MACHINING -LEVEL II

Based on Version 2 February 2017

Occupational Standard (OS)

Training Module –Learning Guide 24-27

Unit of Competence: **Perform Intermediate Lathe Operations**

Module Title: **Performing Intermediate Lathe Operations**

TTLM Code: **IND MAC2 TTLM08 1019v1**

October 2019

Module Title:	**Performing Intermediate Lathe Operations**
TTLM Code:	**IND MAC2 TTLM 1019v1**

LG 24: Determine job requirements
LG Code: IND MAC2 M08 LO1-LG-24
LG 25: Set-up work piece
LG Code: IND MAC2 M08 LO2-LG-25
LG 26: Perform lathe operations
LG Code: IND MAC2 M08 LO3-LG-26
LG 27: Comply with Quality Assurance
LG Code: IND MAC2 M08 LO4-LG-27

INSTRUCTION SHEET	LEARNING GUIDE #24

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

- Interpreting drawings.
- Determining Sequence of operation.
- Selecting cutting tools and work piece materials.

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

- Interpreting Drawings to produce component to specifications.
- Determining Sequence of operation to produce component to specification
- Selecting cutting tools and work piece materials according to the requirements of the operation

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the “Information Sheets”. Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the “Self-checks” which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to “Operation sheets
7. Perform “the Learning activity performance test” which is placed following “Operation sheets” ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to “Operation sheets”.

1.1definition of drawing 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.

1.2 Alphabet of Lines & Their Application

Lines of various forms & thickness are used as alphabets of the graphic language. If these lines are properly & systematically composed, they have the capacity to describe the shape of an object adequately.

It is beneficial to develop the capacity of discriminating each line in shape & thickness. The alphabet of line may be categorized in to three group based of their weight or thickness.

- The Object line, the cutting plain line& the short break lines should be drawn thick.
- The center lines, dimension lines, extension lines, long break lines, & phantom line should be drawn thin.
- The hidden line should have an intermediate thickness between the thin & the thick lines.

In fact, thick lines are (0.5 to 0.8mm) wide, thin lines between (0.3 to0.5mm) wide.

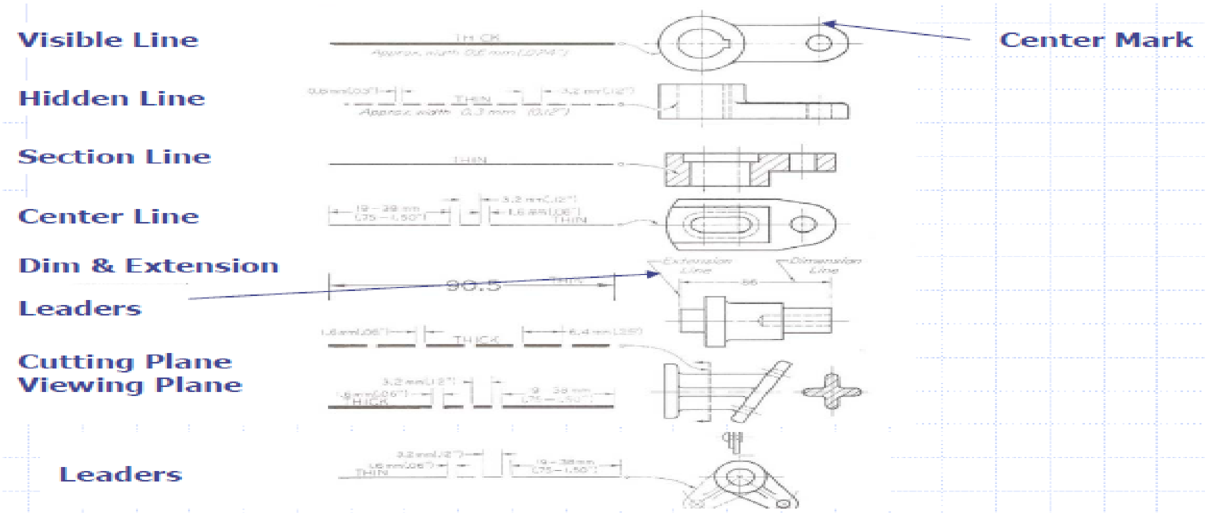


Figure.1 types of lines

1.3 Orthographic projection – Multi view Projection

- 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.

Planes of projection

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

1. Horizontal projection plane.
2. Frontal projection plane.
- Profile projection plane.

The position of the plane is illustrat w

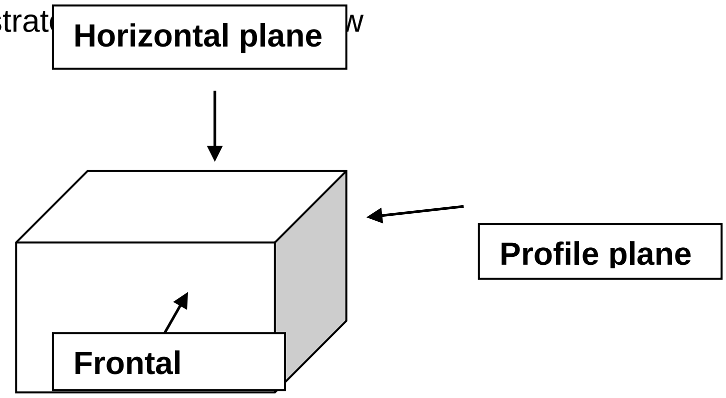
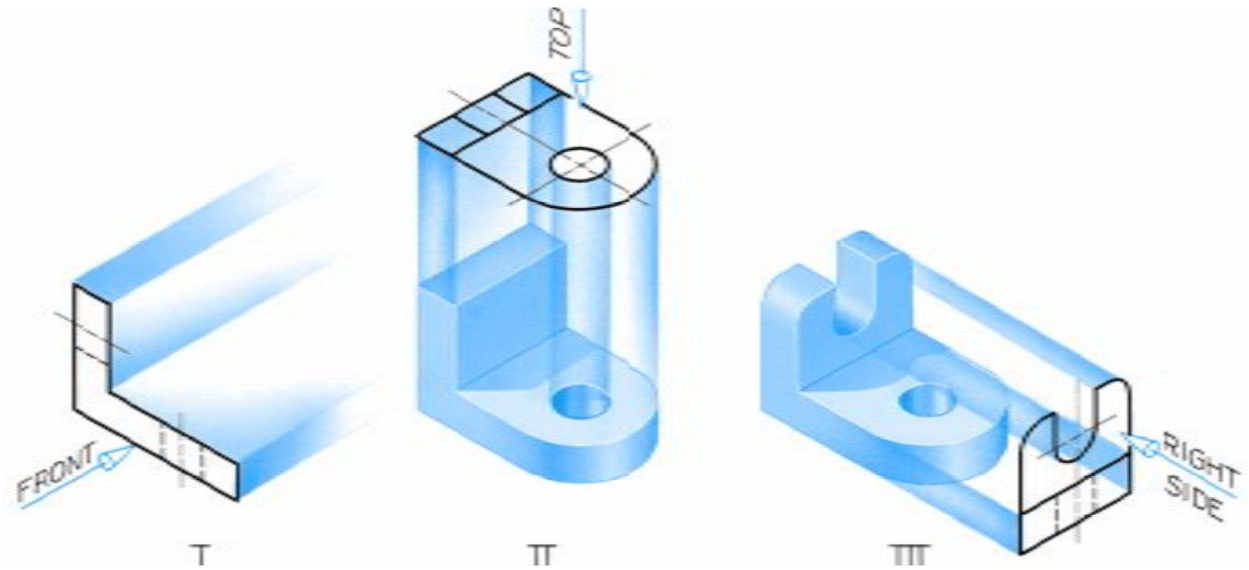


Figure.2 Planes of projection



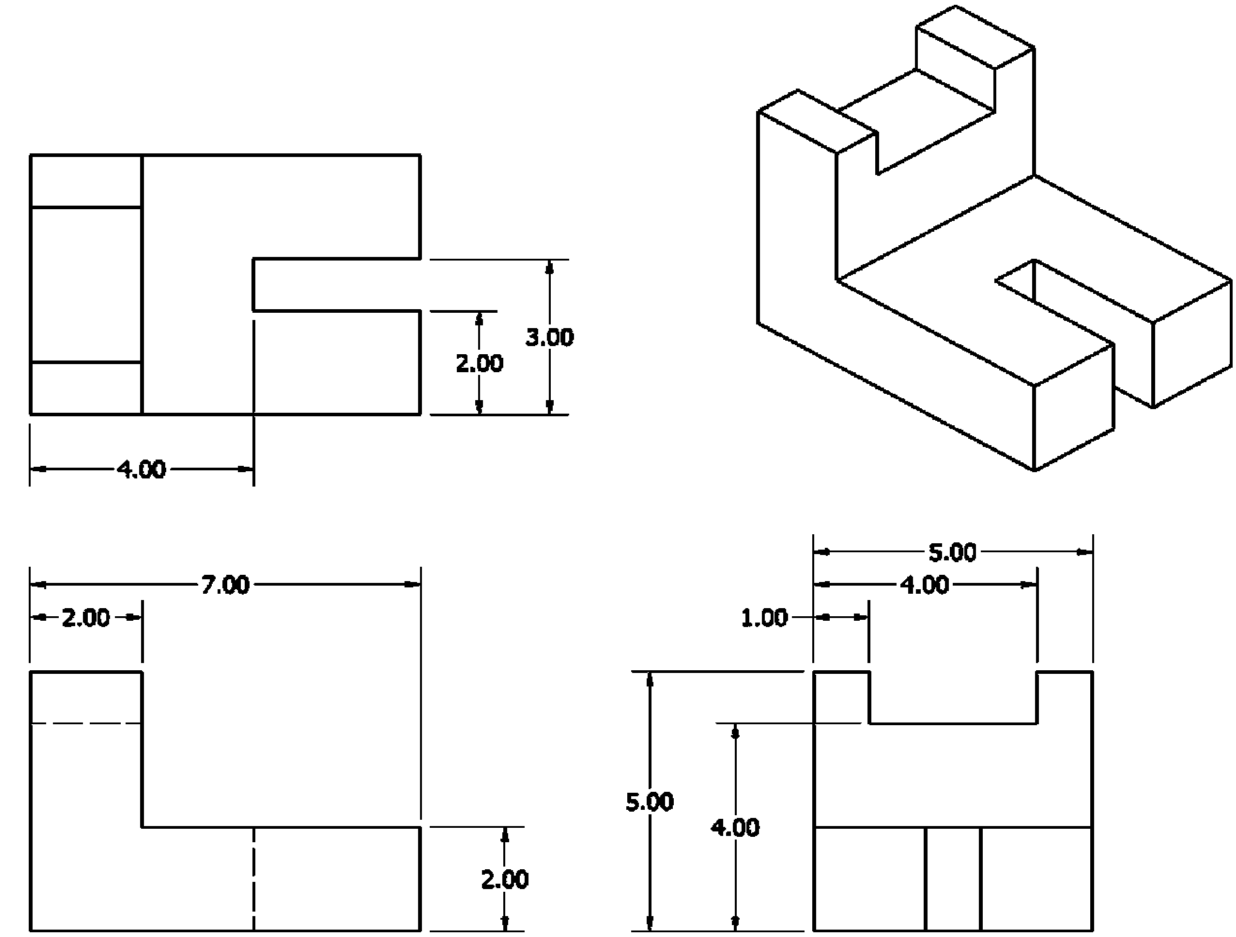


Figure.3 views using 3rd angle projection

SELF-CHECK -1	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1.________________ means reading and understanding the information given by the drawing about the component to be manufactured. (1 point)

a.Projection C. line

B.Drawing interpretation D. all

2.________________ are convectional projections of a three-dimensional object on a two dimensional plane. (1 point)

A. Multi-view drawings C. Isometric Drawing

B. Orthographic Drawing D. A and B

3. Which of the Planes of projection used to locate front view? (1point)

A. Horizontal projection plane. C. Frontal projection plane.

B. Profile projection plane. D. All of the above

4. Which Alphabet of Lines should be drawn with thin lines? (1 point)

A. The center lines C. The short break lines

B. The cutting plain line& D. The Object line,

***Note:* Satisfactory rating – above 2points Unsatisfactory – below 2points points**

You can ask you teacher for the copy of the correct answers.

Information sheet-2	Determining sequence of operation

2.1 Operational sequences

It is a sequence of machining operation plan. Which is prepared based on the information provided on the drawing of the component to be manufactured.

The Plan includes

- Which sequel of machining operation is efficient.
- Tools and equipments required for each machining operation.
- Methods of chucking and examine the component in each machining Operation.

2.2 Definition of lathe machine

The lathe is a versatile machine tool in which the work is held and rotated. a cutting tool is moved along the work to produce cylindrical shapes (turning) or across the work to form flat surfaces (facing).

2.3 Types of Lathe Machine

- **Speed Lathe:** it is so named because of the very high speed of the headstock spindle
- **Engine Lathe :**the most important machine tool in the lathe machines and by far most widely used.
- **Turret Lathe:** it is a production used to perform a large number of operation simultaneously
- **Bench Lathe:** a small lathe which can be mounted on the work bench for doing small precision and light jobs.

2.4 Parts of the lathe machine and their functions

There are 5 parts of lathe machine

- **BED:** The bed is the part of the lathe that provides support for the other components. It is the foundation on which all the other parts are fitted.
- **HEADSTOCK:** The headstock is the lathe feature that provides the means of holding and rotating the work accurately.
- **TAILSTOCK:** The tailstock is used mainly to support the right hand end of the work. It may be moved and clamped in position along the bed. The tailstock spindle is located at the same height and parallel to the headstock spindle.The spindle may be moved in and out of the body by means of a hand wheel and threaded spindle.
- **CARRIAGE:** The carriage is the lathe that provides the method of holding and moving the cutting tool accurately. It consists of two major parts:

  - ✓ **Apron:** The apron is bolted to the front of the saddle. It contains the mechanism for moving and controlling the carriage.
  - ✓ **Saddle:** The saddle is the part of the carriage that fits across and moves along the bed, between the head and tailstock. The **cross slide** is mounted on top of the saddle and provides a cross movement

for the cutting tool. The slide is at right angles to the bed and is moved by means of a screwed spindle fitted with a handle. The **compound rest** is fitted on top and to the front of the cross slide. The compound rest may be swiveled horizontally through 360°.

- **FEED MECHANISM:** Most center lathes have some form of power feed mechanism to provide automatic movement to the cutting tool. Power feed can be applied to both the saddle and cross slide.
  - ✓ **QUICK CHANGE BOX:** This box provides a means of changing the speed of the feed shaft. The rate of feed can be varied in relation to the revolutions of the work to suit different materials and operations.
  - ✓ **FEED SHAFT:** Power is transmitted from the rear end of the headstock spindle through the quick change gear box to the feed shaft located at the front of the lathe bed.
  - ✓ **LEAD SCREW:** Center lathes equipped with power feed also have provision for screw cutting. A special threaded spindle is mounted on the front of the lathe bed adjacent to the feed shaft. It is driven through the quick change gear box. Both the lead screw and the feed shaft pass through the apron of the carriage. Controls on the apron enable the feed shaft or the lead screw to be connected to the carriage.
  - ✓ **FEED LEVER:** When the feed lever is engaged power is connected from the feed shaft to the saddle and cross slide. This provides automatic feeding for facing and turning.
  - ✓ **LEAD SCREW ENGAGEMENT LEVER:** The lead screw is connected and released from the carriage by means of half nuts. The lead screw engagement lever on the apron operates the half nuts. These nuts are halved to enable the lead screw to be engaged or disengaged easily. An interlocking mechanism between the levers prevents the lead screw and the feed shaft from being engaged at the same time. The arrangement of gears in the feed drive mechanism provides a means of reversing the direction of rotation of the feed shaft and the lead screw. Moving the feed directional lever to reverse does this.

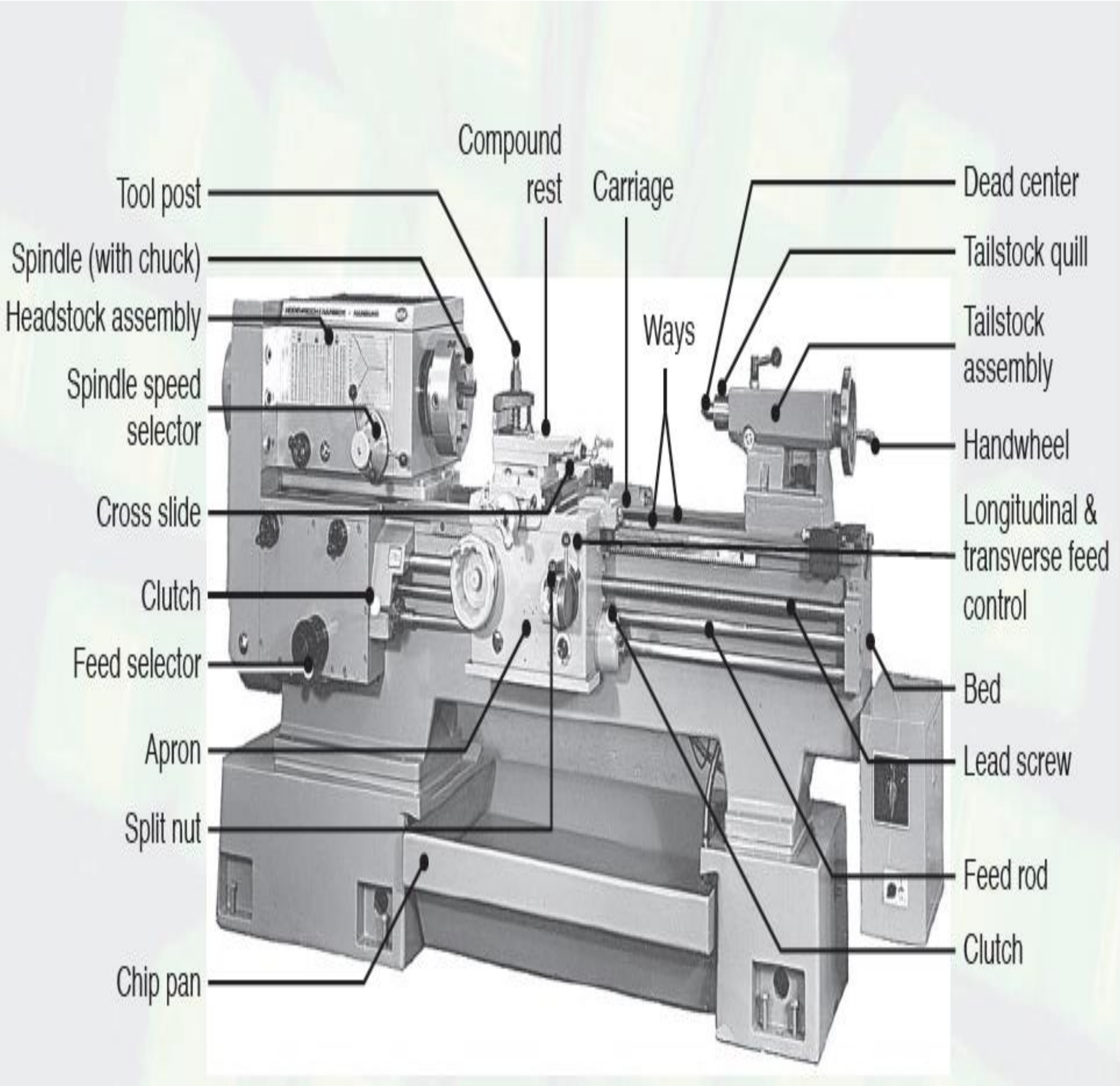


Figure.4 parts of lathe machine

SELF-CHECK -2	WRITTEN TEST

1. Part of the lathe that provides support for the other components. (1point)
   a) Carrier b) Lathe Bed c) Headstock d) None
2. The lathe feature that provides the means of holding and rotating the work accurately. (1point)
   a) Spindle b) lath Chuck c) Headstock d) All
3. Which one is not belongs to the parts of carriage? (1point)
   a) Head stock b)apron c) cross slide d) compound rest
4. The most important types of Lathe Machine tool in the lathe machines and by far most widely used is------?(1point)
   a) Speed Lathe b) Engine Lathe c) Turret Lathe d) Bench Lathe

Note: **Satisfactory rating –above 2 points Unsatisfactory - below 2 points**

Information sheet-3	Selecting cutting tools and work piece materials

3.1 TOOL MATERIAL

A machine tool is no more efficient than its cutting tool. There is nothing in shop work that should be given more thoughtful consideration than cutting tools. Time is always wasted if an improperly shaped tool is used. The cutting action of the tool depends on its shape and its adjustment in the holding device. Lathe cutter bits may be considered as wedges which are forced into the material to cause compression, with a resulting rupture or plastic flow of the material. The rupture or plastic flow is called cutting. To machine metal efficiently and accurately, it is necessary that the cutter bits have keen, well-supported cutting edges, and that they be ground for the particular metal being machined and the type of cut desired. Cutter bits are made from several types of steel, the most common of which are described in the following subparagraphs.

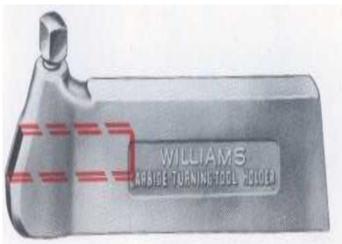
- **High-Speed Steel:** High-speed steel is alloyed with tungsten and sometimes with chromium, vanadium, or molybdenum. Although not as hard as properly tempered carbon steel, the majority of lathe cutting tools are made of high-speed steel because it retains its hardness at extremely high temperatures. Cutter hits made of this material can be used without damage at speeds and feeds which heat the cutting edges to a dull red.
- **Tungsten Carbide:** Tungsten carbide is used to tip cutter bits when maximum speed and efficiency is required for materials which are difficult to machine. Although expensive, these cutter bits are highly efficient for machining cast iron, alloyed cast iron, copper, brass, bronze, aluminum, Babbitt metal, and such abrasive nonmetallic materials as fiber, hard rubber, and Bakelite. Cutter bits of this type require very rigid support and are usually held in open-side tool posts. They require special grinding wheels for sharpening, since tungsten carbide is too hard to be redressed on ordinary grinding abrasive wheels.
- **Drill Bit:** Refers to cutting a circular hole in a material. It has already been mentioned that, before reaming, holes should be drilled. Drilling and reaming are therefore not the same process.
- **Boring tool:** Is ground the same as a left-cut turning tool however the front clearance must be ground at a slightly greater angle .this will prevent the heel of the tool from rubbing in the hole of the work piece.
- **Knurling tool:** The knurling tool is a tool post type tool holder on which a pair of hardened steel rollers is mounted. These rolls may be obtained in diamond and straight line patterns.
- **Parting off tool:** Parting tool should be always set exactly on center. It may sharpened by grinding the end of the cutter blade should have sufficient taper to provide side clearance.
- **Threading tool:** has a 60^0 included angel that confirms with the unified national 60^0 included angle thread. it is used for V-grooving and chamfering .Threading tools for square and acme threads can be ground according to their included angle

3.2 Cutting tool holder

- Straight tool holders: The straight tool holder is generally purpose type. It can be used for taking cuts in either direction and for general machine operation
- Left tool holder: The left hand offset tool holder is designed for machining work close the chuck or face plate for cutting from right to left.
- Right tool holder: The right hand offset holder is designed for machining work close to the tail stock, for cutting from left to right and for facing operations.
- A threading tool holder: A threading tool holder is designed to hold a special form-relieved thread cutting tool.
- The carbide tool holder: The carbide tool holder: has a square hole parallel. To the base of the tool holder, to accommodate carbide tipped tools.
- Cutting-off /parting tools holder: Cutting-off /parting tools: is used to hold the long, thin cutting off blade that can be locked securely in the tool holder by means of either a cam lock or locking nut.
- Boring tool holder: Boring tool holder: are made in several styles. Alight duty tool holder, medium boring tool holder and a heavy-duty boring bar holder.

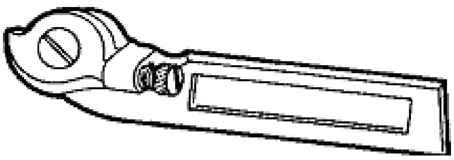
5A. Left hand tool holder

5B. Straight tool holders

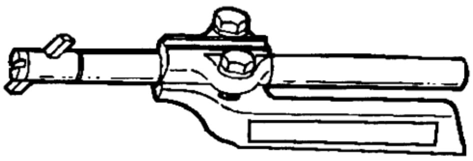


5C. Right hand tool holder

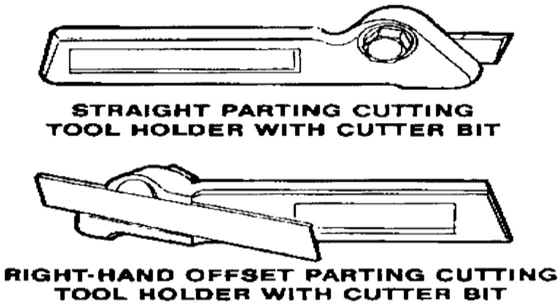
5D. The carbide & ceramic inserts
tool holder



5E. Thread cutting tool holder



5 F. Boring tool holder



5G. Cutting-off /parting tools holder

Figure.5A-5G Cutting tool holder

SELF-CHECK -3	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

Matching item(1 point)

Column ``A``	**Column ``B``**
1. Left tool holder	A. Refers to cutting a circular hole in a material
2. Drill Bit	B. Designing for cutting from left to right and for facing operations.
3. Knurling tool	C. confirms with the unified national 60^0 included angle thread.
4. A threading tool holder	D. be obtained in diamond and straight line patterns.
5. Right tool holder	E. Designing for cutting from right to left and for facing operations.
6. Threading tool	F. designed to hold a special form-relieved thread cutting tool.
	G**.** is used to hold the long, thin cutting off blade

Note: **Satisfactory rating –above 3 points** **Unsatisfactory - below 3 points**

You can ask you teacher for the copy of the correct answers.

OPERATION SHEET 1	TECHNIQUES OF SELECTING AND SETTING CUTTING TOOLS

1.1 PROCEDURES FOR SETTING single point CUTTING TOOLS

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.

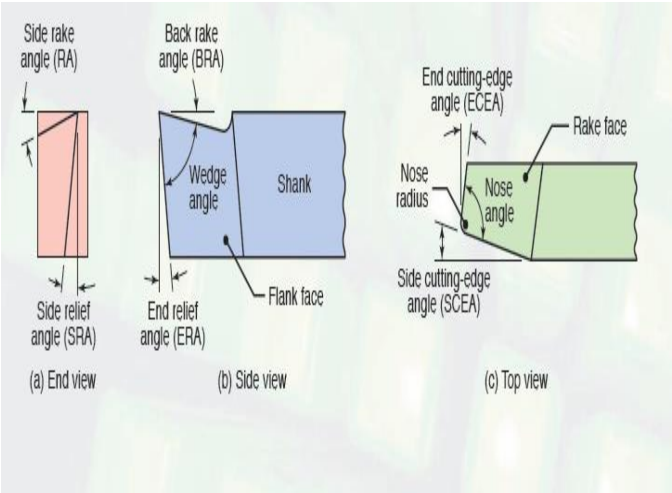
LAP TEST	PRACTICAL DEMONSTRATION

Name: ____________________________ Date: _______________

Time started: ________________________ Time finished: _______________

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

TASK 1: Selecting and setting cutting tools According to grinding



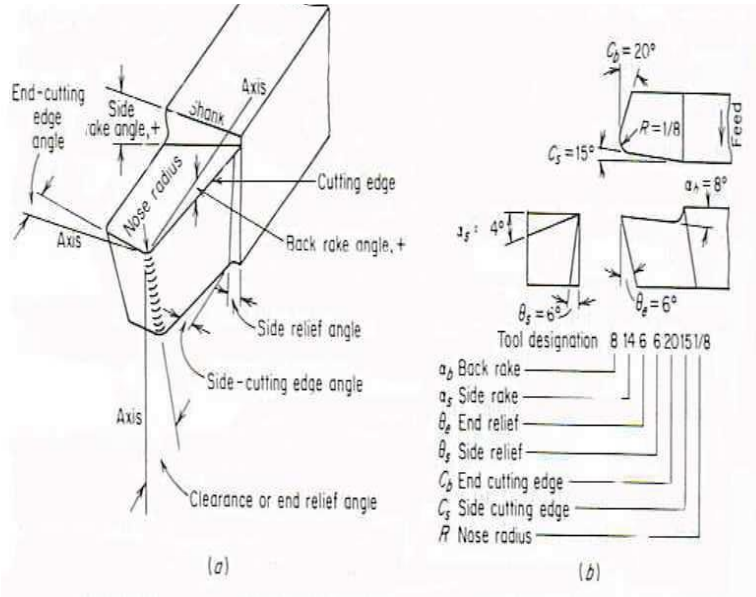


Figure.6. setting cutting tools According to grinding

INSTRUCTION SHEET	LEARNING GUIDE #25

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

- Applying safety precaution.
- Mounting and centering Work piece.
- Setting up work piece using appropriate instruments/equipment.

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

- Mounting and centering Work piece on chuck to required level of accuracy using appropriate tools and equipment and in accordance with worksite procedures.
- Setting up work piece using appropriate instruments/equipment based on standard procedure.
- applying safety procedures and using personal protective devices due to OHS regulations

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the "Information Sheets". Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the "Self-checks" which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to "Operation sheets
7. Perform "the Learning activity performance test" which is placed following "Operation sheets" ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to "Operation sheets".

Information sheet-1	Applying safety precaution

1.1 Safety precaution

- **Personal safety**
  - ✓ dress properly remove necktie, necklace, wrist, watch, rings and other
  - ✓ Jewelries and loose fitting. Sweater wear and apron or a properly shop fitted Coat and safety glasses are a must.
  - ✓ Be sure all guards are in place before attempting to operate the machine
  - ✓ When cleaning the lathe, do not remove chips with bare hands, an air
  - ✓ Hose should never be used to remove chips. The flying particles might injure Your or nearby person.
  - ✓ do not operate machines while taking medication because of possible
  - ✓ drawness.
  - ✓ care must be taken when handling long sections of metal stock. Secure help when moving heavy machine accessories or large piece of metal stock.
  - ✓ Back injuries are usually long term injuries
  - ✓ Avoid horse play

- **Machine safety**
  - ✓ No attempt should be made to operate a lathe until you know the proper procedure and have been checked out in its safe operation by your instructor.
  - ✓ Avoid using compressed air to remove chips and cutting oil from machine.
  - ✓ Keep the machine clear of tools
  - ✓ Turn the face plate or chuck by hand to ensure there is no binding or danger of the work striking any part of the lathe.
  - ✓ keep hand tools in good conditions and store them in such a way that peoples cannot be injured when a tool is taken from a tool panel or storage rack

- **Work place safety**
  - ✓ Keep the shop clean. Metal scraps should be placed in the scrap bin. Never allow them to remain on the bench or floor.
  - ✓ Oily rugs must be placed in approved safety containers.(metal can with metal lid)
  - ✓ Avoid smoking in the work shop

SELF-CHECK -1	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

I. Choose

1. Dress appropriately remove necktie, necklace, wrist, watch & rings are belongs to: (1 points)
   a/ machine safety b/ Work shop safety c/ Personal safety d/ a and b e/ all

II.Matching(each 1 point)

Column "A"	Column "B"
1.Machine safety	A. Keep the shop clean
2.Work shop safety	B. Keep the machine clear of tools
3.Personal safety	C. do not remove chips with bare hands
	D. use compressed air to clean the machine

Note: **Satisfactory rating –above 2 points** **Unsatisfactory - below 2 points**

You can ask you teacher for the copy of the correct answers.

INFORMATION SHEET-2	MOUNTING AND CENTERING WORK PIECE

2.1 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?

2.2 METHODS OF MOUNTING WORK

- **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.

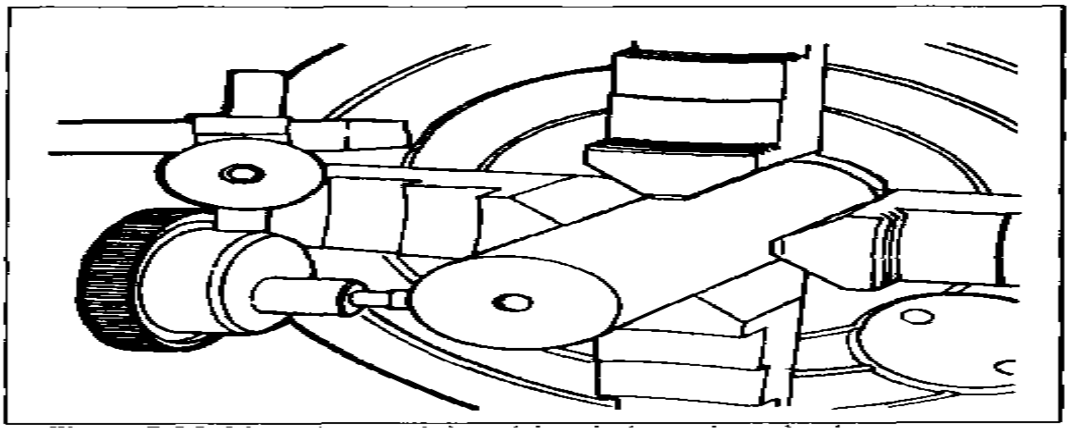


Figure7. Mounting work in a 4-jaw independent chuck

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.

- **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.

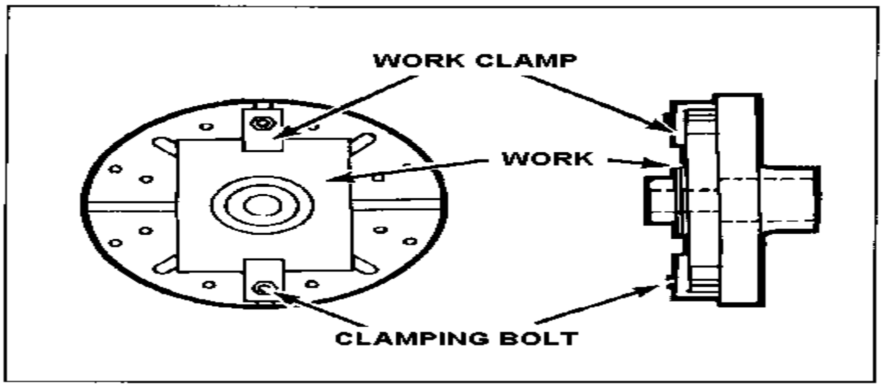
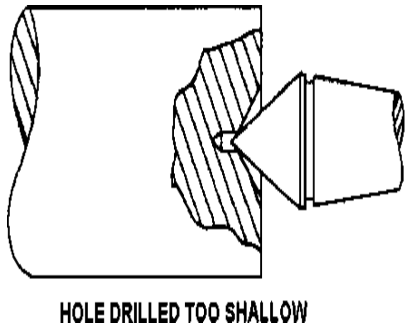
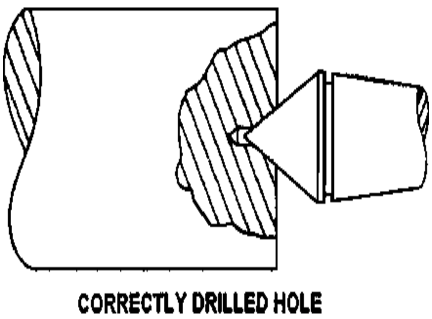


Figure8. Mounting Work to Faceplates

- **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.





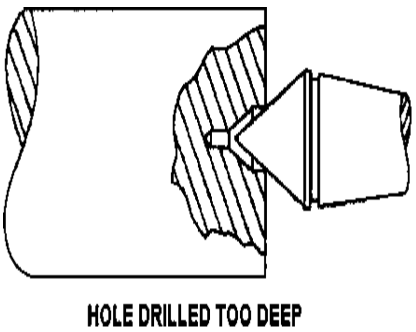


Figure9. Correctly and incorrectly drilled hole

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.

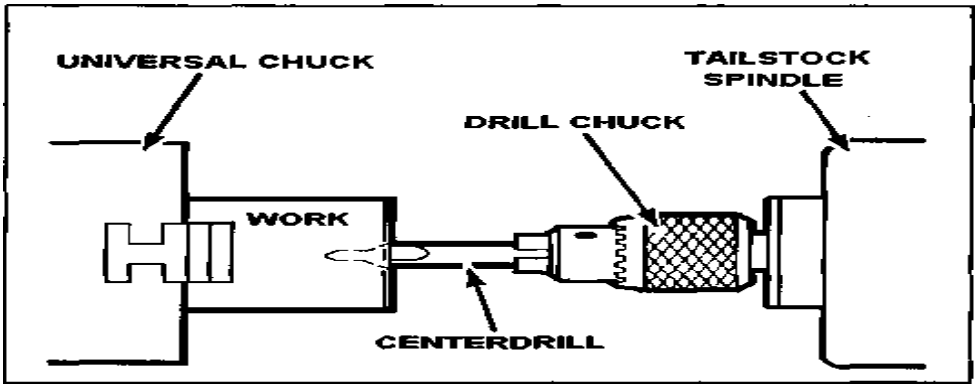


Figure10.center drilling

- **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 (Figure 11). 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.

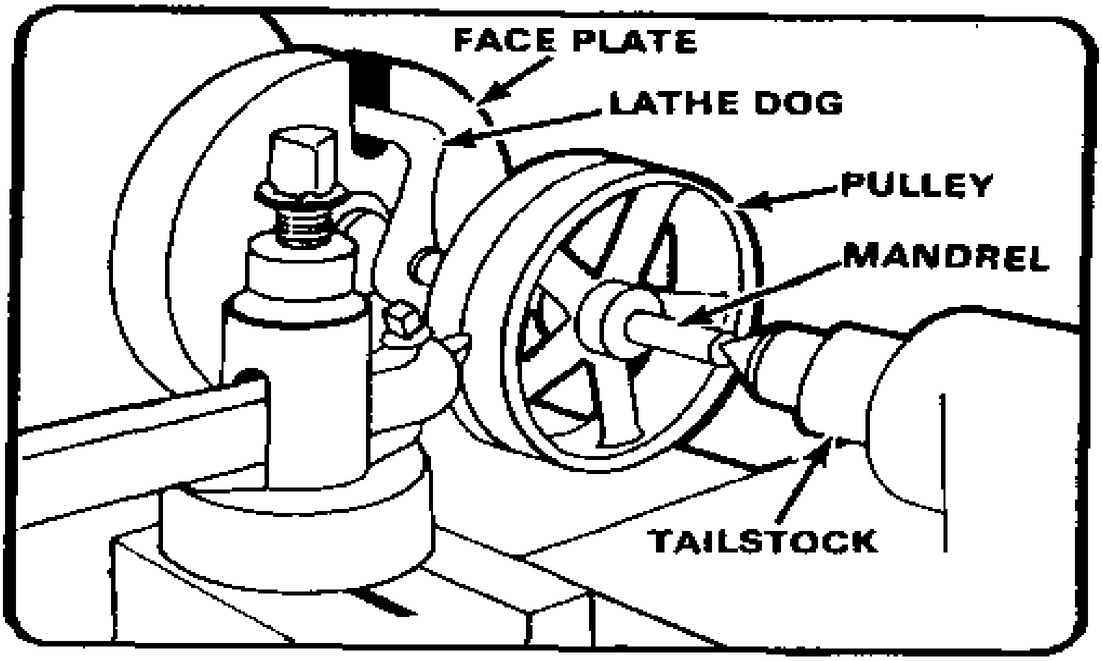


Figure 11.Pulley mounted on a mandrel

Self-Check -2	Written Test

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

I. Choose

1. -------------------must be mounted between centers and a drive plate and lathe dog must be used. (1 points)
   a) tailstock spindles b/ mount the center drill c/ Mount faceplates d/ Mandrel

II. Matching (each 1 points)

Column "A"	**Column "B"**
2. mounting a work- piece between centers	A.3 jaw &independent (4 jaw) chuck
3. Mount faceplates	B. a heavy piece of work is mounted off center
4. Mounting Work pieces in Chucks	C. must be center- drilled and countersunk

***Note:* Satisfactory rating –above 2 points** **Unsatisfactory - below 2 points**

You can ask you teacher for the copy of the correct answers.

3.1 Work piece /metal is an element, compound or alloy that is a good conductor of both electricity and heat.

Types of Metals

- **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 limestonee 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.

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

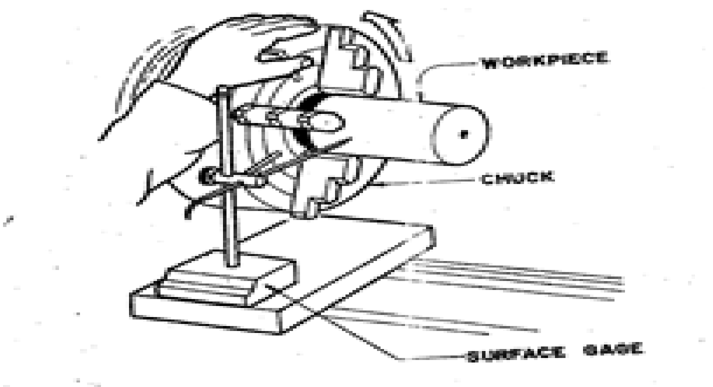
- ✓ **Lead Latin plumbum (Pb)**- atomic number 82, mmetallic lead has a bluish-white color after being freshly cut, but it soon tarnishes to a dull grayish color when exposed to air. Has a shiny chrome-silver luster when it is melted into a liquid.
  - ➢ **Property:**soft, malleable, has little ductility
  - ➢ **usage:** plates for storage batteries, covering for electrical cables

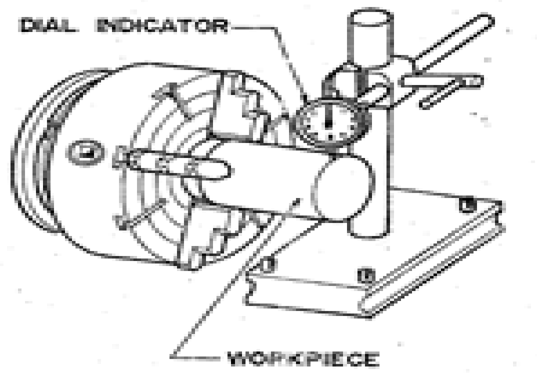
3.2 Performing setup operation

Before starting a lathe machining operation, always ensure that the machine is set up for the job that is to be accomplished.

- 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.

Figure12a. Centering Work in a 4-Jaw





Chuck Figure 12b.Centering Work in a 4-Jaw Using by surface gage
Using by dial indicator

SELF-CHECK -3	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. -------------------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
2. 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

3 -------------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

4 ------------------ Is the mixture of iron and carbon? (1 point)
   a) High speed steel b) Pig iron c) Cast iron d) Steel

***Note:* Satisfactory rating –above 2points** **Unsatisfactory - below 2points**

You can ask you teacher for the copy of the correct answers.

Operation sheet 1	Techniques of setting up work piece using appropriate instruments/equipment

1.1 steps to center work in a 4-jaw chuck:

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.

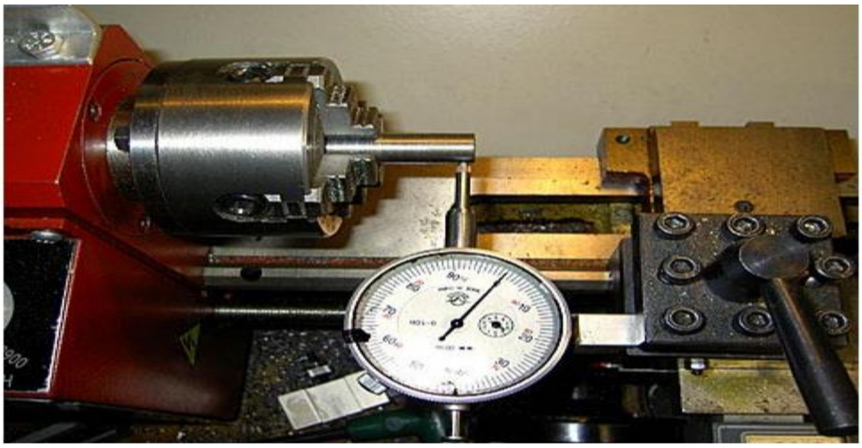


Figure13.center work in a 4-jaw chuck

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	**PRACTICAL DEMONSTRATION**

Name: ________________________________ Date: __________________

Time started: __________________________ Time finished: __________________

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

TASK 1: center work in a 4-jaw chuck using by dial indicator

INSTRUCTION SHEET	LEARNING GUIDE #26

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

- Selecting lathe accessories.
- Calculating depth of thread cut and tapper angle
- Calculating Parameters.
- Performing lathe operations.
- Applying OHS standards

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

- Calculating cutting Parameters using appropriate mathematical techniques and reference material according to standard.
- Calculating depth of thread cut and tapper angle according to standard instructions.
- CalculatingDepth of thread and taper angle according to standard instructions.
- Selecting lathe accessories based on the requirements of the operation.
- Performing lathe operations to produce component in compliance with specifications.
- Applying OHS standards.

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the “Information Sheets”. Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the “Self-checks” which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to “Operation sheets
7. Perform “the Learning activity performance test” which is placed following “Operation sheets” ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to “Operation sheets”.

Information sheet-1	Selecting lathe accessories

1.1 Lathe accessories: are the tools and equipment used in routine lathe machining operations.

1.2 TYPES OF ACCESSORIES

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.

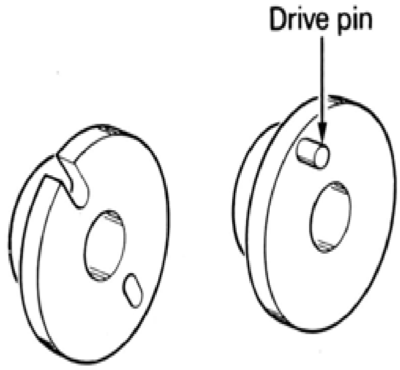


Figure14. Drive plate

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

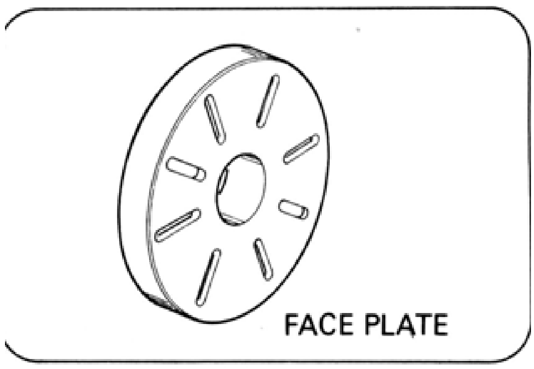


Figure15. Face plate

✓ **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.

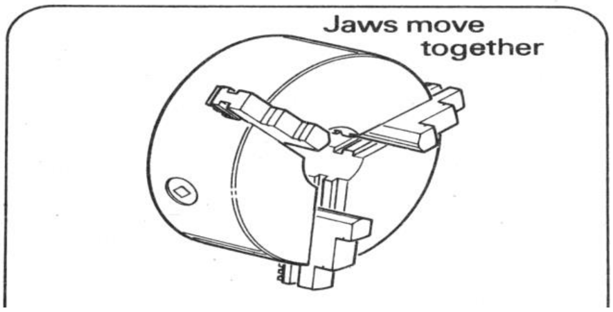


Figure16. Three – Jaw Self Centering Chick

- **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

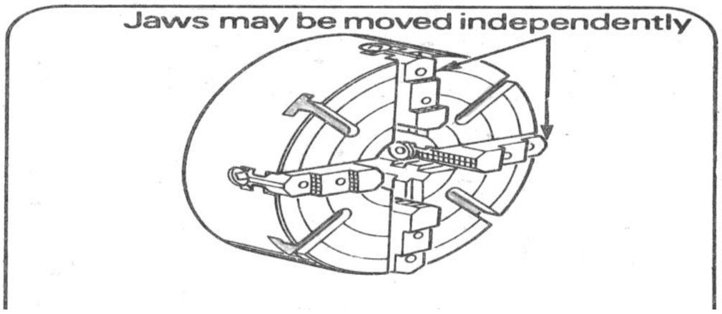


Figure17**.** Four jaw independent chuck

- **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.

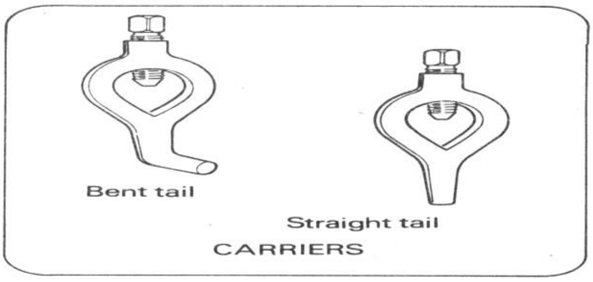
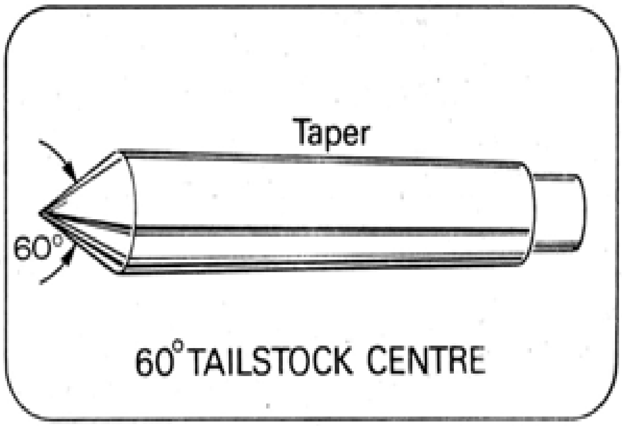


Figure18. Carriers\

- **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.



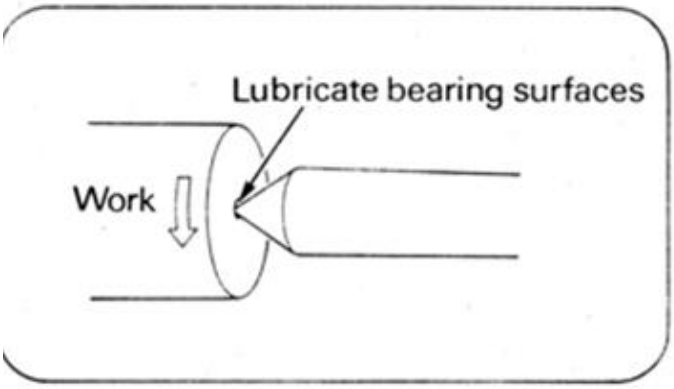


Figure19. Tailstock center

- **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 have the following advantages:
  - ✓ Lubrication is not needed between the center and the work
  - ✓ The work can be rotated at a higher speeds
  - ✓ Heavy cutting loads can be carried by the center

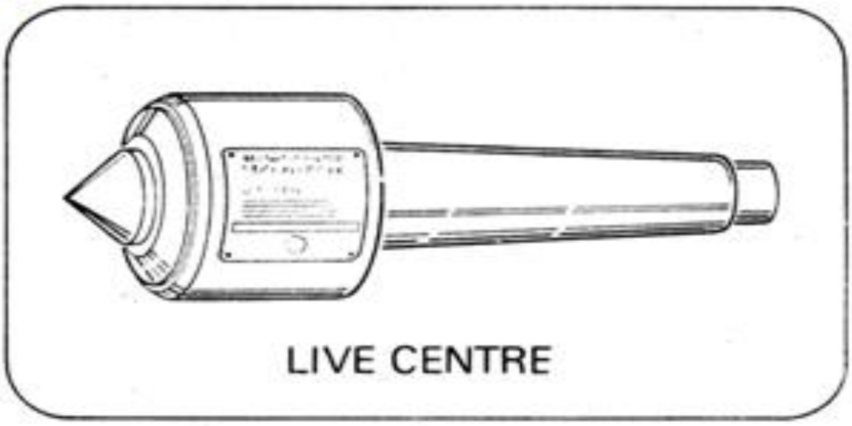


Figure20. Live center

- **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.

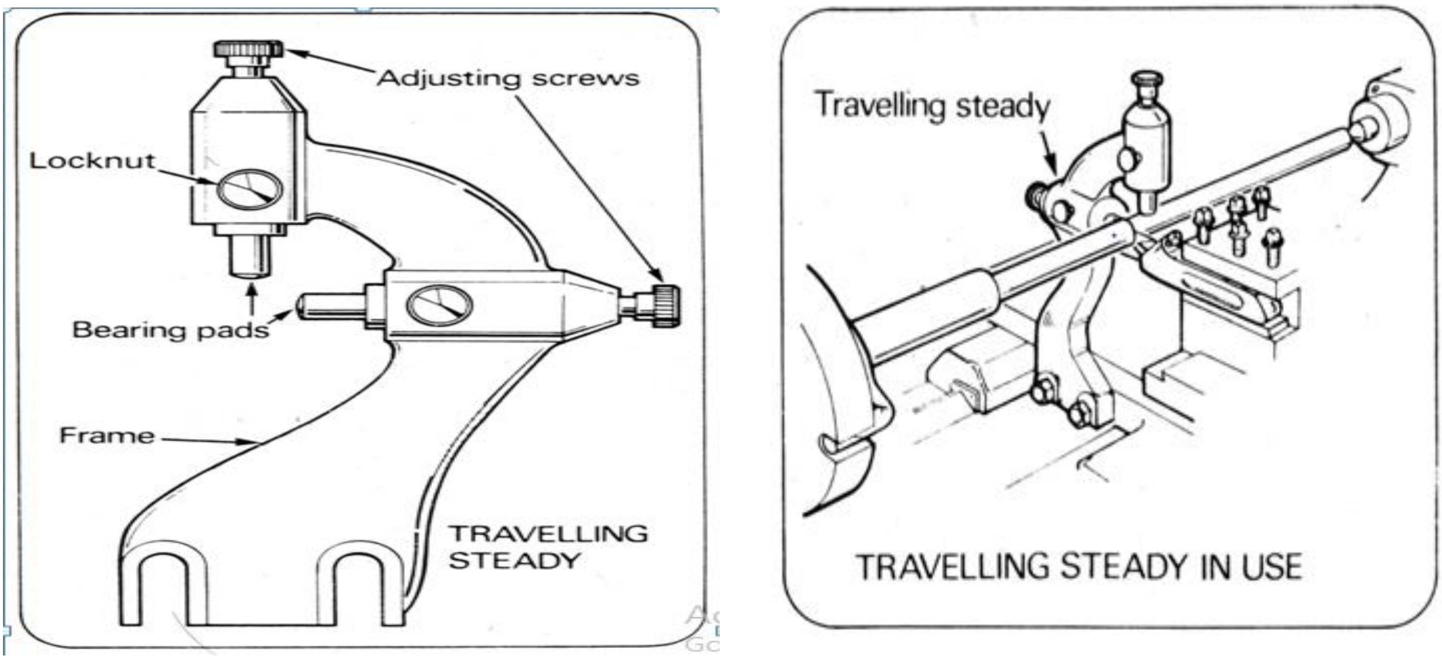


Figure21.Travel steady

- ✓ **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.

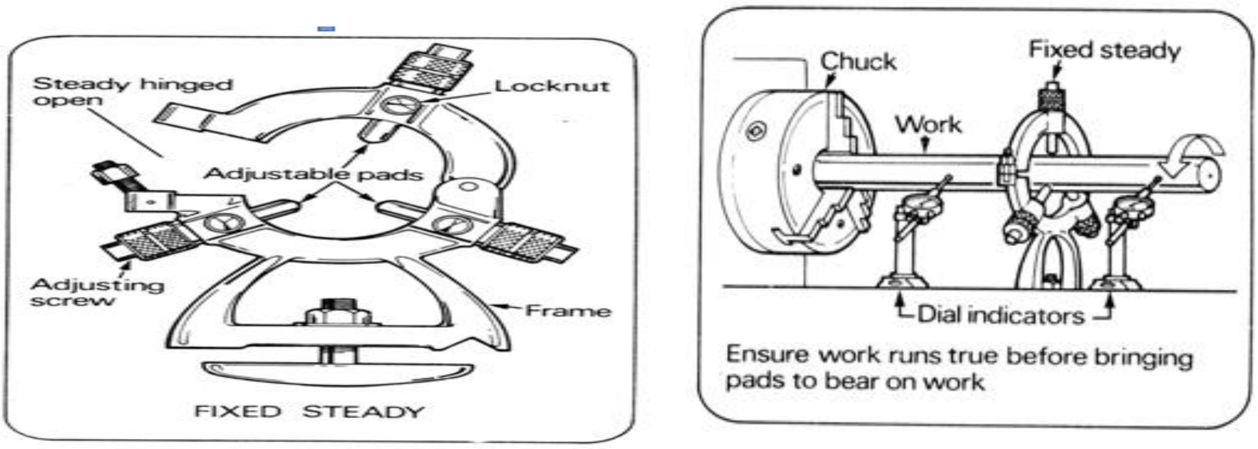
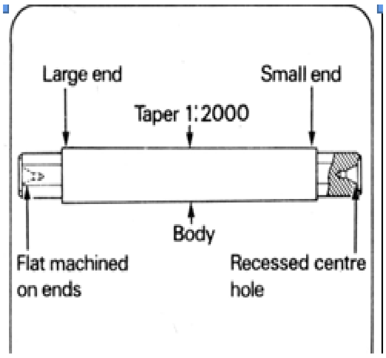


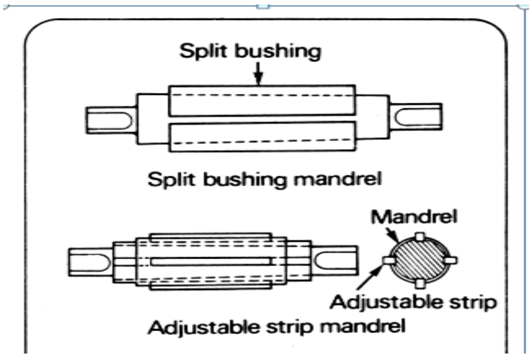
Figure22.Fixed steady

- **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





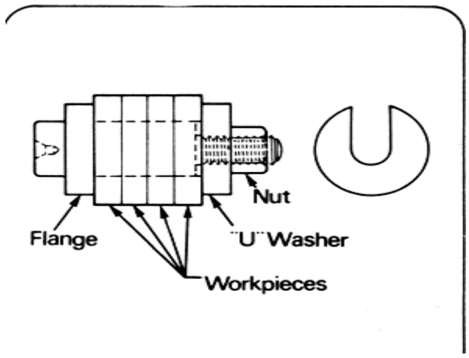
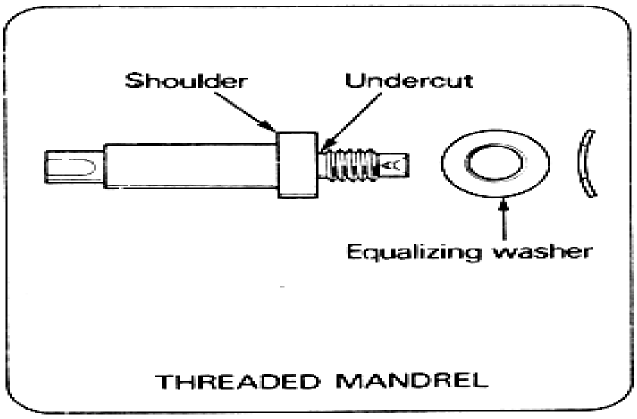


Figure23a. Standard solid mandrel Figure23b. Expansion mandrel Figure23c. Gang mandrel



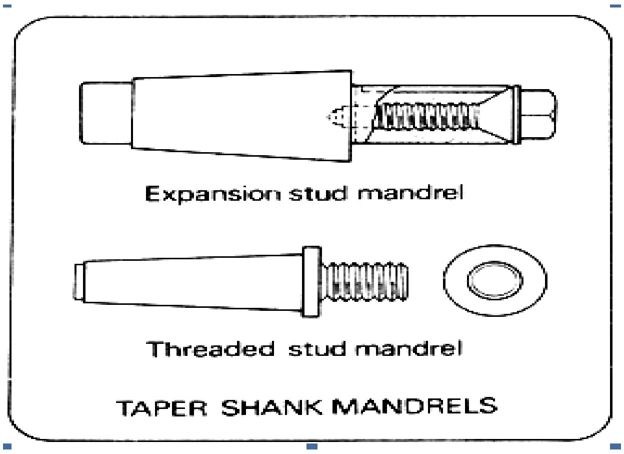


Figure23d. Threaded mandrel

Figure23e. 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.

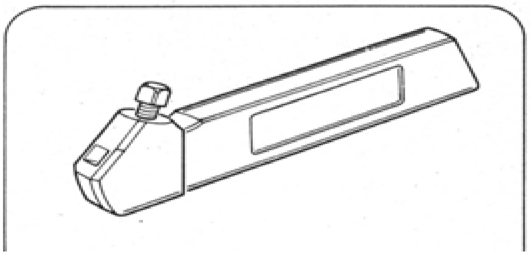


Figure24.Straight tool holder

  - ✓ **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.

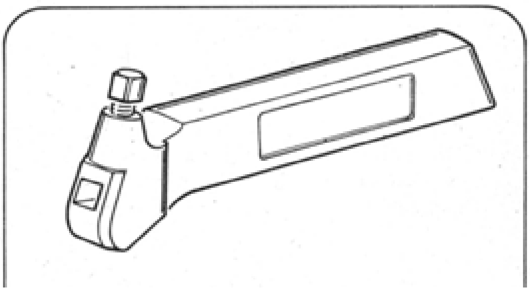


Figure25.Right hand tool holder

  - ✓ **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.

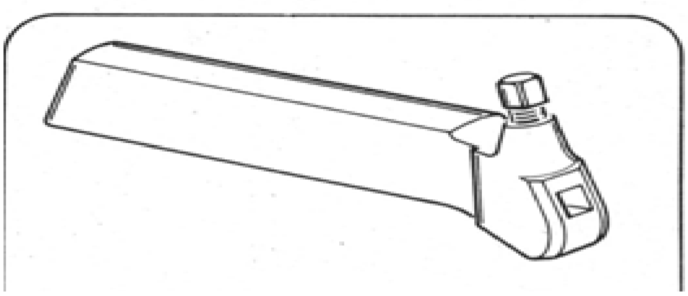


Figure26. Left hand tool holder

SELF-CHECK -1	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

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

Note: **Satisfactory rating –above 2 points Unsatisfactory - below 2 points**

You can ask you teacher for the copy of the correct answers.

Information sheet-2	Calculating depth of thread cut and tapper angle

2.1 thread forms and its type

2.1.1 v -thread

2.1.1.1 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.~~ $\Phi = 60^0$

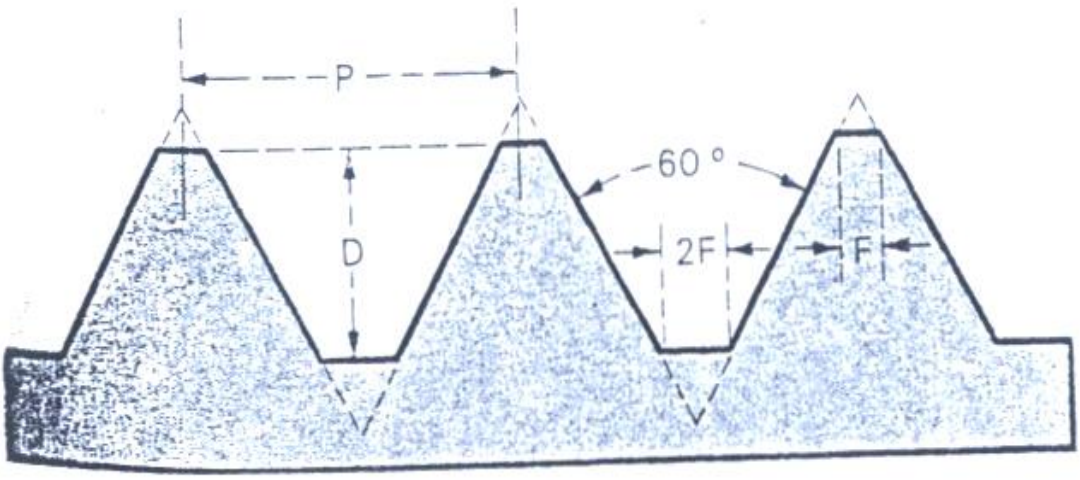
B. D = 0.6134 X P.

C. FC = 0.125XP

D. FR = 2X0.125XP

Table1. 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 = S
  Single Depth of Thread

- P = Pitch
- D (External) = 0.54127 x P
- FC = 0.125 x P
- FR = 0.250 **x** P

Figure 27: ISO Metric Thread

2.1.1.2 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

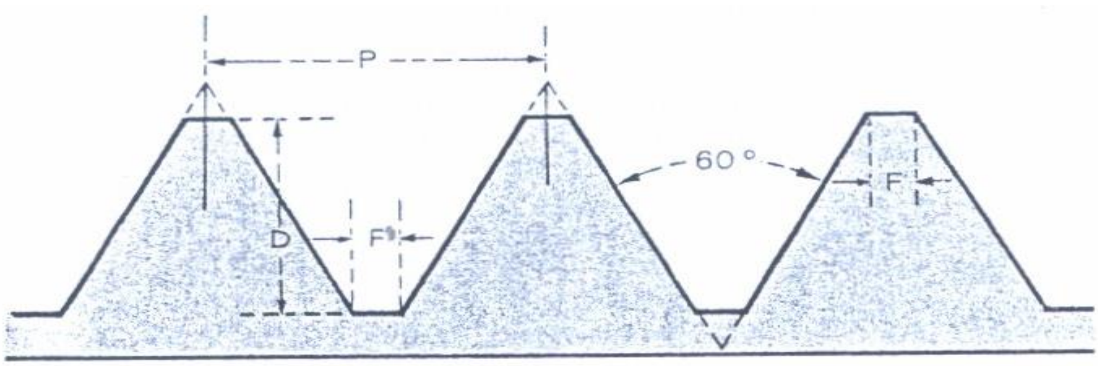


Figure 28: American National Threads

2.1.1.3 The British Standard with Worth Thread (BSW)

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

$$D = 0.6403 \times P \quad OR \quad \frac{0.6403}{N}$$

$$R = 0.1373 \times P \quad OR \quad \frac{0.1373}{N}$$

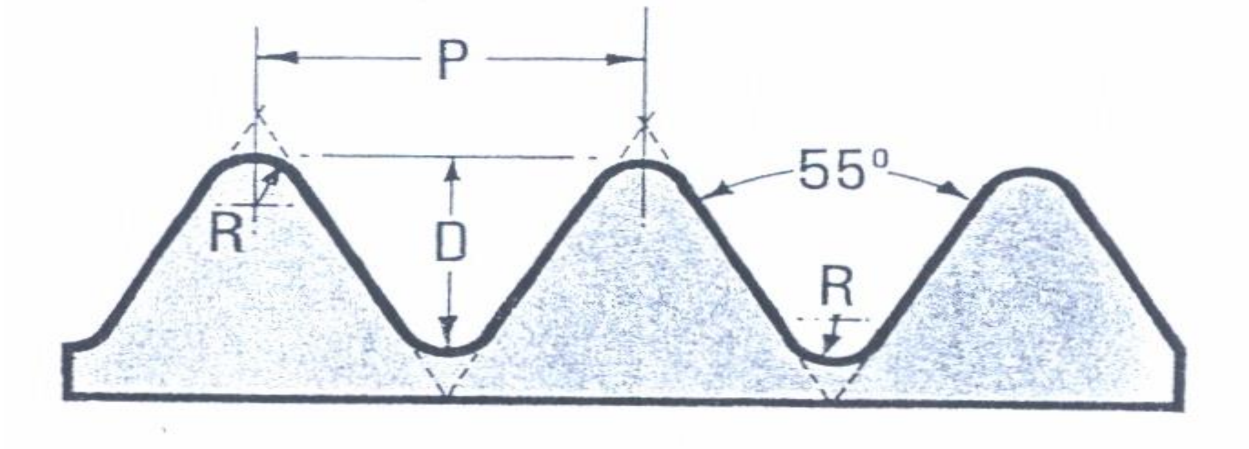


Figure 29: With Worth Thread

2.1.1.4 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)

- $D(External Thread) = 0.6131 \times P \quad OR \quad \frac{0.6131}{N}$
- $D(Internal Thread) = 0.5413 \times P \quad OR \quad \frac{0.5413}{N}$
- $F(External Thread) = 0.250 \times P \quad OR \quad \frac{0.250}{N}$

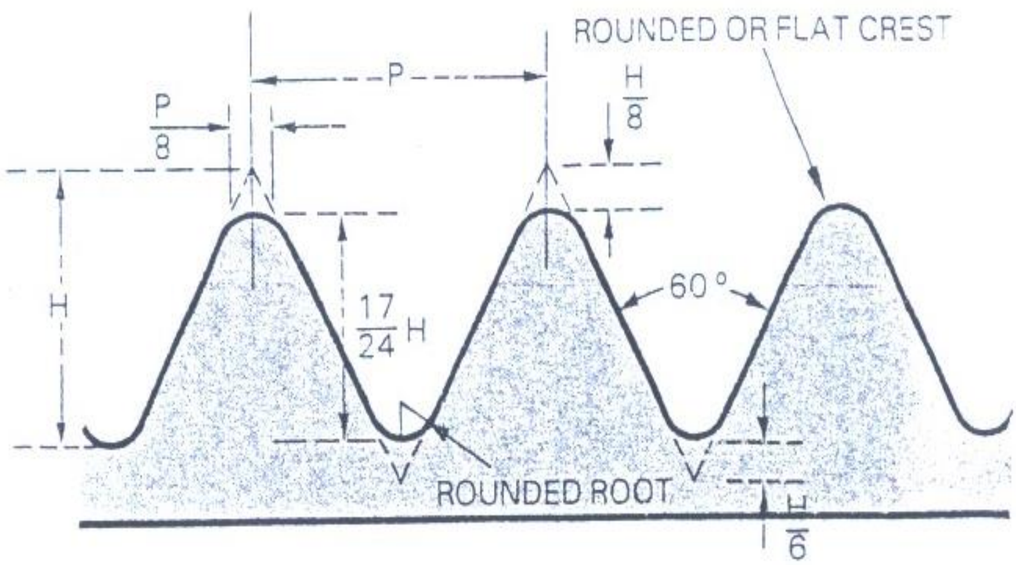
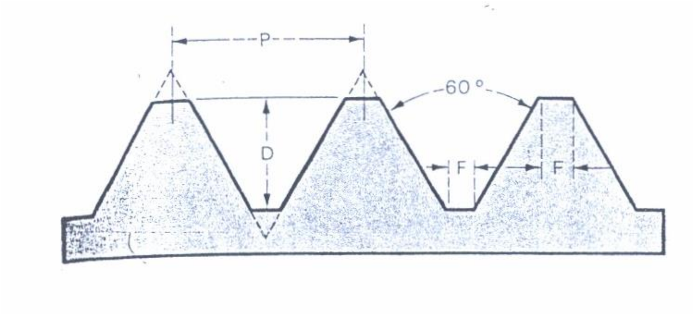


Figure 30: Unified Threads

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 31: International Metric Threads

2.1.1.5 **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)

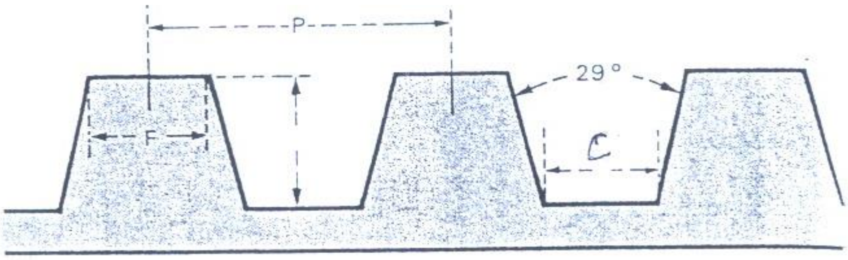


Figure 32 Acme Thread

- 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)

2.1.1.6 **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)

- $D = 0.6866 \times P$
- $F = 0.335 \times P$
- $C = 0.310 \times P$

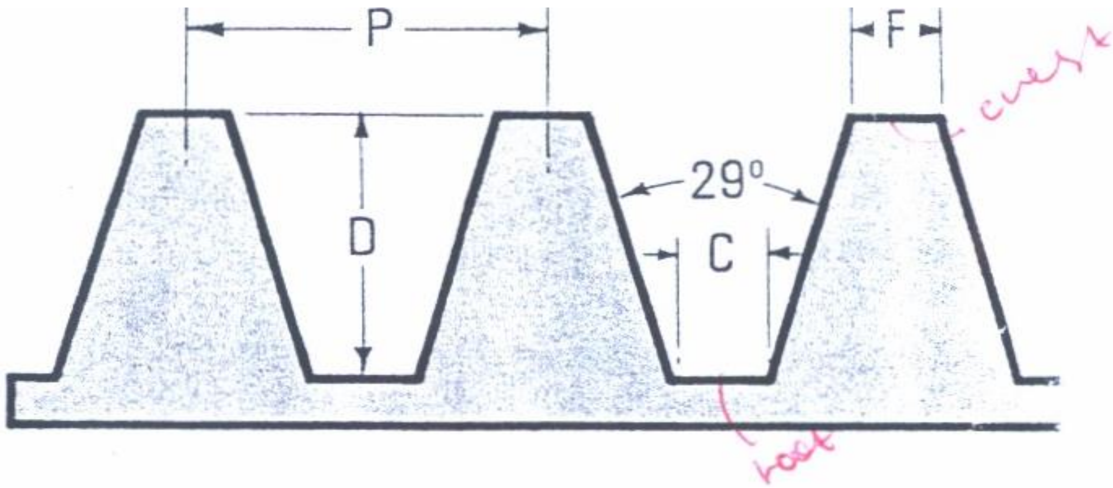


Figure 33: Brown and Sharp Worm Thread

2.1.2 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.

- $D = 0.500P$
- $F = 0.500P$

- $C = 0.500P + 0.002$

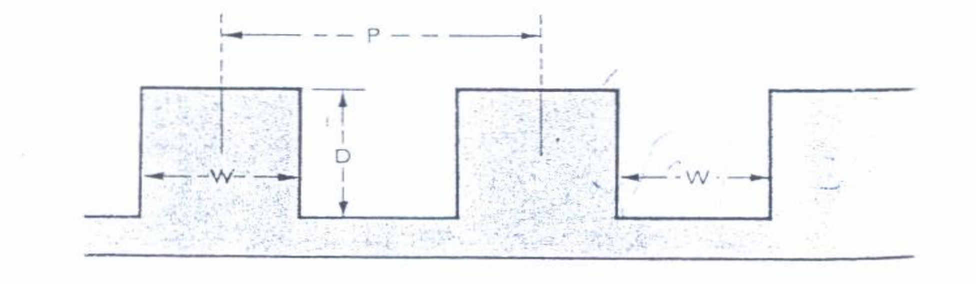


Figure 34: Square Thread

THREAD CALCULATIONS

To cut a correct thread on a lathe, it is necessary first to make calculations so that the thread will have proper dimensions.

Example: What is the pitch, depth, minor diameter, width of crest, and width of root for M6.3 x 1.0 thread?

P = 1 mm

D = 0.54127 x P = 0.54 mm

Minor Diameter = Major Diameter – (D + D)

= 6.3 – (0.54 + 0.54) = 5.22 mm

Width of Crest = 0.125 x P

= 0.125 x 1 = 0.125 mm

Width of Root = 0.250 x P

= 0.250 x 1 = 0.250 mm

2.1 **tapper angle calculating**

- **Taper turning with compound rest method**

$$\tan \Theta = \frac{D - d}{2l}$$

Where: - D= large diameter

d= small diameter

l = tapper length

Example 1:- Calculate the taper angle for a project having 80mm large diameter and 40 mm small diameter with 40 taper length to make its taper using compound rest method.

Given:-D=80mm d=40mm l= 40mm

Solu, $\tan\Theta = \frac{D-d}{2l} = \frac{80-40}{2 \times 40}$

$\Theta = \tan^{-1}(0.5) = 26.5^0$

- **Taper turning with offset tailstock method**

Formula used;

Offset, S = LX (D-d)/2L

Where:- D= diameter of large end

d= diameter of small end

l= length of taper

L= total length of piece

- **Turning a tapper with taper turning attachment method**

Formula used;

$$\Theta = \frac{GL(D-d)}{2l}$$

where . GL = guide bar length

D= diameter of large end

d= diameter of small end

l= length of taper

SELF-CHECK -2	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. Which one is not true about the ISO metric thread ?(1 point)
   A. Φ = 60^{0}
   B. D = 0.6134 X P.
   C. FC = 0.125XP
   D. FR = 2X0.125XP
   E. None
2. 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

Note: **Satisfactory rating –above 1 point** **Unsatisfactory - below 1 point**

You can ask you teacher for the copy of the correct answers.

Information sheet-3	Calculating parameters

3.1 Cutting parameters

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.

3.1.1 **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, $(V) = \frac{\pi DN}{1000}$ m/min.

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).

$$r/\min = \frac{CS \times 320}{D} \quad or \quad r/\min = \frac{1000 \times V}{\pi D} = \frac{1000 \times 40}{3.14 \times 50} = 254.77$$

$$\underline{\underline{r/\min = 255}}$$

3.1.2 **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, V = 25 – 30 m/min, Aluminum V = 60 – 93 m/min and like centers for parallel turning may be applied.

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

Table2.Lathe cutting

speeds in feet and meters per 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

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

Table3. 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

3.1.3**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.

$$t = \frac{D - d}{2} \; mm$$

Calculating machining time

$$Cutting\ Time = \frac{Length\ of\ Cut}{Feed \times r/mm.}$$

Graduated Micrometer Collars

Graduated micrometer collars are sleeves or bushings that are mounted on the compound rest and cross feed screws. They are used, when the diameter of a work piece must be turned to an accurate size. The collars on lathes using the inch system of measurement are usually graduated in thousands of an inch (0.001), and metric system 0.02 mm.

Micrometer collars can be used for:

- ✓ Machine where the work revolves. These include lathes, vertical boring mills (boring machines cylindrical grinders etc.
- ✓ The cutting tool should be moved in only half the amount of material to be removed.
- ✓ Machines where the work does not revolve

Example:
- ➲ Shapers
- ➲ Milling machines
- ➲ Surface Grinders etc.

3.1.4MACHING 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.

$$\text{Time required} = \frac{\text{length of cut}}{\text{Feed X r/min}}$$

SELF-CHECK -3	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. 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
2. Which metal needs high cutting speed during rough turning operation? (1 point)
   a. Bronze b) machine steel c) aluminum d/ cast Iron

3. Calculate the r/min required to turn a 50 mm diameter work piece of machine steel (CS =60 m/min). (1 point)
   a. 456 b) 382 c) 389 d/ none

Note: **Satisfactory rating –above 1.5 points** **Unsatisfactory – below1.5 points**

You can ask you teacher for the copy of the correct answers.

INFORMATION SHEET-4	**PERFORMING LATHE OPERATIONS**

4.1. lathe machine operations

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.

FACING: Facing is machining the ends and shoulders of a piece of stock smooth. Flat and perpendicular to the lathe axis. Facing is used to cut work to the desired length and to produce a surface from which accurate measurements may be taken.

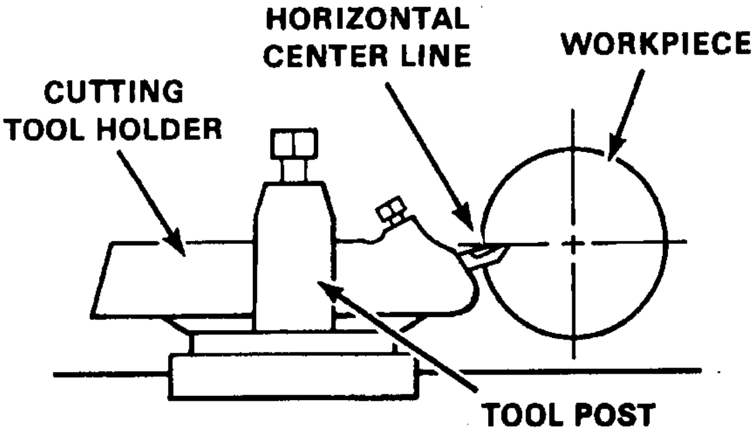


Figure35. Positioning tool bit for facing

1.1.1 TURNING

1.1.1.1 Cylindrical turning: The objectives of cylindrical turning are to produce a circular cylindrical surface in longitudinal cylinder turning the feed is parallel to the rotational axis of the work piece.

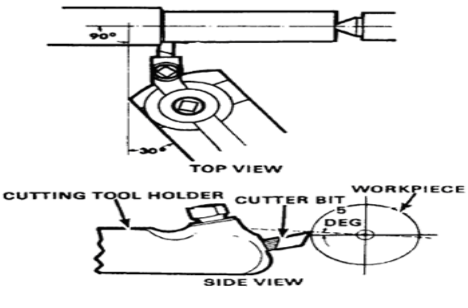


Figure36.Set up for straight turning

1.1.1.2 Taper turning: A section of material is considered to be tapered when it increases or decreases in diameter at a uniform rate.

- **Taper turning with compound rest:** The compound rest method of turning tapers is the easiest .When turning a taper using compound rest, cut is made from small diameter to large diameter.

$$\tan \Theta = \frac{D - d}{2l}$$

where. D= large diameter
d= small diameter
l = tapper length

Example 1. Calculate the taper angle for a project having 80mm large diameter and 40 mm small diameter with 40 taper lengths to make its taper using compound rest method.

Given:-D=80mm..d=40mm l= 40mm

Solution = $\tan\Theta = \frac{D-d}{2l} = \frac{80-40}{2x40}$

$\boldsymbol{\Theta = \tan^{-1} (0.5) = 26.5^{0}}$

Example 2. Calculateϴ if D=100mm. d=60mm and l=80mm

Solution = $\tan \Theta = \frac{D-d}{2l} = \frac{100-60}{2x80} = 0.25$

$\boldsymbol{\Theta = \tan^{-1}(0.25) = 14.04^{0}}$

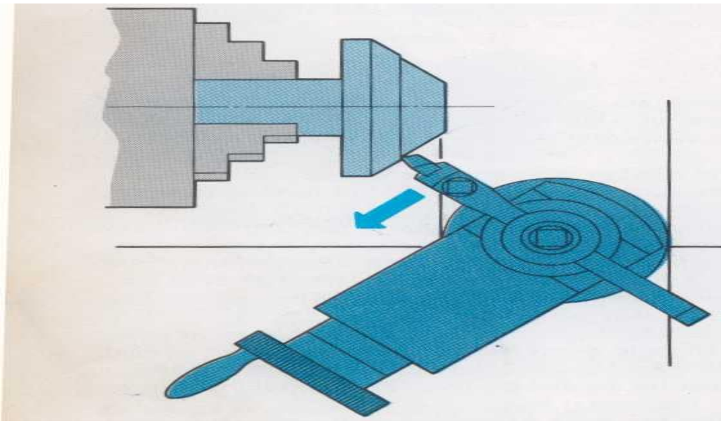


Figure37.Taper turning with compound rest

- **Taper turning by offset tailstock method:** this method also known as the tail stock set-over method is employed for taper turning jobs that can turned between centers. Only external tapers can be machined by using this method.

Calculating tail stock set over(S). Offset(S) must calculate for each job because the length of the piece plays an important part in the calculation. When the length of the piece varies, different will be produced with the same tail stock offset.

Formula used;

Offset, S = LX (D-d)/2L

Where D= diameter of large end

d= diameter of small end

l= length of taper

L= total length of piece

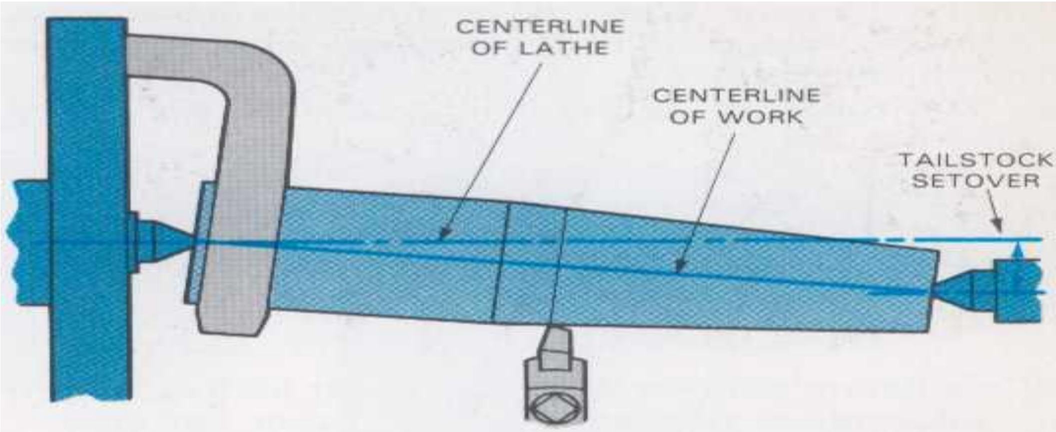


Figure38.Taper turning by offset tailstock method

- **Turning a tapper with taper turning attachment:** it is an accurate way to cut tapers and offers advantages over other methods of machining tapers.
    - ✓ Internal and external tapers can be cut and accurate fit is assured for mating parts.
    - ✓ Work can be held by any conventional means.
    - ✓ The lathe does not have to be altered. The machine can be used for straight turning by locking the taper attachment out. No realignment of the lathe is necessary

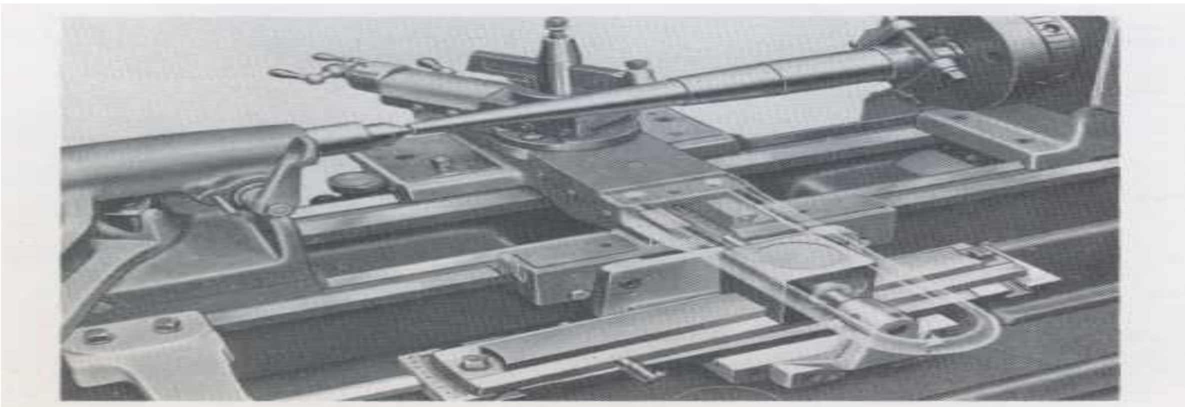


Figure39.Taper attachment

- **Turning a tapper with form tool:** this technique is limited to the production of short tapers.

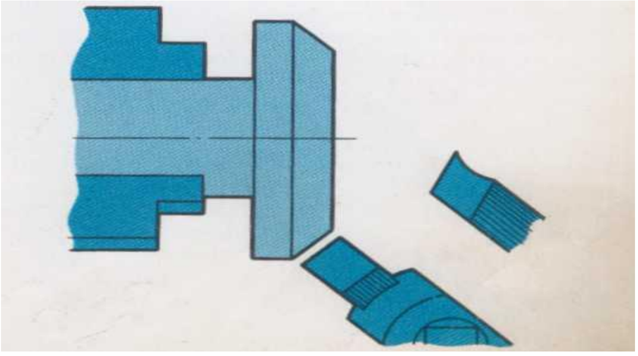


Figure40.Turning a tapper with form tool

1.1.2 Parting operations: Parting is the operation of cutting off material after it has been machined. This is one of the more difficult operations performed on a lathe. The cutting tool must be ground with the correct clearance and held in straight or offset tool holder.

1.1.3 Grooving operations: It is commonly called recessing, undercutting, or necking, is often done at the end of a thread to permit full travel of the nut up to a shoulder, or at the edge of a shoulder to ensure a proper fit of mating parts. Grooves are generally **square, round, or v-shaped.**

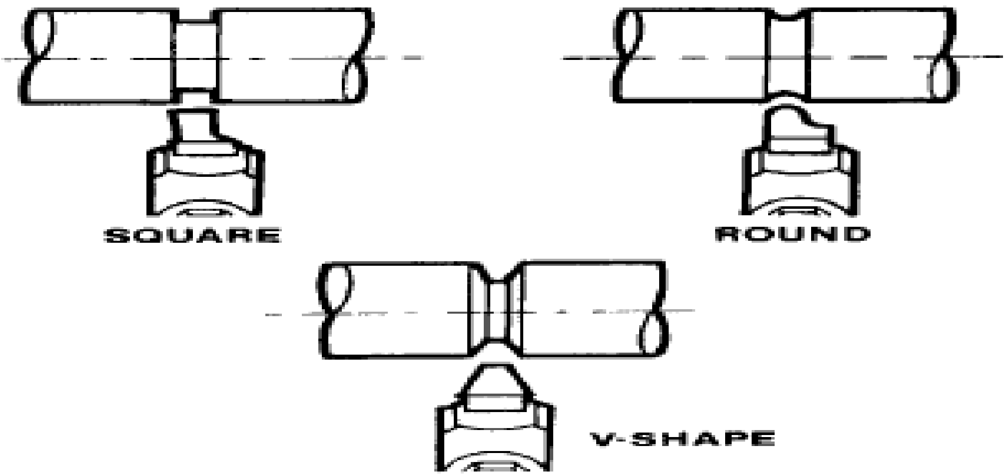


Figure41.Grooving operations

1.1.4 KNURLINGOPERATIONS: is a process of impressing a diamond shaped or straight line patter into the surface of the work piece to improve its appearance or to provide a better grip surface. Straight knurling is often used to increase the work piece diameter when a press fit is required. Diamond and straight pattern rolls are available in three styles: fine, medium, and coarse. The knurling tool is a tool post type tool holder on which a pair of hardened steel rolls is mounted.

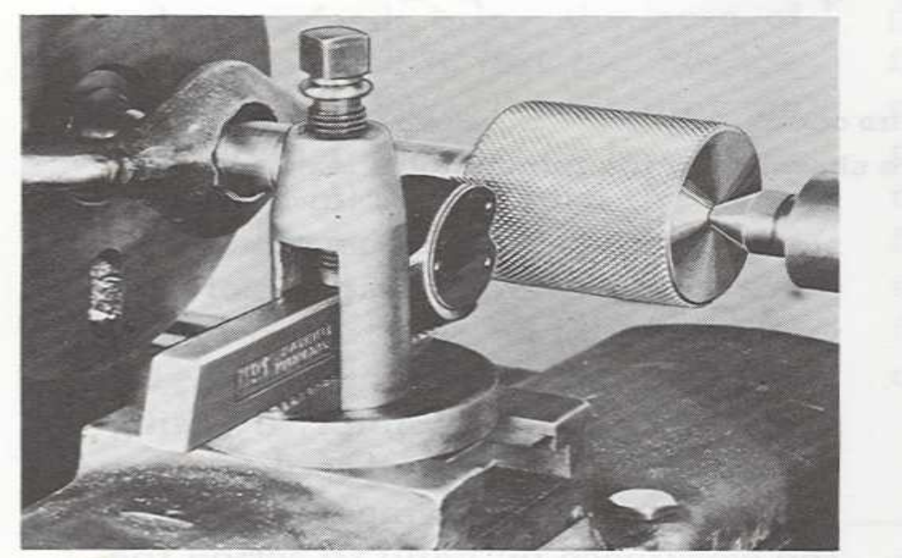


Figure42.Diamond –point knurling of a cylindrical work piece

1.1.5 Drilling: may define as the operation of producing a hole by removing metal from a solid mass using a cutting tool called a twist drill.

1.1.6 Boring: is the operation of enlarging a drilled or bored hole by means of a single or double edge cutting tool held in a boring bar. The hole produced should be concentric, parallel, and perpendicular to the work surface.

1.1.7 Reaming: is the operation of sizing and producing a smooth hole from a previously drilled or bored hole with the use of a cutting tool having several cutting edges.

1.1.8 Counter boring: is the operation of enlarging the end of a hole which has been drilled previously. A hole is generally counter bored to a depth slightly greater than the head of the blot, cap screw, or pin which it is to accommodate.

1.1.9 Thread cutting: A screw thread is a spiral ridge formed on either a straight or tapered cylindrical surface. The thread is cut on an external surface (e.g. bolt). It can also be cut on an internal surface (e.g. nut). The spiral ridge of the thread is of uniform shape or profile throughout its length. Threads may be produced using hand tools, such as taps and dies, or by machine tools. The most common thread is cut on a straight cylindrical surface. It is called a straight thread. One of the most frequently used machine tools for producing threads is the center lathe. The thread is produced by cutting spiral groove in the surface of a rotating work piece.

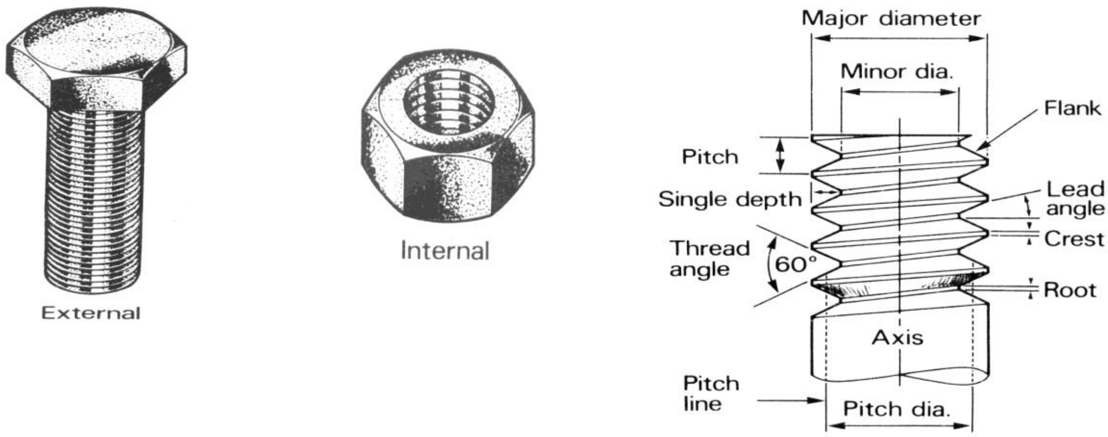


Figure43.Screw Threads

- **Thread Cutting in the Lathe**

Threads are cut in a lathe by rotating a cylindrical work piece at a constant speed and machining the surface of the work with a cutting tool. The cutting tool is moved at a constant speed in a direction parallel to the axis of the work piece. The work piece is held in the lathe using a suitable accessory and is rotated by the lathe spindle. The cutting tool moves with the lathe carriage by engagement of the half nut with the lead screw of the lathe. The shape of the cutting tool determines the shape of the groove cut on the work piece and therefore the profile of the thread. The rotation of the lead screw determines the hand of the thread being cut. The pitch of the lead screw thread and the relative speeds of the lathe spindle and lead screw determine the number of threads cut per unit length of the work piece. By varying the speeds of the lathe spindle and lead screw, a cut of any desired number of threads per unit length may be made. The pitch of the thread is the reciprocal of the number of threads per unit length.

Hence:

$$Pitch = \frac{1}{number of threads per unit of length}$$

For example, a screw with 10 turns per centimeter has a pitch of 1/10 cm.

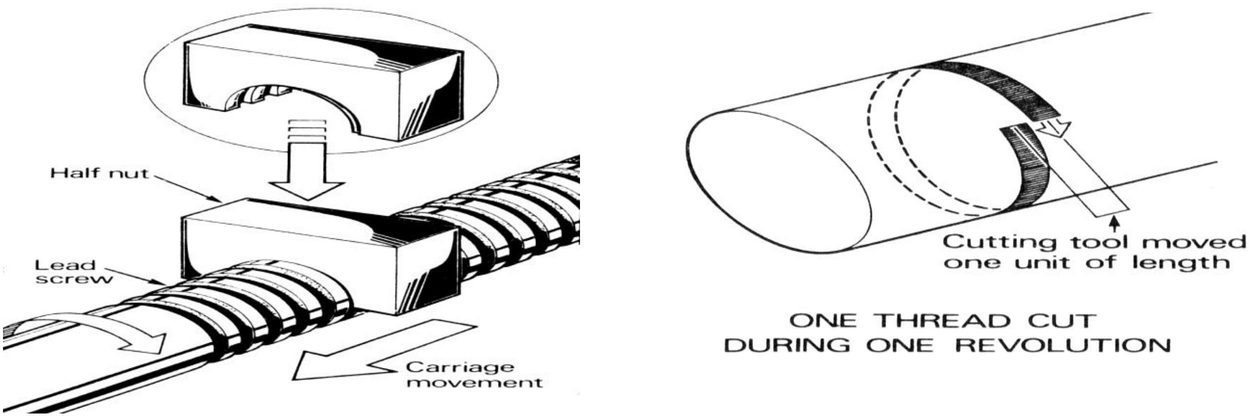


Figure44.Half nut engages lead screw to move carriage

1.1.10 Lathe filing:

is done to remove burrs, round of sharp edges, and to blend-in form cut out lines. A file is not intended to replace a properly sharpened cutting tool and should not be used to improve the surface finish on a turned section.

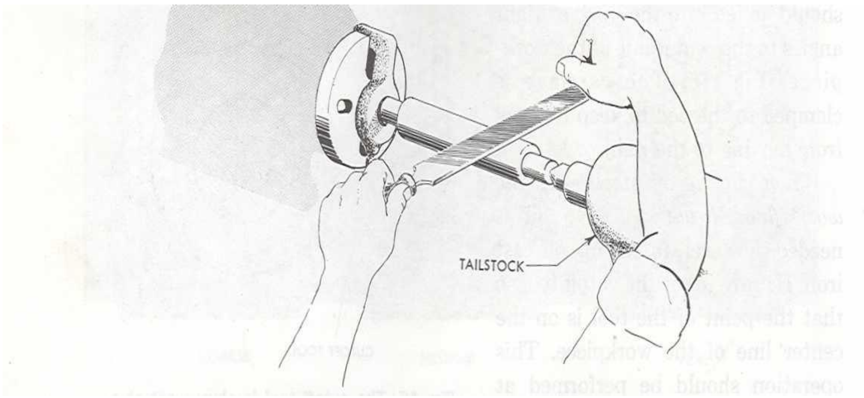


Figure45.Filling a work piece mounted on a lathe

1.1.11 Polishing on a lathe: polishing is sometimes done on a lathe. A strip of abrasive cloth, suitable for the material to be polished. The finer the abrasive used, the finer the resulting finish. A few drops of machine oil on the abrasive will improve the finish.

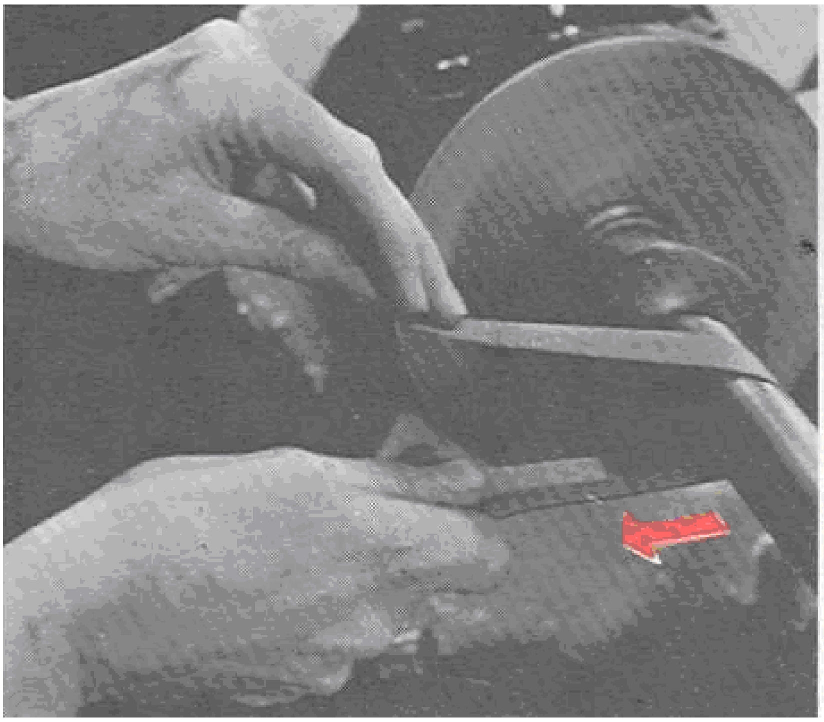


Figure46.Using a strip of aluminum oxide (emery) cloth to polish the turned surface of a work piece.

1.2 CUTTING OILS

General. The chief purpose of cutting oil is to cool the cutter bit and the work piece. The name "coolant" is often given to the oil. A cutter bit will last longer and will be capable of withstanding greater speeds without overheating when cutting oil is used. Cutting oil also helps lubricate the cutter bits, improves the finish of the work piece, guards against rusting, and washes away chips from the cutting area.

- **Use.** In production operations, the practice is to flood the work piece and the cutter bit with cutting oil in order to obtain the full benefit of its use. For effective cooling, it is important that the oil be directed at the exact point of the cutter bit contact. A large stream at low velocity is preferred to a small stream at high velocity. In small shops where pump equipment is not available, cutting oils are used only for finishing and delicate operations. It is general practice in this case to apply the cutting oil only when actually required.

- **Types of Cutting Oils.** Cutting oils most commonly used and their general applications are described in (a) through (g) below. Cutting oils for specific lathe operations are listed in Table 4 on page 64.

✓ **Lard Oil.** Pure lard oil is one of the oldest and best cutting oils. It is especially good for thread cutting, tapping, deep hole drilling, and reaming. Lard oil has a high degree of adhesion or oiliness, a relatively high specific heat, and its fluidity changes slightly with the temperature. It is an excellent rust preventive and produces a smooth finish on the work piece. Because lard oil is expensive, it is seldom used in a pure state but is combined with other ingredients to form good cutting oil mixtures.

✓ **Mineral Oil.** Mineral oils are petroleum based oils that range in viscosity from kerosene to light paraffin oils. Mineral oil is very stable and does not develop disagreeable odors like lard oil; however, it lacks some of the good qualities of lard oil such as adhesion, oiliness, and high specific heat. Because it is relatively inexpensive, it is mixed with lard oil or other chemicals to provide cutting oils with desirable characteristics. Two mineral oils, kerosene and turpentine, are often used alone for machining aluminum and magnesium. Paraffin oil is used, alone or with lard oil, for machining copper and brass.

✓ **Mineral-Lard Cutting Mixture.** Various mixtures of mineral oils and lard oil are used to make cutting oils which combine the good points of both ingredients but prove more economical than and often as effective as lard oil.

✓ **Sulfurized-Fatty-Mineral Oil.** Most good cutting oils contain mineral oil and lard oil with various amounts of sulfur and chlorine, which give the oils good antiweld properties and promote free machining. These oils play an important part in present-

day machining because they provide good finishes on most materials and aid the cutting of tough materials.

✓ **Soluble Cutting Oils.** Water is an excellent cooling medium but has little lubricating value and hastens rust and corrosion. Therefore, mineral oils or lard oils which are mixable with water are often mixed with water to form cutting oil. Soluble oil and water has lubricating qualities depending upon the strength of the solution. Generally, soluble oil and water is used for rough cutting where quick dissipation of heat is most important. Borax and tri sodium phosphate are sometimes added to the solution to improve its corrosion resistance.

✓ **Soda-Water Mixtures.** Salts such as soda ash and tri sodium phosphate are sometimes added to water to help control rust. This mixture is the cheapest of all coolants and has practically no lubricating value. Lard oil and soap in small quantities are sometimes added to the mixture to improve its lubricating qualities. Generally, soda water is used only where cooling is the prime consideration and lubrication is secondary. It is especially suitable in reaming and threading operations on cast iron where a better finish is desired.

- ✓ **White Lead and Lard Oil Mixture.** White lead can be mixed with either lard oil or mineral oil to form cutting oil which is especially suitable for difficult machining of very hard metals.

Table4.cutting oils for lathe operations

Material	Cutting Oil		
	Heavy cutting	Light cutting	Threading
Aluminum	Dry; soluble cutting oil.	Dry	Mineral-fatty-blend cutting oil.
Brass	Dry; soluble cutting oil.	Dry	Sulfurized-fatty-mineral cutting oil.
Bronze	Soluble cutting oil	Soluble cutting oil	Sulfur-treated mineral-lard cutting oil.
Cast iron	Dry	Dry	Sulfurized-fatty-mineral cutting oil.
Copper	Dry; mineral-fatty-blend cutting oil.	Dry; mineral-fatty-blend cutting oil.	Mineral-fatty-blend cutting oil.
Monel metal	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Pure lard cutting oil; sulfurized-fatty-mineral cutting oil.
Steel, machine	Soluble cutting oil	Mineral-fatty-blend cutting oil; soluble cutting oil.	Sulfurized-fatty-mineral cutting oil.
Steel, tool	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Pure lard cutting oil; sulfurized-fatty-mineral cutting oil.
Steel, stainless	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Soluble cutting oil; sulfurized-fatty-mineral cutting oil.	Pure lard cutting oil; sulfurized-fatty-mineral cutting oil.

SELF-CHECK -4	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. ------------- is machining the ends and shoulders of a piece of stock smooth. (1 point)
   a. Facing b/ turning c/grooving d/ threading
2. Which is the taper angle for a project having 80mm large diameter and 40 mm small diameter with 40 taper length to make its taper using compound rest method. (1 point)
   a. 4.8 b/ 20.5 c. 26.56 d/ None

3. it is an accurate way to cut a tapper. (1 point)
   a. using compound rest   c) using tailstock offset
   b. using taper turning attachment   d) None
4. --------------- is the operation of enlarging a drilled or bored hole by means of a single or double edge cutting tool held in a boring bar. (1 point)
   a. Counter boring   b/ Drilling   c/ Reaming   d/ Boring
5. Tool used for the operation of sizing and producing a smooth hole from a previously drilled or bored hole with the use of a cutting tool having several cutting edges. (1 point)
   a. Counter boring   b/Drilling   c/Reaming   d/ Boring

***Note:* Satisfactory rating –above 2.5 points   Unsatisfactory - below 2.5 points**

Information sheet-5	Applying ohs standards

1.1 Safety Precautions For Lathes

General. In machining operations, there is one sequence of events that one must always follow: SAFETY FIRST, ACCURACY SECOND, AND SPEED LAST. With this in mind, let's look at some of the more important safety precautions that should be observed before and during lathe operations.

- Lathe accidents are usually caused by:
  - ✓ Loose clothing snagging on the revolving work piece, the chuck, or the work piece.
  - ✓ Flying chips entering the eye when turning cast iron or nonferrous metals.
  - ✓ Contact of the hands or arms with the lathe dog, chuck or work piece.

- The operator should prepare himself by rolling up his shirt sleeves and removing watches, rings, and other jewelry that might become caught while he is operating the machine.
- The operator should be sure to wear safety glasses or a face shield of the approved type at all times when operating a lathe or when in the area of lathes that are in operation.
- The operator should be sure that the work area is clear of obstructions that one might fall over or trip on.
- On turret lathes, care must be taken not to catch loose or torn clothing on a stock that is supported in the collect with chucks and extends beyond the headstock of the lathe.
- If a coolant or cutting oil is used, the operator should take care when adjusting the splash pans to prevent the liquid from splashing on the floor. The cutting oil or coolant can make the floor beneath the lathe slippery and cause the operator to lose his balance and suffer injury.
- The operator will keep the floor around the machine clear of oil or grease to prevent anyone from slipping and falling into the machine.
- The operator should use assistance when handling heavy or awkward parts, stock, or machine accessories. Never remove chips with your bare hands; a pair of pliers, a hook, or a brush should he used. (Stop the machine while removing the chips.)
- The operator should prevent long chips from being caught in the chuck by using good chip control procedures.
- The operator should never try to stop the machine with his hands or body.
- The operator will turn the machine off before talking to anyone.
- The operator should know how to stop the machine quickly if an emergency arises.
- The operator must be attentive, not only to the operation of the machine, but the events going on around it.
- The operator should be alert to the location of the cutting tool while taking measurements or making adjustments to the machine. He should see that the work and the cutting tools clear each other and that they are clamped securely before starting the machine.
- The operator will remove the centers and the cutting tools when not being used, and always observe the specific safety precautions posted for the machine in which you are operating.

SELF-CHECK -5	WRITTEN TEST

Directions: Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

True or False Item

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)

Note: **Satisfactory rating –above 1 point** **Unsatisfactory - below 1 point**

You can ask you teacher for the copy of the correct answers.

OPERATION SHEET 1	PERFORMING LATHE OPERATIONS

1.1 PROCEDURES 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.

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

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.

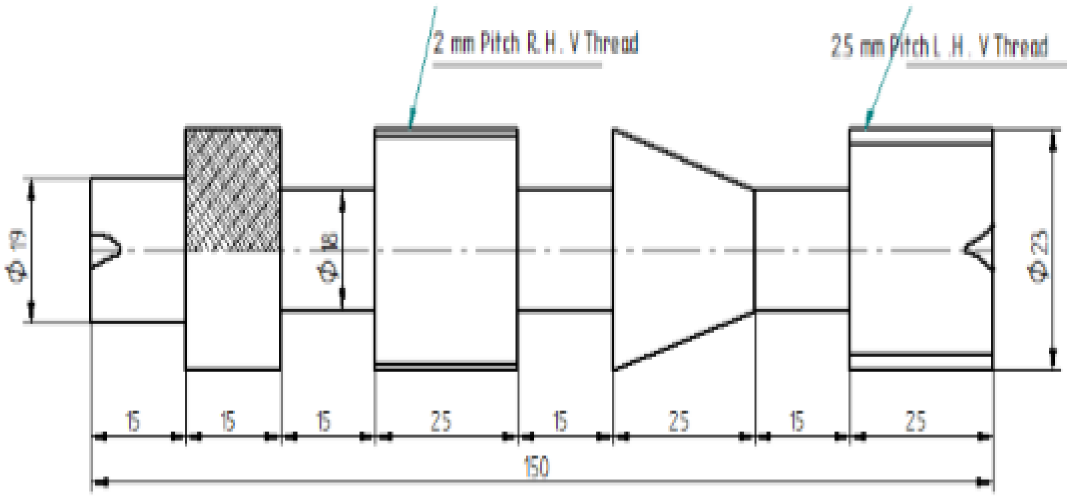


Figure47. Turning Operations

Note:

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

OPERATION SHEET 2	PERFORMING LATHE OPERATIONS

2.1 PROCEDURES FOR ECCENTRIC TURNING

Steps 1-Apply safety precaution

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

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

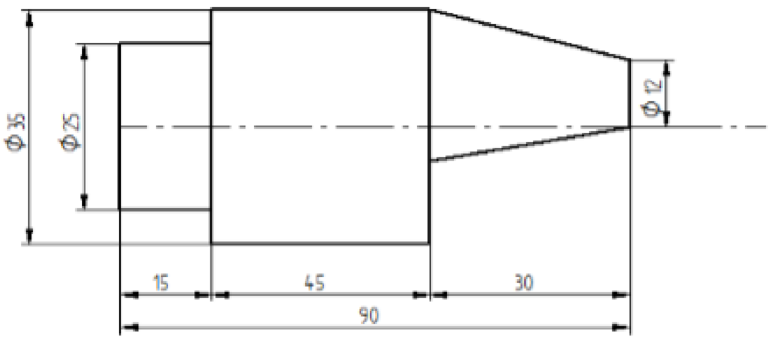
Where $\boldsymbol{\theta}$ = Swivel the compound rest to the calculated angle.

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

Figure48. Eccentric turning

OPERATION SHEET 3	PERFORMING LATHE OPERATIONS

1.3 PROCEDURES OF TURNING OPERATIONS WITH THREAD 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 and centering operation is performed using the surface gauge.
Steps 5- The tool aligned by placing such that, the tip of the tool is perpendicular to the tip of the dead centre. The tool post is tightened using alin key.
Steps 6- Cutting parameters like speed, feed, depth of cut, etcare selected before the machine is turned on.
Steps 7- To carry out thread cutting required pitch is noted down & depth of cut is calculated as shown in the formula. Using a standard chart the auto feed varying mechanism is set for 2mm pitch & the right hand thread cutting operation 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 8- The procedure is repeated 10 times with an increment in depth of cut by 2 divisions i.e. 0.1mm each time. In case of right hand thread cutting operation feed will be from right to left & vice versa for the left hand thread.
Steps 9- For 'V' – thread cutting, Angle of thread is 600, According to the ISO standard metric thread depth = 0.6 to 0.65 X pitch (60 to 65%).
Steps 10- Similarly for Square thread (flat side of the single point cutting tool is used), Depth = 0.5 to 0.55 X pitch (50 to 55%).
Steps 11- 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 12- Finishing operation is done by giving very small depth of cut (say 0.2mm) on medium feed.
Steps 13- Φ8Drill Bit is fixed to the dead centre and advanced towards the work piece.
Steps 14- Drilling is carried out to a depth of 20mm on one circular face of the work piece.
Steps 15- Tapping operation is carried out on the other circular face of the work piece with the help of 3 tapping tools used as rough, medium and fine internal threading to a depth of 30mm.
Steps 16- Knurling operation is carried out by using special knurling tool.

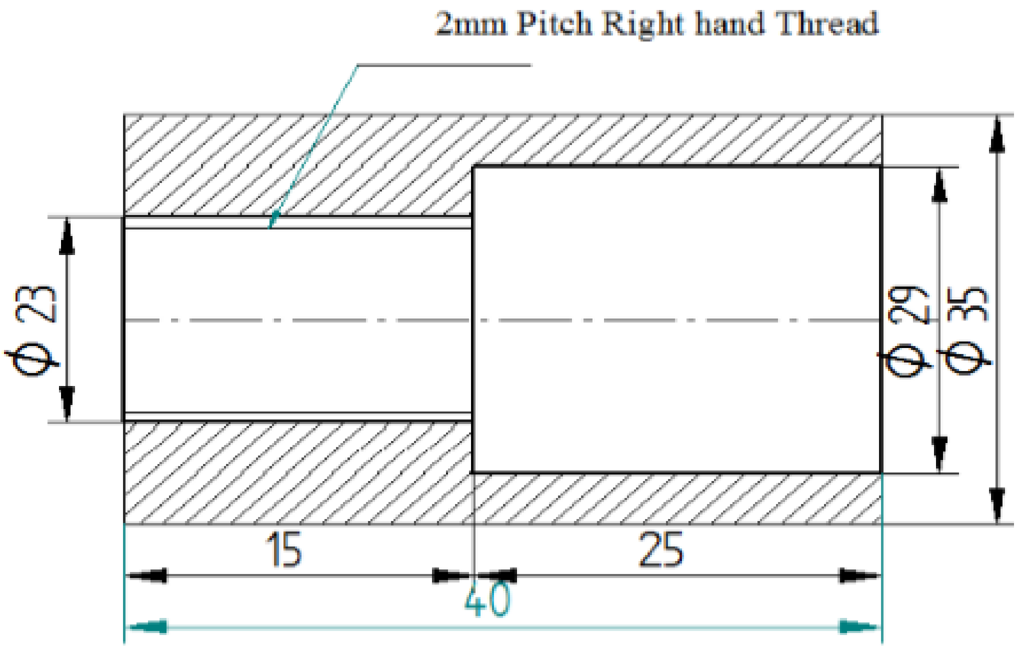


Figure49. Turning operations

Note:

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

OPERATION SHEET 4	PERFORMING LATHE OPERATIONS

4.1 PROCEDURES OF LATHE SET – UP FOR CUTTING V- THREADS

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.

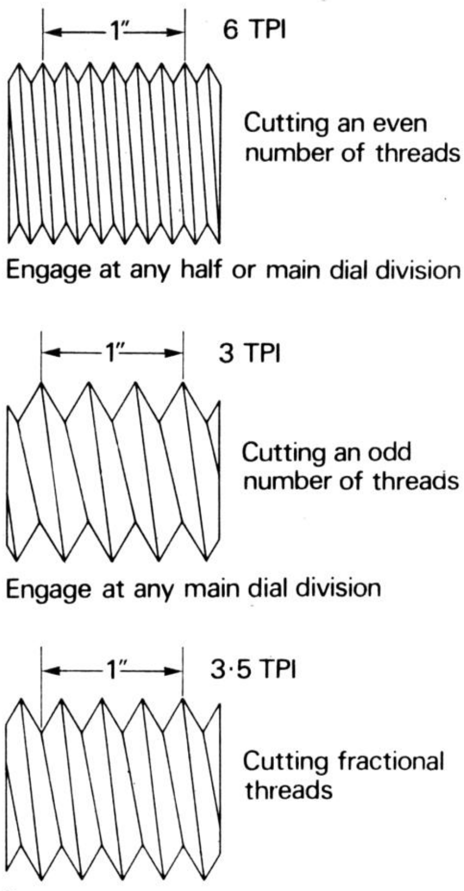
Steps 2- set the lathe spindle speed at about quarter of the normal turning speed

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.

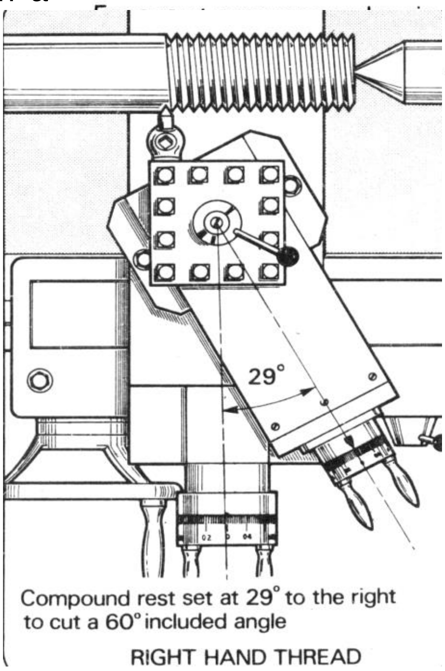


Figure50. Set – up for cutting v- threads

OPERATION SHEET 5	PERFORMING LATHE OPERATIONS

5.1. PROCEDURES OF 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
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

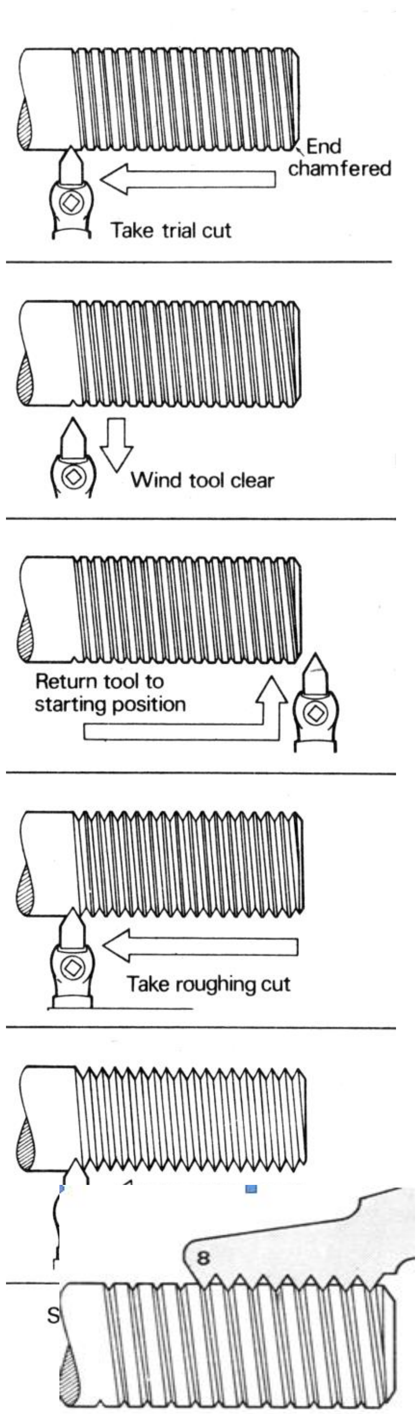


Figure51.Cutting external threads

OPERATION SHEET 6	PERFORMING LATHE OPERATIONS

6.1. PROCEDURES OF CUTTING AN INTERNAL THREAD

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

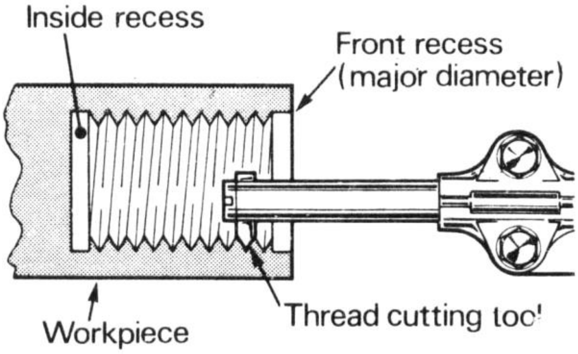
Steps 2- drill a hole in the work piece slightly smaller than the root diameter (tap drill size = nominal size – pitch for V-threads) of the thread to be cut

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.

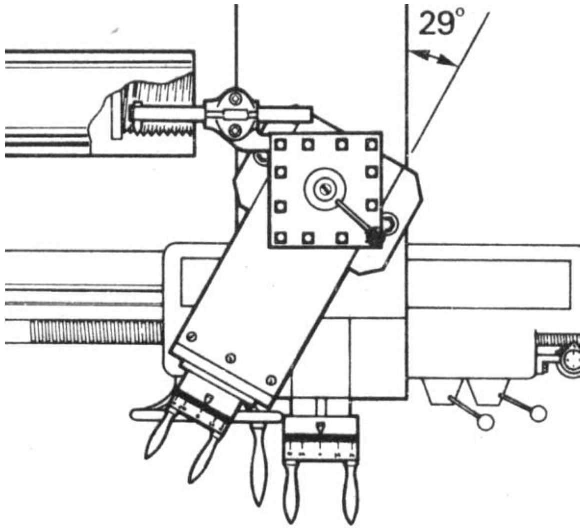
Steps 3- cut a recess at the inner end of the bore wide enough to allow the cutting tool to clear the thread. The recess should be slightly deeper then the major diameter of the thread

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.



Cutting an internal thread



COMPOUND REST SET AT 29° TO THE LEFT TO CUT A 60° INCLUDED ANGLE RIGHT HAND INTERAL THREAD

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

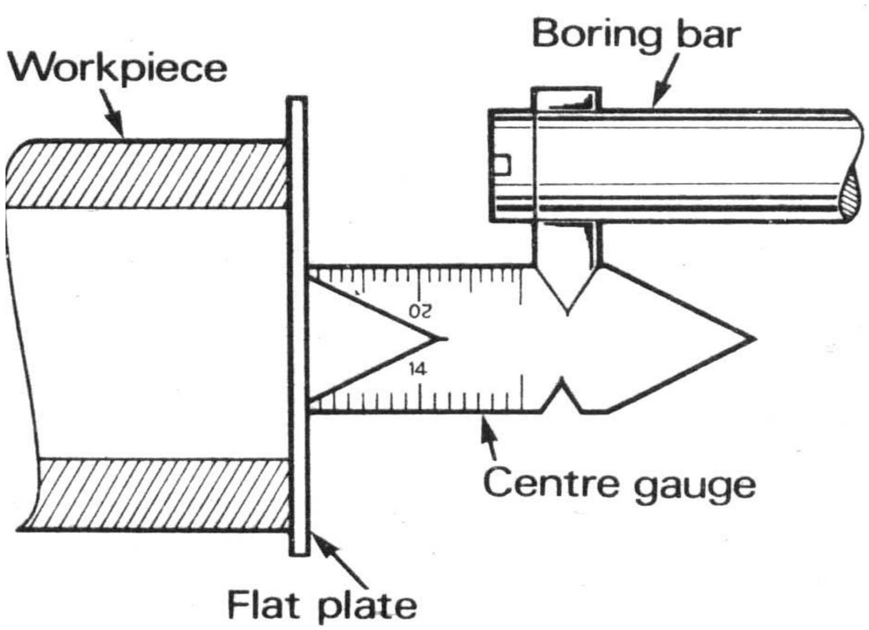


Figure52.Squaring the cutting tool with a center gauge

OPERATION SHEET 7	PERFORMING LATHE OPERATIONS

7.1 PROCEDURES OF CUTTING AN EXTERNAL ACME 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

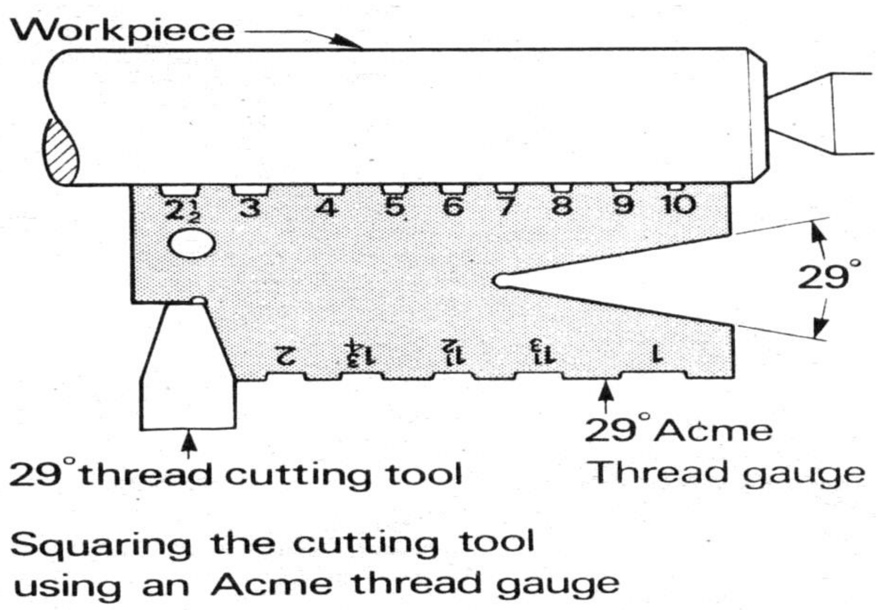


Figure53.Square the cutting tool using an Acme thread gauge

OPERATION SHEET 8	PERFORMING LATHE OPERATIONS

8.1 PROCEDURES OF CUTTING 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

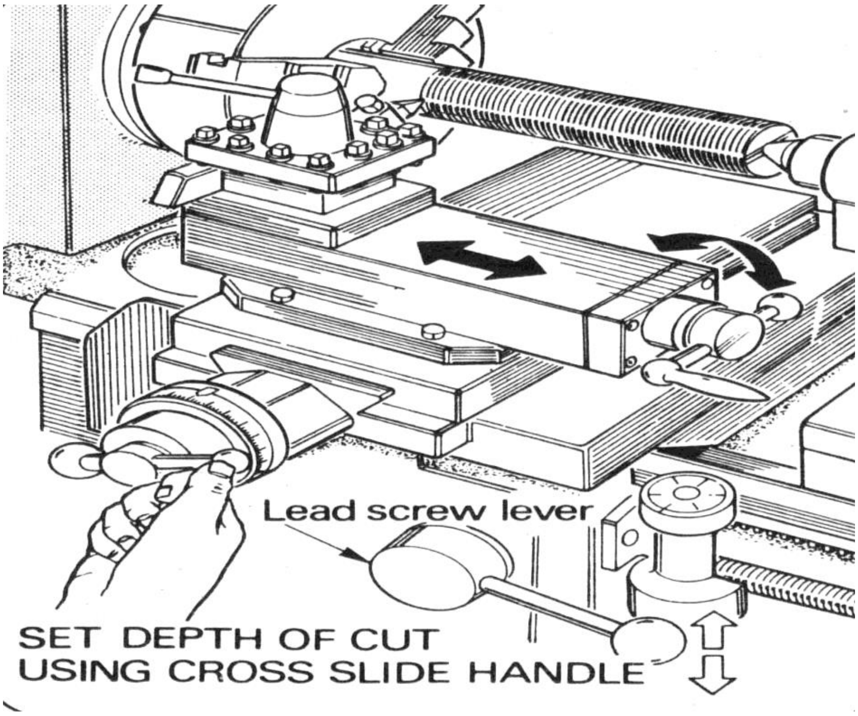


Figure54. Cutting an Internal Square Thread

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 --- 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

INSTRUCTION SHEET	LEARNING GUIDE #27

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

- Measuring tools and equipment
- Checking techniques of work pieces
- Handling deviations

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

- measuring tools and equipment using appropriate techniques, in conformance with specification
- Checking techniques of work pieces in conformance with finished quality
- Handling deviations appropriately in accordance with organization procedures and standard

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below.
3. Read the information written in the “Information Sheets”. Try to understand what are being discussed. Ask your trainer for assistance if you have hard time understanding them.
4. Accomplish the “Self-checks” which are placed following all information sheets.
5. Ask from your trainer the key to correction (key answers) or you can request your trainer to correct your work. (You are to get the key answer only after you finished answering the Self-checks).
6. If you earned a satisfactory evaluation proceed to “Operation sheets
7. Perform “the Learning activity performance test” which is placed following “Operation sheets” ,
8. If your performance is satisfactory proceed to the next learning guide,
9. If your performance is unsatisfactory, see your trainer for further instructions or go back to “Operation sheets”.

Information sheet-1	Measuring tools and equipment

1.1. **Measuring tools**

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.

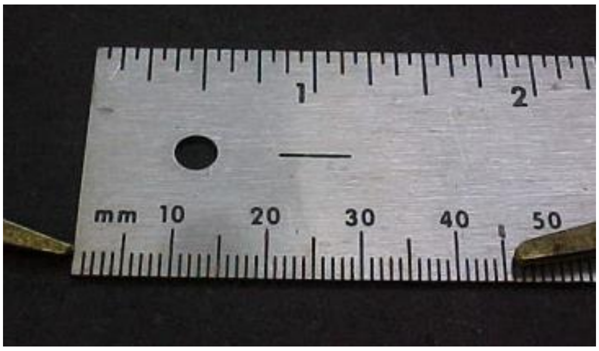
- The means of measurement could be classified as follows:
  - ✓ **Standards** (reference masters or setting standards)-these are used to reproduce one or several definite values of a given quantity.
  - ✓ **Fixed gauges**-these are used to check the dimensions, form, and position of product features.
  - ✓ **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).

1.1.1. 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.



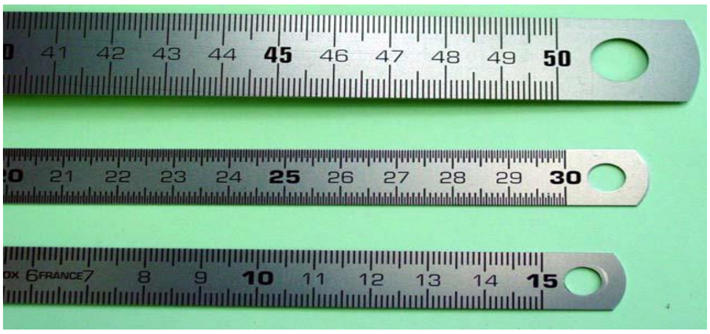


Figure55. Steel Rule

1.1.2. Vernier 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 56A. 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 57B. Vernier Calipers

- **Reading:**

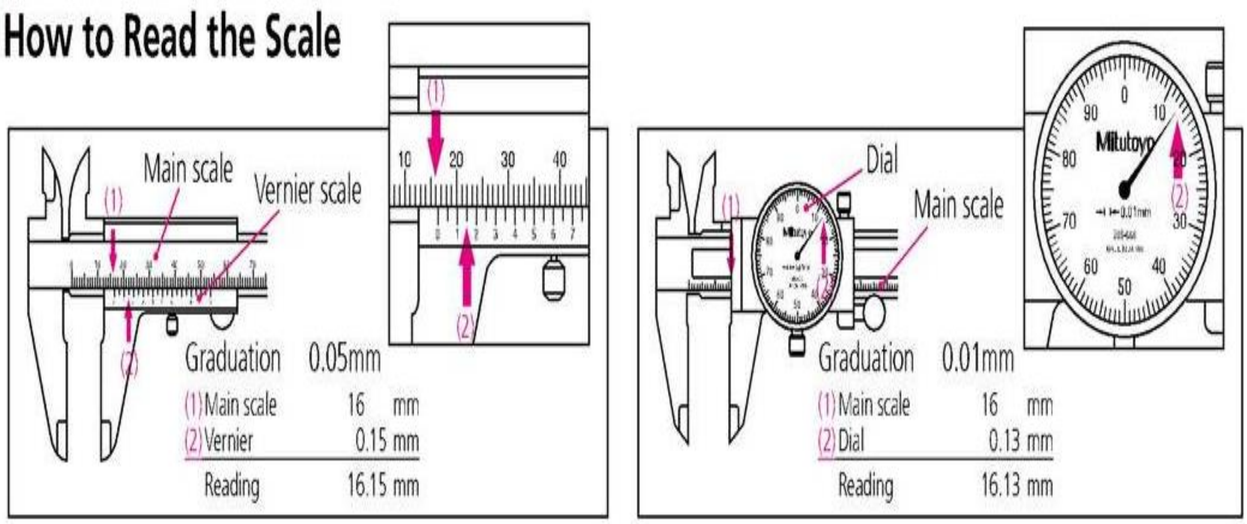


Figure 58. Vernier Reading system

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.

MAIN AND VERNIER SCALE

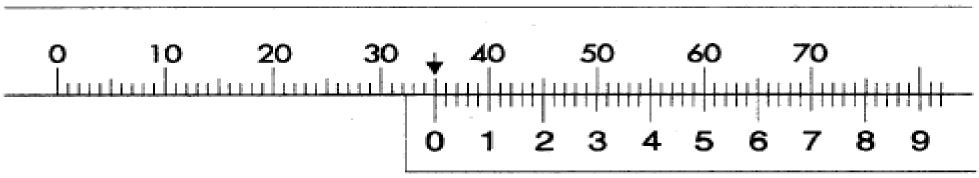


Figure 59. caliper reading

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.

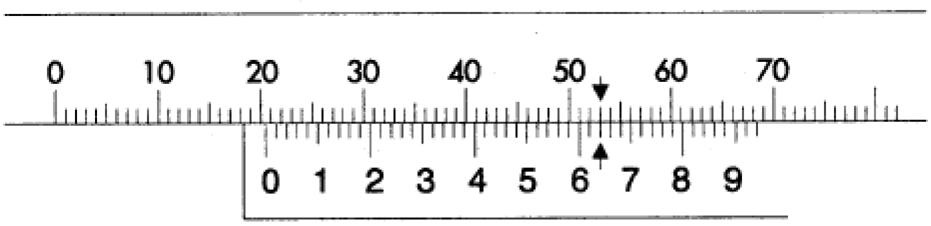


Figure60. A non-integer reading on the main scale

Figure 52 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**

1.1.3. 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.

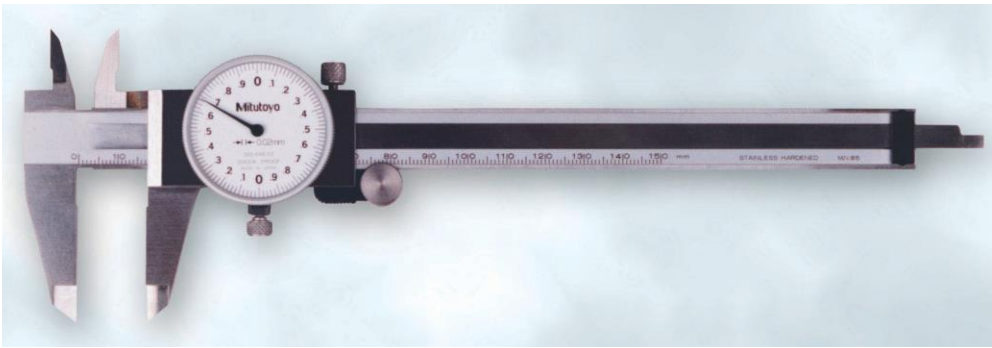


Figure61. Dial caliper

1.1.4. 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.

Figure62. vernier height gauge

1.1.5. 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 63. 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 64. 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 65. Depth Micrometer

- **Metric system Micrometer Reading**

Figure66. 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.

1.1.6. 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 67. Dial Indicator

1.1.7. 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.

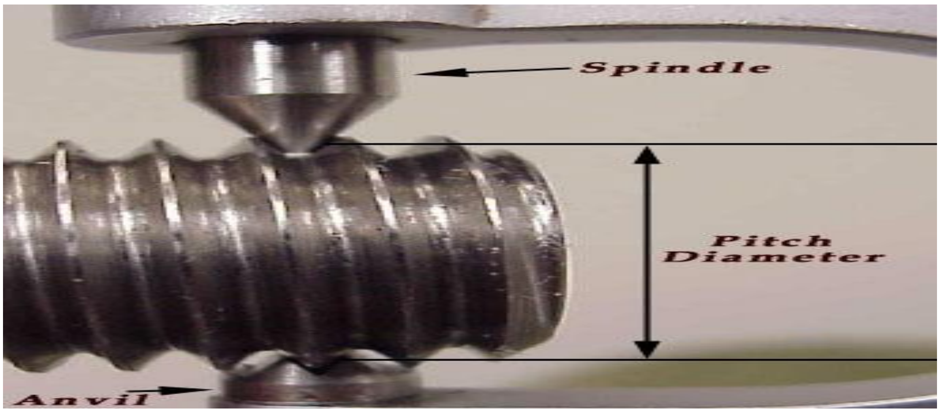


Figure 68. Screw thread micrometer

1.1.8. 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 69. Slip Gauge

1.1.9. 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 70. Different size Plug gage

1.1.10. comparator

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

Figure 71. Comparator

SELF-CHECK -1	WRITTEN TEST

Directions: Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

I. **Choose the best answer(each 1 point)**

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
7. -------------- used for measuring height of an object or for marking lines onto an object of given distance from a datum base
   a. Meter tap b. Micrometer c. vernier height gauge D. None

Note: **Satisfactory rating –above 3.5 points** **Unsatisfactory - below 3.5 points**

You can ask you teacher for the copy of the correct answers.

INFORMATION SHEET-2	CHECKING TECHNIQUES OF WORK PIECES

2.1. **Checking techniques of work pieces**

2.1.1 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:

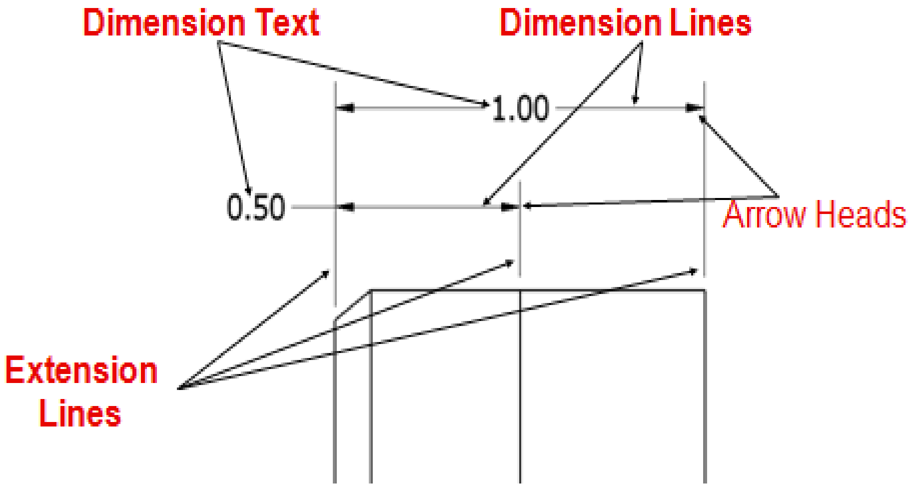


Figure 72. Linear dimensions

- **Dimensioning Methods**

  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.

Dimensioning Symbols

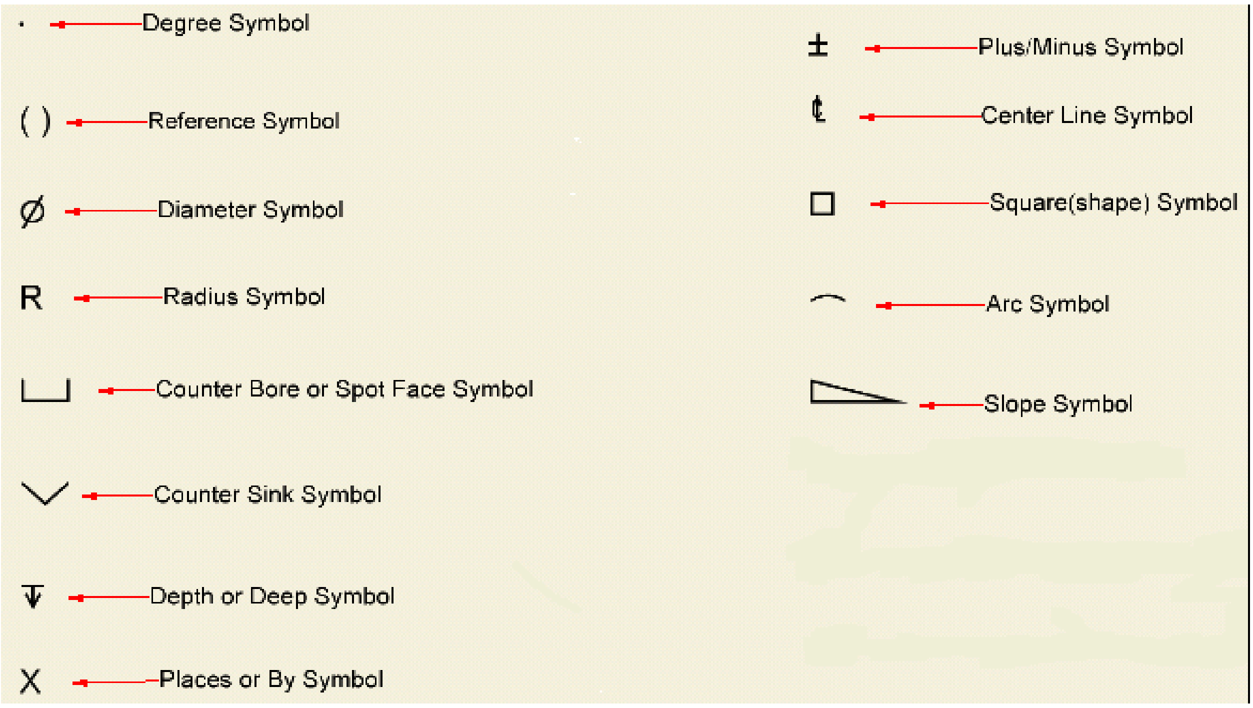


Figure 73. Dimensioning Symbols

2.1.2 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,**
  2.500 +0.010, -0.005

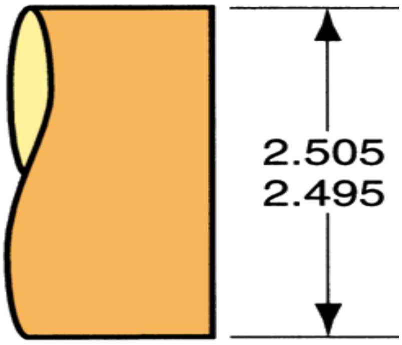


Fig74A. Limit Dimensions

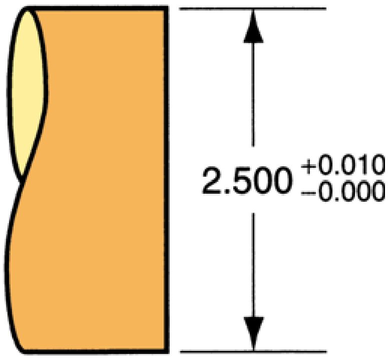


fig74B. Unilateral Tolerance

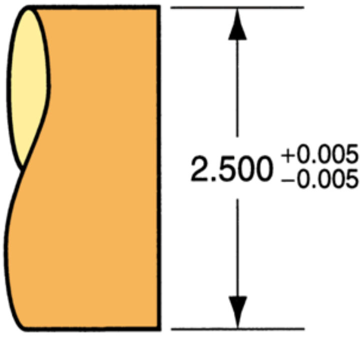


fig74C. Bilateral Tolerance

2.1.3 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

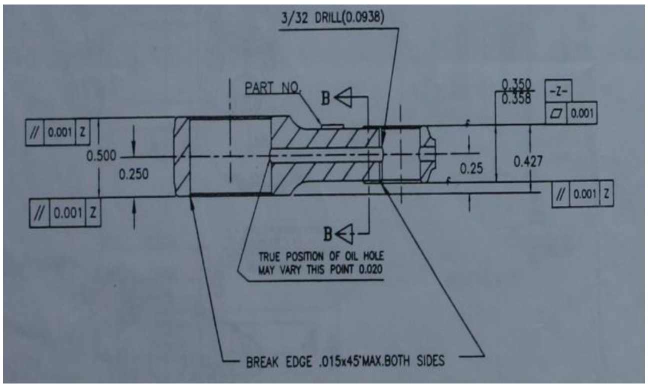


Figure75. Surface Finishing

- 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.

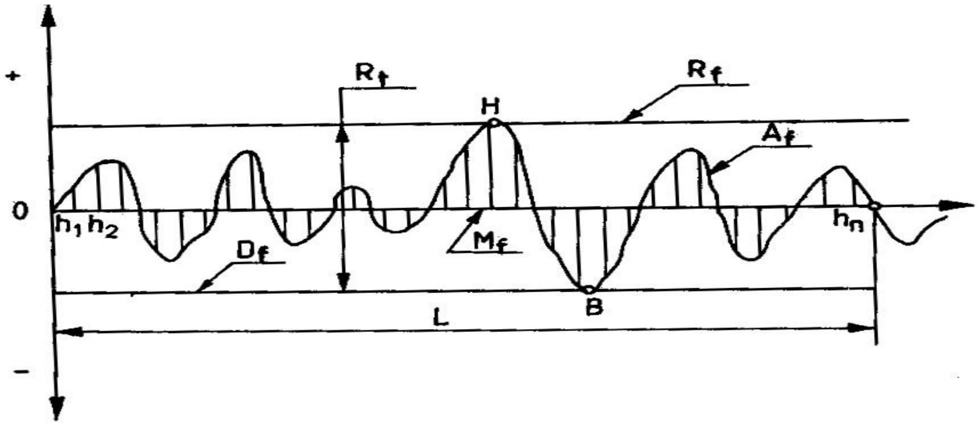


Figure76.surface roughness

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,

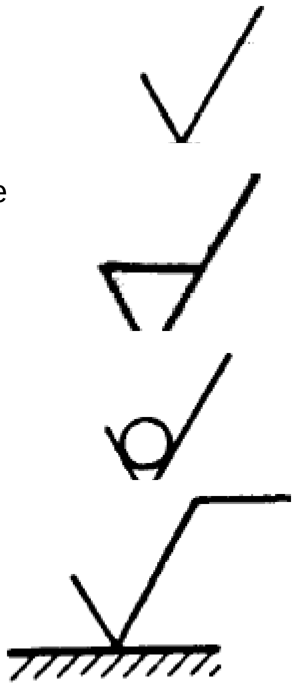


Figure77.surface texture

Symbols with Additional indications.

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

Table 5.surface roughness

SELF-CHECK -2	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

I. Choose the best answer(each 1 point)

1. The geometrical characteristics of a surface include,
   a. Macro-deviations b)Surface waviness C) Micro-irregularities d) All
2. Which Variation shows from the specified dimension is permitted in only one direction, either positive or negative, but not both.
   a) Unilateral Tolerance B)Bilateral Tolerance b)Limit Dimensions d) None
3. -------------- : are used to describe the sizes and relationships between features in your drawing
   a) Surface finish b. Tolerance c. Dimensioning d. all
4. The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size.
   a. Fit b/ Tolerance c. Allowance d/ None

Note: **Satisfactory rating –above 2 points Unsatisfactory - below 2 points**

You can ask you teacher for the copy of the correct answers.

INFORMATION SHEET-3	HANDLING DEVIATIONS

1.1 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 the corresponding basic size. It is denoted by letters 'EI' for a hole and 'ei' for a shaft..

SELF-CHECK -3	WRITTEN TEST

Directions: Answer all the questions listed below. Use the Answer sheet provided in the next page:

I. Matching column A with Column B(each 1 point)

1. Deviation: A. It is the algebraic difference between the minimum limit of size and the corresponding basic size.
2. Upper Deviation: B. It is the algebraic difference between the maximum limit of size and the corresponding basic size.
3. Lower Deviation: C. It is the algebraic difference between a limit of size and the corresponding basic size.

Note: **Satisfactory rating –above 1.5 points** **Unsatisfactory - below 1.5 points**

You can ask you teacher for the copy of the correct answers.

OPERATION SHEET 1	MEASURING TOOLS AND EQUIPMENT

1.1 PRINCIPLE AND USE VERNIER CALIPER

Steps 1- Apply safety precaution.
Steps 2- Move the sliding jaw to be in contact with the faces being measured.
Steps 3- Tighten the locking screw on the clamp.
Steps 4- Make fine adjustment using the fine-setting screw.
Steps 5- Move the jaws so that they just touch the work; do not apply any force.
Steps 6-Tighten the head lock.
To read (for example) 25.44 mm from the caliper, look for the number of the millimeter division below the venire zero: in Figure 65, for example, it is 25. Next find the line on the venire scale that coincides with a line on the main scale: in this case 22. To calculate the total measurement, multiply 20 by 0.2 and add to 25:
That is:
Main scale reading = 25.00 mm
Venire scale reading = 22 X 0.2 (0.44 mm) Final reading = 25.44 mm

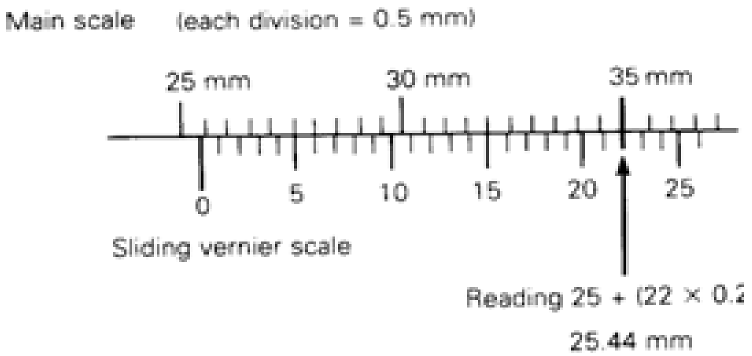


Figure 78.Taking a measurement with the vernier caliper.

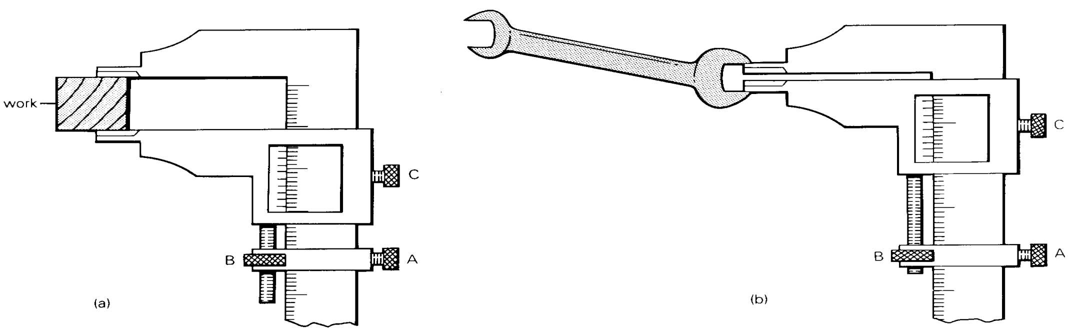


Figure 79.reading internal and external venire caliper

OPERATION SHEET 2	MEASURING TOOLS AND EQUIPMENT

1.1 PROCEDURE OF MICROMETER READING

Steps 1- Apply safety precaution.
Steps 2- Hold the plastic insert to prevent thermal expansion.
Steps 3- Keep the measuring faces square with the surfaces that you are measuring, to ensure an accurate measurement.
Steps 4- Turn the thimble until the faces touch the work.
Steps 5- Use the ratchet (if there is one) to obtain the correct pressure when turning the thimble, and prevent the jaw from moving further when it comes into contact with the work.

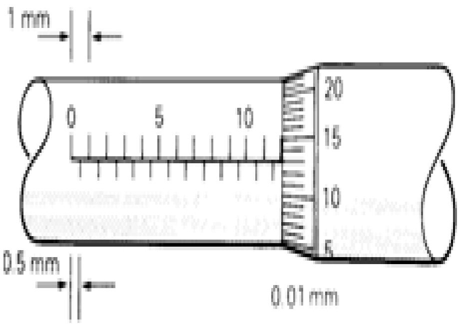


Figure 80.Micrometer reading

The reading in Figure 67 is:
Upper main scale 12.00 mm
Lower main scale (no half mm) 0.00 mm
Circular thimble scale 0.13 mm
13 X 0.01 mm = 0.13mm
Total reading **12.13 mm**

Note: The micrometer is an expensive tool, and you must take the utmost care when using it. The following points should help you.

- Make sure that you store the micrometer away from dust.
- Always clean the measuring faces for good results.
- Oil the micrometer regularly to avoid rust.
- Pack the micrometer in its box when not in use.
- Do not use force on the thimble or ratchet.

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 --- hours.

TASK 1: principle and use vernier caliper

TASK 2: micrometer reading

LIST OF REFERENCE MATERIALS

1. lements of work shop technology 7th Edition 1996 by S.K Hajrachoudhury, S.K Bose
2. Metal work Technology 1st 2007 J.K. SV.SACKEY AND S.K AMHAKOBENE
3. Manufacturing Technology 3rd edition 1998 M.HASLEHURST
4. Engineering drawing 1st ,2nd&3rd Impression 2007,2009 ,M.B.shah B.c Rana
5. Elements of work shop technology 7th Edition 1996 by S.K Hajrachoudhury, S.K Bose
6. Metal work Technology 1st 2007 J.K. SV.SACKEY AND S.K AMHAKOBENE
7. Manufacturing Technology 3rd edition 1998 M.HASLEHURST
8. Engineering drawing 1st ,2nd&3rd Impression 2007,2009 ,M.B.shah B.c Rana
9. Machine drawing 3rd edition K.L. Narayana,P.Knnaiah,k.Venkata Reddy
10. Elements of work shop technology 7th Edition 1996 by S.K Hajrachoudhury, S.K Bose
11. Metal work Technology 1st 2007 J.K. SV.SACKEY AND S.K AMHAKOBENE
12. Manufacturing Technology 3rd edition 1998 M.HASLEHURST
13. Engineering drawing 1st ,2nd&3rd Impression 2007,2009 ,M.B.shah B.c Rana
14. Machine drawing 3rd edition K.L. Narayana,P.Knnaiah,k.Venkata Reddy

The trainers who developed the learning guide

No	Name	Qualification	Region	E-mail
1	Wakweya Tolera	A	Oromia	Wakweya.tolera@yahoo.com
2	Mohammed Abdela	A	Dire Dewa	Mahammedabdella49@gmail.com
3	Waliyi Bena	A	Oromia	waliyibena@gmail.com
4	Elias Zemenu	A	Somali	Elijahze26@gmail.com
5	Gurmu Mulatu	A	Adis Ababa	gurmumuulaatuu@gmail.com
6	Muluneh Negussie	B	Amhara	Muluneh3000@gmail.com
7	Teshome W/Yuhanis	A	SNNPR	Tesho1921@gmail.com
8	Solomon Aderaw	B	Amhara	Solo19@gmail.com