little ductility

usage: plates for storage batteries, covering for electrical cables

Self-Check -2.1

Directions: I Choose the best answer for the given alternative listed below.

1.-----must be mounted between centers and a drive plate and lathe dog must be used. (1 points)

A. tailstock spindles B. the center drill C. faceplates D. Mandrel

2. ----is the device inserted to the tail stock and used to mount between center of the work piece. (1 point)

A. live center B. dead center C. half center D. All

3.The first step in preparing the centers is to see that they are accurately mounted in the headstock and tailstock spindles. (1 point)

A. true B. false

4.---is the process of applying a protective zinc coating to steel or iron, in order to prevent rusting. (1 point)

A. Galvanization B. Tin C. Lead D. Copper

5. ---------- Is the mixture of iron and carbon? (1 point)

A. High speed steel B. Pig iron C. Cast iron D. Steel

B. Matching Column A With Column B (1 Point for each)

Column ``A``	**Column ``B``**
Column "A"	**Column "B"**
1. mounting a work- piece between centers	A.3 jaw &independent (4 jaw) chuck
2. Mount faceplates	B. a heavy piece of work is mounted off center
3. Mounting Work pieces in Chucks	C. Must be center -drilled and counter sunk
4. Ferrous Metals	D. Zinc (Zn),
5. Non-Ferrous Metals	E. Iron (Fe)

Directions: III. Write if the statement is correct say True or if the statement wrong says False

1. Magnetic Base Indicator Stands are precision tools used to hold and position measuring instruments such as dial indicators

2. probe indicator or comparator) is a precision measuring instrument used for several applications across many industries where precision is necessary,
3. A chuck is a device used to hold the workpiece securely in place on the lathe's spindle
4. The surface gauge is usually used with a V Block and angle plate as well.
5. Steady rest is a flat, circular plate that attaches to the lathe's spindle. It is used for holding large, irregularly shaped workpieces that cannot be held by a chuck.

Operation Sheet 2.1

Operation title: steps to center work in a 4jaw check and Set-Up Work Piece

Purpose: To practice and demonstrate the knowledge and skill required in putting center work in a 4-jaw chuck.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 3Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

- 4-Jaw chucks
- Lathe machine
- Dial indicator

Quality Criteria: must have good accuracy with dial indicator

Precautions: check and save all activity

Procedures in doing the task

Steps 1-Put the work piece in the chuck and center it by eye. 4-jaw chucks have grooves in the face of the chuck body to make this easier.

Steps 2- Set a dial indicator against the work piece.

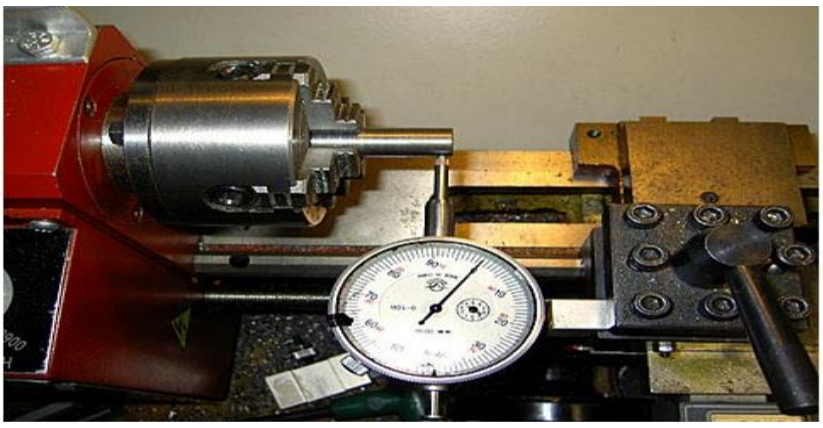


Figure13.center work in a 4-jaw chuck

Steps3-By hand turn the chuck through one complete revolution. Write down the high reading and the low reading from the indicator.

Steps 4-Calculate the average reading by adding the high reading to the low reading and then dividing by two. Average = (High Reading + Low Reading)/2

Steps 5-Turn the chuck until the indicator needle is at the average reading.

Steps 6- Turn the bezel on the dial indicator so that the zero is under the needle.

Steps 7-Rotate the chuck so that one jaw is aligned with the plunger of the dial indicator.

Steps 8-Adjust the jaw that is aligned with the dial indicator plunger and the opposite jaw so the indicator reads zero.

Steps 9-Rotate the chuck 90 degrees so another jaw is aligned with the indicator plunger.

Steps 10-Adjust the jaw that is aligned with the dial indicator plunger and the opposite jaw so the indicator reads zero.

Lap Test –2: Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

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

Task 1: Center work in a 3-jaw chuck using vernier caliper

TASK 2: center work in a 4-jaw chuck using by dial indicator

Unit Three. Advanced Lathe Machine Operations

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

- Cutting parameter
- Lathe machine accessories
- Advanced lathe operations

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 Cutting parameter.
- Identify lathe accessories
- Perform lathe operations

3.1. Cutting Parameter

General operations on the lathe include straight and shoulder turning, facing, grooving, parting, turning tapers, and cutting various screw threads. Before these operations can be done, a thorough knowledge of the variable factors of lathe speeds, feeds, and depth of cut must be understood. These factors differ for each lathe operation, and failure to use these factors properly will result in machine failure or work damage. The kind of material being worked, the type of tool bit, the diameter and length of the work piece, the type of cut desired (roughing or finishing), and the working condition of the lathe will determine which speed, feed, or depth of cut is best for any particular operation. The guidelines which follow for selecting speed, feed, and depth of cut are general in nature and may need to be changed as conditions dictate.

A. Cutting Speed (CS):

Cutting Speed (CS): It is the rate at which a point on the work circumference travels past the cutting tool. It is always expressed in feet per minute (ft/min) or in meters per minute (m/min).

Mathematically, cutting speed, (V) = ∏DN m/min.

1000

Where D= is diameter of a job in mm

N= is spindle or job speed in rpm

Example: Calculate the r/min required to turn a 50 mm diameter work piece of machine steel (CS =40 m/min).

B. Lathe Feed

Lathe Feed: It is the distance that the cutting tool advances along the length of the work for every revolution of the spindle. For example, if the lathe is set for 0.40 mm feed, the cutting tool will travel along the length of the work 0.40 mm for every complete turn that the work makes. The feed of an engine lathe is dependent upon the speed of the lead screw or feed rod. The speed is controlled by the change gears in the quick-change gear box. For general-purpose machining a 0.25 to 0.40 mm for roughing, 0.07 to 0.25 mm for finishing is used. For mild steel, = 25 □ 30 m/min, Aluminum = 60 □ 93 m/min and like centers for parallel turning may be applied.

Lathe cutting speeds meters per in feet and minute using a high-speed tool bit (cutting tool.

Material

Turning and Boring

Threading

(m/min)

	Rough Cut (m/min)		Finish Cut (m/min)
Machine Steel	27	30	11
Tool Steel	21	27	9
Cast Iron	18	24	8
Bronze	27	30	8
Aluminum	61	93	18

Table 3.2. Lathe cutting speeds in feet and meters per minute using a high-speed tool bit (cutting tool)

Feeds for various materials (Using a High-Speed Cutting Tool)

Table.3.3. Feeds for various materials (Using a High-Speed Cutting Tool)

Material	Rough Cuts (mm)		Finish Cuts (mm)	
Machine steel	0.25	0.50	0.07	0.25
Tool steel	0.25	0.50	0.07	0.25
Cast iron	0.40	0.65	0.13	0.30
Bronze	0.40	0.65	0.07	0.25
Aluminum	0.40	0.75	0.13	0.25

C. Depth of Cut (t):

Depth of Cut (t): The depth of cut may be defined as the depth of the chip taken by the cutting tool and in one-half of the total amount removed from the work piece in one cut.

When rough turning, the depth of cut will depend up on the following factors:

- The condition of the machine,
- The type and shape of the cutting tool used
- The rigidity of the work piece
- The rate of feed

When finish turning, the depth of cut will depend up on the type of work and the surface finish required. In any case, it should not be less than 0.13 mm.

D. Machining Time

To calculate the time required to machine any work piece, factors such as

- Speed,
- Feed and
- Depth of cut must be considered.

By applying the following formula, the time required to take a cut can be readily calculated.

Time required = length of cut

Feed X r/min

3.2. Lathe Accessories

There are several accessories commonly used with lathes to enhance their functionality and enable a wider range of machining operations. Here are some commonly used lathe accessories:
These are just a few examples of the many accessories available for lathes. The specific accessories used will depend on the type of lathe and the intended machining operations. Lathes can be equipped with a wide range of additional tools and attachments to expand their capabilities and meet specific machining requirements.

Types of accessories

Lathe accessories: are the tools and equipment used in routine lathe machining operations.
Many lathe accessories are available to increase the versatility of the lathe and the variety of work which can be machined. Lathe accessories may be divided in to two parts:
Work-Holding, Supporting and Driving Devices. It includes;
Drive plate: The plate is used to drive work that has been set up between centers.

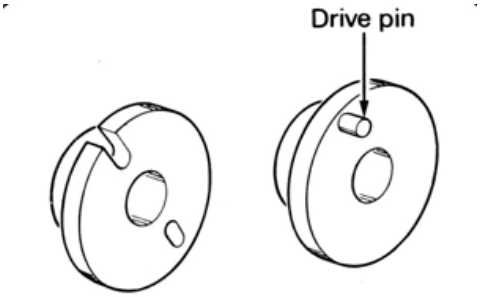


Figure 3.1. Drive plate

Face plate: The face plate is used to hold and drive work that cannot be held between centers or in a chuck

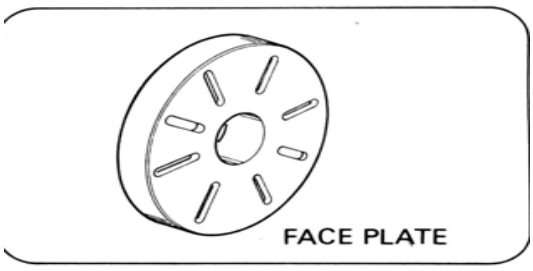


Figure 3.2. Face plate

Chucks: Chucks are work holding devices, which grip work of various size and shape by means of adjustable jaws.

The most commonly used chucks are

Three jaw self - centering chuck is used to hold round and hexagonal work. A chuck key is used to rotate a scroll that moves the three jaws simultaneously.

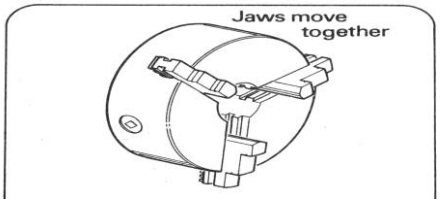


Figure 3.3. Three – Jaw Self Centering Chick

Four jaw independent chucks: are used to hold regular and irregular or odd shaped work.

Advantages:

- Each jaw can be moved independently
- Work can be held very firmly because each jaw opposes the others
- Work can be set up to run very true by adjusting the jaws and checking the work with a dial indicator
- Each jaw can be reserved independently to enable odd shaped work to be held
- Concentric circles marked on the face assist in locating work centrally

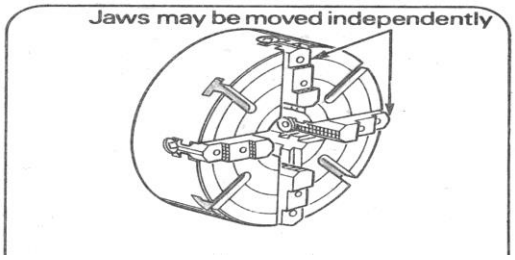


Figure 3.4. Four jaw independent chuck

Headstock center: Work to be turned between centers is driven by a clamp attached to the end of the work. The clamp called a carrier has a leg or tail that locates against the pin

of the driving plate fitted to the lathe spindle. Bent tail carriers engage into a slot in the drive plate.

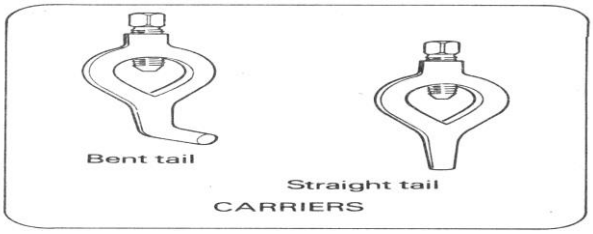


Figure 3.5. Carriers

Tailstock Centers: is held in the tapered bore of the tailstock spindle. It supports the right-hand end of work to be turned between centers.

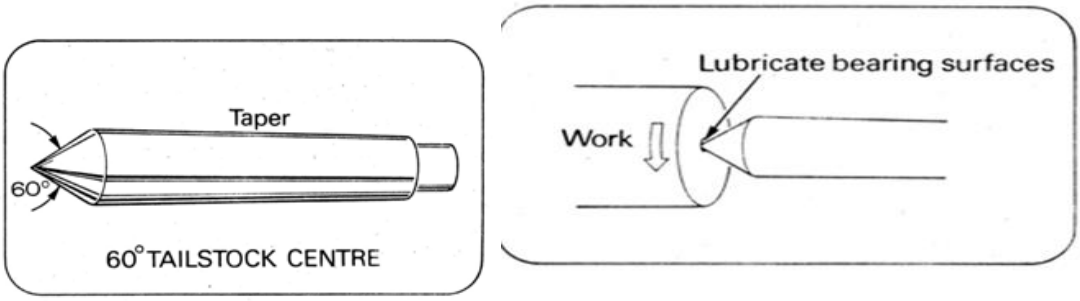


Figure 3.6. Tailstock center

Live center: are tailstock center that run on roller or ball bearings. This enables the point of the center to rotate with the work. Excessive pressure on the center would cause the bearings to overheat and become damaged.

Live center has the following advantages:

- Lubrication is not needed between the center and the work
- The work can be rotated at a higher speed
- Heavy cutting loads can be carried by the center

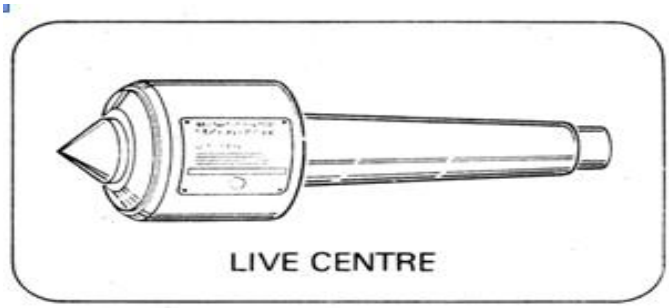


Figure 3.7. Live center

Steady Rests: are lathe attachment used to support long or slender work. They prevent the work from being bent or deflected by the cutting tool during machining.

Two types of steadies are used. They are:

Travelling steady: The travelling steady is bolted to the saddle of the lathe and moves along with the cutting tool. The steady consists of a frame holding two adjustable bearing pads. The pads are positioned 90° apart. One pad is situated behind the work, directly opposite the cutting tool. The other pad is situated on top of the work. Each pad is adjusted by means of a screw thread and can be locked in the required position.

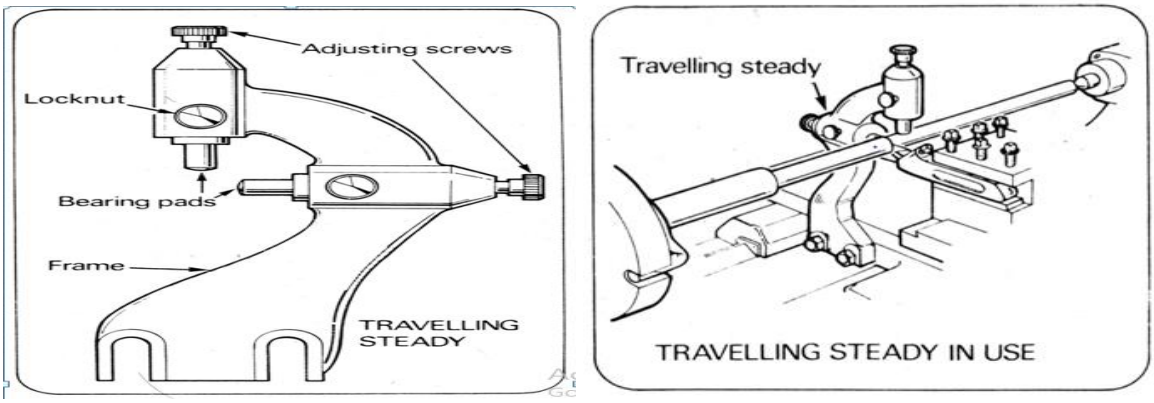


Figure 3.8. Travel steady

Fixed steady: The fixed steady is bolted directly on the top of the lathe bed. It can be positioned anywhere along the bed. The steady is used to support long slender work that is held between centers, or to support one end of long work that has its other end held in a chuck. The steady consists of a frame that has three adjustable bearing pads. Each pad is operated by a screw thread and can be locked in the required position.

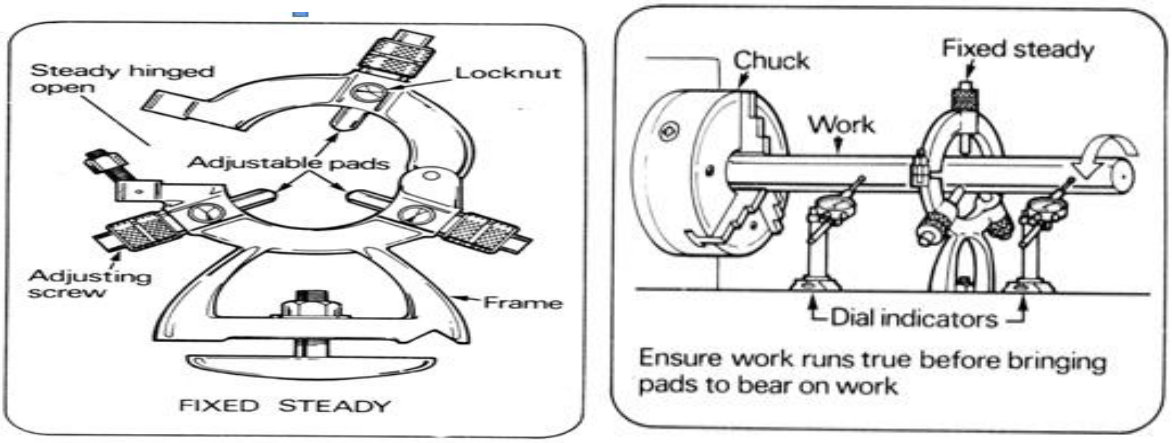


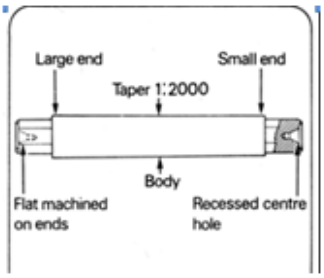
Figure 3.9. Fixed steady

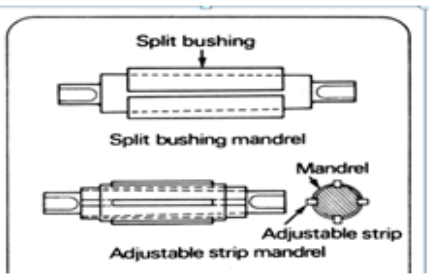
Mandrels: A work piece that cannot be held between centers because its axis has been drilled or bored, and which is not suitable for holding in a chuck or against a faceplate, is usually machined on a mandrel. A mandrel is a tapered axle pressed into the bore of the work piece to support it between centers. A mandrel should not be confused with an arbor, which is a similar device used for holding tools rather than work pieces. To prevent damage to the work, the mandrel should always be oiled before being forced into the hole. When turning work on a mandrel, feed the cutting tool toward the large end of the mandrel, which should be nearest the headstock of the lathe.

The mandrel is provided with center holes to permit mounting between lathe centers.

Types of mandrels:

- Standard solid mandrel:
- Expansion mandrel
- Gang mandrel
- Threaded mandrel
- Taper shank mandrel





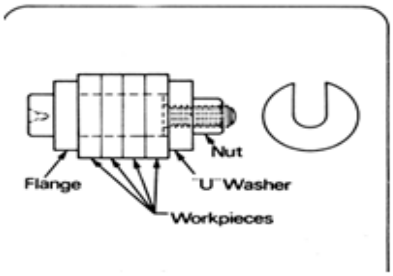
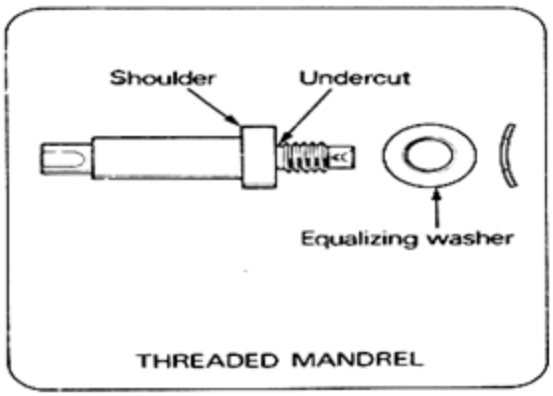


Figure 3.10.a. Standard solid mandrel **Figure 3.11.b.** Expansion mandrel **Figure 3.12c.** Gang mandrel



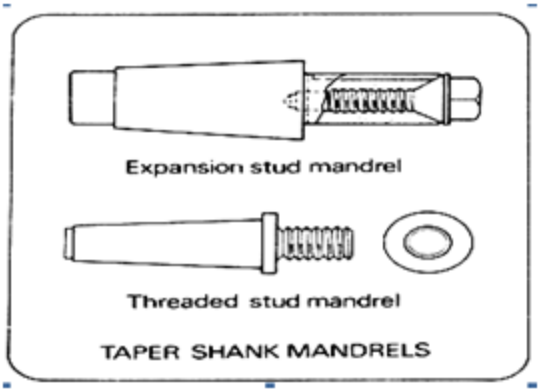


Figure 3.13 d. Threaded mandrel **Figure 3.14.e.** Taper shank mandrel

Cutting Tool-Holding Devices. It includes various types of straight and offset tool holders

Straight tool holder is used for general purpose machining and for thread cutting. Some makes of holders have the letter S stamped on their bodies to make them easily identifiable.

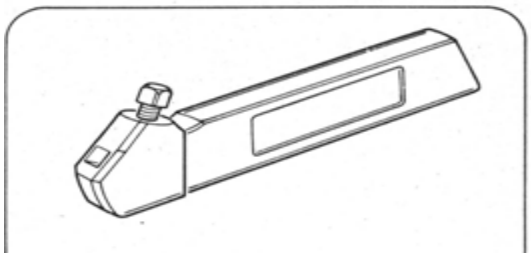


Figure 3.15. Straight tool holder

Right hand tool holder is used for facing operation and for machining work close to the tailstock. Some makes of holders have the letter R stamped on their bodies to make them easily identifiable.

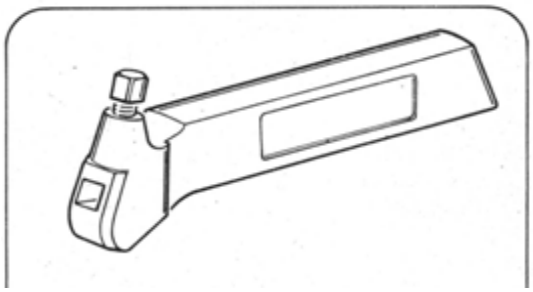


Figure 3.16. Right hand tool holder

Left hand tool holder is used for facing operations and for machining work close to the headstock. Some makes of holders have the letter L stamped oh their bodies to make them easily identifiable.

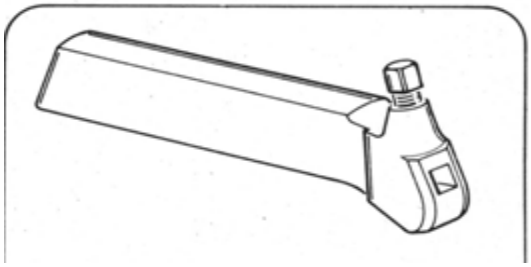


Figure 3.17. Left hand tool holder

3.3. Advanced Lathe Operations

advanced lathe operation involves utilizing advanced techniques and features to achieve complex machining tasks. Here are some advanced lathe operations you can explore:

Taper Turning: Taper turning involves machining a workpiece with a gradually changing diameter. This can be done using the compound rest or taper attachment. By adjusting the compound rest or using a taper attachment, you can create precise tapers on the workpiece.

Thread Cutting: Lathe machines can be used to cut external or internal threads. External threading is done using a threading tool, while internal threading is achieved using a tap or a threading insert. You'll need to calculate the correct pitch, depth, and angle for the threads and use appropriate gears or thread-cutting tools.

Knurling is a process of creating a pattern of ridges on the surface of a workpiece to improve grip or aesthetics. Knurling tools with specially designed wheels are used to press the pattern onto the workpiece. Proper alignment and feed rate control are crucial for achieving consistent knurling results.

Boring: Boring involves enlarging existing holes or creating internal features with precise dimensions. It is accomplished using boring bars with cutting inserts or specialized boring tools. The process requires careful setup, alignment, and control over feed rates and cutting depths.

Parting-off: Parting-off is the process of cutting off a finished workpiece from the main stock. It is achieved using a parting tool or a cutoff tool. Proper alignment, tool selection, and control of feed rates and cutting depths are essential to prevent damage to the workpiece and the lathe.

Face Turning: Face turning refers to machining the end face of a workpiece. It can be done using a facing tool or a facing mill. Face turning is used to create flat surfaces, remove material, or achieve precise dimensional control on the workpiece's end face.

Eccentric Turning: Eccentric turning involves machining workpieces with off-center features or irregular shapes. This technique requires adjusting the workpiece's position in the chuck or using specialized eccentric turning attachments. It allows you to create eccentric shafts, camshafts, and other non-symmetric parts.

Multi-Axis Machining: Some advanced lathes have multiple axes, such as live tooling or Y-axis capabilities. This enables simultaneous machining operations, such as turning, milling, drilling, and tapping, on different surfaces of the workpiece without the need for additional setups.

CNC Lathe Programming: If you're working with a computer numerical control (CNC) lathe, advanced operations involve programming the lathe to execute complex machining sequences. This includes writing G-code or using CAM software to control tool movements, spindle speed, feed rates, and other parameters.

Operation of Eccentric turning

Eccentric turning is a technique used to machine workpieces with off-center features or irregular shapes. Here's a step-by-step guide on how to perform eccentric turning on a lathe:

Understand the Design: Begin by reviewing the design or blueprint of the workpiece to identify the specific areas that require eccentric turning. Determine the amount of eccentricity (distance from the center) and the location where the off-center feature needs to be created.

Prepare the Lathe: Set up the lathe by securing the workpiece in the chuck. Ensure that the workpiece is held securely and aligned properly.

Determine the Eccentricity: Use precision measuring tools such as calipers or a dial indicator to determine the required eccentricity of the turning operation. Measure the distance between the center of the workpiece and the desired off-center position.

Adjust the Workpiece Position: To achieve eccentricity, you have a few options:

A. **Off-Center Chucking:** Move the workpiece off-center within the chuck. Loosen the chuck jaws and shift the workpiece to the desired off-center position. Then, tighten the chuck jaws to secure the workpiece.
B. **Eccentric Turning Attachment:** Some lathes have specialized attachments designed for eccentric turning. These attachments allow you to adjust the position of the workpiece relative to the lathe's spindle, creating the desired eccentricity.

Set the Tool: Install a suitable cutting tool in the tool post or tool holder. Ensure that the tool is properly aligned and positioned to make contact with the workpiece at the desired location.

Adjust the Tool Height and Depth of Cut: Position the cutting tool at the correct height to engage the workpiece effectively. Adjust the tool post or tool holder to achieve the desired cutting depth based on the design requirements.

Start the Lathe: Turn on the lathe and set the spindle speed and feed rate based on the material and the diameter of the workpiece.

Perform the Eccentric Turning: Slowly traverse the cutting tool along the workpiece surface, following the desired cutting path. Take light cuts and gradually increase the depth of cut as needed. Pay close attention to the cutting forces and vibrations to ensure the tool and workpiece remain stable.

Check the Results: Periodically stop the lathe to inspect the workpiece and verify the accuracy of the eccentric turning. Use measuring tools to check the dimensions and alignment of the off-center feature.

Repeat if Necessary: If multiple eccentric features are required, repeat the process for each feature, adjusting the workpiece position accordingly.

Crankshaft on Lathe

How to Machining Crankshaft on Lathe

for Turning Lathe Machinery Tips: Depending on its construction and the type of engine in which it will be fitted, the single crankshaft can be made in various ways. In a first-attempt small steam-engine everything may be arranged in the easiest manner, taking what might be regarded as a mechanical liberty, but excusable in the circumstances. At the other extreme, in the case of a fast high-compression petrol engine, “correct” engineering principles in regard to the type of component must be followed-for to do otherwise would be courting disaster. Making the crankshaft as simply as possible, flat plate material can be employed for the web, holes can be drilled and, if necessary, tapped for the main shaft and crankpin. Then these two parts, to fit in the web, can be shouldered by turning in a lathe-from which it follows the essential aliment of the finished crankshaft will depend partly on the truth of the holes? but mostly on the shoulders abutting firmly to the parallel web material. Using thick plate material for the web, basic alignment of main shaft and crankpin holes is vital; so the holes should be drilled and bored with

the material clamped to the Lathe Machine Faceplate, or sub-faceplate--and this principle is extended to the more complicated constructions with separate webs.

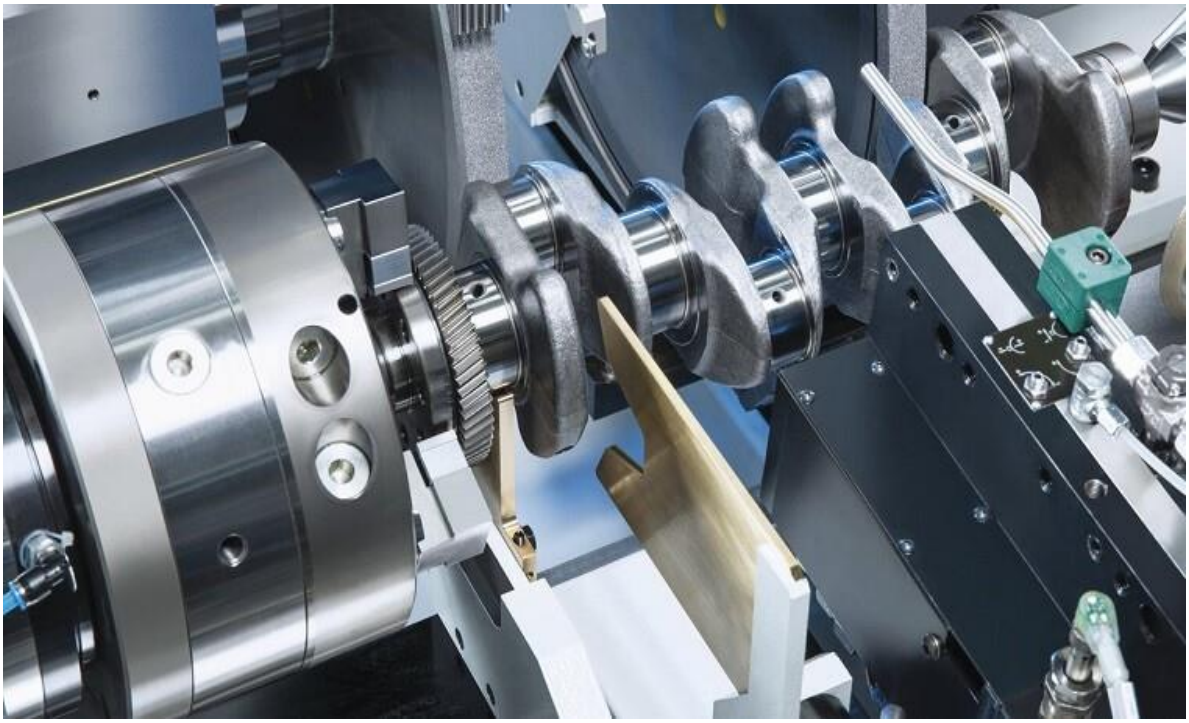


Figure 3.18 Crank shaft

Thread cut and tapper angle

- v -thread
- ISO Metric thread

One of the world's major industrial problems has been the lack of international thread standards. In April 1975, the ISO drew up an agreement covering a standard metric thread standard. The new series has only 25 thread sizes ranging in diameter from 1.6 to 100 mm identified by letter M, the nominal diameter, and the pitch.

The *ISO metric thread* has a 60^0 included angle and a crest equal to 0.125 times the pitch D = 0.6134 times the pitch. Because of these dimensions, the flat on the root of the thread (FR) is wider than the crest (FC). (Refer fig. 27)

INGENERAL

A. $\Phi = 60^{\theta}$

B. D = 0.6134 X P.

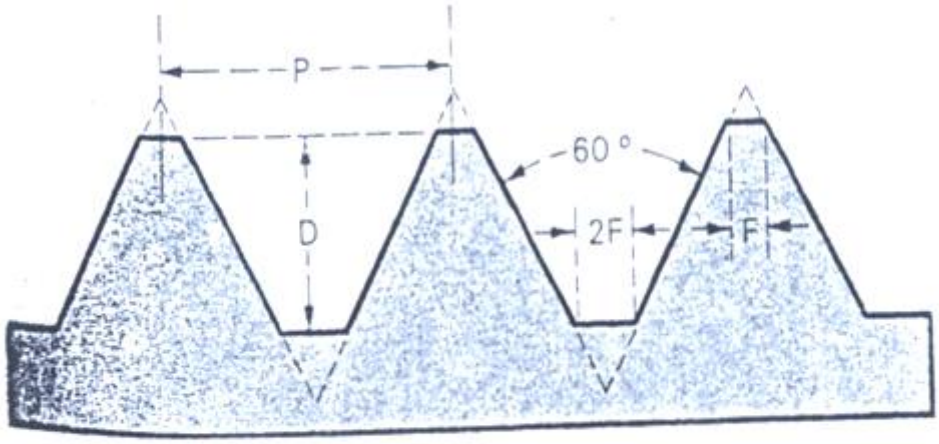
C. FC = 0.125XP

D. FR = 2X0.125XP

ISO Metric Pitch and Diameter Combinations

Table 3.1. ISO Metric Pitch and Diameter Combinations

Nominal Diameter (mm)	Thread Pitch (mm)	Nominal Diameter (mm)	Thread Pitch (mm)
1.6	0.35	20	2.5
2	0.40	24	3.0
2.5	0.45	30	3.5
3.0	0.50	36	4.0
3.5	0.60	42	4.5
4.0	0.70	48	5.0
5.0	0.80	56	5.5
6.3	1.0	64	6.0
8.0	1.25	72	6.0
10.0	1.50	80	6.0
12.0	1.75	90	6.0
14.0	2.00	100	6.0
16.0	2.00		



- *D* = Single Depth of Thread
- *P* = Pitch
- *D* (External) = 0.54127 x *P*
- *FC* = 0.125 x *P*
- *FR* = 0.250 x *P*

Figure 3.18.: ISO Metric Thread

The American National Standard Thread

The American National Standard thread is divided into four main series, all having the same shape and proportions. (Refer fig 3.7)

- National Course (NC)
- National Fine (NF)
- National Special (NS) and
- National Pipe (NPT)

This thread has 60^0 angles with a root and crest truncated to one-eighth the pitch. It is used:

- In fabrication
- Machine construction and assembly, and
- For components where easy assembly is desired

Depth of thread for 100 percent thread $= \frac{0.866}{N}$

Since this thread would be very difficult to cut (especially internally), the will use the formula which given 75 percent in industry.

$D = 0.6495 \times P$ **OR** $\frac{0.6 + 95}{N}$

$F = 0.125 \times P$ **OR** $\frac{0.125}{N}$

F = Width of Crest

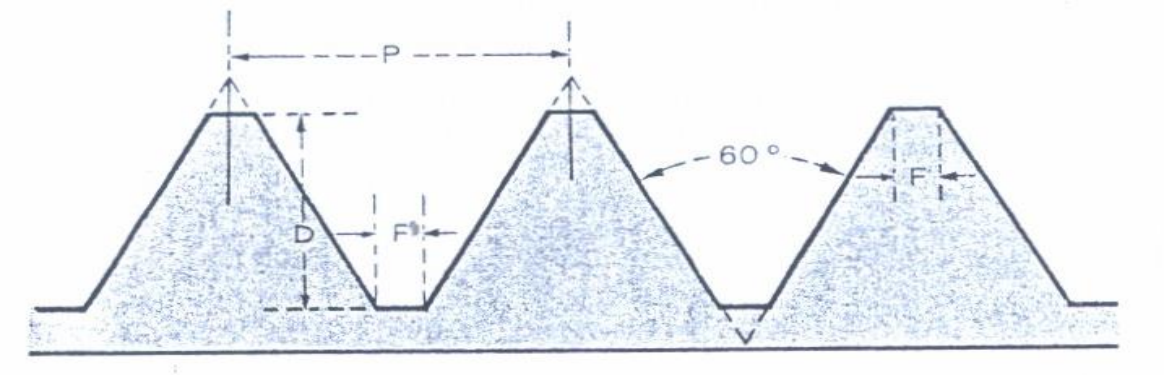


Figure 3.19: American National Threads

The British Standard with Worth Thread (BSW)

It has a 55^0 V-form with rounded crests and roots. (Refer fig 29)

$D = 0.6403 \times P$ *OR* $\frac{0.6403}{N}$ $R = 0.1373 \times P$ *OR* $\frac{0.1373}{N}$

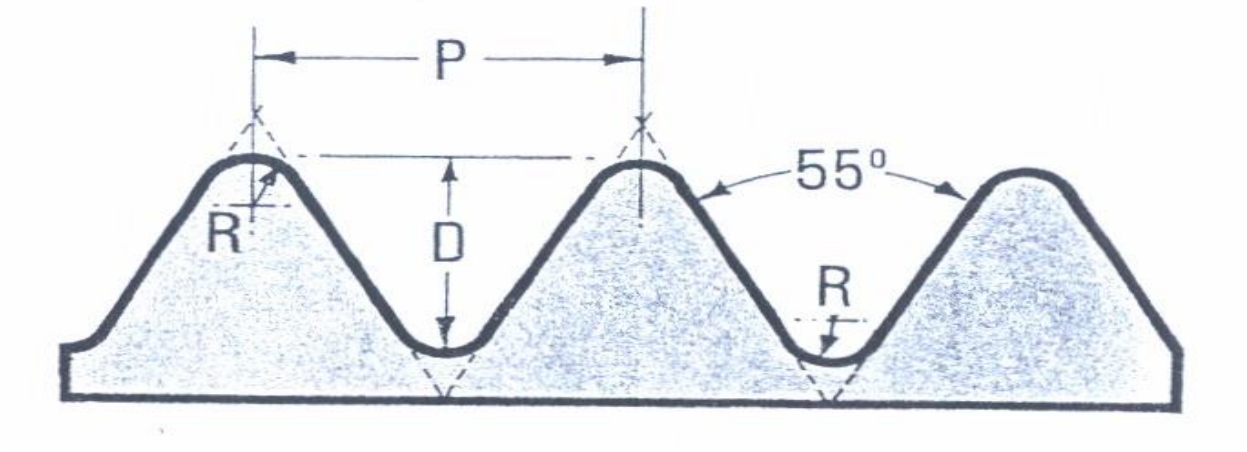


Figure .3.20: With Worth Thread

The Unified Thread

It was developed by United States, Britain and Canada, which would have a standardized thread system. It is a combination of the British standard with worth and the American National form thread. This thread has a 60^0 angle with a rounded root, and the crest may be rounded or flat. (Refer fig 30)

$$D(ExternalThread) = 0.6131 \times P \quad OR \quad \frac{0.6131}{N}$$

$$D(InternalThread) = 0.5413 \times P \quad OR \quad \frac{0.5413}{N}$$

$$F(ExternalThread) = 0.250 \times P \quad OR \quad \frac{0.250}{N}$$

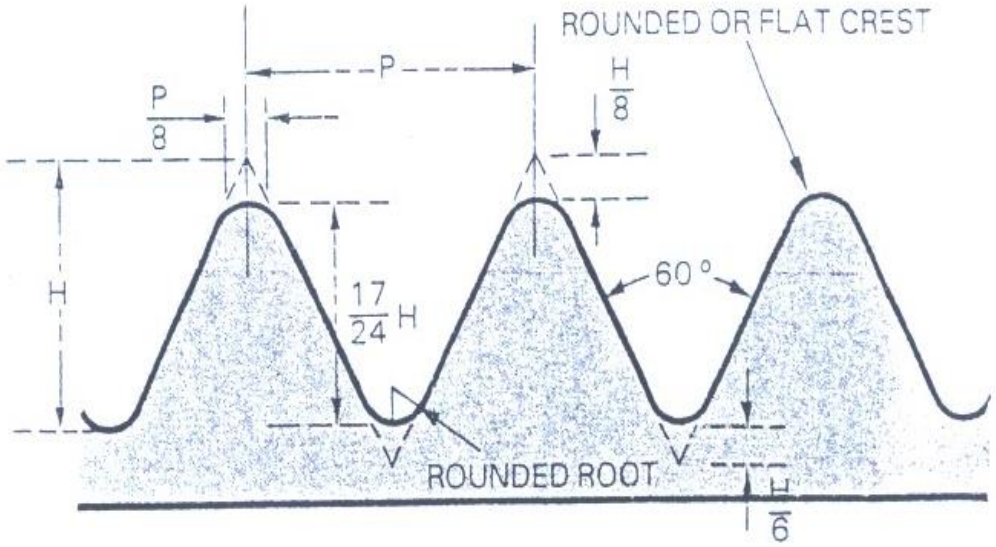


Figure 3.21.: Unified Threads

The International: It is a standardized thread used in Europe. It has 60^0 included angles with crest and root truncated to one eight of the depth. It is used for spark plugs and manufacture of instruments.

$$D = 0.7035P \ (Maximum)$$

$$= 0.6855P \ (Minimum)$$

$F = 0.125P$

$R = 0.0633P\,(Maximum)$

$= 0.05AP\ \ (Minimum)$

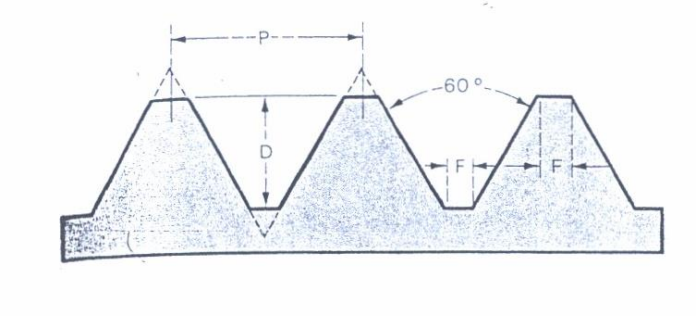


Figure.3.22.: International Metric Threads

The American National Acme Thread: It is replacing the square thread in many cases. It has 29^0 angles and is used for feed screws, jacks and vises. (Refer fig 32)

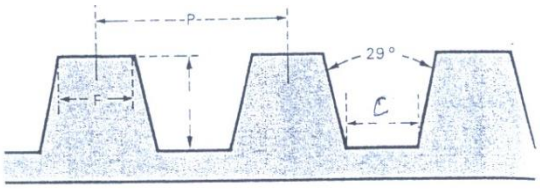


Figure.3.23. Acme Thread

D = Minimum 0.500 x P

D = Maximum 0.500 x P + 0.010

F = 0.370 x P

C = 0.370 x P – 0.0052 (for maximum depth)

The Brown and Sharp Worm Thread:

It has 29^0 included angle as the Acme thread; but the depth is greater and the widths of the crest and he root are different.

This thread is used to mesh with worm gears and transmit motion between two shafts at right angles to each other but not in the same plane. (Refer fig 33)

$D = 0.6866 \times P$

$$F = 0.335 \times P$$

$$C = 0.310 \times P$$

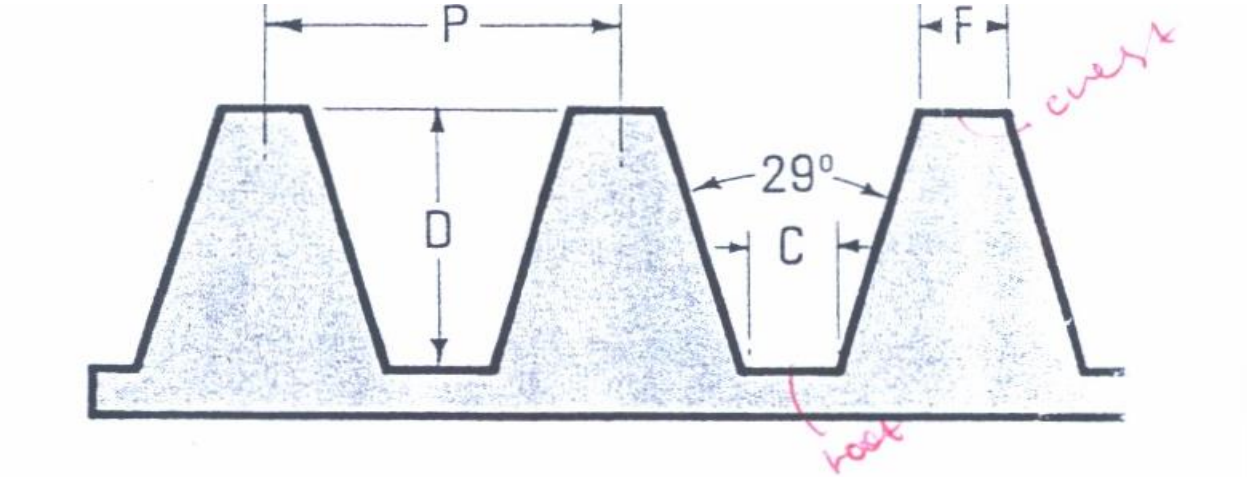


Figure 3.24. Brown and Sharp Worm Thread

The Square Thread

It is frequently replaced by the Acme thread due to the difficulty of cutting it. (Practically with taps and dies) (Refer fig 33)

They were often found on rises and jacks screws.

$$D = 0.500P$$

$$F = 0.500P$$

$$C = 0.500P + 0.002$$

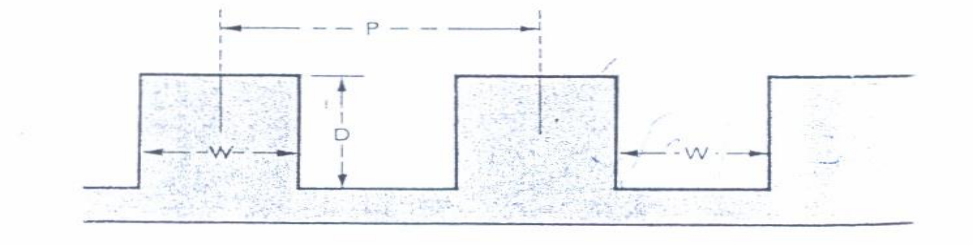


Figure 3.25.: Square Thread

Directions I : Choose the best answer for the following question the listed below

Self-Check -3.1

1. Which one is not belongs to work holding device?(1 point)
   a) Drive plate b) Face plate c) Chucks d) Straight tool holder
2. ---------- are used to hold regular and irregular or odd shaped work. (1 point)
   a) Three jaw self - centering chuck c) Magnetic Chucks
   b) Four jaw independent chucks d) All of the above
3. **----------:** is used to hold and drive work that cannot be held between centers or in a chuck. (1 point)
   a) Face plate b) Drive plate c) Chucks d) Straight tool holder
4. Which one is used for facing operation and for machining work close to the tailstock(1 point)
   a) Left hand tool holder b) Fixed steady c) Right hand tool holder d) all
5. Which one is not true about the *ISO metric thread* ?(1 point)
   A. $\Phi = 60^{\theta}$
   B. $D = 0.6134 \text{ X } P.$
   C. FC = 0.125XP
   D. FR = 2X0.125XP
   E. None
6. Which one is true not about the pitch, depth, minor diameter for `` M14 x 2`` thread?(1 point)
   A. Pitch = 2mm
   B. Depth = 1.08254mm
   C. Minor Diameter = 12.91746
   D. None
7. It is one-half of the total amount removed from the work piece in one cut.(1 point)
   A. Depth of cut B. machining time C. cutting speed D. RPM
8. Which metal needs high cutting speed during rough turning operation? (1 point)
   A. Bronze B. machine steel C. aluminum D. cast Iron

Directions: II. **Write If the statement is correct say True or if the statement is wrong say False**

1. In machining operations, the sequence of events that we follow is safety`` first, accuracy second, and speed last``(1 point)

2. Lathe accidents my usually caused by Loose clothing snagging on the revolving work piece(1point)
3. Eccentric turning is a technique used to machine workpieces with off-center features or irregular shapes.
4. Facing is the process of cutting off a finished workpiece from the main stock

Directions: III fill the blank space for the following question

5. ----------is often used to increase the work piece diameter when a press fit is required.
6. _____is a process of creating a pattern of ridges on the surface of a workpiece to improve grip or aesthetics
7. _____is used for general purpose machining and for thread cutting.
8. _____ is held in the tapered bore of the tailstock spindle.

Operation Sheet 3.1

Operation title: Turning Operations

Purpose: To practice and demonstrate the knowledge and skill required in turning, grooving, facing.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 3Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

- 3-Jaw chucks
- Lathe machine
- Dial indicator
- outside caliper
- micrometer
- knurling tools
- pitch gauge
- vernier caliper

Quality Criteria: must have good accuracy with in Tolerance is ± .025

Precautions: Check and care all activity

Procedures in doing the task; turning operations performed on a given mild steel work piece

Steps 1-Apply safety precaution

Steps 2- Using the Outside Calipers the diameter of the given work piece is measured.

Steps 3- The facing operation is carried out on both sides of the work piece and a counter hole is drilled at the center.

Steps 4- The work piece is placed in the 3 jaw chuck.

Steps 5- The tail stock is moved so that the work piece is held between the chuck and the dead center

Steps 6- The tool aligned by placing such that, the tip of the tool is perpendicular to the tip of the dead center. The tool post is tightened using align key.

Steps 7- The tool is made to touch the work piece & the depth of the cut is adjusted to get the diameter of 23mm using a micrometer provided in compound rest or in the carriage.

Steps 8- Cutting parameters like speed, feed, depth of cut, etcare selected before the machine is turned on.

Steps 9- Plain turning operation is carried out for required length of 150mm.

Steps 10- Markings are made to perform step turning operation.

Steps 11- Step turning operations are made to get the desired shape of the work piece by giving depth of cut on each run.

Steps 12- Groove turning operations are made to get the desired shape of the work piece by giving depth of cut on each run.

Steps 13- Taper turning angle is calculated using the taper angle formula and the compound rest is swiveled to the angle "$\boldsymbol{\theta}$" in degrees (as per the calculation) with the help of align key. The taper turning at this angle is performed for a length of 25 mm and depth of cut of 1 mm on each run.

$$\boldsymbol{\theta} = \mathbf{tan}^{-1}\left(\frac{\boldsymbol{D} - \boldsymbol{d}}{\mathbf{2}\boldsymbol{l}}\right)$$

Where $\boldsymbol{\theta}$ = Swivel the compound rest to the calculated angle.

D = Major diameter.

d = Minor diameter.

l = Length of the taper.

Steps 14- Knurling operation is carried out by using special knurling tool.

Steps 15- To carry out thread cutting required pitch is noted down & depth of cut is calculated using formula. Using a standard chart the auto feed varying mechanism is set for 2mm pitch for the right hand thread cutting operation & 2.5mm pitch for left hand thread is carried out by engaging lead screw which rotates at required speed & direction for the mentioned thread along with suitable gears in gear train.

Steps 16- The procedure is repeated 13 times with an increment in depth of cut by 0.2 mm divisions each time. In case of right hand thread cutting operation feed will be from right to left.

Steps 17- The procedure is repeated 16 times with an increment in depth of cut by 0.2 mm divisions each time. In case of left hand thread cutting operation feed will be from left to right.

Steps 18- For 'V' – thread cutting, Angle of thread is 600, According to the ISO standard metric thread depth = 0.65X pitch (60 to 65%)

Steps 19- After turning with the help of pitch gauge check the accuracy of the thread pitch and with the help of Vernier caliper, check the diameter of turned part.

Steps 19- Finishing operation is done by giving very small depth of cut (say 0.2mm) on medium feed.

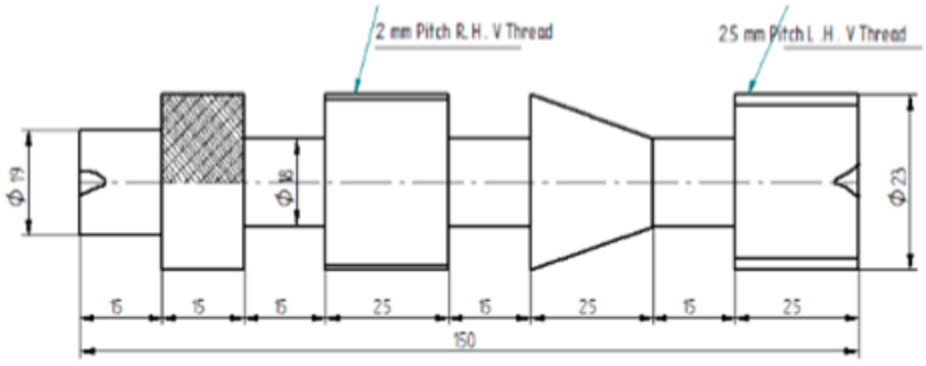


Figure.1. Turning Operations

Note:

- All dimensions are in mm
- Tolerance is ± .025

Operation Sheet 3,2

Operation title: eccentric Turing

Purpose: To practice and demonstrate the knowledge and skill required in turning, Eccentric shape.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 3Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

- 4-Jaw chuck
- Lathe machine
- Dial indicator
- outside caliper
- micrometer
- Center punch
- pitch gauge
- vernier caliper

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: : Check and care all activity

Procedures in doing the task; Eccentric turning operations

Steps 1- Using the Outside Calipers, the diameter of the given work piece is measured. **Steps** 2. The work piece is placed in the 4-jaw chuck and centering operation is performed using the surface gauge.

Steps 3- The facing operation is carried out on both sides of the work piece.

Steps 4- The tool is made to touch the work piece & the depth of the cut is adjusted using a micrometer provided in compound rest or in the carriage.

Steps 5- Cutting parameters like speed, feed, depth of cut, etc are selected before the machine is turned on.

Steps 6- Now the work piece is removed and required marking on circular face of the work piece is made with the help of marking device like V-block, vernier height gauge, scriber, surface plate, etc

Steps 7- Markings are made to perform Eccentric turning operation.

Steps 8- The marked line is punched with the help of center punch.

Steps 9- Fix the work piece to the 4-jaw chuck & centering operation is carried out such that center of the circular face is exactly coinciding with axis of the lathe.

Steps 10- Eccentric turning operations are made to get the desired shape of the work piece by giving depth of cut 1mm on each run.

Steps 11- Taper turning angle is calculated using the taper angle formula and the compound rest is swiveled to the angle "$\boldsymbol{\theta}$ " in degrees (as per the calculation) with the help of align key. The taper turning at this angle is performed for a length of 30mm and depth of cut of 0.5mm on each run.

$$\boldsymbol{\theta} = \mathbf{tan}^{-}\mathbf{1}\frac{(\boldsymbol{D} - \boldsymbol{d})}{\mathbf{2}\boldsymbol{l}}$$

Where $\boldsymbol{\theta}$ = Swivel the compound rest to the calculated angle.

D = Major diameter.

d = Minor diameter.

l = Length of the taper.

Steps 12-. Step turning operations are made to get the desired shape of the work piece by giving depth of cut on each run.

Note:

- All dimensions are in mm
- Tolerance is ± .025

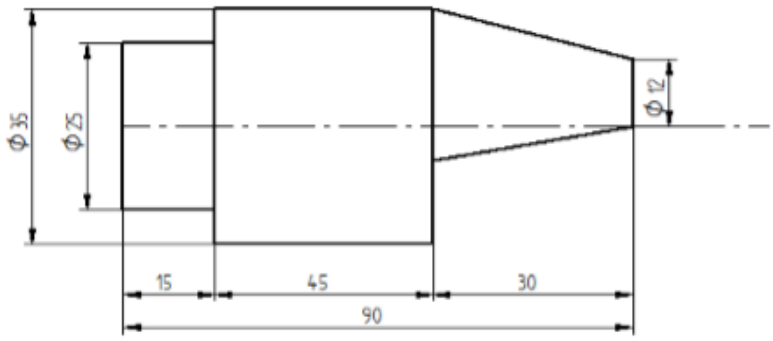


Figure.2. Eccentric turning

Operation Sheet 3.3

Operation title: Cutting internal & external, ACME and Multi-start threads

Purpose: - to let the trainee understand cutting internal & external, ACME and multi-start threads.

Condition for the operation: - Materials, equipment and all the necessary PPE should be provided.

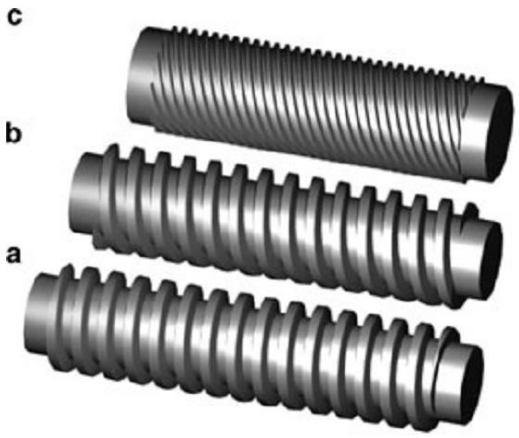
Equipment, tools and materials: - , goggle, safety shoe, coolant, cutter, etc.

Procedure: -

1. Select bar stock with diameter of 1.2in and length 10in.
2. Turn between centers to the required outside diameter (i.e1in).
3. Perform thread making with the following parameters

Parameter	Symbol	Value
Designation	--	1-0.2P-0.4L-ACME-2G
Nominal Diameter	D	1
Threads per inch	n	5
Thread Tolerance	--	2G
LIMITS OF SIZES FOR ACME THREADED ROD		
Maximum Major Diameter	d_{max}	1.0000
Minimum Major Diameter	d_{min}	0.9900
Maximum Pitch Diameter	d_{2max}	0.8920
Minimum Pitch Diameter	d_{2min}	0.8726
Maximum Minor Diameter	d_{1max}	0.7800
Minimum Minor Diameter	d_{1min}	0.7509

Thread length is 7in.



Sample	Nom. Dia.	Pitch	Starts	Lead
A	1"	0.25"	1	0.25"
B	1"	0.25"	2	0.50"
C	1"	0.10"	10	1.00"

Precaution: -

- Use the proper tools and equipment including the PPE
- Keep the machine and working area clean

Quality Criteria: -

- Take appropriate measurement (pitch, length, finish, depth, etc.).
- Always use the right equipment for the right work

Operation Sheet 3.4

Operation title: procedures of lathe set – up for cutting v- threads

Purpose: To practice and demonstrate the knowledge and skill required in turning Thread on the given object.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 4Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

3-Jaw chuck, Lathe machine, Pitch gauge, Mild steel micrometer Center punch, vernier caliper Alin key drill bit tapping tools knurling tool.

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: Check and care all activity

Procedures in doing the task; Thread turning operations

Steps 1- check the diameter of the work piece to be threaded by reference to the working drawing

To provide clearance, it is good practice to turn the diameter of the work piece about 0.05mm undersize.

Steps 2- set the lathe spindle speed at about quarter of the normal turning speed

Steps 3- determine the number of threads per inch to be cut

Steps 4- set up the gearbox to give the appropriate lead screw speed

Steps 5- set the compound rest to 1 degree less than half of the desired thread angle

- in the case of a 60 degree thread angle, the compound rest would be set at 29 degrees to the right for a right hand thread and to the left for a left hand thread.
- The angle to which the compound rest is set affects the cutting action of the cutting tool by producing a shaving action on the trailing edge of the tool. This produces a cleaner cut.

Steps 6- Select the cutting tool and check its accuracy with a center gauge. If necessary, regrind the tool to the correct angle

Steps 7- set the thread cutting tool in the tool holder which is mounted to the tool post

N.B. To prevent the tool from chattering, it must not extend more than 12 mm from the tool holder.

Steps 8-set the height of the tool to the center of the lathe

Steps 9-mount the work piece accurately between the chuck and the live center

Steps 10-set the cutting tool square to the axis of the work

Steps 11- Mark out the exact length of the work piece to be threaded.

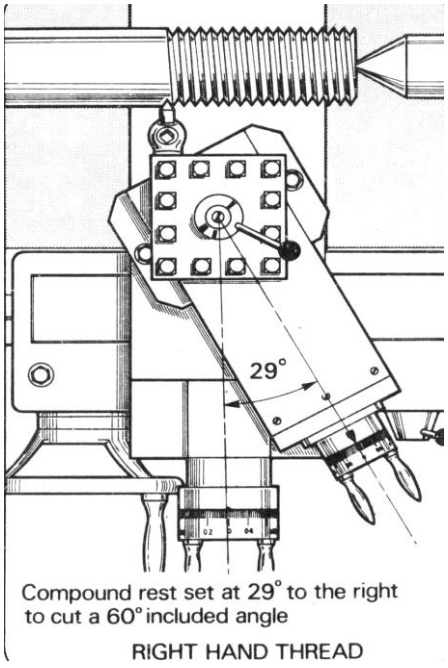


Figure.4. Set – up for cutting v- threads

Operation Sheet 3. 5

Operation title: procedures of cutting an external thread

Purpose: To practice and demonstrate the knowledge and skill required in turning Thread on the external part of object.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 4Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

3-Jaw chuck, Lathe machine, Pitch gauge, Mild steel micrometer

Center punch,

vernier caliper

Alin key drill bit tapping tools knurling tool.

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: Check and care all activity

Procedures in doing the task; Cutting an external thread

Steps 1- put on safety goggles

Steps 2- start the lathe machine

Steps 3- chamfer the end of the work piece with the leading edge of the cutting tool to a depth just greater than the minor diameter of the thread to be cut

Steps 4- move the cutting tool to the work piece surface by operating the cross-slide handle

Steps 5- stop the cross slide when the tip of the cutting tool just touches the work piece surface

Steps 6- set the cross slide and top slide handle graduated collars to zero

Steps 7- move the carriage to the right until the end of the

tool clears the work piece

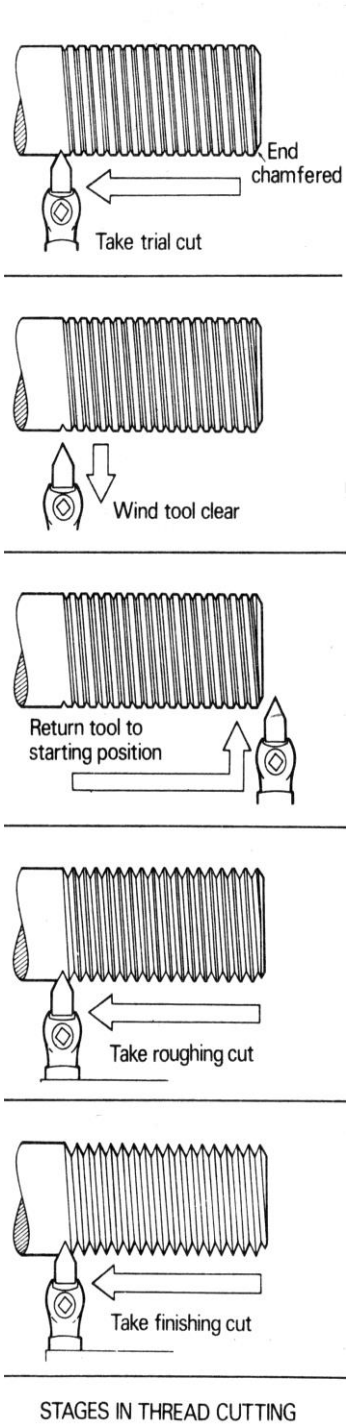


Figure.5.Cutting external threads

Steps 8- feed the tool just about 0.1 mm using the top slide handle

Steps 9- watch movement of the thread chasing dial and engage the lead screw with the carriage half nut at the appropriate time

Steps 10- take a trial cut along the length of the work piece to be threaded

Steps 11- at the end of the trial cut, immediately wind (withdraw) the cutting tool clear of the work piece by operating the cross-slide handle

Steps 12- disengage the half nut

When cutting METRIC threads, using an English lead screw, do not disengage the half nut until the thread is complete.

Steps 13- stop the lathe and check the thread cut with a pitch gage, rule or center gage

Steps 14- return the carriage to the right with the cutting tool clear of the work piece

Steps 15- set the depth of cut with the top slide feed handle

Steps 16- engage the half nut

The first two cuts should be about 0.4 mm deep, and the succeeding cuts around 0.25 deep, and finishing cuts about 0.03 mm deep.

Steps 17- take cuts as necessary to produce a thread of the desired depth

Steps 18- check the finished thread with gage

Operation Sheet 3.6

Operation title: procedures of cutting an internal thread

Purpose: To practice and demonstrate the knowledge and skill required in turning Thread on an internal part of object.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 4Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

3-Jaw chuck, Lathe machine, Pitch gauge, Mild steel micrometer, Center punch, vernier caliper Alin key drill bit tapping tools knurling tool.

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: Check and care all activity

Procedures in doing the task; Cutting an internal thread

Steps 1- mount the work piece accurately using the appropriate accessory

Steps 2- drill a hole in the work piece slightly smaller than the root diameter (tap drill size = nominal size – pitch for V-threads) of the thread to be cut

For a blind hole, the depth of the hole must be longer than the length of the thread to be cut so that the end of the boring bar clears the thread.

Steps 3- cut a recess at the inner end of the bore wide enough to allow the cutting tool to clear the thread. The recess should be slightly deeper than the major diameter of the thread

Steps 4- for a 60-degree V-form thread, set the compound rest 29 degrees to the left

For cutting a left-hand internal thread, the compound rest must be swung 29 degrees to the right. Other threads will require the rest to be set to a different angle.

Steps 5- set the gearbox to turn the lead screw at the speed necessary to produce the required number of threads per inch

Steps 6- mount a cutting tool ground to the correct thread angle in a boring bar

Steps 7- mount the boring bar parallel to the lathe centerline and set the point of the cutting tool on center

Steps 8- Mark the boring bar to indicate the required depth of entry into the bore. To do this, add half the recess width to the length of the thread to be cut, then mark the boring bar this distance back from the center line of the cutting tool.

Steps 9- move the point of the cutting tool until it lightly touches the work surface at the outside end of the bore

Steps 10- set the cross slide and top slide graduated collars to zero

Steps 11- withdraw the cutting tool from the bore

Steps 12- start the lathe

Steps 13- by turning the top slide handle counter clockwise feed the cutting tool 0.1 mm and take a trial cut engaging the half nut at the appropriate dial graduation

Steps 14- at the end of the cut, disengage the half nut and clear the cutting tool from the thread by turning the cross-slide handle clockwise and moving the carriage to the right

Steps 15- check the pitch of the thread

Steps 16- take further cuts starting with feed of 0.4mm, reducing to about 0.03mm for the final cut

Steps 17- check the finished thread with a thread plug gage or a bolt

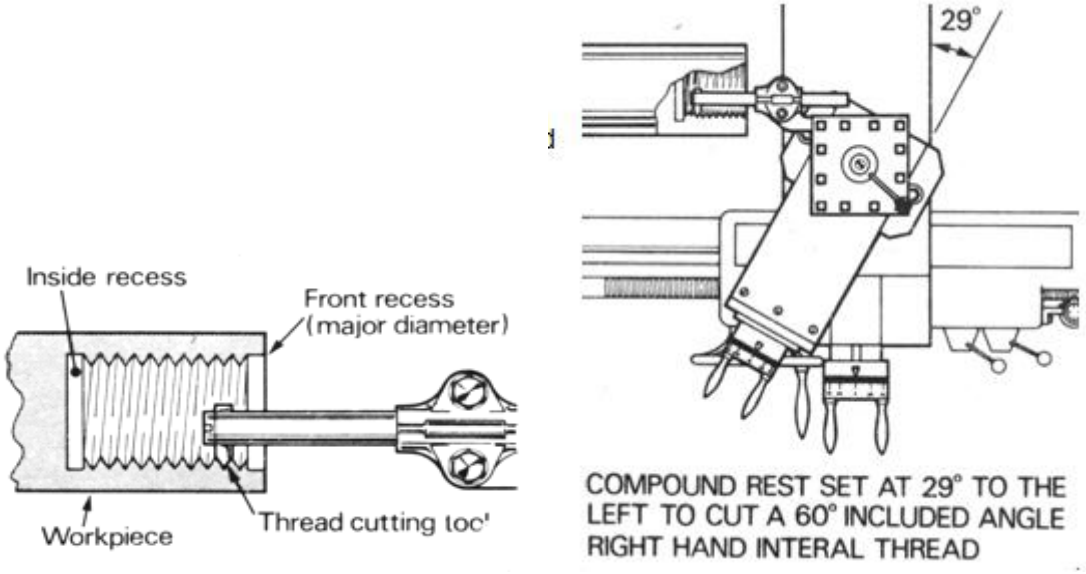


Figure.6. Cutting an internal thread

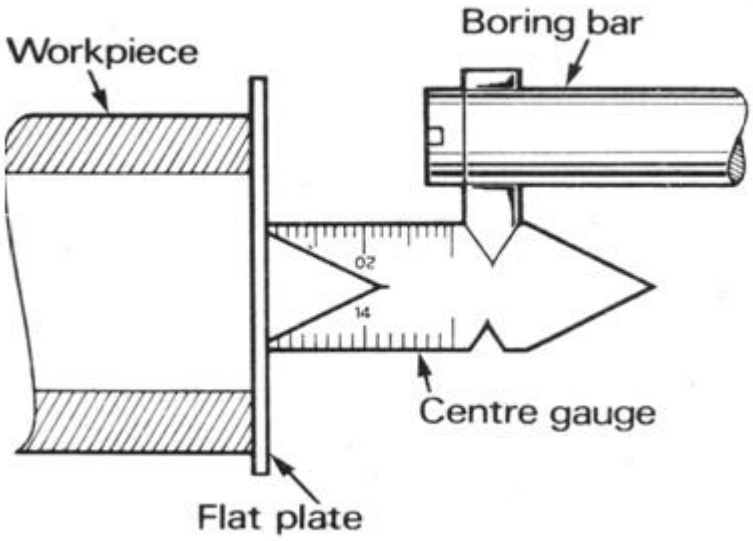


Figure.7. Squaring the cutting tool with a center gauge

Operation Sheet 3.7

Operation title: procedures of cutting an external acme thread

Purpose: To practice and demonstrate the knowledge and skill required in turning Thread on an external acme thread part of object.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 4Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

3-Jaw chuck, Lathe machine, Pitch gauge, Mild steel micrometer

Center punch, vernier caliper Alin key drill bit tapping tools knurling tool.

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: : Check and care all activity

Procedures in doing the task; Cutting an internal thread

Steps 1- select a cutting tool ground to a 29-degree angle and of the correct width for the pitch of the thread to be cut

Steps 2- check the cutting tool with an Acme thread gage

Steps 3- set the compound rest to an angle of $14\frac{1}{2}$ degrees

Steps 4- square the tool with the work using the Acme thread gage

Steps 5- complete the cut as in V-form thread

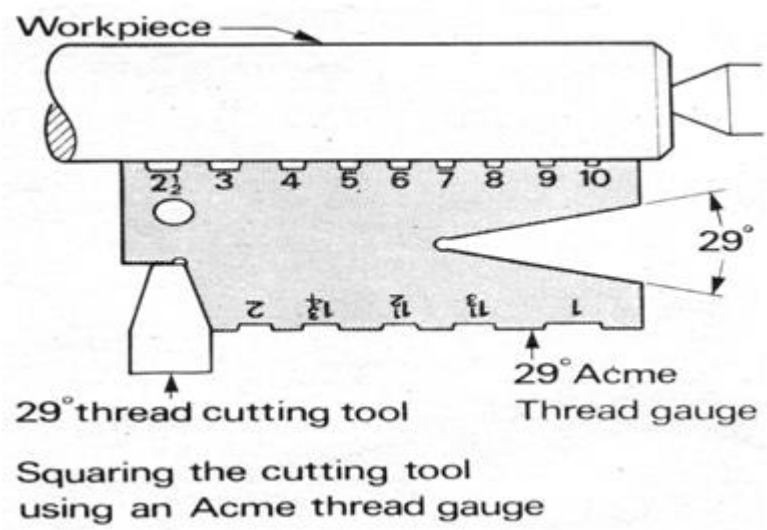


Figure.8. Square the cutting tool using an Acme thread gauge

Operation Sheet 3. 8

Operation title: procedures of cutting an external square thread

Purpose: To practice and demonstrate the knowledge and skill required in turning Thread on the an external square thread part of object.

Instruction: Use the given figure below, the tools and equipment and then follow given steps. For this operation you have given 4Hour and you are expected to show the shape on the Lathe machine

Tools and requirement:

3-Jaw chuck , Lathe machine, Pitch gauge, Mild steel micrometer

Center punch, vernier caliper Alin key drill bit tapping tools knurling tool.

Quality Criteria: must have good accuracy with in Tolerance is ± .025 and all dimensions are in mm.

Precautions: : Check and care all activity

Procedures in doing the task; An external square thread

Steps 1- calculate the leading and following angles for the cutting tool

Steps 2- grind the tool 0.05 mm wider than half the pitch of the square thread

Steps 3- set up the lathe for thread cutting

Steps 4- set the top slide 90 degrees to the right of the cross slide

Steps 5- mount and center the cutting tool

Steps 6- mount the work piece

Steps 7- set the cutting tool square to the work piece

Steps 8- proceed as for a V-form thread cutting using the cross-slide handle to set the depth of each cut

Steps 9- use cutting fluid for each cut

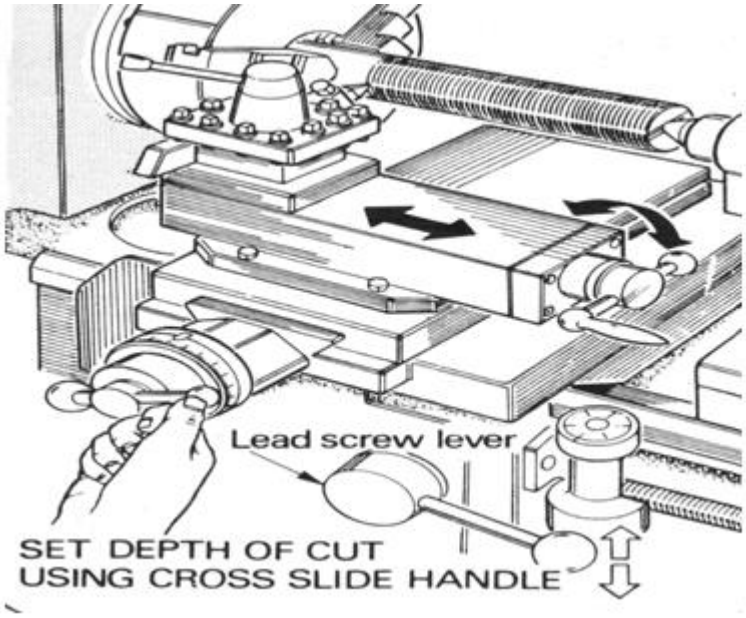


Figure.9. Cutting an Internal Square Thread

LAP Test: Practical Demonstration

Name: _____ Date: _____

Time started: _____ Time finished: _____

Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within 6 hours.

TASK 1: Turning operations

TASK 2: Eccentric turning

TASK 3: Turning with thread operations

TASK 4: Lathe set –up for cutting V- thread

TASK 5: Lathe cutting an external thread

TASK 6: Lathe cutting an internal thread

TASK 7: Lathe cutting an external acme thread

TASK 8: Lathe cutting an external square thread

Unit Four: Product Quality Assurance

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

- . Measuring tools and equipment
- . Checking techniques of work pieces
  - Dimension
  - Tolerance
  - Surface Finish
- Deviations.

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:

- Use measuring tools and equipment
- Identify checking techniques of work pieces
- Handling deviations.

4.1. Measuring Tools and Equipment

Is defined as the measurement of dimensions: length, thickness, diameter, taper, angle, flatness, profiles, and others.

For example, consider the slide ways for machine tools, these **components must have specific dimensions, angles, and flatness** in order for the machine to function properly and function with the desired dimensional accuracy.

The means of measurement could be classified as follows:

Standards (reference masters or setting standards)-these are used to reproduce one or several definite values of a given quantity.

Fixed gauges-these are used to check the dimensions, form, and position of product features.

Measuring instrument-these are used to determine the values of the measured quantity.

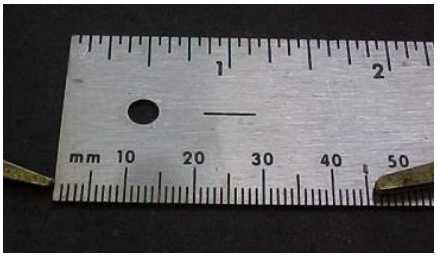
Measuring instruments are measuring devices that transform the measured quantity or related quantity into an indication or information. Measuring instrument can either indicate directly the value of the measured quantity or only indicate its equality to a known measure of the same quantity. (e.g. equal arm balance, or null detecting galvanometer). They may also indicate the value of the small difference b/n the measured quantity and the measure having a value very near to it (comparator).

Steel Rule

It is the simplest and most commonly used linear measurement instrument. It is the part replica of the international prototype meter. The common lengths are 150 mm and 300 mm. Metric rules are graduated in millimeters (mm) and centimeters (cm).

To increase its versatility in measurement certain scales are marked with 0.5 millimeter in some portion. Some steel rules carry graduation in centimeters on one sides and inches on the other side.

A steel rule should be made of good quality spring steel and should be chrome plated to prevent corrosion.



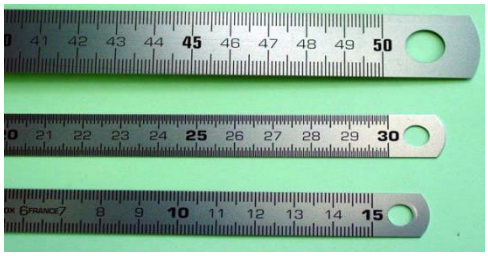


Figure.4.1. Steel Rule

Verneer Calipers

Vernier Calipers (figure 56A) are more precise tools capable for measuring external dimensions, internal dimensions, and depths. Besides the two pairs of measuring jaws and the depth gauge, its main features also include a main scale and a vernier scale.



Figure .4.2.A. Vernier Calipers

The resolution of a vernier scale is determined by the difference on the distance of one division on the main scale and one division on the vernier as shown in figure 57. For example: A vernier scale of length 49mm is divided into 50 equal divisions. That means ONE division on the vernier represents 49/50=0.98 mm while ONE division on the main scale represents 1mm. Then, the resolution of the vernier is 1mm - 0.98mm = 0.02mm.



Figure 4.3.B. Vernier Calipers

- **Reading:**

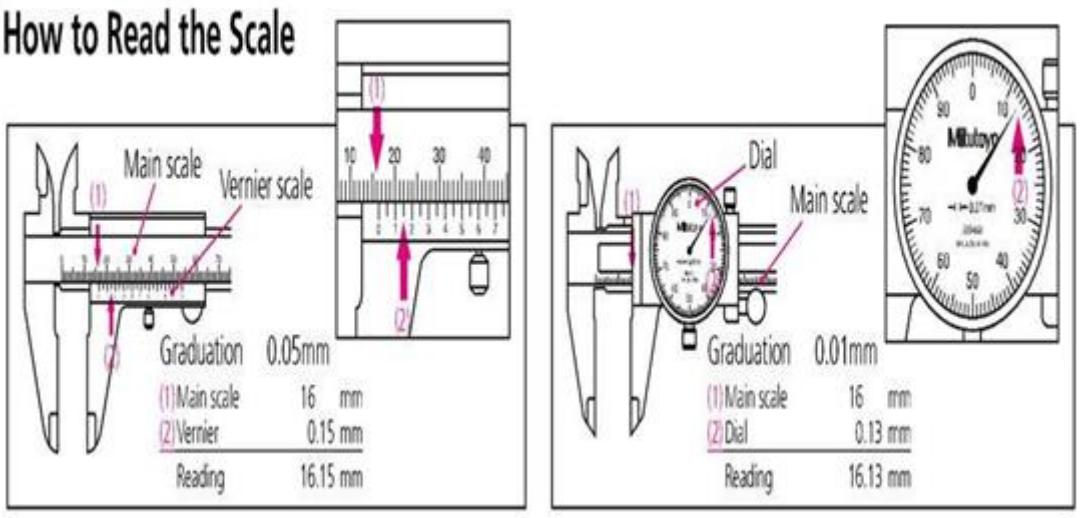


Figure 4.4. Vernier Reading system

Worked Examples

The example below explains how a reading is taken using a metric vernier caliper. Figure 51illustrates the case where the zero graduation on the vernier scale aligns perfectly with a graduation on the main scale.

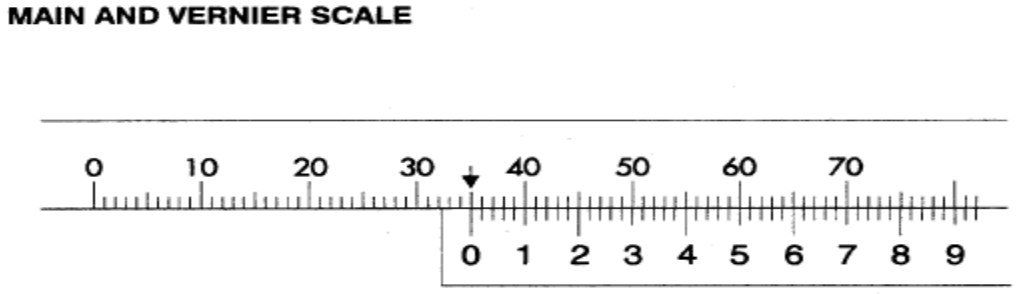


Figure 4.5 . caliper reading

In this case, the zero on the vernier scale and the 35 mm graduation on the main scale align. The correct caliper reading is therefore 35.00 mm.

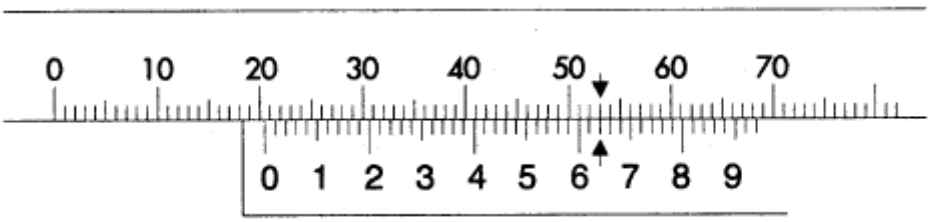


Figure 4.6. A non-integer reading on the main scale

illustrates the case where the zero on the vernier scale does not line up with a graduation on the main scale In this case the procedure for finding the correct reading is as follows.

Look along the main scale and determine the smaller of the two graduations that most closely line up with the zero on the vernier scale. In the example above (Figure52) the zero on the vernier scale lies between 20 and 21 on the main scale. The correct main scale reading to use is therefore 20.

Now look along the vernier or sliding scale and see which graduation lines up with a main scale division. In Figure this is 64 on the vernier scale.

The correct reading in the above example is therefore 20 mm + 0.64 mm = **20.64 mm**

Dial Caliper

Dial calipers are used extensively because they provide an easier method of reading when compared with vernier calipers thus allowing quicker measurement. The dial caliper uses the amplification mechanism of a dial gauge. Measurements are taken by adding the position of the pointer on the dial to the reading on the main scale. Before using dial calipers the jaws should be cleaned. The jaws should then be closed and the zero adjusted by rotating the scale bezel. With dial calipers it is essential that the toothed rack is kept clean. Contamination can cause the pinion gear to jump and lead to the pointer not returned to zero.

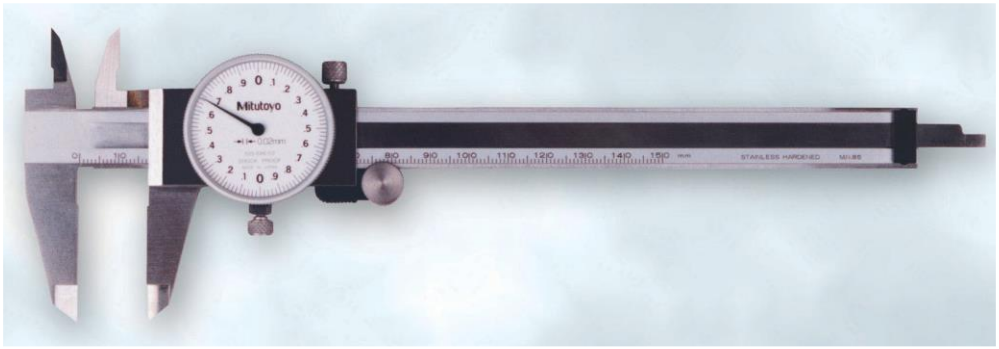


Figure.4.7 . Dial caliper

Vernier Height Gauge

A vernier height gauge (figure 62) is used for measuring height of an object or for marking lines onto an object of given distance from a datum base.



Figure 4.8. vernier height gauge

Micrometer

A micrometer is a more precise measuring instrument than the vernier calipers. The accuracy is come from the fine thread on the screw spindle. The ratchet prevents excess force from being applied. Generally, the screw spindle has a pitch of 0.5mm. The thimble is divided into 50 equal divisions.

Common types of micrometers used in the workshops are: -

Outside Micrometer

An outside micrometer (figure 55) is used for measuring external dimensions. The work to be measured is placed between the anvil and the tip of the spindle.



Figure 4.9. Outside Micrometer

Inside Micrometer

This is similar in structure to an outside micrometer and is used for measuring internal dimensions as shown in figure5 6.



Figure 4.10. Inside Micrometer

Depth Micrometer

A depth micrometer (figure 57) is used for measuring the depth of a hole, slot and keyway etc. A complete set of depth micrometer is equipped with spindles of different lengths, which can be interchanged to suit different measuring ranges.



Figure 4.11. Depth Micrometer

Metric system Micrometer Reading



Figure 4.12. Micrometer thimble reading

The spindle of an ordinary metric micrometer has **2 threads per millimeter**, and thus one complete revolution moves the spindle through a distance of **0.5 millimeter**. The longitudinal line on the frame is graduated with **1-millimeter divisions and 0.5-millimeter** subdivisions. The thimble has 50 graduations, each being 0.01 millimeter (one-hundredth of a millimeter). Thus,

the reading is given by the number of millimeter divisions visible on the scale of the sleeve plus the particular division on the thimble which coincides with the axial line on the sleeve.

Suppose that the thimble were screwed out so that graduation 5, and one additional 0.5 subdivision were visible (as shown in the image), and that graduation 28 on the thimble coincided with the axial line on the sleeve. The reading then would be 5.00 + 0.5 + 0.28 = 5.78 mm.

Dial Indicator

The principle of dial indicator (dial gauge) is that the linear mechanical movement of the stylus is magnified and transferred to the rotation of pointer as shown in figure 60. The accuracy of dial indicator can be up to 0.001mm. It is usually used for calibration of machine.



Figure 4.13 . Dial Indicator

Magnetic Base Indicator Stands

Magnetic Base Indicator Stands are precision tools used to hold and position measuring instruments such as dial indicators, test indicators, and dial test indicators. The stands consist of a magnetic base, an articulated arm, and a holder for the indicator.

The Magnetic Base is typically made of a strong magnet that securely attaches to ferromagnetic surfaces. The articulated arm is made up of a series of adjustable joints that can be moved and

locked into position. The indicator holder is mounted on the end of the arm and can be rotated and adjusted to accommodate different measuring instruments.

Magnetic Base Indicator Stands at GreatGages.com

Magnetic Base Indicator Stands are widely used in manufacturing, machining, and engineering industries to ensure precise and accurate measurements. They are particularly useful in situations where a measuring instrument needs to be held in a specific position while taking measurements, such as when inspecting machined parts, aligning machine tools, or checking for runout.

Overall, Magnetic Base Indicator Stands offer a stable and flexible platform for measuring instruments, enabling precise and accurate measurements to be taken with ease. They are durable, reliable, and easy to use, making them an essential tool in many industries.

Screw thread micrometer

It is used for accurate measurement of pitch diameter of screw threads. This micrometer has **a pointed spindle** and a **double V-anvil**, both correctly shaped to contact the screw thread of the work being gauged.

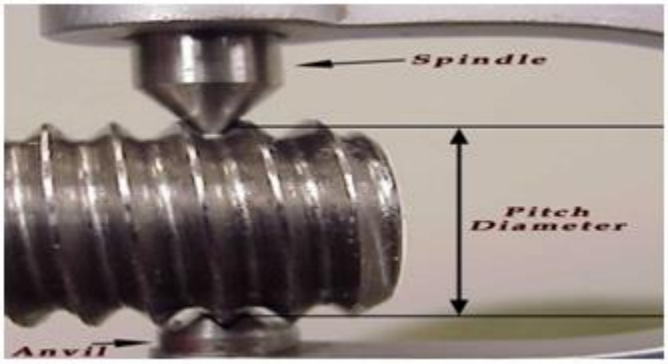


Figure 4. 14.. Screw thread micrometer

Slip Gauge

These are practical end standards and can be used in linear measurements in many ways like to **set the required length** to be used as a reference dimension.

Slip gauges are rectangular blocks of hardened and stabilized high grade cast steel or ceramic compound and are available with a **9mm wide**, **30 to 35mm long cross section**. The length of a

slip gauge is strictly the dimension which it measures- in some slip gauges it is the shortest dimension and in the larger slip gauges, it is the longest.



Figure 4.15. Slip Gauge

plug gage

A type of "Go" or "No-Go" gage used to admit or refuse the inside dimension of an object.

There are three types:

- plain cylindrical
- cylindrical taper
- thread plug gage



Figure 4.16. Different size Plug gage

comparator

Comparator graphically displays and measures dimensions and shapes that would be difficult to measure with regular tools.



Figure 4.17. Comparator

4.2. Checking Techniques Of Work Pieces

Dimensioning: are used to describe the sizes and relationships between features in your drawing. Dimensions are used to manufacture parts and to inspect the resulting parts to determine if they meet the drawing's specifications.

Linear dimensions are comprised of four components:

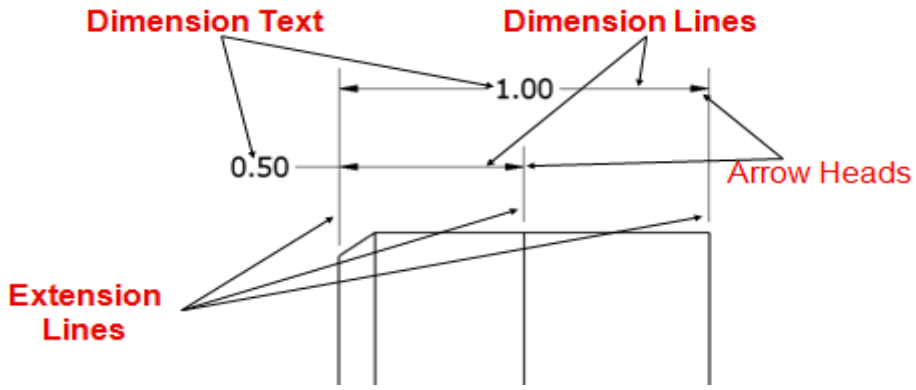


Figure 4.18. Linear dimensions

Dimensioning

Dimensions are represented on a drawing using one of two systems, unidirectional or aligned.

The **unidirectional method** means all dimensions are read in the same direction.

The **aligned method** means the dimensions are read in alignment with the dimension lines or side of the part, some read horizontally and others read vertically.

Types of Dimensions

There are two classifications of dimensions: size and location.

Size Dimensions are dimensions which indicate the overall size of the object and the various features which make up the object.

Location dimensions are used to locate various features of an object from some specified datum or surface.

Tolerance:

Tolerance: The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size. If the variation is provided on one side of the basic size, it is termed as unilateral tolerance. Similarly, if the variation is provided on both sides of the basic size, it is known as bilateral tolerance.

Types of Tolerances

Dimensional tolerance is the total amount a specific dimension is permitted to vary, which is the difference between maximum and minimum permitted limits of size.

Geometric tolerance is the maximum or minimum variation from true geometric form or position that may be permitted in manufacture.

Geometric tolerance should be employed only for those requirements of a part critical to its functioning or interchangeability.

How Is Tolerance Specified?

Tolerances can be expressed in several ways.

General Tolerances :- A note may be placed on the drawing which specifies the tolerance for all dimensions except where individually specified.

N.B All Decimal Dimensions To Be Held To ±0.020

Specific Tolerances:- The tolerance for a single dimension may be specified with the dimension based on one of the following methods.

Limit Dimensions:- Permissible variation in a part feature size, consisting of the maximum and minimum dimensions allowed.

Unilateral Tolerance:- Variation from the specified dimension is permitted in only one direction, either positive or negative, but not both.

Bilateral Tolerance:- Variation is permitted in both positive and negative directions from the nominal dimension. It is possible for a bilateral tolerance to be unbalanced;

for example,

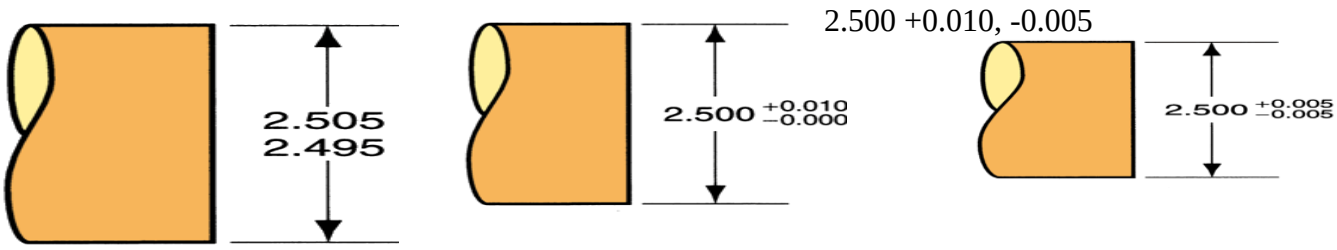


Fig4.20 A. Limit Dimensions **fig4.20 B**. Unilateral Tolerance **fig 4.20 C**. Bilateral Tolerance

Surface Finish

- An engineering component may be cast, forged, drawn, welded or stamped, etc.
- All the surfaces may not have functional requirements and need not be equally finished
- Some surfaces (owing to their functional requirements) need additional machining that needs to be recorded on the drawing

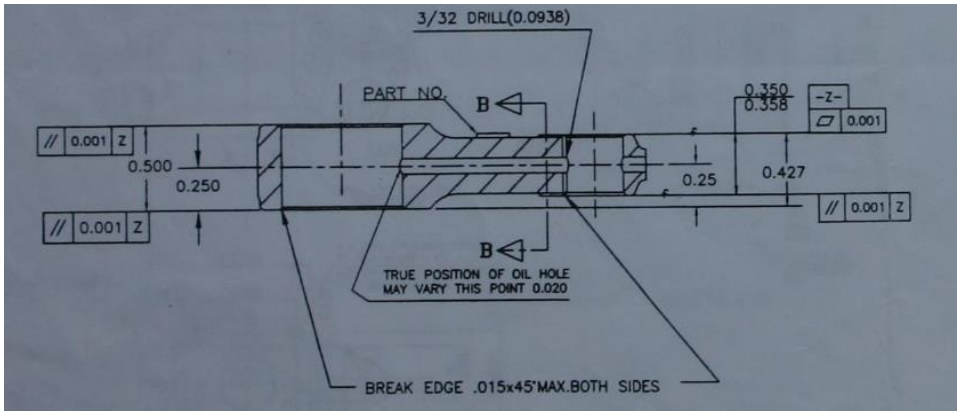


Figure 4.21. Surface Finishing

Surface Roughness

The geometrical characteristics of a surface include,

- Macro-deviations,
- Surface waviness, and

Micro-irregularities

The surface roughness is evaluated by the height, R_t and mean roughness index R_a of the micro-irregularities.

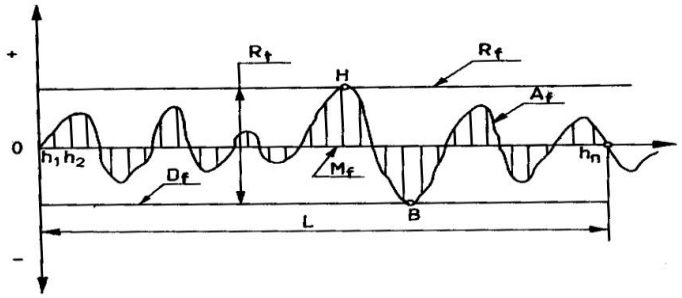


Figure4.22. surface roughness

Indication Of Surface Texture

The **basic symbol** consists of two legs of **unequal** length inclined at approximately **60'** to the line representing the considered surface The **symbol must be represented by thin line**

If the removal of material by **machining** is required, a bar is added to the basic symbol,

If the **removal of material is not permitted**, a circle is added to the basic symbol

When **special surface characteristics** have to be indicated, a line is added to the longer arm of any of the above symbols,

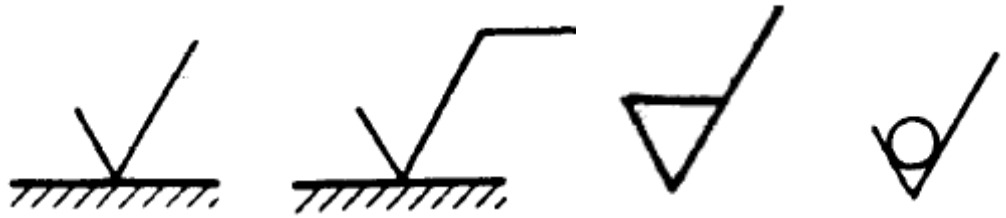


Figure 4.23. . surface texture

Symbols with Additional Indications.

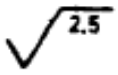
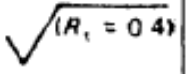
Symbol	Meaning
	Production method: milled.
	Sampling length: 2.5 mm
	Direction of lay: perpendicular to the plane of projection of the view.
	Machining allowance: 2 mm
	Indication (in brackets) of a criterion of roughness other than that used for R_a for example $R_t = 0.4\ \mu m$

Table 4. 24.. surface roughness

4.3. Handling Deviation

Deviation: It is the algebraic difference between a limit of size and the corresponding basic size.

a) Upper Deviation: It is the algebraic difference between the maximum limit of size and the corresponding basic size. It is denoted by letters ‘ES’ for a hole and ‘es’ for a shaft.

b) Lower Deviation: It is the algebraic difference between the minimum limit of size and

c) the corresponding basic size. It is denoted by letters ‘EI’ for a hole and ‘ei’ for a shaft.

C) Fundamental deviation. The minimum difference in size between a component and the basic size. This is identical to the upper deviation for shafts and the lower deviation for holes. If the fundamental deviation is greater than zero, the bolt will always be smaller than the basic size and the hole will always be wider

4.3.1. Calculation of Fundamental Deviation

Fundamental Deviations can be calculated with the help of some formulas given below.

1. Calculation Fundamental Upper and Lower Deviation of Shaft:

2. es = ei +IT, ei = es-IT.

3. Calculation Fundamental Upper and Lower Deviation of Hole:

4. ES=EI+IT, EI=ES-IT

Explanation

For making hole and shaft assembly two types of systems is adopted internationality.

There are

- Shaft basis system
- Hole basis system

Shaft basis system

In this system , the size of the shaft is kept constant, and the diameter of the hole is varied to given the various fits.

The basic size of the shaft is taken as the upper deviation of the shaft is zero.

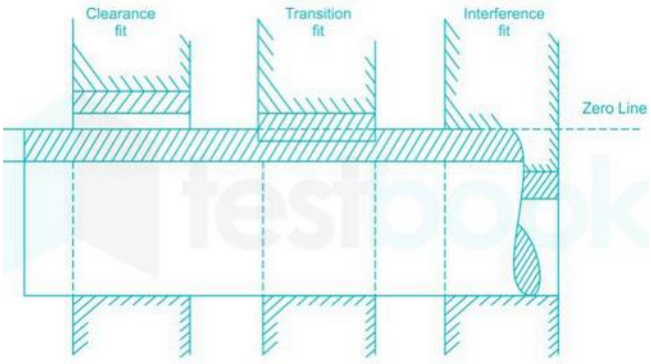


Figure. 4.26. Shaft basis system

Hole basis systems

- In this system, the designation of tolerance of the hole is kept constant and the shaft diameter is varied to obtain the various types of fits.
- The basic size of the hole is taken as the lower size limit of the hole i.e. lower deviation of the hole is zero.
- In general practice, the hole basis system is adopted.
- Various tools like standard drills, reamers, broachers are easily available and shafts are later given tolerance to produce the desired fit.

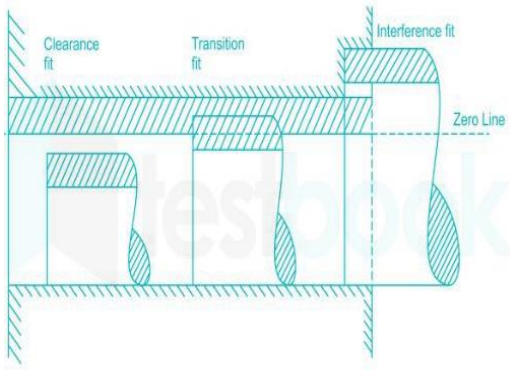


Figure. 4.26. Hole basis system

Self-Check -4.1

Directions: I. Answer all the questions listed below.

Choose the correct answer for the following alternative

1. Graphically displays and measures dimensions and shapes that would be difficult to measure with regular tools..

   A. Comparator B. plug gage C. Slip Gauge D. None

2. ----------**:** it is used for accurate measurement of pitch diameter of screw threads

   A, Slip Gauge B. Dial Indicator C. Screw thread micrometer D. None

3. A type of "Go" or "No-Go" gage used to admit or refuse the inside dimension of an object.

   A. Comparator B. plug gage C. Slip Gauge D. None

4. A types of Micrometer used for measuring the depth of a hole, slot and keyway

   A. Inside Micrometer C. vernier Caliper

   B. Outside Micrometer D. A depth micrometer

5. Which one of the following is used for measuring internal dimensions

   A. Outside Micrometer B. Inside Micrometer C. Depth Micrometer D. All

6. Which one is a more precise measuring instrument ?

   A. Micrometer B. Caliper C. steel rule D. All

Directions: II if the statement is correct say true or if the statement is wrong say False

7. Dimensioning: are used to describe the sizes and relationships between features in your drawing.
8. The basic size of the shaft is taken as the upper deviation of the shaft is zero
9. The basic size of the hole is taken as the lower size limit of the hole i.e. lower deviation of the hole is zero.
10. The surface roughness is evaluated by the height, R_t and mean roughness index R_a of the micro-irregularities.

Directions: III Write the correct answer for the following question

11. ---------- used for measuring height of an object or for marking lines onto an object of given distance from a datum base

12. :For making hole and shaft assembly two types of systems is adopted internationality.

There are A, ---------- B.----------

13. . ----------It is the algebraic difference between a limit of size and the corresponding basic size

Operation Sheet 4.1.

Operation title: Polishing on a lathe operation of shaft

Purpose: To fine the surface roughness

Instruction: Use tools and equipment and then follow given steps. For this operation you have given 2 Hour and you are expected to show the shape of the shaft is very fine

Tools and requirement:

- Cutter
- strip of abrasive cloth
- suitable for the material
- sandpaper
- polisher
- machine oil
- micrometer
- vernier caliper

Quality Criteria: must have good accuracy with in Tolerance is ± .025

Precautions: Check and care all activity

Procedures in doing the task; turning operations performed on a given mild steel work piece

Steps 1-Apply safety precaution

Steps 2- Using the Outside Calipers the diameter of the given work piece is measured.

Steps 3- The facing operation required

Steps 4- The work piece is placed in the 3 jaw chuck.

Steps 5- The tail stock is moved so that the work piece is held between the chuck and the dead center

Steps 6- The tool aligned by placing such that, the tip of the tool is perpendicular to the tip of the dead center. The tool post is tightened using align key.

Steps 7- The tool is made to touch the work piece & the depth of the cut is adjusted to get the diameter of 30 mm using a micrometer provided in compound rest or in the carriage.

Steps 8- Cutting parameters like speed, feed, depth of cut, etc are selected before the machine is turned on.

Steps 9- Plain turning operation is carried out for required length of 160mm.

Steps 10- Markings are made to perform step turning operation.

Steps 11- Step turning operations are made to get the desired shape of the work piece by giving depth of cut on each run.

Steps 12- turning operations are made to get the desired shape of the work piece by giving depth of cut on each run.

Steps 13 polishing by sand paper

Steps 14 The finer of the abrasive should used

Steps 15 the finer of the result is finish.



- Tolerance is ± .025

List Of Reference Materials

1. Department Of mechanical Engineering Workshop and Machine shop lab manual
2. Elements of work shop technology 7th Edition 1996 by S.K Hajrachoudhury, S.K Bose
3. Introduction to Basic Manufacturing Processes and Workshop Technology
4. Metal work Technology 1st 2007 J.K. SV.SACKEY AND S.K AMHAKOBENE
5. Manufacturing Technology 3rd edition 1998 M.HASLEHURST
6. Engineering drawing 1st ,2nd&3rd Impression 2007,2009 ,M.B.shah B.c Rana
7. Machine drawing 3rd edition K.L. Narayana,P.Knnaiah,k.Venkata Reddy
8. Poe.com

Developers profile

No	Name	Qualification (Level)	Field of Study	Organization/ Institution	Mobile No.	E-mail
1	Tsegaye Mulat	A	Manufacturing Technology	A.A T/ P/T/C	0910411511	ttshgm97@gmail .com
2.	Abebe Hailay	A	Manufacturing Technology	M/G/M/B/P/T/college	0913175634	aabebe52@gmail.com
3.	Mesret woldemariam	A	Manufacturing Technology	W/P/T/college	912117416	bmesi2007@gmail.com
4.	Boka Terefe	A	Manufacturing Technology	Ambo/P/T/college	0917038870	bokaterefe@gmail.com
5.	Mesay Teshome	B	Manufacturing Technology	Hawassa P/T/college	0905456888	mesayy2018@gmail.com
6.	Mohammed Bedri	A	Manufacturing Technology	Robe/P/T/college	0913897297	
7.	Mohammed Abdela	A	Manufacturing Technology	Ethio italy poly technic college	0907755560	Mahammedabdella49@gmail.