BASIC METAL WORK

Level - I

Learning Guide -40

Unit of Competence: -Operate Basic Workshop Machinery

Module Title: - Operating Basic Work shop Machinery

LG Code: IND BMW1 M12 LO1-LG-40

TTLM Code: IND BMW1 TTLM 0919v1

LO1: Determine and plan job requirements

Instruction Sheet	Learning Guide 40

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

- **Analyzing and selecting work requirements**
- **Selecting appropriate machine and tools**
- **Determination of sequence of operations**

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 –**

- **Analyze and select Work requirements from work instructions and drawings/diagrams**
- **Select appropriate machine and tools based on work requirements**
- **Determine sequence of operations for maximum efficiency and to meet work requirements and specifications**

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described in number 3 to 13.
3. Read the information written in the "Information Sheets 1". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
4. Accomplish the "Self-check 1" in page 5.
5. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 1).
6. If you earned a satisfactory evaluation proceed to "Information Sheet 2". However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
7. Submit your accomplished Self-check. This will form part of your training portfolio.
8. Read the information written in the "Information Sheet 2". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
9. Accomplish the "Self-check 2" in page 22.
10. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 2).

11. Read the information written in the “Information Sheets 3. Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
12. Accomplish the “Self-check 3” in page 25.
13. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 3).

Information Sheet-1	Analyzing and selecting work requirements

1.1. Work instructions and drawings/diagrams

A work instruction is a tool provided to help someone to do a job correctly. A Work Instruction is the most detailed description of a task. Its purpose is to explain step by step how to do a specific task or work. Work instructions are key to reducing variation, allowing manufacturers to improve quality and meet demand.

a. Working drawings/Diagrams

The term working drawing is used to describe the complete set of drawing information needed for the machining and assembly of a product based on its design. An essential element of a working drawing is the parts list, or bill of materials (abbreviated BOM).

A working drawing is a drawing or set of drawings produced by the designer, manufacturer, or fabricator. Shop drawings are typically required for prefabricated components. The working drawing normally shows more detail.

The primary role or function of working drawings is to convert design data into finished part information and to clearly communicate that information to building industry, code officials, product manufacturers, suppliers and fabricators.

Assembly drawings can be used to represent items that consist of more than one component. ... Assembly drawings may include instructions, lists of the component parts, reference numbers, and references to detail drawings or shop drawings, and specification information.

It helps streamline the manufacturing process. The precise details of the technical drawing helps builders create objects without errors, delays or other costly issues. This alone is the most important duty of the drawings.

Working drawings will develop in detail from block and massing drawings and sketches to very detailed technical drawings describing every component in a way that will enable them to be constructed and operated.

1.2 **Safety Requirement in machine shop**

Safety

The importance of safety was realized a century ago because of the occurrence of millions of industrial and other accidents, which resulted in death, or disablement of industrial employees every year.

1.1 Causes of Accidents

The accidents may take place due to

- Human causes,
- Environmental causes and
- Mechanical causes.

a. ***Human Causes***

- Accidents may occur while working on unsafe or dangerous equipment or machineries possessing rotating, reciprocating and moving parts.
- Accidents occur while operating machines without knowledge, without safety precautions, without authority, without safety devices
- Accidents generally occur while operating or working at unsafe speed
- Working for long duration of work, shift duty etc.
- Accidents commonly occur during use of improper tools
- Accidents may occur while working with mental worries, ignorance, carelessness, nervousness, dreaming etc.
- Accidents occur because of not using personal protective devices

b. ***Environmental Causes***

- Working at improper temperature and humidity (causes fatigue to the workers so a chance of accidents increases with workers having fatigue).
- The presence of dust fumes and smoke in the working
- Poor housekeeping, congestion, blocked exits; bad plant layout etc. may cause accidents.
- Accidents occur due to inadequate illumination.
- Improper ventilation in the plant may also leads to industrial accidents**.**

c. **Mechanical Causes**

- Continued use of old, poor maintained or unsafe equipment may result in accidents.
- Accidents commonly occur due to use of unguarded or improper guarded machines or equipment
- Unsafe processes, unsafe design and unsafe construction of building structure may lead to accidents in the plant.
- It can occur due to improper material handling system and improper plant layout.
- May occur due to not using of safety devices such as helmets, goggles, gloves, masks etc.

a. Other general causes of accidents in workshops

- Because of ignorance to work with
  -Equipment,
  - Hand tools,
  -Cutting tools and
  -Machine tools.
- Operating machine and equipment's without knowledge
- Extra curiosity to work without knowing
- Poor working conditions. Because of speedy work
- Improper method to work
- Due to use of improper tools
- Because of lack of discipline
- Uninterested in work
- Due to carelessness
- Due to over confidence
- Bad working environment
- Excessive over times duty by industrial workers
- Dangerous materials with which to work
- Lack of cleanliness
- Due to poor planning

As always we should be aware of safety requirements and attempt to observe safety rules in order to eliminate serious injury to ourselves or others.

Wear glasses, short sleeves, no tie, no rings, no trying to stop the work by hand. Stop the machine before trying to check the work. Don't know how it works? –"Don't run it." Don't use rags when the machine is running.

Unguarded moving parts of machines/equipment and the sudden or uncontrolled release of their power systems can result in serious injuries.

Personnel working with machines must be aware of the risks involved and follow safe work practices.

b. Causes of accidents while working with machinery

- Loose clothing, hair, jewelry being caught in moving parts.
- Materials ejected from the machine when it is operational.
- Unplanned starting of the machine.
- Slipping and falling into an unguarded nip.
- Contact with sharp edges, e.g., cutting blade.
- Making adjustments while the machine is operational.
- Unauthorized operation of machines.
- Lack of preventive maintenance.

c. Hazards parts of machines

Rotating machine parts give rise to damages or injuries. Examples are

- Rotating gears
- Belt and its pulley
- Chain and sprocket
- Between grinding wheel and tool rest
- Between rotating and fixed parts-Rotating parts operating alone
- Shafts
- Couplings-Reciprocating and sliding motions

d. SAFETY RULES

a) Personal safety

- Dress appropriately remove necktie, necklace, wrist, watch & rings
- Wear apron or a properly shop fitted over coat and safety glasses
- place all guards before attempting to operate the machine
- To clean the lathe, do not remove chips with bare hands.
- Care must be taken when handling long sections of metal stock.
- Keep hand tools in good conditions and store them in such a way that peoples cannot be injured.

b) Machine safety

- No attempt should be made to operate a lathe until you know the proper procedure
- Don`t use compressed air to remove chips and cutting oil from machine.
- Keep the machine clear of tools.

c) Work shop safety

- Avoid horse play
- Keep the floor around your machine clear of chips and wipe up spilled cutting fluid
- Oily rugs must be placed in approved safety containers.

1.3 Safety when Machining Work Piece

- Do not operate any machine before understanding its mechanism.
- Always stop a machine before measuring, cleaning or making any adjustments. It is dangerous to do any type of work around moving parts of a machine.
- Never operate a machine unless all safety guards are in place.
- Keep hands away from moving parts. It is dangerous practice to "feel" the surface of the revolving work or to stop a machine by hand
- Never use a rag near the moving parts of a machine rag may be drawn into the machine, along with the hand that is holding it.
- Never have more than one person's operate a machine at the same time. Not knowing what the other person would or would not do has caused many accidents.

Get first aid immediately for any injury, no matter how small

1.4 FACTORS DETERMINING JOB REQUIREMENTS

1. Materials to be used
2. Surface finished required
3. Tolerance to be allowed
4. Quantity of units
5. Scale of drawing
6. Name of the object and how it is used
7. Types of drawings

Self-Check -1	Written Test

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

1. Write three examples of Hazards parts of machines? (3points)
2. Loose clothing, using jewelry while operating machine which has moving parts not so much causes of accidents. (1point)

   A. True B. False

3. Essential causes of accidents a. ____________________ ? (5point)

   b. ---

   c. --

4. Write three types of ***Human Causes*** accidents? (5point)

5. Write the Good working habits of a skilled machinist (5point)

6. Name the seven (7) factors to be considered in determining the job requirements

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

Answer Sheet

Score = ____________
Rating: _____________

Name: ___________________________ Date: ________________

Short Answer Questions

Information Sheet-2	Selecting appropriate machine and tools

2.1 Introduction

Machining is a manufacturing process in which a sharp cutting tool is used to cut away material to leave the desired part shape. Machining is one of the most important manufacturing processes. To perform the operation, relative motion is required between the tool and work. This relative motion is achieved in most machining operations by means of a primary motion, called the cutting speed, and a secondary motion, called the feed. The shape of the tool and its penetration into the work surface, combined with these motions, produces the desired geometry of the resulting work surface.

Types of machines

2.2 Lathe machine

The lathe removes undesired material from a rotating work piece in the form of chips with the help of a cutting tool which is traversed across the work and can be feed deep in work. 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). The lathe machine is used principally for shaping of metal (and sometimes wood or other materials). The basic lathe that was designed to cut cylindrical part, to produce screw threads, tapered work, drill holes, knurled surfaces, and crankshafts and etc. Various designs and constructions of lathe have been developed to suit different Machining conditions and usage.

A. TYPES OF LATHE MACHINE

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

B. The five Major Parts of Lathe Machine

- The Bed
- The Headstock
- The Tailstock
- The Carriage
- The Feed Mechanism

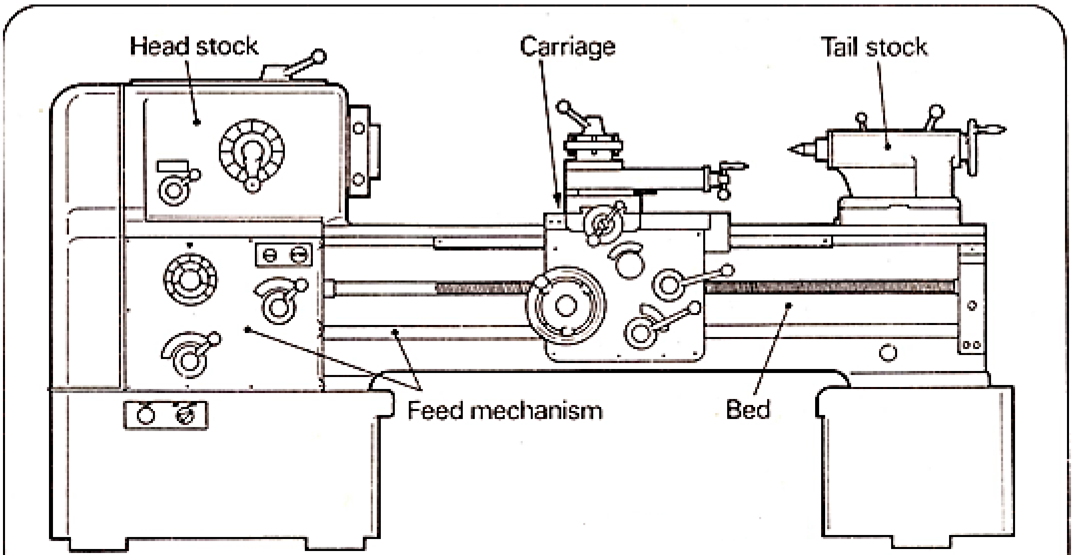
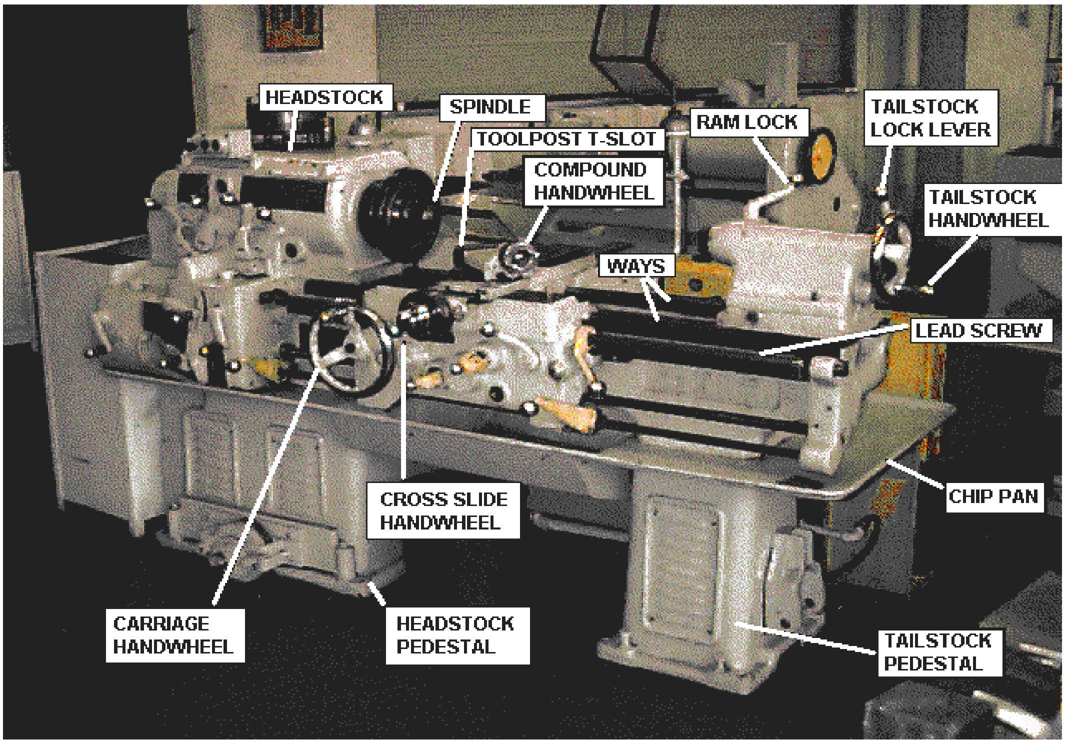
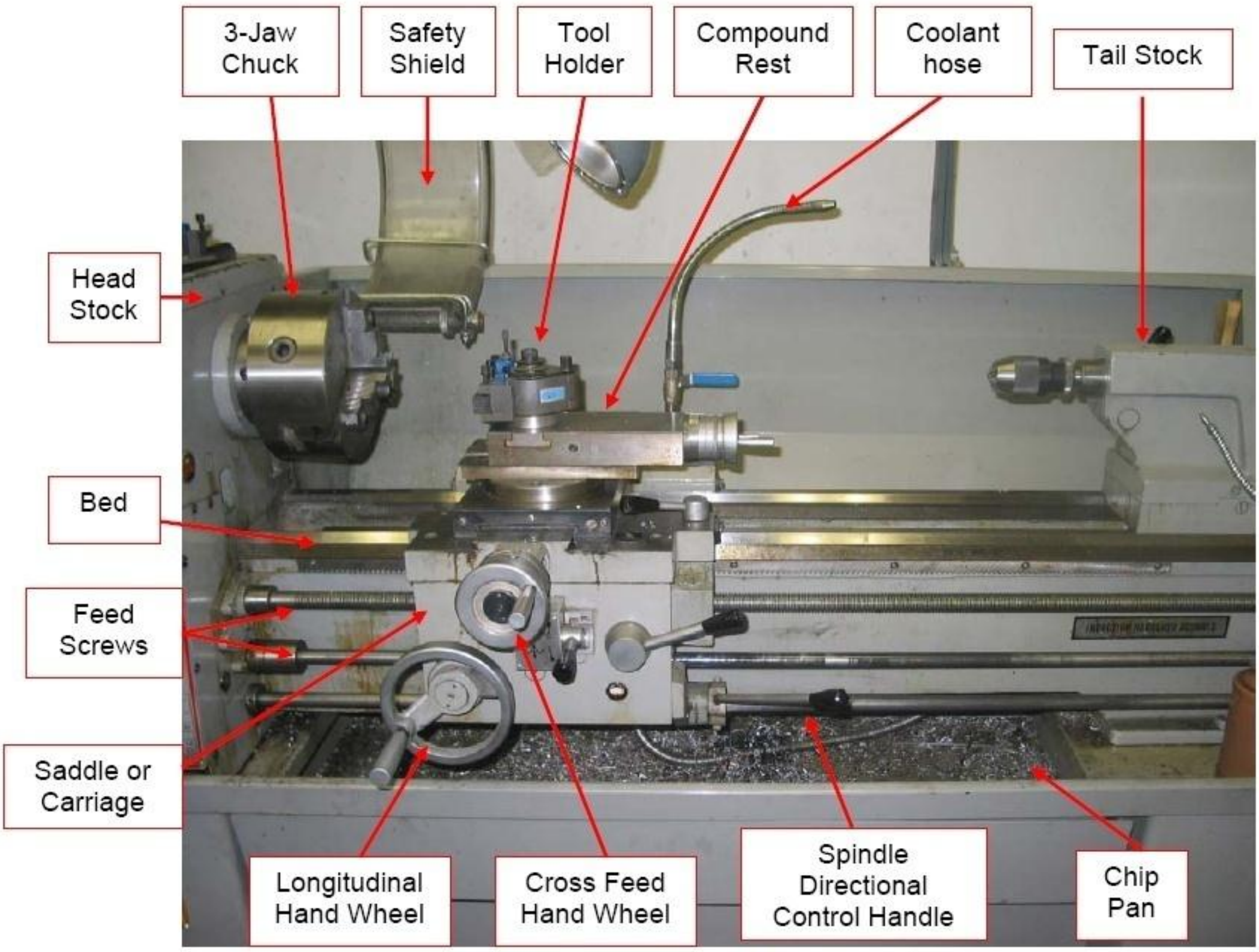


Fig 1.1. Parts of Lathe machine





1. **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.

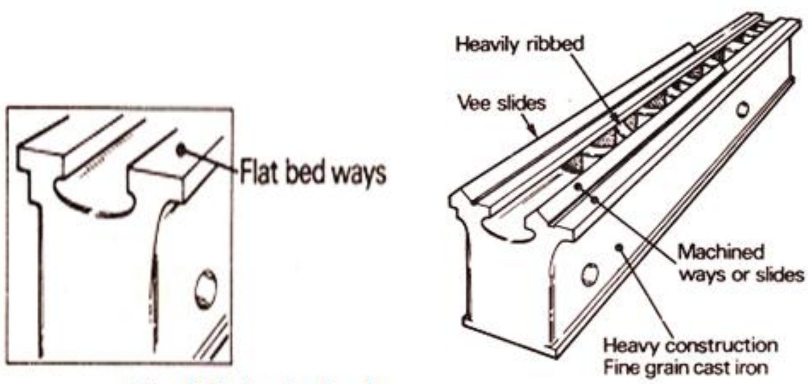


Fig 1.2: Lathe bed

Fig. 1.1

2. **HEADSTOCK:** The headstock is the lathe feature that provides the means of holding and rotating the work accurately.

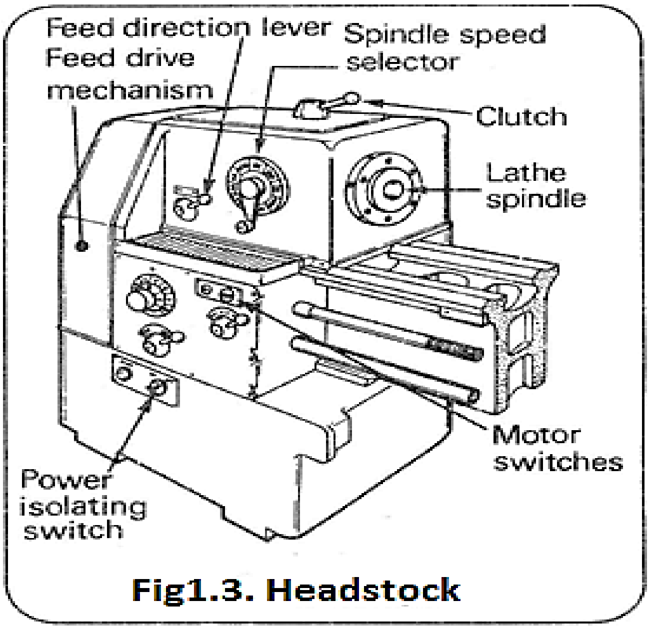


Fig1.3. Headstock

Fig. 1.2

Fig. 1.3

3. **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.

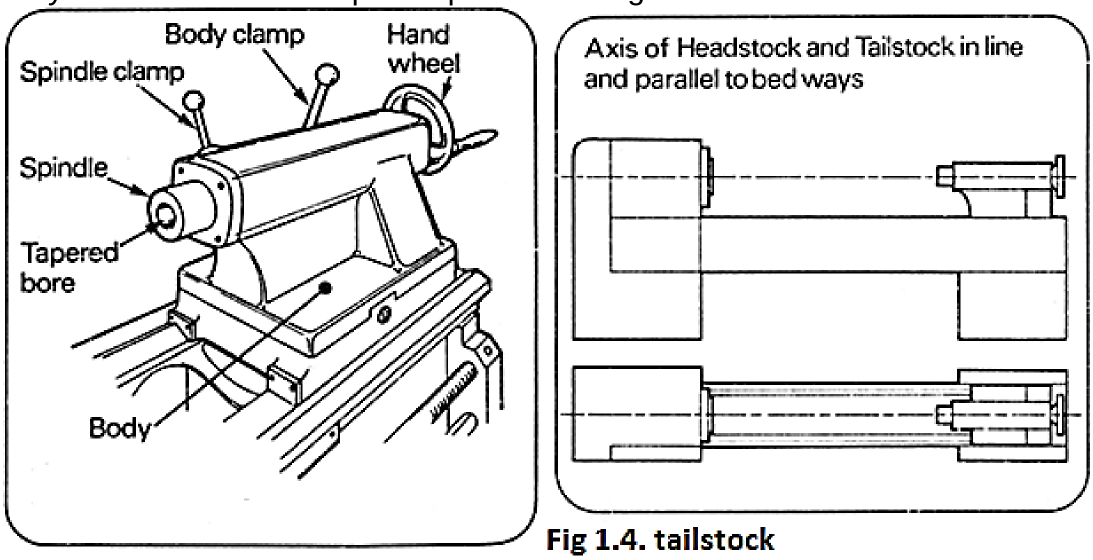


Fig 1.4. tailstock

Fig. 1.3

4. **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**
    - **Saddle**

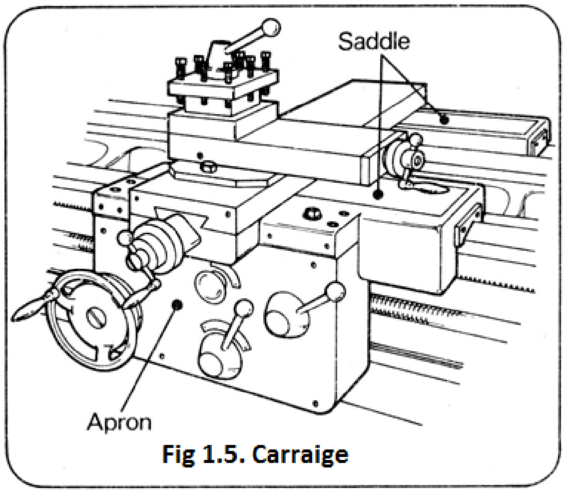


Fig 1.5. Carraige

Fig. 1.4

a. **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°.The **top slide** is fitted on top of the compound rest. It provides a means of supporting the tool post, which holds the cutting tool. The top slide provides a limited horizontal movement for the cutting tool. By swiveling the compound rest, the top slide may be set an angle to the cross slide. Normally the compound rest is set so that the top slide is at the right angles to the cross slide.

b. **Apron** is bolted to the front of the saddle. It contains the mechanism for moving and controlling the carriage.

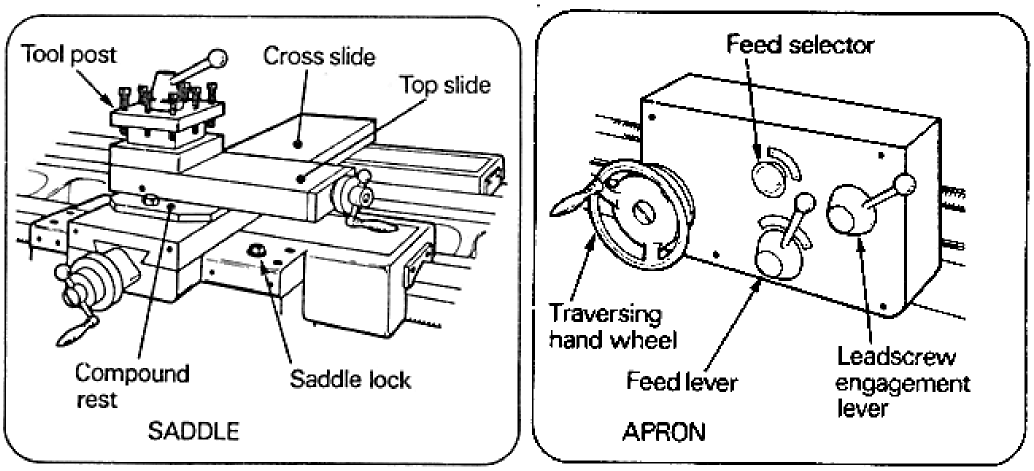


Fig1.6. Saddle and apron of lathe machine

5. 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 slid

a. 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.

b. 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.

c. 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.

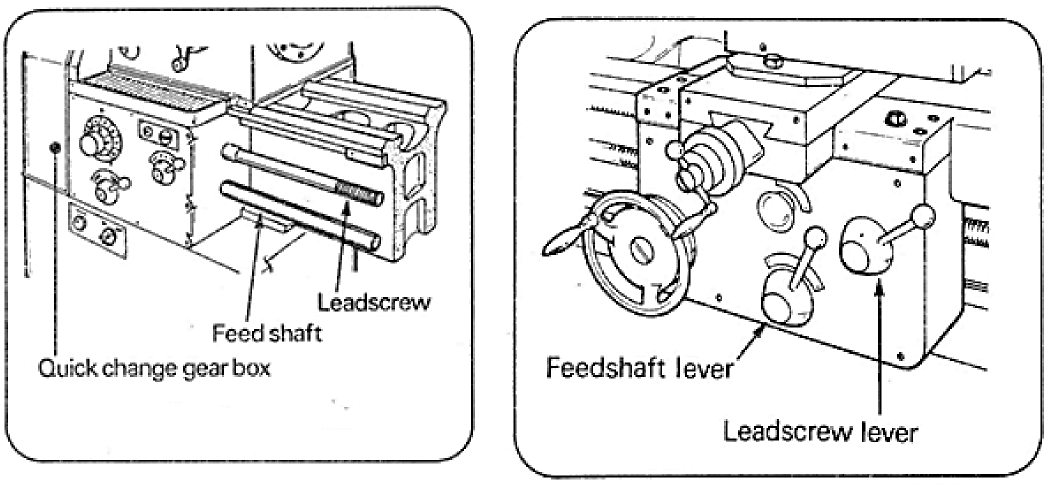


Fig. 1.7. Feed Mechanism

d. Feed engagement 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.

e. 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.

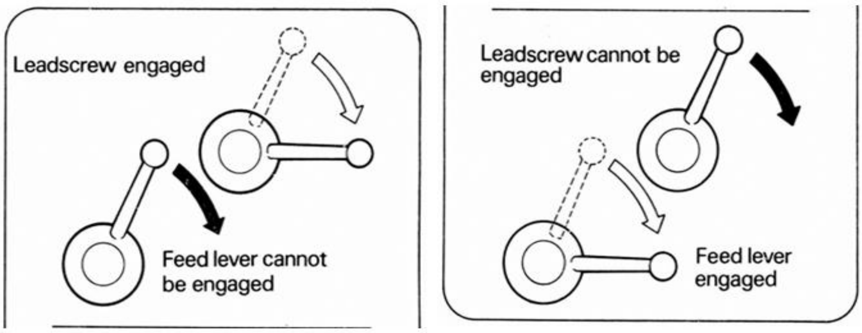


Fig1.8. Lead screw and Feed engaged/disengaged levers

6. TOOL POST

The tool post is fixed on the top of the top slide.
Several types are used. They all have the features of firmly supporting and holding the cutting tool. The most widely used **types** are:

- **Square tool post**
- **Rigid tool post**
- **Standard tool post**

The **square tool post** is a common type. It provides positions for holding four different cutting tools. The post can be indexed to position a selected tool against the work

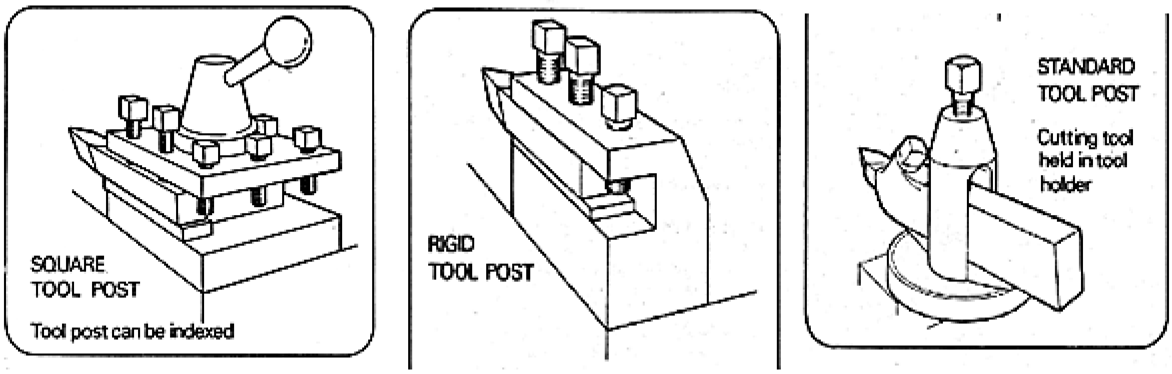


Fig: 1.9. Types of Tool posts

7. Lathe Machine Accessories

A. lathe chucks

- ✓ 3-Jaw universal chuck- is designed to permit all jaws to operate at one time.

It will automatically center round or hexagonal shaped stock.

- ✓ 4-Jaw independent chuck- Each jaw of the 4-jaw independent chuck operates individually.

This permits square rectangular and odd shaped work to be centered.

- ✓ Magnetic chuck – it has magnetic character to hold the work.

- Note: - Never turn on the lathe until checking that you did not accidentally leave the chuck key in the chuck.

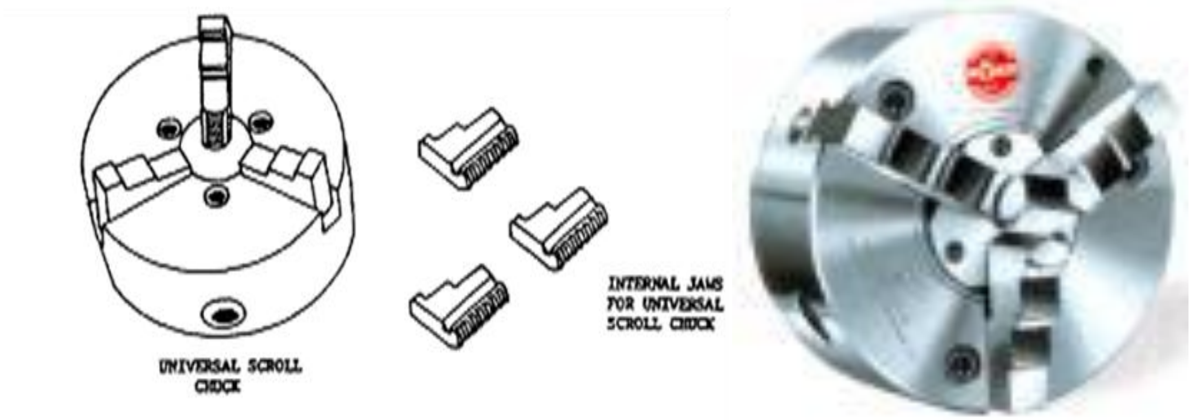


Fig7.1a 3-Jaw universal chuck

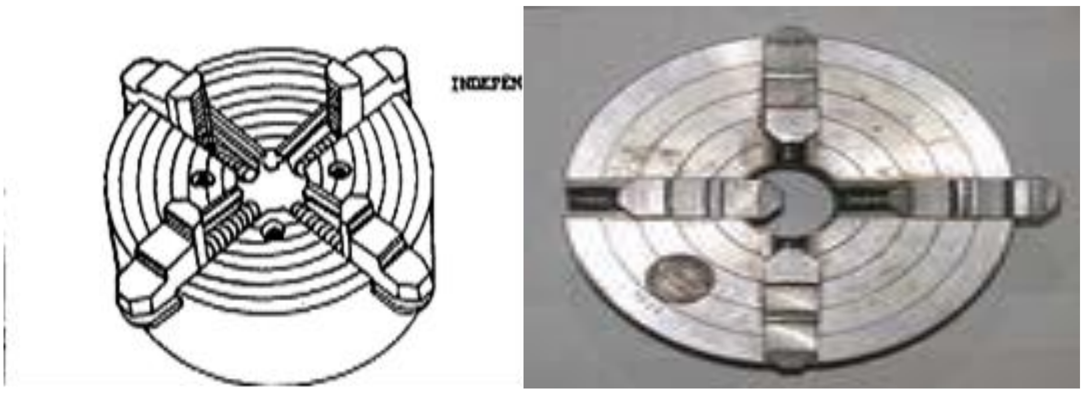


Fig7.1b 4-Jaw independent chuck

Fig7.1c Magnetic chuck

b. Face plate: - a circular plate that fits to head stock spindle and drives or carries work to be machined. Used to turn eccentric turn on the work.

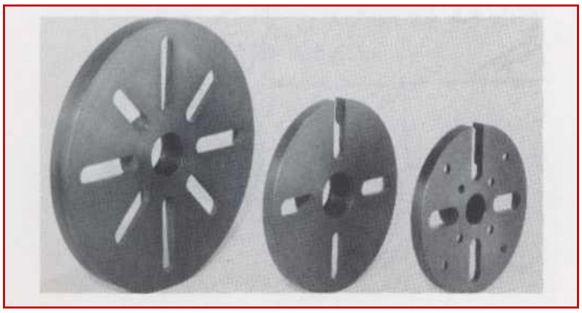


Fig. b. Lathe faceplates come in various sizes

c. Lathe centers:-a part fitted to the tail stack

These are:-

a. **Live center or revolving center**: a heavy duty ball bearing or a rotating center. Fig. 10.65
b. **Dead center**: this is a center which does not revolve.
c. **Half center**: widely used in milling machine.

_Lathe centers are used for:-

- Supporting long work piece.
- Centering cutting tool.

d. **Taper attachment**:

- This attachment is confined to cut external tapers_only.
- It is bolted on the back of the lathe and has a guide bar which may be set at the desired angle of taper.

e. Mandrel

Mandrel is used to hold work pieces with previously machined holes.

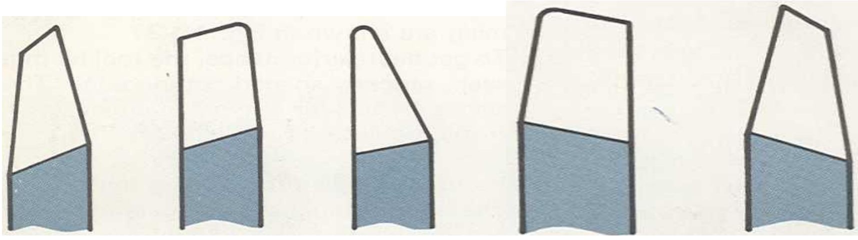
f. Lathe dog:

Lathe dog is a device for clamping work so that it can be machined between centers.

8. LATHE CUTTING TOOLS

1. Turning and facing tools

- ❖ Left- cut roughing tools. The left cut roughing tool cuts are most efficient when it travel from left to right.
- ❖ Right- cut roughing tools:- operates just the opposite of left cut roughing tool



LEFT –CUT TOOLS RIGHT-CUT TOOL

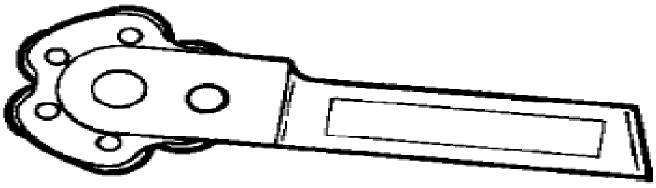
Fig8.1

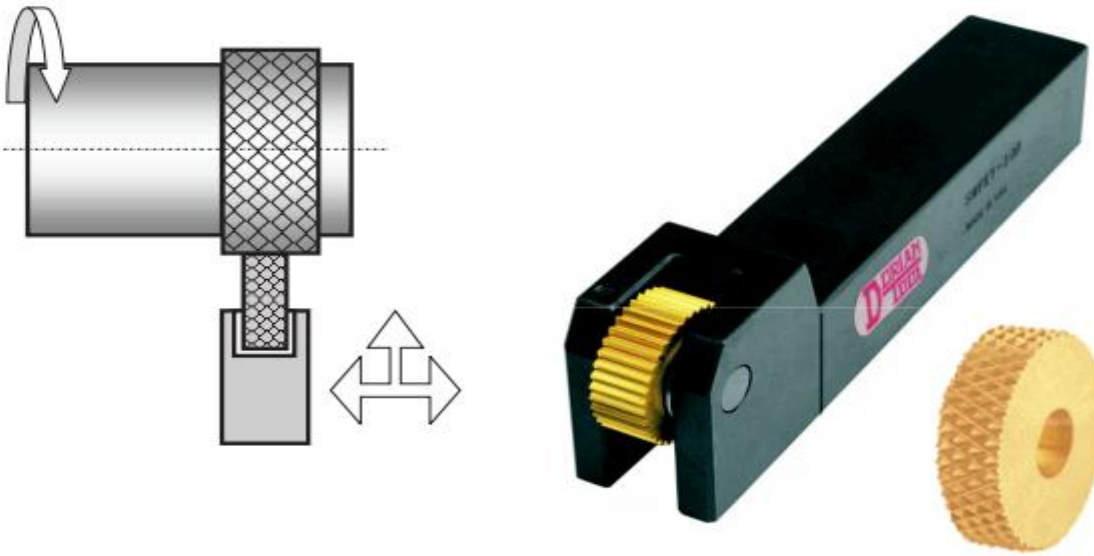
2. Threading tool

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. 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.





(*Left*) Knurling operation; (*Right*) Knurling tool and knurling wheel. Wheels with different patterns are easily available.

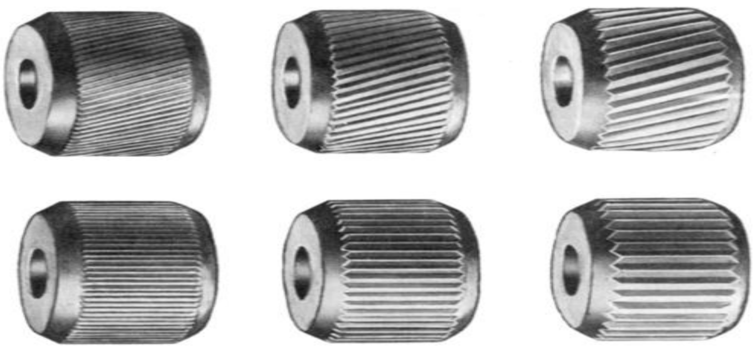


Fig the common knurling tool

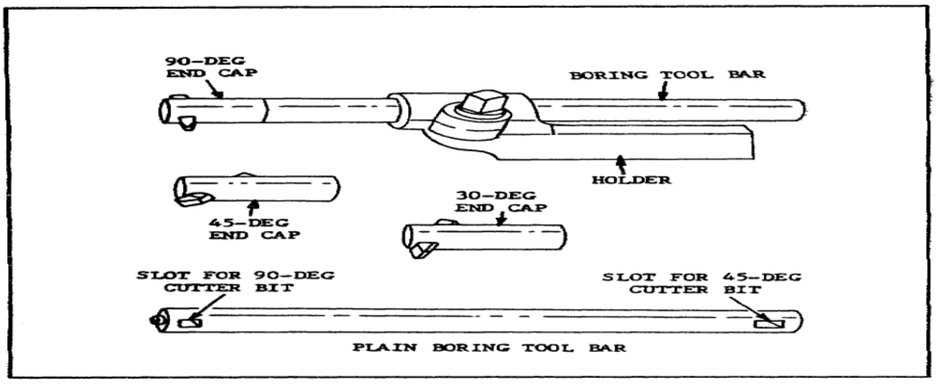
4. Parting off tool

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.

- The parting cutter bit has its principal cutting edge at the end.
- Both sides must have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge.
- The bit is convenient for machining necks and grooves, square

5. Boring tool

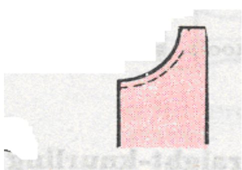
Is ground the same as a left-cut turning tool. However the front clearance must be ground at a slightly greater angle .this will prevent the heel of the tool from rubbing in the hole of the work piece.

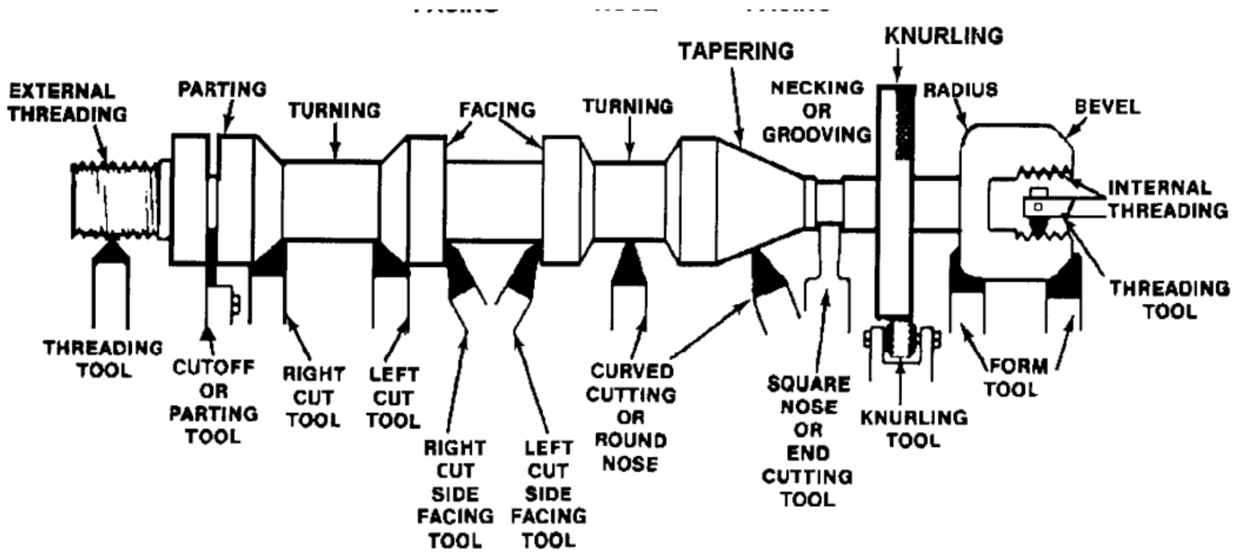


- Is ground the same as a left-cut turning tool.
- However the front clearance must be ground at a slightly greater angle this will prevent the heel of the tool from rubbing the work piece.

6. FORMING TOOLS

- They have no rake angle. In order to retain to their shape they are reground only on the cutting face.





2.3 Characteristics of Cutting Tools

• The cutting tool should have the following characteristics.

- High hardness
- High hardness temperature, hot hardness
- Resistance to abrasion, wear, chipping of the cutting edge.
- High toughness (impact strength)
- Strength to resist bulk deformation

2.4 Cutting Tool Properties:

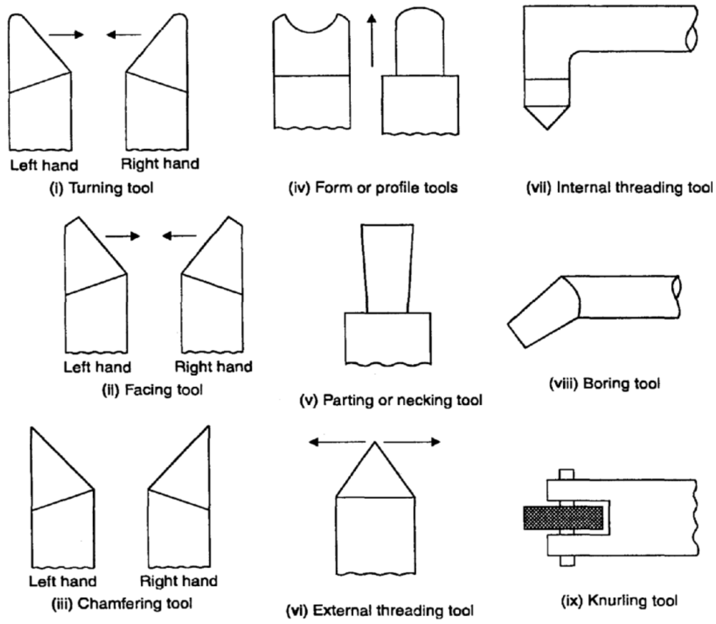
- Good chemical stability
- Adequate thermal properties
- High elastic modulus (stiffness)
- Consistent tool life
- Correct geometry and surface finish

2.5 Lathe's Tool shapes

- The basic tool shapes required are not many but the few there are can be applied to a variety of tools for both internal and external work. The first of these is the roughing tool which probably needs less power (despite its ability to take heavy cuts) than any other.
- In a lathe, for a general purpose work, the tool used is a single point tool (one cutting edge), but for special operations multi-point tools may be used.

Depending upon the nature of operation done by the tool, the lathe tools are classified as follows

- ✓ Turning tool (left hand or right hand
- ✓ Facing tool (left hand or right hand)
- ✓ Chamfering tool (left hand or right hand)
- ✓ Form or profile tool
- ✓ Parting or necking tool
- ✓ External threading tool and Internal threading tool
- ✓ Boring tool
- ✓ Knurling tool



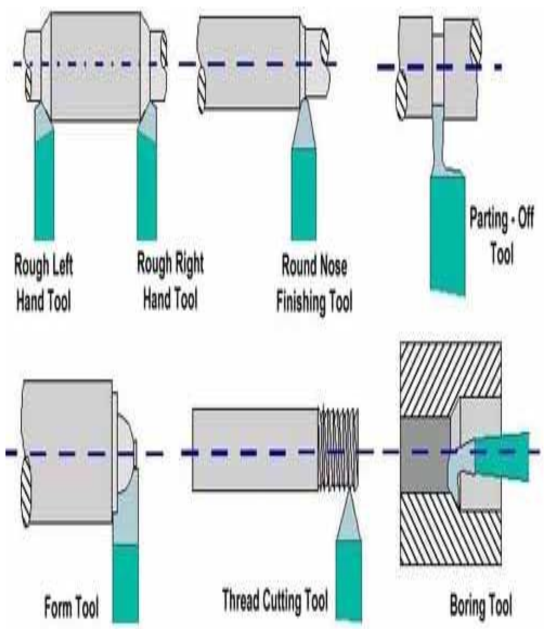


Fig basic Lathe tool shapes

9. Cutting tools Terminology and Tool Geometry

• For cutting tools, geometry depends mainly on the properties of the tool material and the work material.
• The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end and side relief angles

a. RAKE ANGLE

Rake angle is the angle between the top face of the tool and the normal to the work surface at the cutting edge
• A **large** rake angle will improve cutting action, but would lead to early tool failure, since the tool wedge angle is relatively weak
. The **back** rake angle affects the ability of the tool to shear the work material and form the chip. It can be positive, negative or neutral
• **Positive** rake angles reduce the cutting forces resulting in smaller deflections of the work piece, tool holder, and machine.
• If the back rake angle is too large, then strength of the tool is reduced as well as its capacity to conduct heat.
• In machining hard work materials, the back rake angle must be small, even negative for carbide and diamond tools.
• The higher the hardness, the smaller the back rake angle.
• For high-speed steels, back rake angle is normally chosen in the positive range.
. ***Side rake*** must be given to the tool point, to provide the correct cutting ***angle*** for the material being machined

. Negative ***rake is*** when the top surface of the tool is inclined at an angle above the center-line of the work. It employed with success when machining the harder bronze alloys.
Table 4.2 Typical value for top rake angle

Metal Being Cut	Cast Iron	Hard Steel /Brass/	Medium Carbon Steel	Mild Steel	**Aluminum**
Top Rake Angle	0°	8°	14°	20°	**40°**

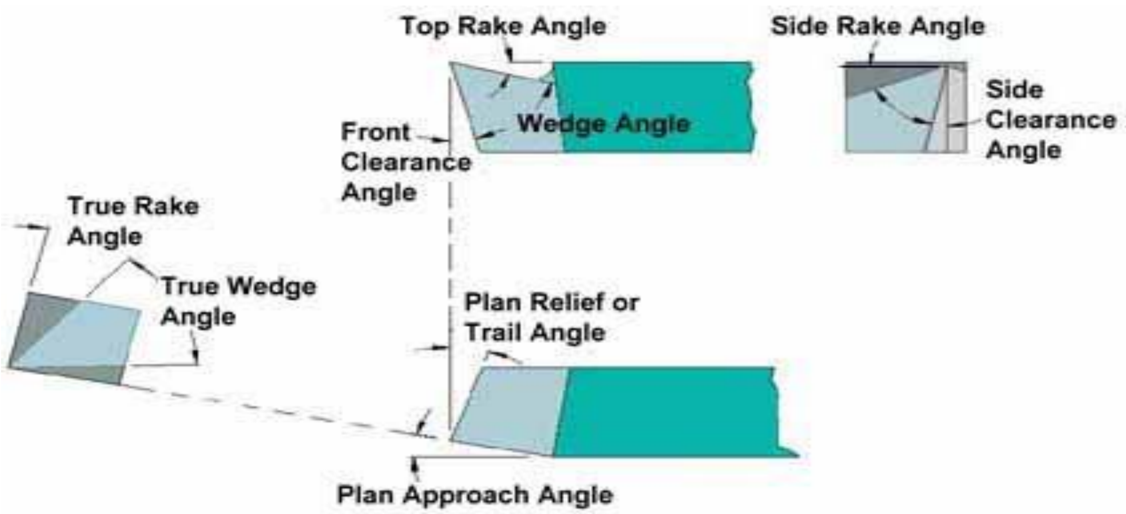


Fig4.21 Main Features of a Single Point Cutting Tool

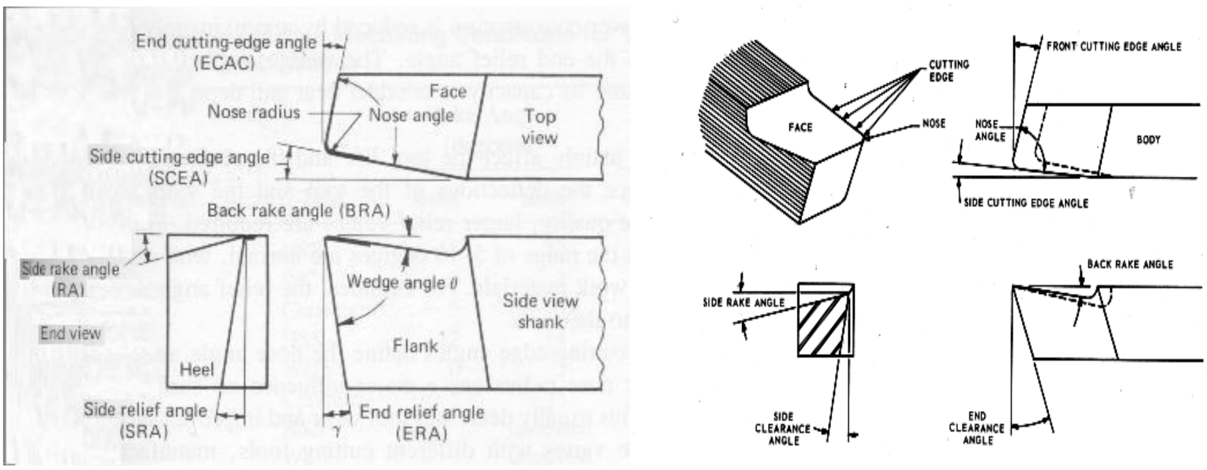


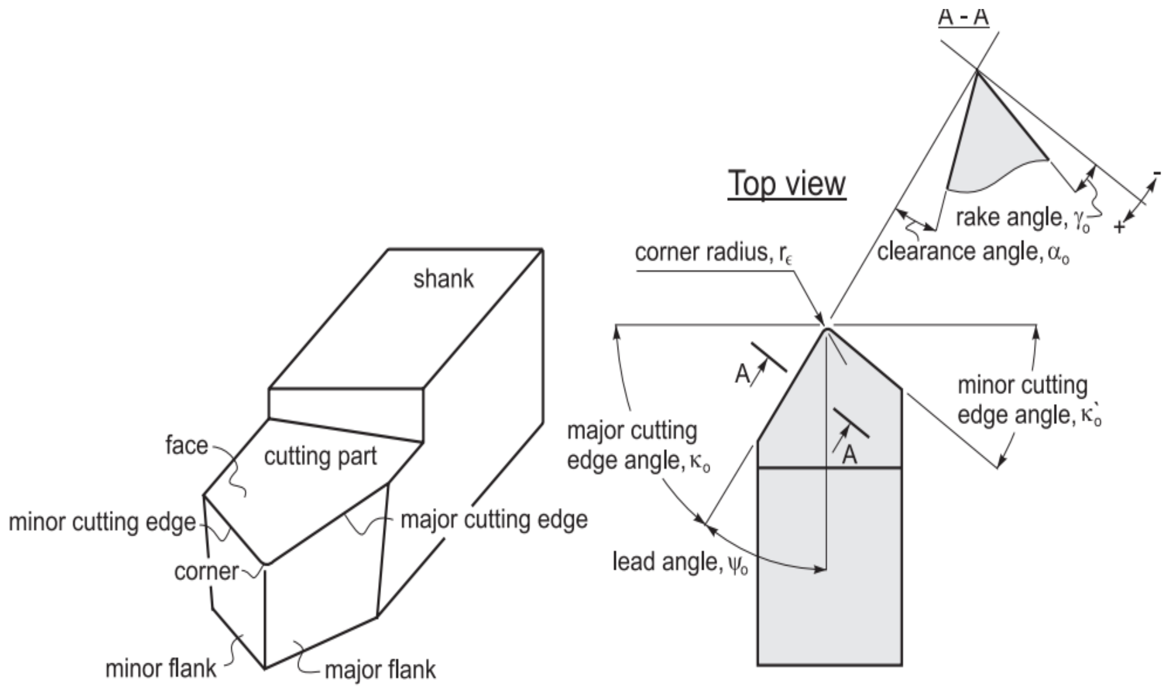
Fig 4.22 single point cutting tool terminology and tool geometry

b. Clearance Angle

• Clearance angle is the angle between the flank or front face of the tool and a tangent to the work surface originating at the cutting edge.
• All cutting tools must have clearance to allow cutting to take place.
• Clearance should be kept to a minimum, as excessive clearance angle will not improve cutting efficiency and will merely weaken then tool.
• Typical value for front clearance angle is 6° in external turning
. ***Side clearance*** must be given to allow the tool to cut or the tool will rub on the work.

. ***Front clearance*** must also be given to the tool in order to provide a satisfactory cutting angle.
Angle of relief

This is needed to prevent "chatter" or vibration of the tool, the frontal area of the tool in contact with the work being reduced to the minimum consistent with obtaining a good finish to the work surface.



Cutting edges, surfaces and angles on the cutting part of a turning tool

2.6 Cutting speed, feed and depth of cut

c. Cutting Speed/Surface Velocity

• Cutting speed is defined as the speed at which the work moves with respect to the tool

- It is the peripheral speed of the work past the cutting tool, or the speed at which the metal is removed by the tool from the work.

- It is expressed in meters / min or measured in feet per minute

Cutting speed, V = πdn /1000 m / min. =πdn/ft/min

Where; n= r.p.m., and d = diameter of w/p in mm or in feet

d. Feed:

Feed is the distance the tool advances for each revolution of the work piece

- Expressed in mm / revolution or inch/revolution

• Feed rate is defined as the distance the tool travels during one revolution of the part.

e. Depth of Cut

- The perpendicular distance measured from the machined surface to the uncut surface of work Or the distance the tool is plunged into the surface

- expressed in mm or in inch

If d1 = diameter of work before machining, and

d2 = diameter of work after machining,

Then, Depth of cut = d1 - d2 / 2

10. METAL REMOVAL RATE

• For turning, MRR values range from 0.1 to 600 in^3 per minute.

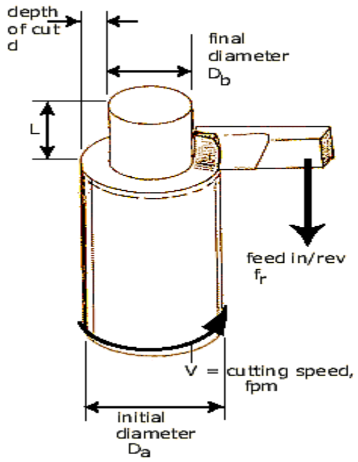
• Most processes have MRR's that can be expressed as the volume of metal removed divided by the time needed to remove it:

MRR = (volume of cut)/(cutting time)

• MRR can be used to estimate the power required to sustain the cutting operation.

For most Aluminum alloys,

- On a roughing cut (.010 to .020 inches depth of cut) run at 600 fpm.
- On a finishing cut (.002 to .010 depth of cut) run at 1000 fpm



$$DOC = (\text{Depth Of Cut}) = d = \frac{D_a - D_b}{2}$$

$$V = \frac{\pi D_a N_s}{12}$$

N_s = rpm of workpiece

D_a = initial diameter

D_b = final diameter

MRR = Metal Removal Rate $\cong 12 V f_r d$ (in^3/min for turning) f_r = feed (e.g. in/rev, in/cycle, in/min, in/tooth)

Fig4.26 metal removal rate

Cutting time

• With turning, the cutting time can be expressed as the following:
• The allowance is an estimation factor which is added to the L term to allow for the tool to enter and exit the cut.

$$\text{CUTTING TIME} = \frac{L + \text{Allowance}}{f_r N_s}$$

.

2.7 CUTTING FLUIDS

- The purposes of using cutting fluids on the lathe are to cool the tool bit and work piece that are being machined, increase the life of the cutting tool, make a smoother surface finish, deter (discard) rust, and wash away chips.
- Cutting fluids can be sprayed, dripped, wiped, or flooded onto the point where the cutting action is taking place.
- Generally, cutting fluids should only be used if the speed or cutting action requires the use of cutting fluids.

Descriptions of some common cutting fluids used on the lathe follow:

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 only slightly with temperature.
- It is 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 commonly mixed with lard oil or other chemicals to provide cutting oils with desirable characteristics.
- Two mineral oils are
  - 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 Oil 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 pure lard oil.

Sulfur zed Fatty-Mineral Oil

- Most good cutting oils contain mineral oil and lard oil with various amounts of sulfur and chlorine which give the oils well anti weld 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 material.

Soluble Cutting Oils

- Water is an excellent cooling medium but has little lubricating value and hast rust and corrosion.
- Therefore, mineral oils or lard oils which can be mixed with water are often used to form cutting oil.
- A soluble oil and water mix has lubricating qualities dependent 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 (TSP) are sometimes added to the solution to improve its corrosion resistance.

Soda-Water Mixtures

- Salts such as soda ash and TSP 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 a secondary consideration.
- It is especially suitable in reaming and threading operations on cast iron where a better finish is desired.

2.8 Milling Machine

- Milling machines are machine tools used to rotate single or multiple cutting edges and removes metal when work is feed against a rotating cutter
- Functions of Milling machine :-capable of machining flat or contoured surfaces, slots, grooves, recesses threads, gears, spirals and other configurations.

Milling machine is one of the most versatile conventional machine tools with a wide range of metal cutting capability. The number of operations which may be carried out on a given machine depends on the type of machine, type of cutter, and the accessories or attachments available for use with the machine. A large number of accessories are available for expanding the variety of operations which can be performed on these machines.

In a milling machine operation the work piece is fed against a rotating cutting tool called a milling cutter. Cutters of many shapes and sizes are available for a wide variety of milling operations. Milling machines cut flat surfaces, grooves, shoulders, inclined surfaces, dovetails, and T-slots. Various form-tooth cutters are used for cutting concave forms and convex grooves, for rounding corners, and for cutting gear teeth.

Milling machines are available in a variety of designs that can be classified as the following: (1) standard knee-and-column machines, including the horizontal and the vertical types; (2) bed-type or manufacturing machines; and (3) machines designed for special milling jobs.

2.8.1 Types of Milling Machine

Milling machine may be grouped into two large families:-

A. **Fixed bed type milling machine**:-are characterized by very rigid work table construction and support. The work moves only longitudinal direction.
   It is sub grouped in two;-

- ❖ Horizontal ,
- ❖ Vertical and
- ❖ Planer type

B. **Column and knee type milling machine:-**the parts that provide movements to the work consists of a column that support and guides the knee in vertical movements But both group are made with horizontal and vertical spindle

There are three basic types of Knee and column type MM

a) **Plane (horizontal spindle) MM:-**

- ❖ Contains the drive motor and gearing and a fixed position HMM spindle.
- ❖ The work table on this machine has three movements vertical ,cross and longitudinal or (z ,y & x axis)

The plain horizontal milling machine’s column contains the drive motor and gearing and a fixed position horizontal milling machine spindle. An adjustable overhead arm containing one or more arbor supports projects forward from the top of the column. The arm and arbor supports are used to stabilize long arbors. Supports can be moved along the overhead arm to support the arbor where support is desired depending on the position of the milling cutter or cutters.

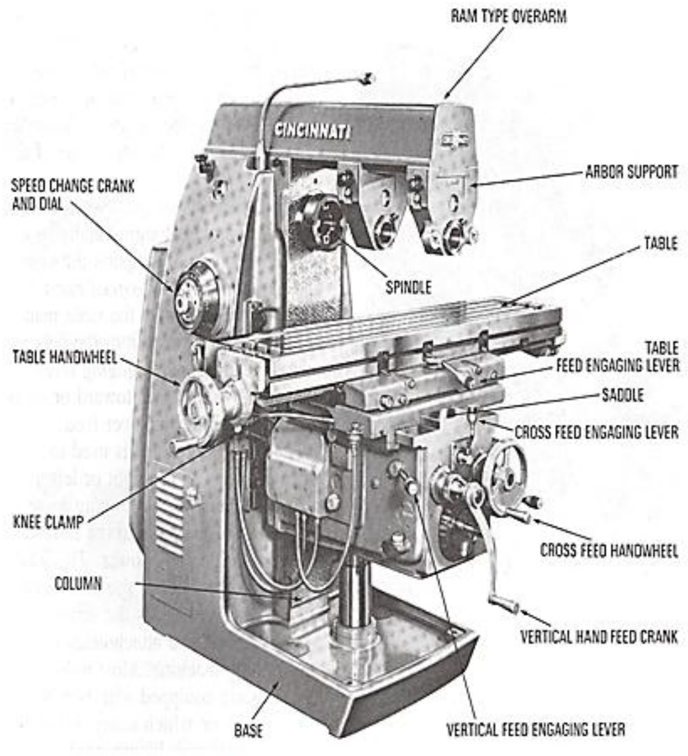


Fig 2.1. Horizontal milling machine

- **Column:** the column houses the spindle, the bearings, the gear box, the clutches, the shafts, the pumps, and the shifting mechanisms for transmitting power from the electric motor to the spindle at a selected speed.
- **Knee is** mounted in front of the column is for supporting the table and to provide an up or down motion along the Z axis.
- **Saddle** is consists of two sideways, one on the top and one at the bottom located at 90º to each other, for providing motions in the X or Y axes by means of lead screws.
- **Table** is mounted on top of the saddle and can be moved along the X axis. On top of the table are some T-slots for the mounting of work piece or clamping fixtures.
- **Arbor** is an extension of the spindle for mounting cutters. Usually, the thread end of an arbor is of left hand helix.
- **Swivel table housing** - is fastened to saddle on a universal milling machine, enables the table to be swiveled 45^0 to either side of the center line.
- **Feed dial**- is used to regulate the table feeds.
- **The spindle** - provides the drive for arbors, cutters and attachments used on a milling machine.
- **Over arm** - provides correct alignment and support of the arbor and various attachments.
- **Arbor support** - is fitted to the over arm and can be clamped at any location on the over arm.

- **Spindle speed dial** - is set by a crank that is turned to regulate the spindle speed

b) **Universal (horizontal spindle) MM:-** It is similar to the Plane (horizontal spindle) MM but have table to swing up to 45° in either direction for angular and helical milling operations

Universal milling machine is a milling machine having a table fitted with all motions and a dividing head with change gears so that it can perform any type of milling operation.

The basic difference between a universal horizontal milling machine and a plain horizontal milling machine is the addition of table swivel housing between the table and the saddle of the universal machine. This permits the table to swing up to 45° in either direction for angular and helical milling operations. The universal machine can be fitted with various attachments such as the indexing fixture, rotary table, slotting and rack cutting attachments, and various special fixtures.

c) Vertical spindle MM: - differ from other type of MM by having the cutter spindle in a vertical position or at right angle to the work **.**It also include

i. **A swivel head MM**:-Spindle can be swiveled for angular cut
ii. **Sliding head MM:-**The head can be moved in vertical position
iii. **The rotary head MM**:- Spindle can move vertically in a circular line
A vertical milling machine which is of similar construction to a horizontal milling machine except that the spindle is mounted in the vertical position.

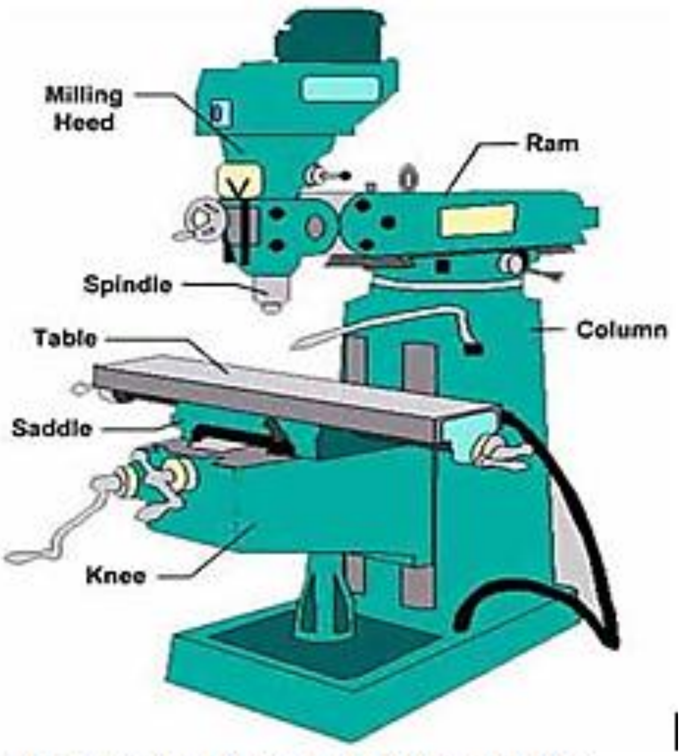


Fig 2.2. Vertical Milling machine

.Methods of controlling MM

- ✓ Manual:- All movement are made hand lever control
- ✓ Semi-automatic :- movements are controlled by hand and /or by power feed
- ✓ Fully Automatic :- A complex hydraulic feed arrangements follow two or three dimensional templates to guide the cutter automatic

2.8.2. MAJOR PARTS OF MILLING MACHINE

A. Column: a column is a hollow casting with thick walls and supports other parts of the machine such as spindle, sliding k nee and over arm.
B. Base is made of ribbed cast iron and it may contain a coolant reservoir.
C. Knee: moves up and down the face of the column and supports the saddle and table.
D. Over arm: the over arm may round or of the more common ram type. It may be adjusted to ward or away from the column to increase the capacity of the machine.

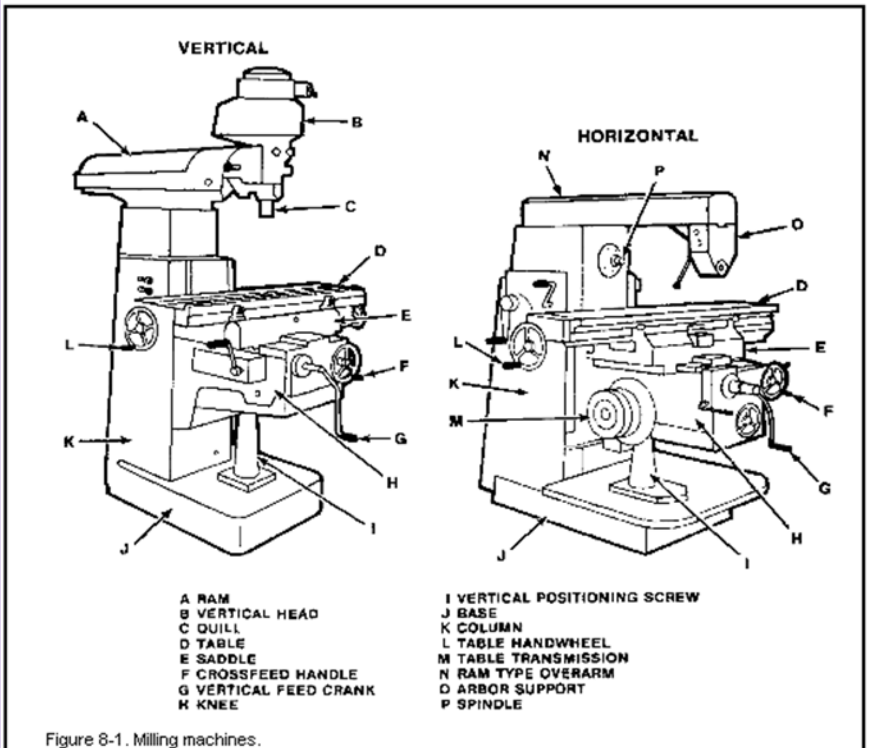
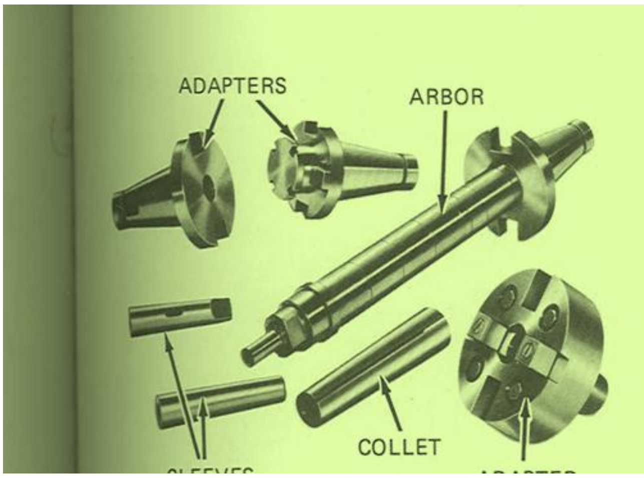


Figure 8-1. Milling machines.

Milling machine accessories

- **Cutter holding device**
  - ✓ **Arbors** are used to hold and drive cutters
  - ✓ **A collet holder** is a form of sleeve bushing for reducing the size of the tapered hole in the spindle.
  - ✓ **Adapter** is used for holding face milling cutters which center on the outer face of the spindle.



- **Work holding device**

  There are several devices used in machine shop for holding work

  - ✓ **Vise** is used for holding square, round, and rectangular pieces for the cutting of keyways, grooves, flat surfaces, angles, gear racks, and T-slots.
  - ✓ **Angle plates** are used for holding large work or special shapes when machining one surface square with another.
  - ✓ **Fixture** is a special holding device made to hold a particular work piece for one or more milling operations.
  - ✓ **Clamps** are held in position by a T- bolt that fits in to the T- slots of the milling machine.

- **Milling Machine Attachments**

I. **Vertical attachment** which may be mounted on the face of column or the over arm

II. **Dividing head**: is used to permit the cutting of bolts heads, gear teeth

III. **Rack milling attachment**

IV. **Slotting attachment**: converts the rotary motion of the spindle in to reciprocating motion

2.9 The indexing fixture or dividing head

The indexing or dividing head is one of the most important attachments for the milling machine. It is used to divide the circumference of a work piece into equally spaced divisions when milling gears, splines, squares, hexagons, etc. It is also used to rotate the work piece at a predetermined ration to the table feed rate to produce cams and helical grooves on gears, drills, reamers etc.

This very useful accessory permits the cutting of bolt heads, gear teeth, and ratchets, etc. When connected to the lead screw of the milling machine.

Indexing equipment is used to hold the work piece, and to provide a means of turning it so that a number of accurately located speed cuts can be made, such as those required in cutting tooth spaces on gears, milling grooves in reamers and taps, and forming teeth on milling cutters.

There are many variations of the indexing fixture. The name universal index head is applied to an index head designed to permit power drive of the spindle so that helixes may be cut on the milling machine. "Gear cutting attachment" is another name for an indexing fixture; in this case, one primarily intended for cutting gears on the milling machine.

a. Index head parts

The universal dividing head set consists of:

- Head stock with index plate

- Head stock change gears and quadrant
- Universal chuck
- Foot stock and
- Center rest

The index head and the footstock are attached to the worktable of the milling machine by T-slot bolts. An index plate containing graduations is used to control the rotation of the index head spindle. The plate is fixed to the index head, and an index crank, connected to the index head spindle by a worm gear and shaft, is moved about the index plate.
The center rest can be used to support long slender work.
The foot stock is used in conjunction with the head stock to support work held between centers or the end work held in a chuck. The center of the footstock can be raised or lowered for setting up tapered work pieces that require machining.
The work piece is held in a chuck, attached to a indexing head spindle, or mounted in between a live center in the indexing head and dead center in the footstock or may be fitted directly into the taper spindle recess of some indexing fixtures.

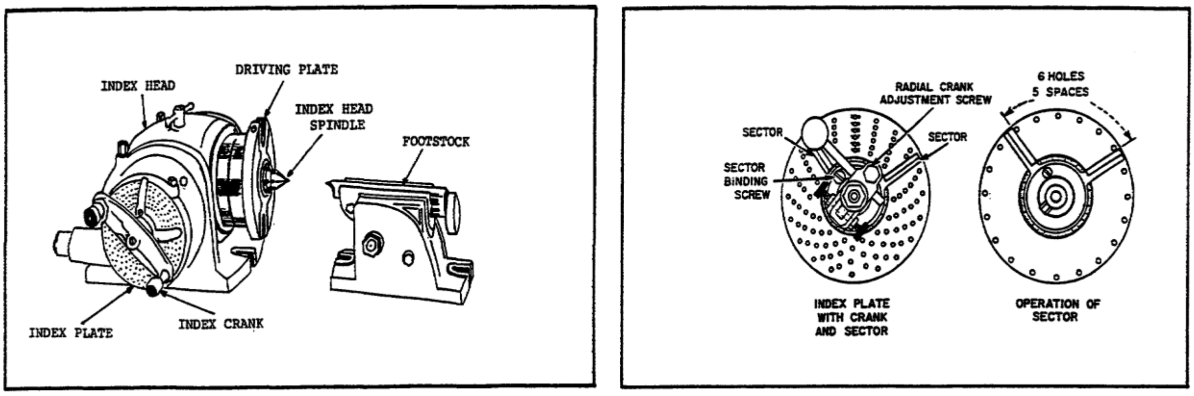


Fig 6.34 indexing fixture

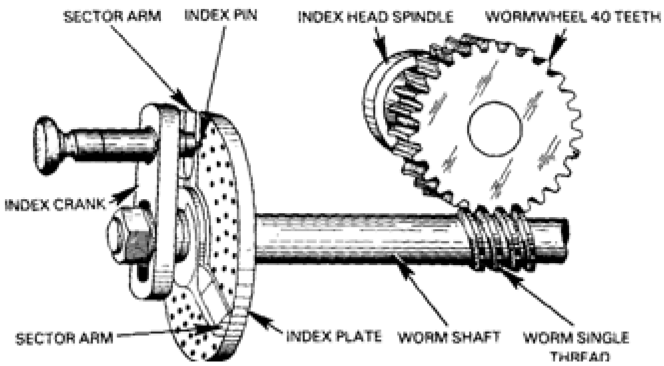


Fig. 6.35 section through dividing head showing the worm wheel & warm shaft

Head stock

The head of the indexing fixture contains an indexing mechanism, used to control the rotation of the index head spindle in order to space or divide a work piece accurately. It consists of a 40 tooth worm wheel fastened to the index head spindle engaging with a single threaded worm attached to the index crank. Since there are 40 teeth in a worm wheel one complete turn of the index crank will cause a spindle to rotate 1/40 of the turn. There for, 40 turns of the index crank revolve the spindle complete turn the making a ratio of 40:1 or (5:1 in some causes).

Index Plate

The index plate is a round metal plate with a series of six or more circles of equally spaced holes; the index pin on the crank can be inserted in any hole in any circle. With the interchangeable plates regularly furnished with most index heads, the spacing necessary for most gears, bolt heads, milling cutters, splines, and so forth, can be obtained.

Sector

The sector indicates the next hole in which the pin is to be inserted and makes it unnecessary to count the holes when moving the index crank after each cut. It consists of two radial, beveled arms which can be set at any angle to each other and then moved together around the center of the index plate. Assume that it is desired to make a series of cuts, moving the index crank 1 1/4 turns after each cut. Since the circle has 20 turns, the crank must be turned one full turn plus 5 spaces after each cut. Set the sector arms to include the desired fractional part of a turn, or 5 spaces, between the beveled edges of its arms. If the first cut is taken with the index pin against the left hand arm, to take the next cut, move the pin once around the circle and into the hole against the right-hand arm of the sector. Before taking the second cut, move the arms so that the left-hand arm is again against the pin; this moves the right-hand arm another five spaces ahead of the pin. Then take the second cut; repeat the operation until all the cuts have been completed.

NOTE It is a good practice always to index clockwise on the plate.

b. Methods of indexing

The main purpose of the indexing or dividing head is to divide the work piece circumference accurately into any number of divisions.

They are the following indexing methods:

- Direct
- Simple
- Angular and
- Differential

1. Direct indexing

- It is the simplest form of indexing. It is performed by disengaging the worm shaft from the worm wheel by means of an eccentric device in the dividing head. Some direct dividing heads do not have a worm and worm wheel but rotates on bearings.
- It is used for quick indexing of the work piece when cutting flutes, hexagons, squares etc.
- The work is rotated the required amount and held in place by a pun which engages is to a hole or slot in the direct indexing plate mounted on the end of the dividing head spindle. It usually contains three sets of hole circles or slots: 24, 30 & 36. The number of divisions to index is limited to the numbers which are factors of 24, 30, or 36.

Example 1 What direct indexing is necessary to mill eight flutes on a reamer blank?

Plate hole circle	Factors
24	2, 3, 4, 6, 8, 12, 24 - -
30	2, 3, 5 , 6,10, 15, 30 --
36	2, 3, 6, 9,12, 18, 36---

As the **24** - Hole circle is the only one divisible by 8 (the required number of divisions), it is the only circle which can be used in this case.

$$Indexing = \frac{24}{8} = 3\, holes\ on\ a\ 24 - hole\, circle.$$

NOTE: never count the hole or slot in which the index pin is engaged.

Example2. Explain how to mill a square by direct indexing method.

1. Simple indexing

In simple indexing, the work is positioned by means of the crank, index plate; and sector arms. Since there are to teeth on the worm wheel, one complete turn of the index crank will causes the spindle and the work to rotate one-fortieth of a turn so 40 turns of the crank will revolve the spindle and the work one turn. That means with a ratio of 40:1 between the turns of the index crank and the dividing head spindle.

For most divisions, it is necessary to divide 40 by the number of divisions (N) to be cut.

$$Indexing = \frac{40}{N}$$; N- number of divisions

Example1 The indexing required to cut eight flutes would be:

$$Indexing = \frac{40}{8} = 5\, full\ turns\ of\ the\ index\ crank$$

Example2 Find the indexing required cutting 7 flutes

$$Indexing = \frac{40}{7} = 5\,{}^{5}\!/_{7}\ turns$$

To get five-sevenths of a turn, choose any hole circle, which is divisible by the denominator 7, such as **21,** then take five-sevenths x 3/3 = 15 holes on a 21 hole circle.

There for $\frac{40}{7} = 5\,{}^{5}\!/_{7}$ turns or **5 complete turns** plus **15 holes** on a **21** - hole circle.

*****When extreme accuracy is required for indexing, choose the circle with the most holds.

Table: Index-plate hole circles (Brown and sharp)

Plate 1 =	15, 16, 17, 18, 19, 20
Plate 2 =	21, 23, 27, 29, 31, 33
Plate 3 =	37, 39, 41, 47, 49

Cincinnati standard plate

One side	-	24, 25, 28, 30, 34, 37, 38, 39, 41, 42, 43
Other side	-	46, 47, 49, 51, 53, 54, 57, 58, 59, 62, 66

***2.* Angular indexing**

When the angular distance between divisions is given instead of the number of divisions, the set up for simple indexing may be used, however, the method of calculating the indexing is changed.

One complete turn of the index crank turns the work one-fortieth f a turn or $\frac{360^0}{40} = 9^0$

Indexing in degrees = $\frac{No.of\ \deg rees\ required}{9}$

Example 1 Calculate the indexing for 45^0 (use **18** hole circle)

Indexing = $\frac{45^0}{9} = 5\ complete\ turns$

Example 2 Calculate the indexing for 60^0 (use 18 hole circle)

Indexing = $\frac{60}{9} = 6\frac{2}{3}$ = 6 complete turns, 12 holes on a 18 holes circle.

If the dimensions are given in degrees and minutes, convert the degrees in to minutes and add this sum to the minutes required.

Indexing in minutes = $\frac{no.of\ \min utes\ required}{540}$

Example 1. Calculate the indexing necessary for 24',

Indexing = $\frac{24}{540} = \frac{4}{90} = \frac{1}{22.5}$ would be 1 hole one the 22.5 hole circle. As the 23 - hole circle is the nearest hole circle, it will be 1 hole on the 23 - hole circle (there is slight error)

Example 2 Calculate the indexing for 24^0 30' (use **18** hole - circle)

1) Convert 24^0 into minutes; 24 x 60^0 = 1440'
2) Add 30' = 30' ; Total = 1470'
3) Indexing = $\frac{1470}{540} = 2\frac{390}{540} = 2$ full turns and $\frac{390}{540}$ $on\ 18 - hole\ circle, which\ is\ \frac{390}{540}\ x18 = 13$; that means **2** full turns and **13** holes **on 18-hole circle.**

4. Differential indexing

When it is impossible to calculate the required indexing by the simple indexing method that is, when the fraction $\frac{40}{N}$ cannot be reduced to a factor of one of the available hole circles, it is necessary to use differential indexing. With this method of indexing, the index plate must be revolved either for ward or backward a part of a turn while the index crank is turned to attain the proper spacing or indexing.

The method of calculating the change gears required to rotate the plate the proper amount is as follows:

Change gear ratio = $(A - N)\,x\frac{40}{A} = \frac{driver\,(spindle)\,gear}{driven(worm)\,gear}$

Where; A - approximate number of divisions
N - Required number of divisions

When the approximate number of divisions is larger than the required number, the resulting fraction is plus and the index plate must move in the same direction as the crank (clock wise) or positive, which is accomplished by using the idler gear. If the approximate number is smaller than the required number, the resulting fraction is minus and the index plate must move in a counter clock-wise direction, which needs two idler gears. The gearing may be simple or compound.

- Simple gearing: one idler for a positive rotation of the index plate. Two idlers for a negative rotation of the index plate

- Compound gearing: **one idler** for a ***negative rotation*** of the index plate, **two idlers** for a ***positive rotation*** of the index plate.

Example: Calculate the indexing and change gears required for **57** divisions.

Change gears supplied with dividing head

24, 24, 28, 32, 40, 44, 48, 56, 64, 72, 86, 100

Procedure:

1. Indexing = $\frac{40}{N} = \frac{40}{57}$ $There\,is\,not\ 57\ hole\ circle$
2. Let approximate number of divisions equal 56
3. Indexing for 56 divisions = $\frac{40}{56} = \frac{5}{7}\ or\,15\ holes\,on\ the\ 21-hole circle$
4. $Gear\,ratio\ = (A - N) = \frac{40}{A} = (56-57)\,x\,\frac{40}{56} \quad = -1\ x\ \frac{40}{56} = \ -\frac{5}{7}$

Change gears = $-\frac{5}{7}\ x\ \frac{8}{8} = -\frac{40}{56}\left(\frac{spindle\ \ gear}{worm\ gear}\right)$

There for, indexing 57 divisions, a 40 - tooth gear is mounted on the dividing head spindle, and a 56-tooth gear is mounted on the worm shaft. Since the fraction is negative quantity simple gearing is to be used, the index rotation is **negative (counter clockwise)** and **two idlers** must be used. After the proper gears are installed, the simple indexing procedure for 56 divisions should be followed.

.2.9.1 **MILLI NG CUTTERS**

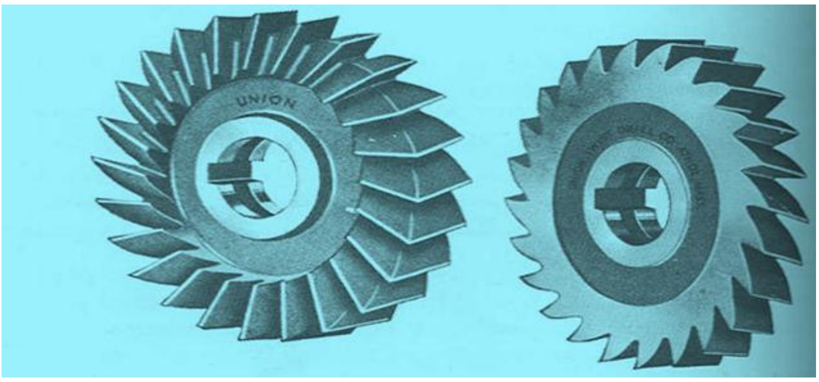
Types of milling cutters

1. **plain milling cutters**: are cylindrical with teeth located around the circumference.

   It is used to produce a flat surfaces.

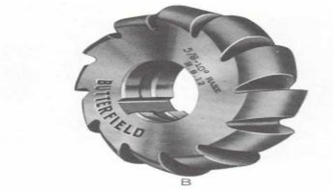
2. Side milling cutters:

- have cutting edges on their circumference and on one or both sides.
- It is used for cutting slots and for face and straddle milling operations



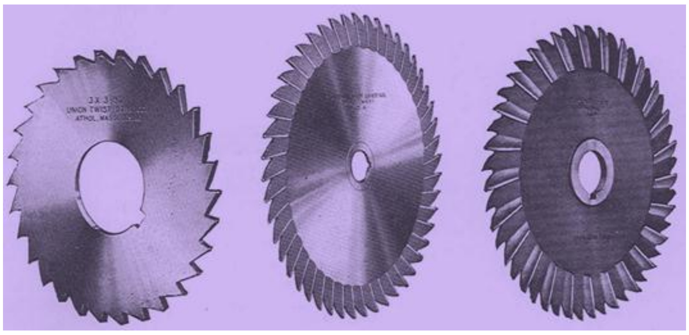
3. Formed cutter

- Desinged to accurately duplicate
- It is used for the production of small part



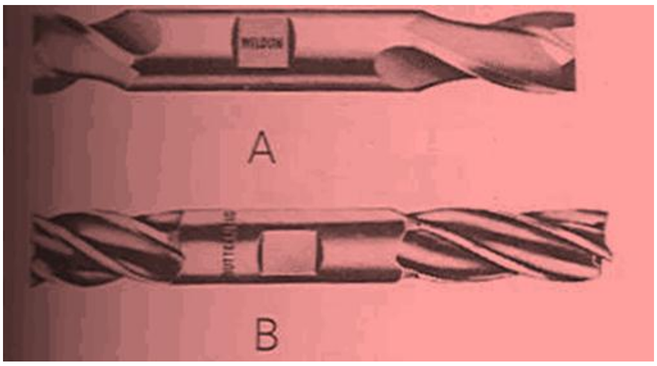
4.Metal saws:

- are thin milling cutters that resemble circular saw blades.
- It is used for narrow sloting and cut-off operations



5.End mills:

- have cutting teeth on the end as well as on the periphery and fitted to the spindle by a suitable adaptor.
- It is used for drilling a hole, slot cutting and facing



6. T-slot cutter:

- It consists of a small side milling cutter with teeth on both sides and an integral shank for mounting.
- It is used to cut the wide horiziantal groove at the bottom of T-slot after the narrow vertical has been machined with an end mill

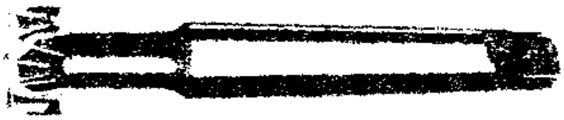


Fig T-slot cutters

1.4. Drilling Machines:

Drilling is a process of producing round holes in a solid material or enlarging existing holes with the use of multi tooth cutting tools called *drills* or *drill bits*. Various cutting tools are available for drilling, but the most common is the *twist drill***.**

Drilling machines, also called drill presses, cut holes in metal with a twist drill. They also use a variety of other cutting tools to perform the following basic hole-machining operations: (1) reaming, (2) boring, (3) counter boring, (4) countersinking, and (5) tapping internal threads with the use of a tapping attachment.

Drilling is an operation of producing **circular hole** by using drill bits. The drilling operation accomplished by Drilling machine, lathe in which the drill is held in tailstock and the work is held by the chuck. The most common drill used is the twist drill.

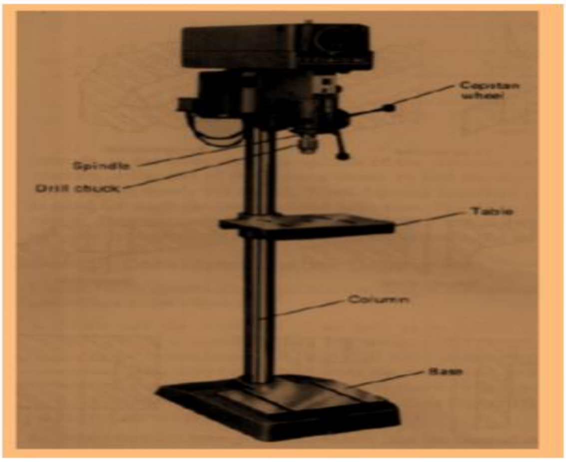


Fig 2.3. Drilling machine

2.10 PORTABLE DRILLS

The portable drill is a hand-supported, power-driven machine tool that rotates twist drills, reamers, counter bores, and similar cutting tools. The portable drill may be electrically powered by means of an internal electric motor.

Portable drills are rated by the maximum size hole that can be drilled in steel without overtaxing the motor or drill.

Therefore, a 1/4-inch-capacity drill is capable of drilling a 1/4-inch diameter hole or smaller in steel. Portable electric and pneumatic drills rated at 1/4 to 1/2-inch maximum capacities are usually equipped with geared drill chucks for mounting straight, round shank twist drills or other similar tools by using a chuck key.

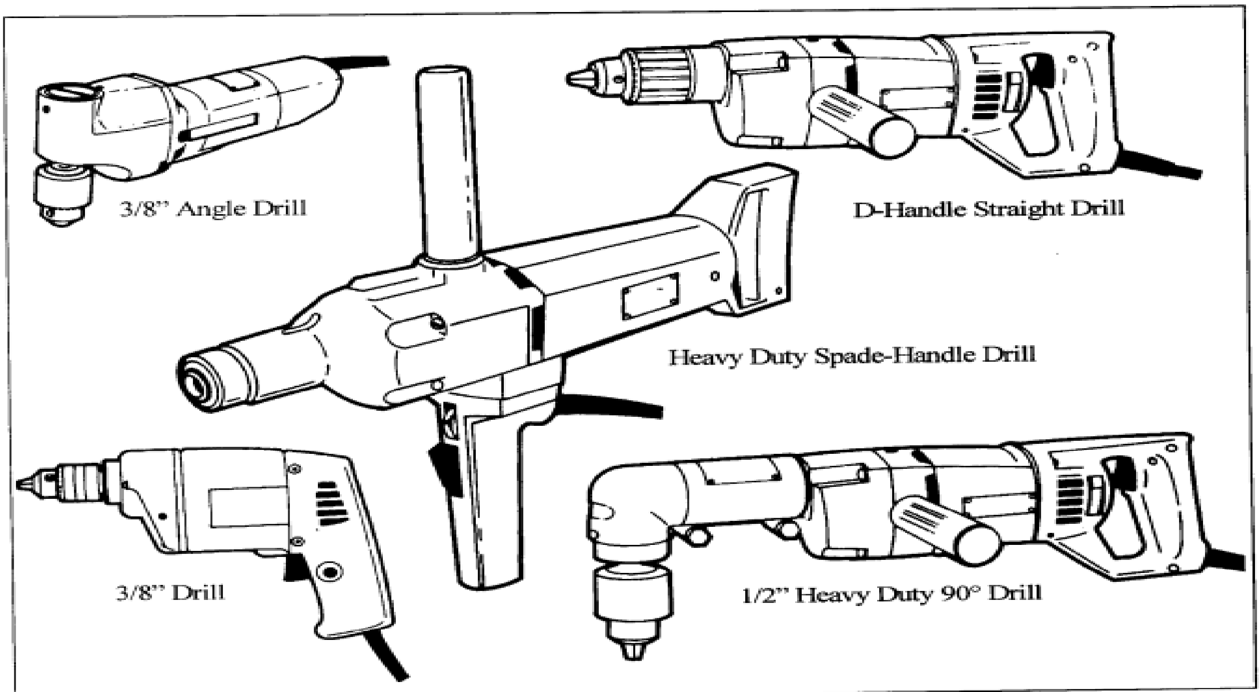


Figure 3-1. Portable electric hand drills.

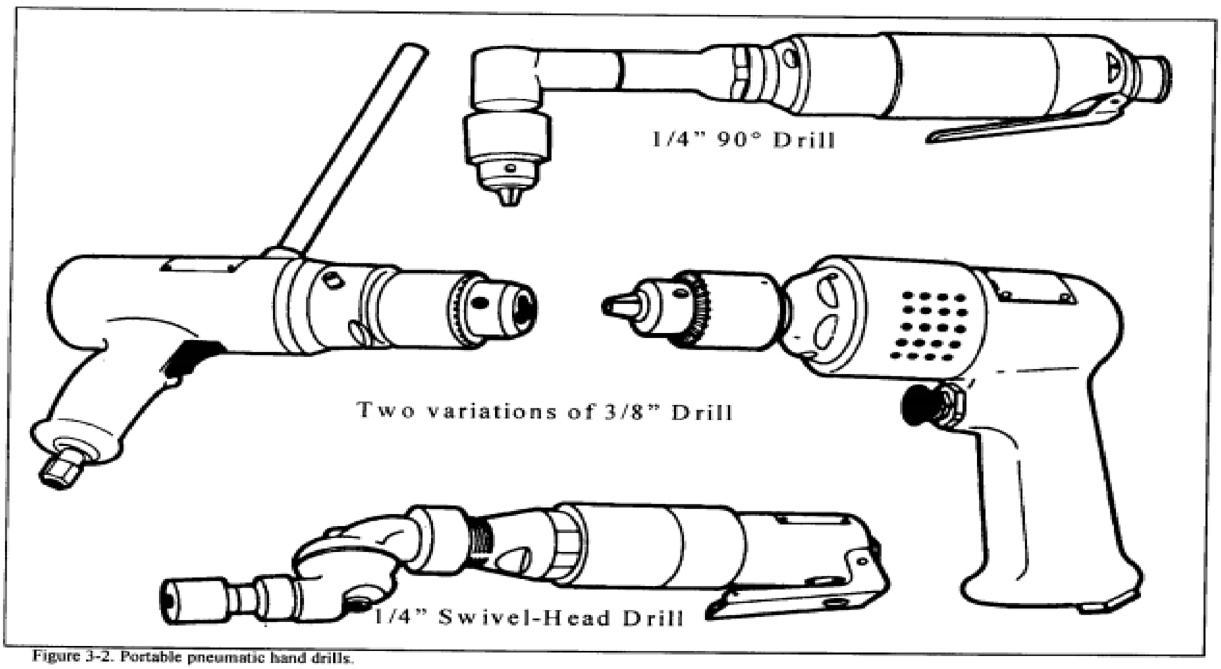


Figure 3-2. Portable pneumatic hand drills.

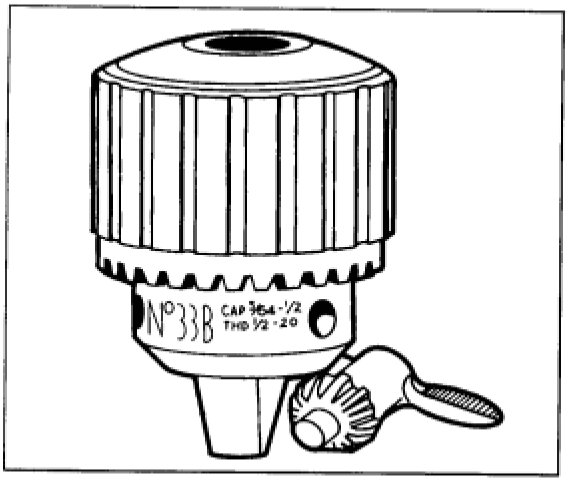


Figure 3-3. Geared drill chuck and chuck key.

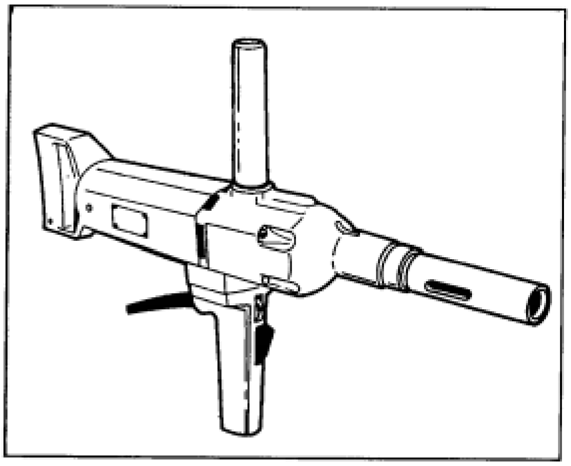


Figure 3-4. 1-inch capacity portable electric drill.

1.4.1 Parts of drilling machine

1. **Spindle:** the spindle holds the drill or cutting tools and revolves in a fixed position in a sleeve.
2. **Column:** the column is cylindrical in shape and built strong and solid. The column supports the head and the sleeve or quill assembly.
3. **Head:** The head of the drilling machine is composed of the sleeve, a spindle, an electric motor and feed mechanism. The head is bolted to the column.
4. **Worktable:** The worktable is supported on an arm mounted to the column. The worktable can be adjusted vertically to accommodate different heights of work or it can be swung completely out of the way. It may be tilted up to 90 degree in either direction, to allow long pieces to be end or angle drilled.
5. **Base:** The base of the drilling machine supports the entire machine and when bolted to the floor, provides for vibration-free operation and best machining accuracy. The top of the base is similar to the worktable and may be equipped with **T- slot** for mounting work too larger for the table.

2.3.1 Cutting conditions in drilling

The twist drill is a cutting tool with two symmetrical opposite cutting edges, each removing part of the material in the form of chip.

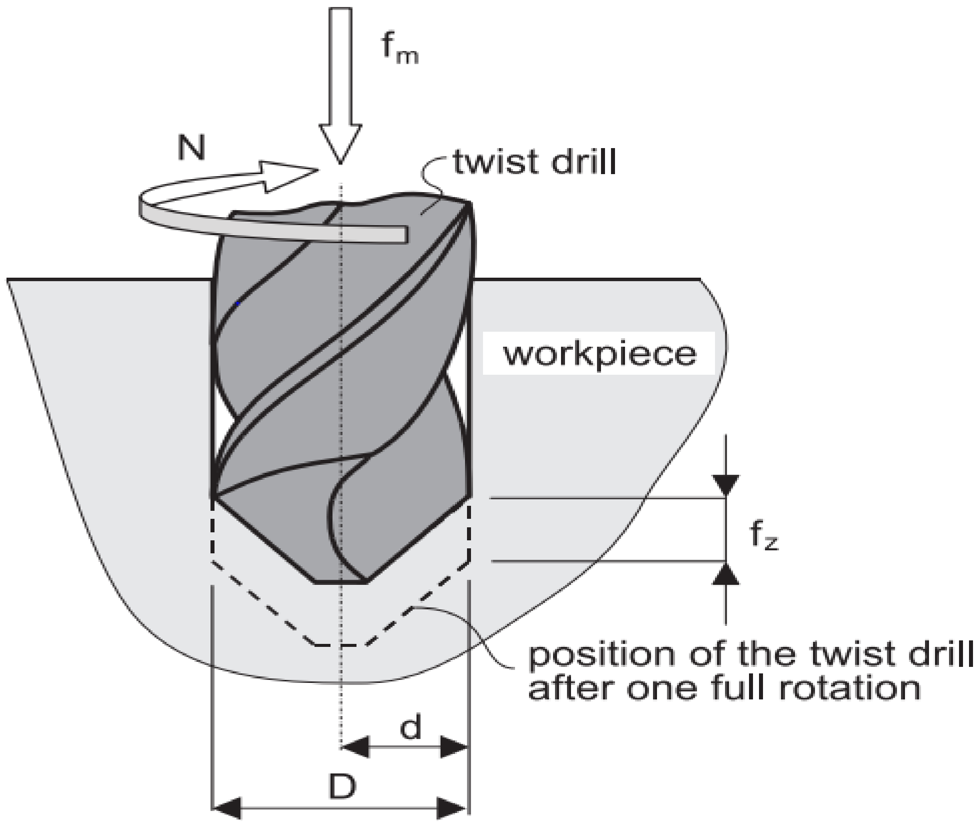


Fig. Basics of a drilling operation.

2.3.2 **Cutting velocity** V in drilling is not a constant along the major cutting edge as opposed to the other Machining operations. It is zero at the center of the twist drill, and has a maximum value at the drill Corner. The maximum cutting speed is given by

$V = \pi DN$

Where D is the drill diameter, and N is the rotational speed of the drill.

2.3.3 Types of drilling machine

- Portable drilling machine
- Bench drilling machine
- Up-Right Drilling Machine
- Radial drilling machine
- Pillar drilling machine
- Gang drilling machine
- Multiple drilling machine

Drill press

Although a hand drill is commonly used for drilling of small holes, a drill press is preferable when the location and orientation of the hole must be controlled accurately. A drill press is composed of a base that supports a column; the column in turn supports a table. Work can be supported on the table with a vise or hold down clamps, or the table can be swiveled out of the way to allow tall work to be supported directly on the base. Height of the table can be adjusted with a table lift crank than locked in place with a table lock. The column also supports a power head containing a motor. The motor turns the spindle at a speed controlled by a variable speed control dial. The spindle holds a

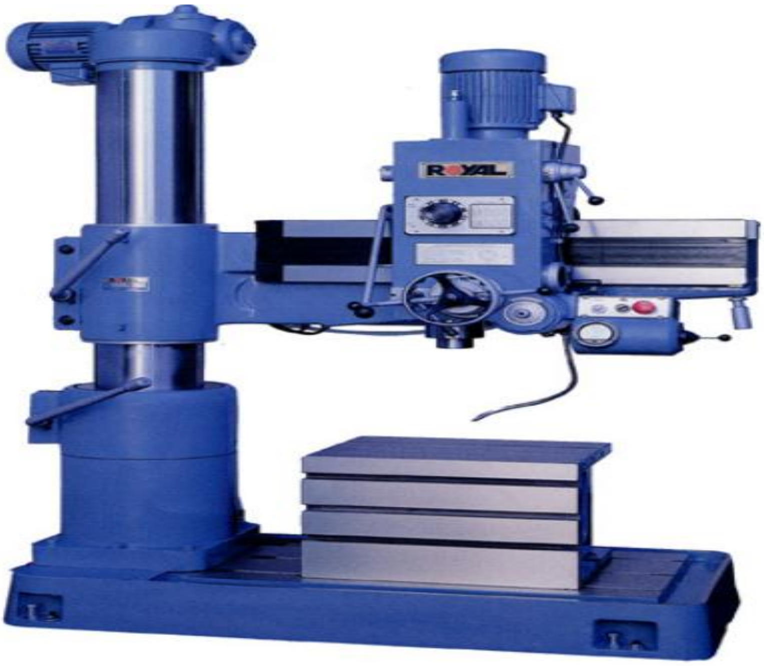
Drill chuck to hold the cutting tools (drill bits, center drills, reamers, etc.).The machine tool described is a typical upright drill press. The smaller modifications, mounted on a table rather than the floor are known as bench drills.

Fig. Upright drill press.

Fig. Bench drill press.

Radial drill

This is the largest drill press designed to drill up to 100-mm diameter holes in large work parts. It has a radial arm along which the drilling head can be moved and clamped.

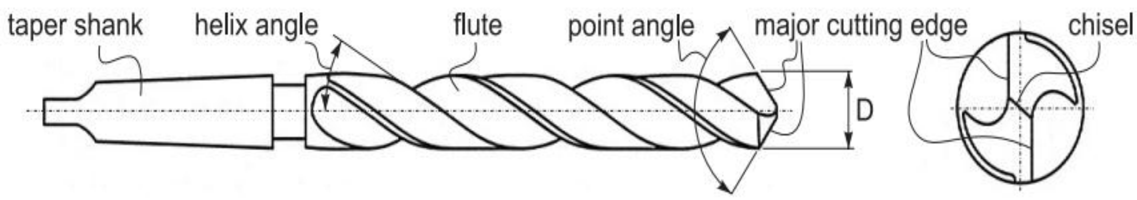


2.3.4 Work holding equipment

Work holding on a drill press is accomplished by clamping the part in a vise, fixture, or jig. **A vise** is a general-purpose work holding device possessing two jaws that grasp the work in position.

A fixture is a work holding device that is usually custom designed for the particular work part. The fixture can be designed to achieve higher accuracy in positioning the part relative to the machining operation, faster production rates, and greater operator convenience in use.
A jig is a work holding device that is also specially designed for the work part. The distinguishing feature between a jig and a fixture is that the jig provides a means of guiding the tool during the drilling operation. A fixture does not provide this tool guidance feature. A jig used for drilling is called a drill jig.

Twist drill Drills
The twist drill does most of the cutting with the tip of the bit. It has two flutes to carry the chips up from the cutting edges to the top of the hole where they are cast off. The standard drill geometry
Is shown in the figure:



Standard geometry of a twist drill.

The typical helix angle of a general purpose twist drill is 18~30o while the point angle (which equals two times the major cutting edge angle, see page 101) for the same drill is 118o. Some standard drill types are,

- **straight shank**: this type has a cylindrical shank and is held in a chuck;
- **Taper shank**: his type is held directly in the drilling machine spindle.

Drills are normally made of HSS but carbide-tipped drills, and drills with mechanically attached carbide inserts are commonly used in many operations, especially on CNC drilling machines:

Coated HSS twist drills. Carbide-tipped twist drills. Indexable inserts twist drills.

Twist Drill Parts: A drill may be divided into 3 main parts: shank, body, point.

1. **Shank**

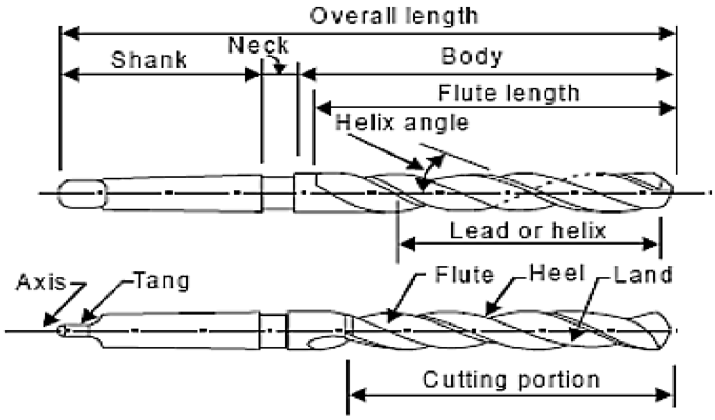
Generally, drills up to ½ “ or 13mm in diameter have straight shanks, while those over this diameter usually have tapered shanks. Straight shank drills are held in a drill chuck; tapered shank drills fit into the internal taper of the drill press spindle. A tang is provided on the end of the tapered-shank drills to prevent the drill from slipping while it is cutting and allow the drill to be removed from the spindle or socket without the shank being damaged.

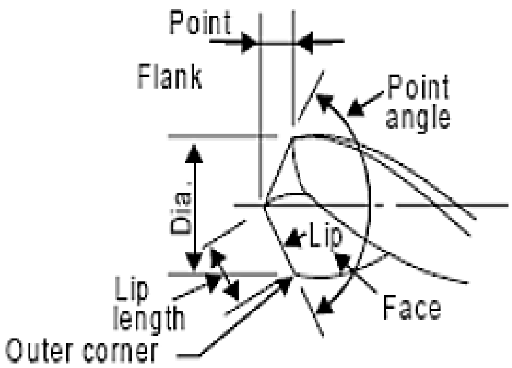
2. **Body**

The body is the portion of the drill between the shank and the point.

3. **Point**

The point of the twist drill consists of the chisel edge, lips, lip clearance, and heel. The chisel edge is the chisel-shaped portion of the drill point. The lips or the cutting edges are formed by the intersection of the flutes. The lips must be of equal length and have the same angle so that the drill will run true and will not cut a hole larger than the size of the drill.





CARE OF DRILLING MACHINES

Lubrication is important because of the heat and friction generated by the moving parts. Follow the manufacturer's manual for proper lubrication methods. Clean each machine after use. Clean T-slots. Grooves. And dirt from belts and pulleys. Remove chips to avoid damage to moving parts. Wipe all spindles and sleeves free of grit to avoid damaging the precision fit. Put a light coat of oil on all unpainted surfaces to prevent rust. Operate all machines with care to avoid overworking the electric motor.

2.4.1 Grinding machines

Safety in grinding

any unsafe practices in grinding can be hazardous for operation and deserve careful attention. Various important aspects are on: —

(i) Mounting of grinding wheels. The wheel should be correctly mounted in the spindle and enclosed

by a guard. Wheel bore should not be a tight fit on the sleeve.
(ii) Wheel speed. The maximum wheel speed is determined by the

Ultimate bursting strength of the wheel and it depends on the abrasive used, grit size, bond, structure, grade, shape and size of the wheel. Its value is specified by the manufacturers which should never be exceeded.
(iii) Wheel inspection. Wheels before mounting should be checked for damage in transit, cracks and other defects. Ringing test is good enough for vitrified bond. Sound wheels, when tapped lightly at 45 from the vertical line with a plastic hammer sound like a clear metallic ring but the cracked wheel will not ring.
Wheels, when not is use, should be stored in a dry place and placed on their edges in racks.
(**iv) Wheel guards**. These should always be used during grinding, and periodically adjusted to compensate for wheel wear.
(v) Dust collection and health precaution. When grinding dry, provision for extracting grinding dust should be made. Protective covers of machine should never be removed while machine is in use. Operator should wear safety devices to protect his eyes and body from flying abrasive particles and dust.
(vi) Wheel operation. Adequate power is essential in grinding machines. If power is not adequate then wheels will slow down and develop flat spots, making the wheel to run out-of-balance.
During wet grinding, the wheel should not be partly immersed, as this would seriously throw the wheel out of balance

Grinding is the process of removing metal by the application of abrasive which are bonding to form a rotating wheel. When the moving abrasive particle contacts the work piece, they act as tiny cutting tools, each particle cutting a tiny chip from the work piece.

The grinding machine is used for roughing and finishing flat, cylindrical and sharpening cutting tools.

2.5.2 Classification of grinding machine operation

The grinding machine may be classified according to the:

1. Types of operations

A. tool grinding b. cut off grinding

2. Quality of surface finish

A. Precision grinding

B. rough grinding

- Bench
- Floor stand
- Portable

3. Type of surface generated

A. cylindrical grinders –center type

- Plain & universal
  - ✓ Center less
  - ✓ Chucking type
  - ✓ Tool post

B. Internal grinders – chuck Vertical type

- center less

C. surface grinders – plain type

- Horizontal spindle & vertical spindle
- reciprocating table
- Rotating table – vertical table
- Horizontal table

D. Tool grindings – universal &special

E. Special grinding –cutting off –sawing

- Portable - off hand grinding
- Honing and lapping - accurate finish

Grinding machines remove small chips from metal parts that are brought into contact with a rotating abrasive wheel called a grinding wheel or an abrasive belt. Grinding is the most accurate of all of the basic machining processes. Modern grinding machines grind hard or soft parts to tolerances of plus or minus 0.0001 inch (0.0025 millimeter).

The common types of grinding machines include the following: (1) plain cylindrical, (2) internal cylindrical, (3) center less, (4) surface, (5) off-hand, (6) special, and (7) abrasive-belt.

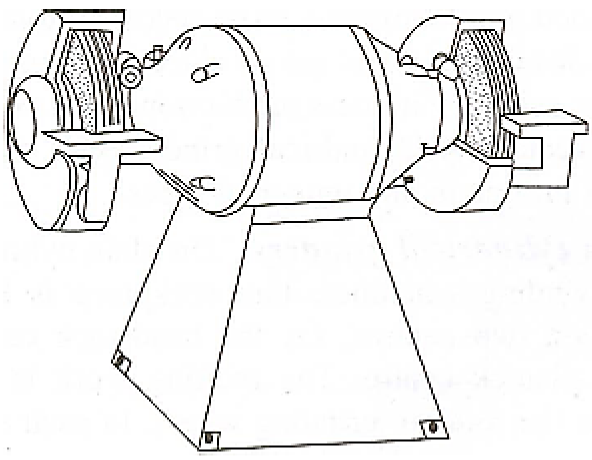


Fig.2.4. Grinding Machine

PORTABLE GRINDERS

The portable grinder is a lightweight, hand-operated machine tool. It can be powered electrically or pneumatically, depending on the model selected. The portable grinder is used in the field or maintenance shop to grind excess metal from welds, remove rust, and for special finishing operations around the work area. Since this tool is hand operated, the quality of the work depends upon the ability and experience of the operator.

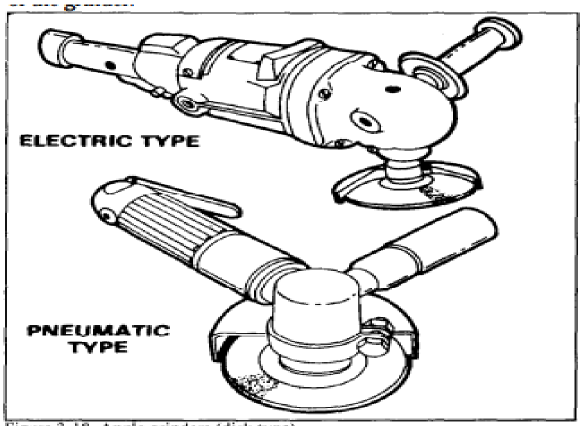


Figure 3-18. Angle grinders (disk type).

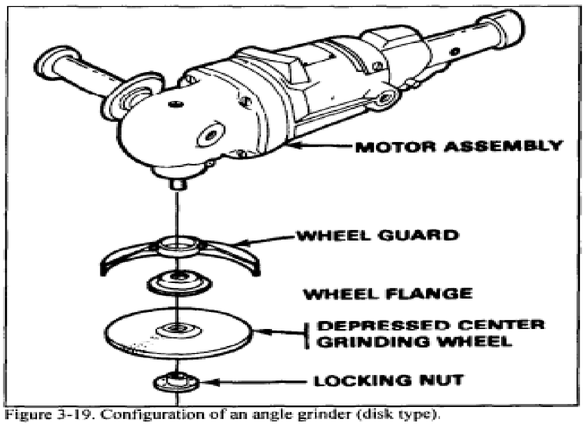
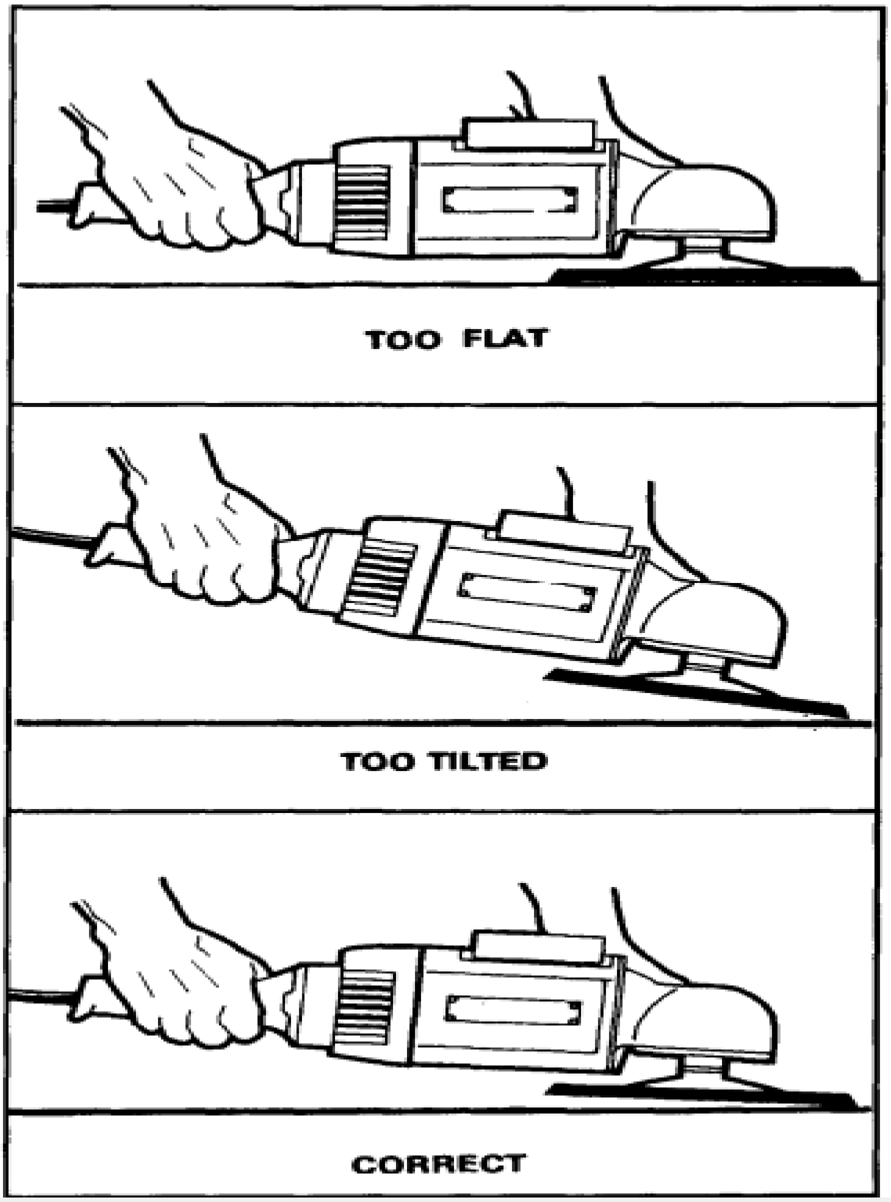


Figure 3-19. Configuration of an angle grinder (disk type).

OPERATIONS WITH PORTABLE GRINDERS



Off – hand grinding is the term used in engineering to describe the process where the work is held by hand material is removed using an abrasive wheel.

This type of grinding is carried out in the workshop for such work as:

a. Removing excess materials

b. Smoothening surfaces

c. Preparing plates for welding

d. Sharpening cutting tools (drills, chisels, punches, shaper and lathe tools)

Off – hand grinding must be performed with great regard of safety. The principle of operation requires an exposed portion of the abrasive wheel to be in close proximity to the operator.

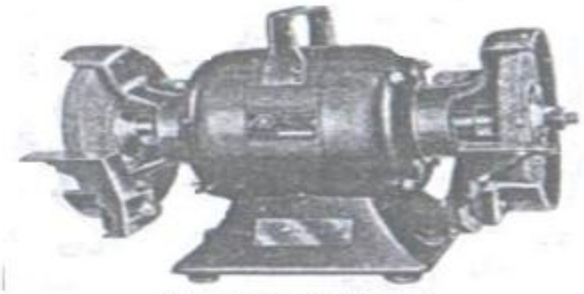
Hazard may be created by having relatively heavy abrasive wheels rotating a high speed. The wheels on all types of machines must be heavily guarded.

The guard exposes enough of the wheel surface to enable the operator to perform the work required

NOTE: Wear safety goggles when performing any grinding operation.

Portable grinder

Pedestal



Bench grinder

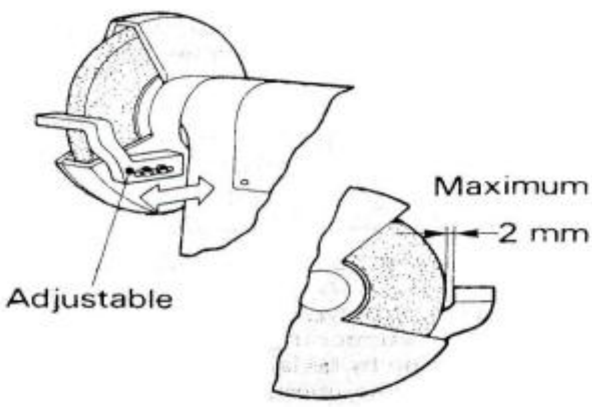
2. Parts and Functions (features of machines)

a. Work Rests

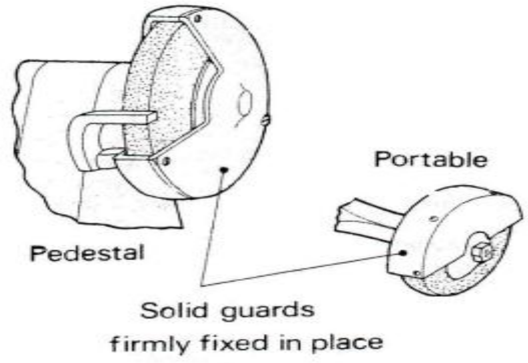
b. Wheel Guards

c. Wheel Speed

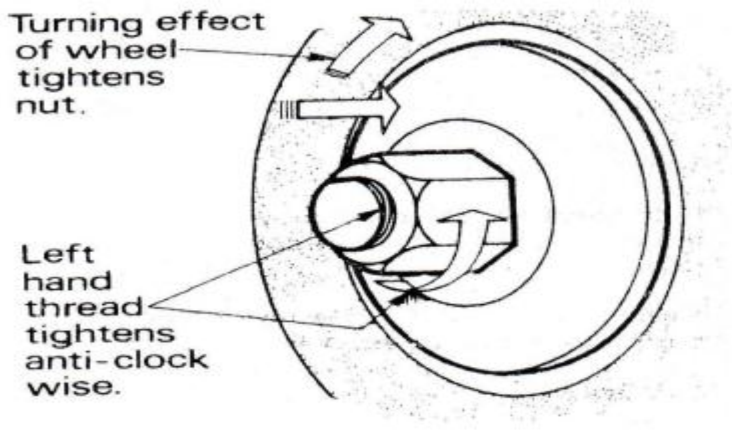
d. Wheel Rotation



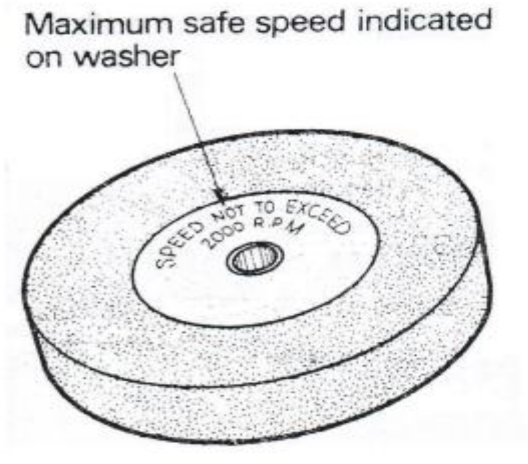
WORK RESTS



WHEEL GUARDS



LEFT HAND SIDE



3. Types of Abrasives Wheels

Two abrasives are used in the manufacturer of abrasives wheels.

a. Aluminum oxide

b. Silicon carbide

Abrasives wheels are made from abrasives grains held together by a suitable materials called a bond.

a. Vitrified bond

b. Silicate bond

c. Organic bonds

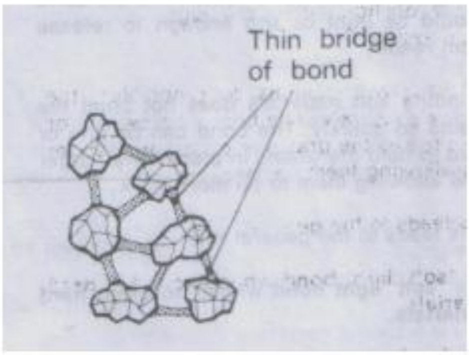
3.1 Amount of Bonds

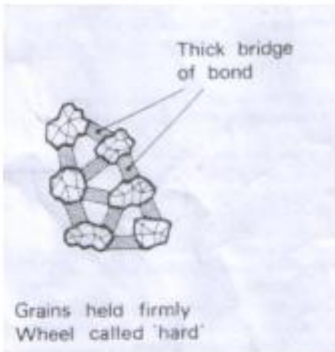
a. The bonds hold the abrasive grains together.

b. It supports them while they cut. The amount of grade bond determines whether the abrasives grains are held lightly or firmly.

c. If a thick bridge holds grains firmly. The wheel is said to be "hard."

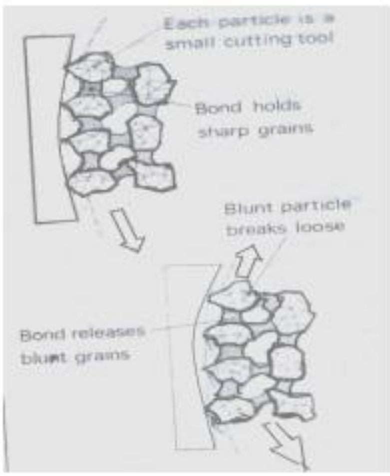
d. If a thin bridge is said to be "soft", the grains can break easily

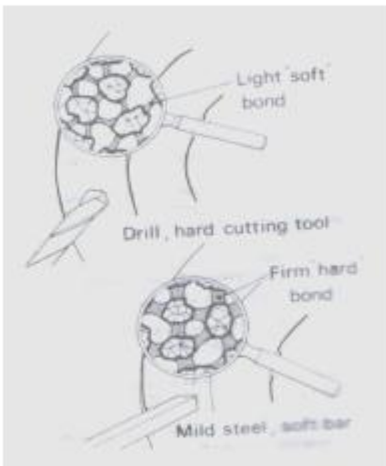




3.2 Cutting action of an Abrasive Wheel

- Use "soft" light bond wheels to grind hard materials.
- Use "hard" firm bond wheels to grind soft materials.





3.3 Identification of Abrasive Wheel

a. Abrasive type

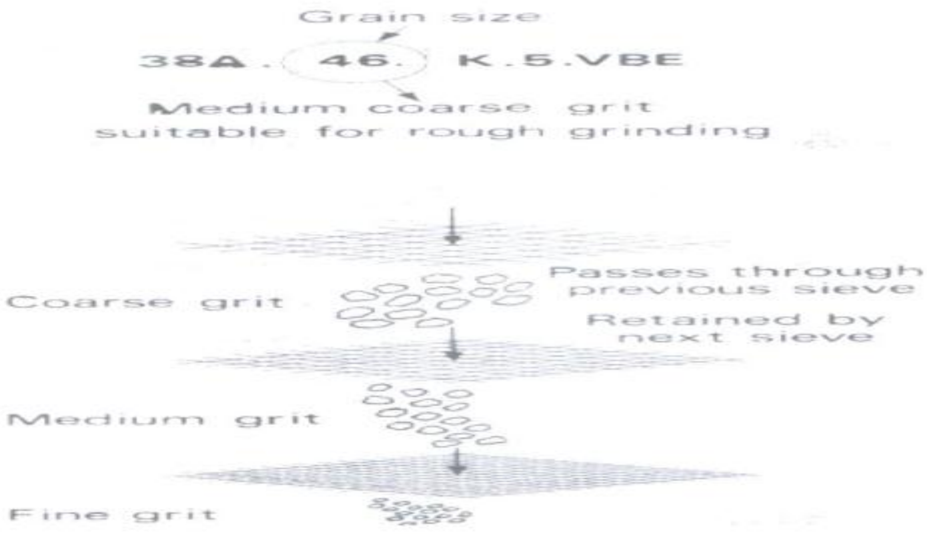
b. Grain size

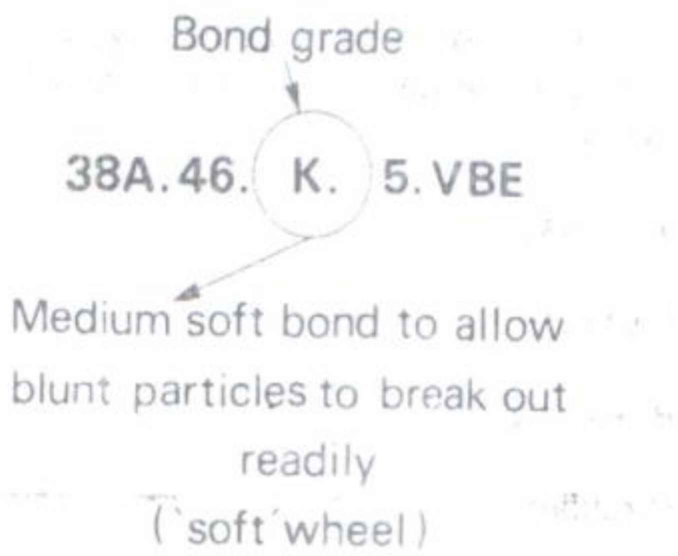
c. Grade of bond

d. Structure

e. Bond type

f.





3.4 An example of an abrasive wheel marking.

a. A aluminum oxide abrasive

b. 16 coarse grain size

c. P medium to hard grade of bond

d. 5 medium to dense structure

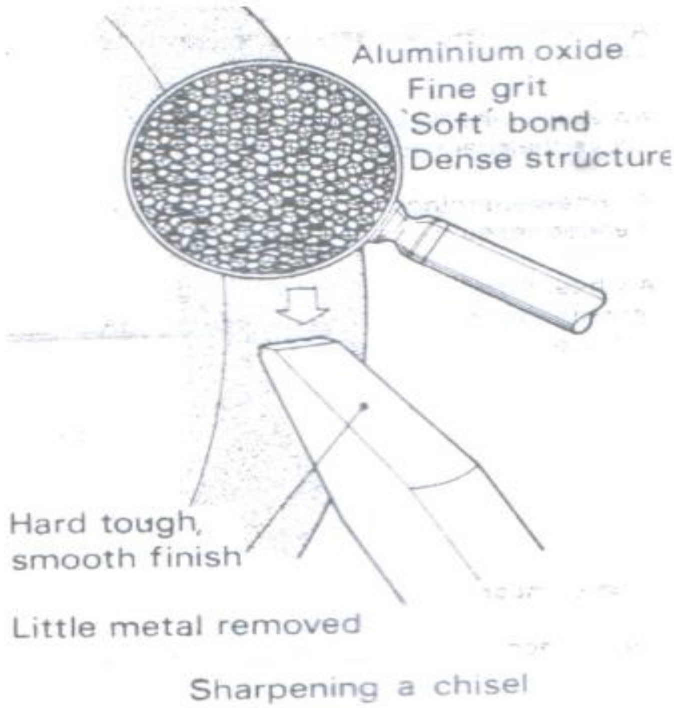
e. V vitrified bond

f. BE manufacturer's particular bond character

3.5 Selecting the Abrasive Wheel

a. The kind of material to be ground

- Strong tough materials are ground with aluminum oxide wheels.
- Hard materials are ground with silicon carbide wheels.
- Hard materials are ground with soft bonded wheels.



STANDARD SYSTEM FOR MARKING GRINDING WHEELS

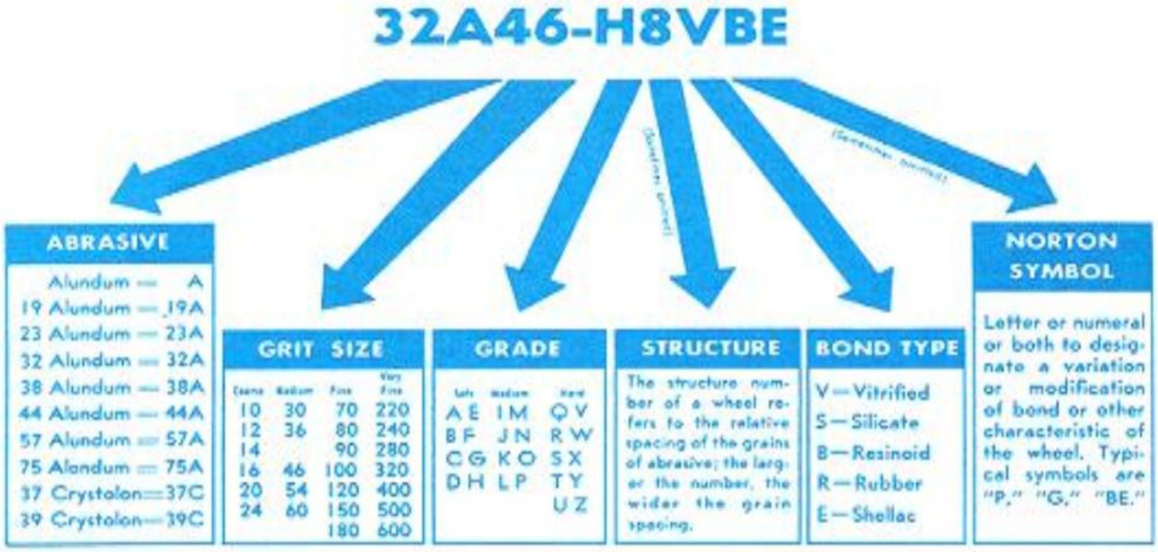
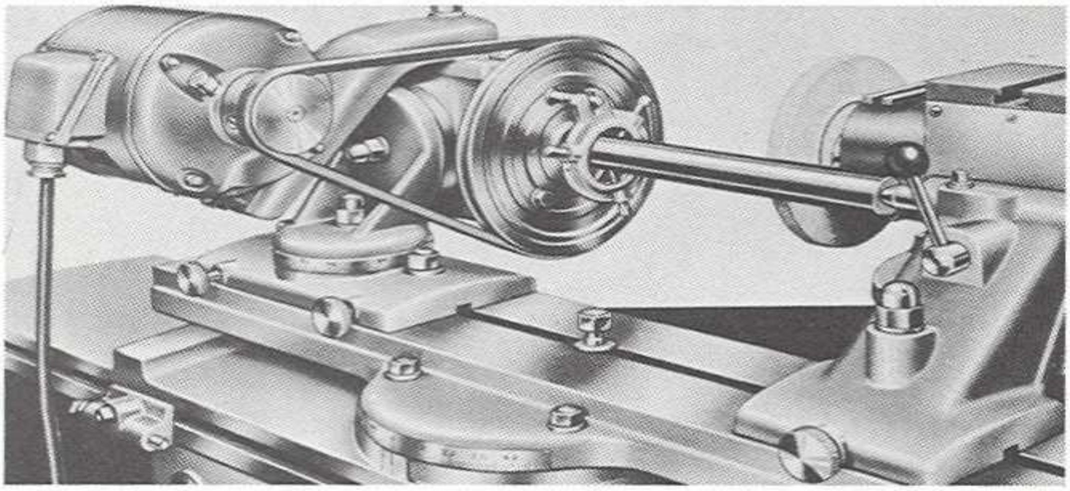


Fig. 21-39. Standard system for marking grinding wheels.

1**. Surface grinding** – it is used for grinding flat surfaces. The work piece is supported on rectangular table which moves back forth and reciprocating beneath the grinding wheel.

2. **Cylindrical grinding -** this process used for grinding external and internal cylindrical surface.



3**. Center less grinding** – it is used for grinding of external and internal cylindrical surfaces.

4. Form grinding - is generally used grinding of gears, thread grinding and grinding of spines etc.

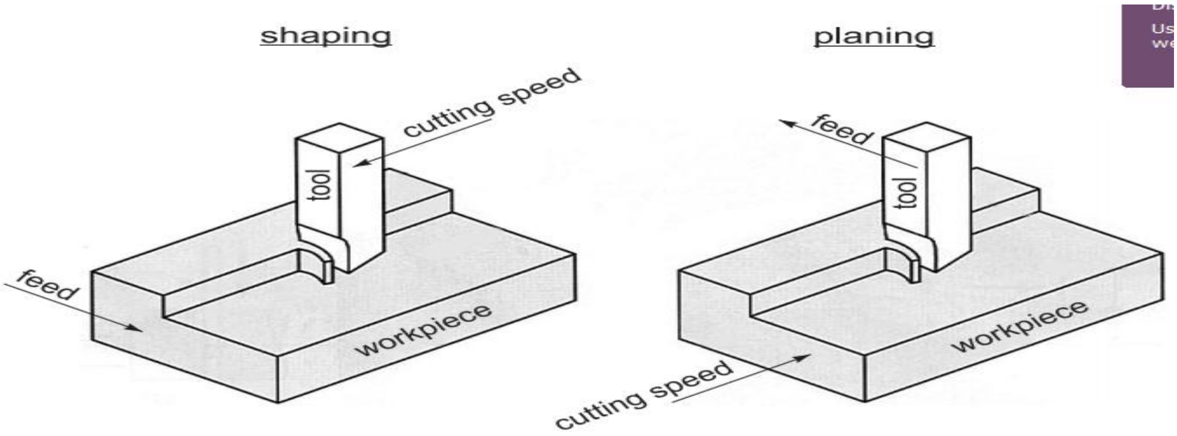
5. Abrasive cut- off - used for severing metallic and nonmetallic materials with the help of thin grinding wheel rotating at a high speed.

6. Manual grinding – process are those in which work piece or grinding wheel in held by hand.

7. Special grinding process – this machine have been designed such as crankshaft grinding, cam shaft grinding, gear grinding etc.

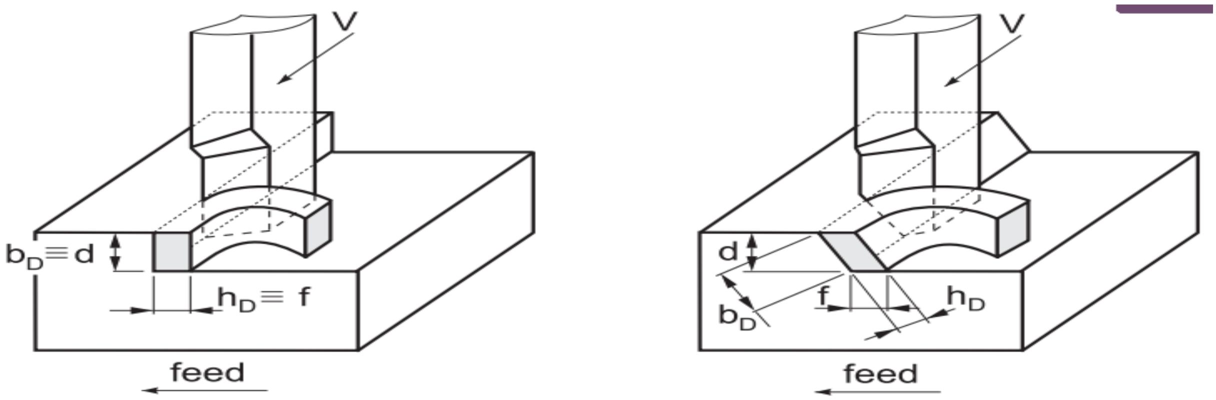
2.5.2 PLANING AND SHAPING

Planning and *shaping* are similar operations, which differ in the kinematics of the process. Planning is a machining operation in which the primary cutting motion is performed by the work piece and feed motion is imparted to the cutting tool. In shaping, the primary motion is performed by the tool, and feed by the work piece:



Kinematics of shaping and planing.

The *cutting conditions* in planning and shaping are illustrated in the figure. Only the shaping operation is portrayed but the cutting conditions are essentially the same and for planning:



Cutting conditions in orthogonal (*Left*) and oblique (*Right*) shaping.

Cutting velocity V in planning is linear and constant along the cutting path. In shaping, the picture is more complicated. The cutting tool is held in the tool post mounted in the ram, which reciprocates over the work with a forward stroke, cutting at velocity V and a quick return stroke at higher velocity. The cutting velocity is therefore not constant along the cutting path. It increases from zero to maximum in the beginning of the stroke and gradually decreases to zero at the
end of the stroke. The cutting speed V is assumed to be twice the average forwarded ram velocity.
Feed f in planning and shaping is in mm per stroke and is at right angles to the cutting direction.
Depth of cut d is defined as usual as the distance between the work and machined surfaces.

Shapers

Shapers are used to produce primarily flat surfaces by means of single or multiple cutting tools. Shapers use a reciprocating motion

- For ward stroke (cutting motion)
- Backward stroke (non cutting stroke)

The size of the shaper is determined by the largest cube which it can machine. For example, a 300mm shaper can machine 300mm x 300mm x 300mm.

Use of Shaper operations

- flat surfaces,
- a block square and parallel,
- angular surfaces,
- Serrating - operation performed on a surface to form a series of equally spaced grooves, which are often used on wide jaws, plates, washers and other parts that are intended to provide a good grip

Main parts of shaper

- **Frame.**
- **Ram** moves in adjustable slide rails. The stroke length and position can be adjusted.
- **Cross-rail**- It is itself adjustable side rails and carries the table on horizontal slide rails.
- **Table** has T-slots and holes to clamp works piece. The setting motion is vertical and the feed motion is horizontal.
- **Main gear box** - The rotary movement of the motor is transmitted the crank gear by change-over gear box (3 to 8 r.pm). The direction of motion is changed by a crank and rocker

2.3 Types of shapers

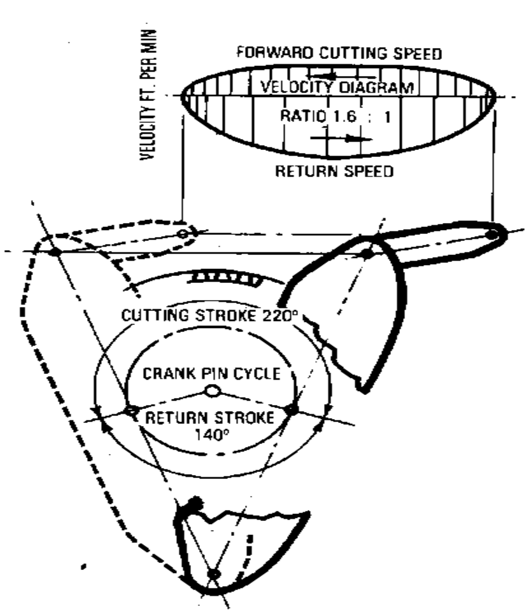
The most common type of shaper is horizontal shaper they can be:

- Mechanically driven (crank and rocker)
- Hydraulically driving (cylinder and piston

1. **Principle of the crank-type shaper**

There are several mechanical methods of driving the reciprocating shaper ram, the crank-type being the most commonly used

- The quick return of the ram of the back or non-cutting stroke is obtained when the crank pin is traveling through 140 degree of the cycle.
- The crank pin will pass through 220 degree of the cycle during the cutting stroke or forward stroke.
- The direction of motion is changed by a crank and rocker



- The motor drives the crank wheel (driving wheel) over the main gear box. The crank is seated in the drilling wheel and can be shifted radially. This carries with it the sliding block which guides in the rocker arm and sets its' reciprocating motion.
- . The crank pin runs in its circular path at uniform speed.
- It must travel a longer distance in its working stroke α (longer time), and
- a shorter distance in the return stroke β (less time), the speed is alternately higher and lower. The longer the stroke the greater is the difference in speeds

Changing the stroke length

If the crank pin is drawn away from the center of the wheel, it describes a crank circle of greater radius, and the rocker arm swings through a greater distance, the stroke becomes longer

- Analogously, if it is drawn towards the center of the wheel, a shorter stroke results.
- *Changing the stroke position must be set to match the position of the work piece.*

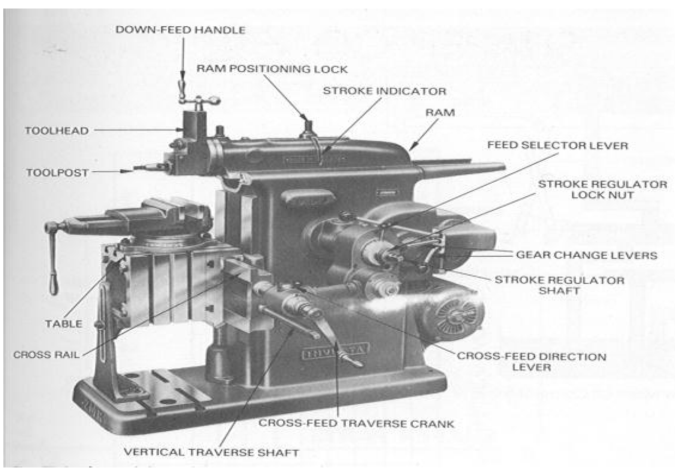


Fig 2.1 parts of shaper

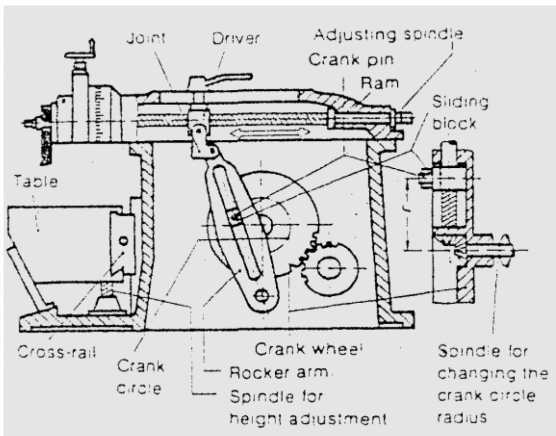


fig 2.2 principle of the crank type shaper

The cycle of the crank-type shape

Setting the rate of feed

- The rotating driving wheel sets the pawl into reciprocating motion by means of the connecting rod and crank pin.
- The pawl carries the ratchet with it in one direction, and moves the table.

- ***Crank pin moved outwards***: Greater distance traveled by the pawl and ratchet result in a **higher feed rate.**
- ***Crank pin drawn inwards*** - smaller distance traveled by the pawl and ratchet results in lower **feed rate**.

Fig 2.3 the cycle of the crank type shaper

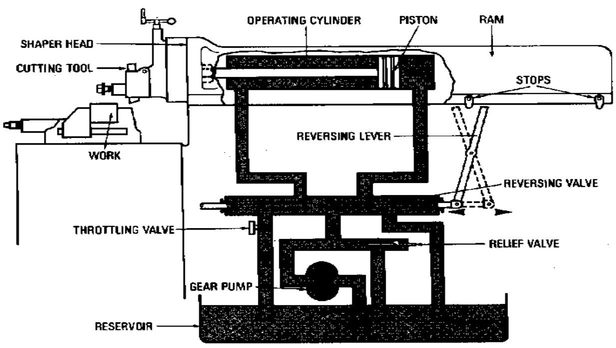
2. **Principle of Hydraulic shaper**
   Hydraulically driven shapers are similar in nearly all respects to the crank; only the drive principle is different

Main advantage of a hydraulic shaper

- It provides a uniform positive motion
- A wide range of speeds and feeds is available
- Speeds and feeds are independent and constant during the cutting stroke
- The speeds and feed can be adjusted without stopping the shaper
- Cutting speeds and feeds are shown on a dial

Hydraulic ram drive

- The motor drives the pumps. The quantity of oil changes the speed of the ram, which is changed by swiveling the pump.
- The reversal of its **direction of movement, stroke length** and **position are changed by a trip dogs**.
- ***Working stroke*** - when the oil flows into the cylinder chamber V1, the ram speed is low, because a large cylinder volume has to be filled; the thrust force is high, because of the oil presses is against the large piston surface.
- ***Return stroke*** - The oil flows into the cylinder chamber V2. The ram speed is high, because a smaller cylinder volume has to be filled the thrust force is low because the oil pushed against a smaller piston surface.



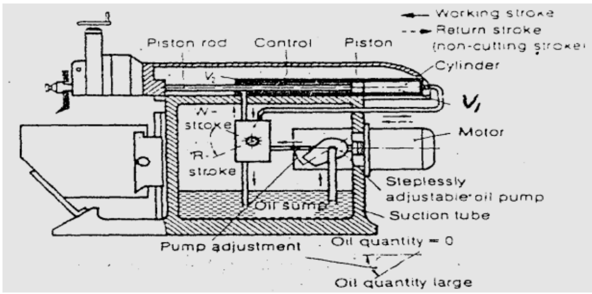


Fig 2.4 principle of hydraulic shaper

2.4 Shaper speeds

Depends up on three factors:

- The type of material being cut
- Its length and
- The type of cutting tool used

Cutting speeds for materials are expressed in either feet per minute (ft./min) or meters per minute (m/min)

- Since the recommended cutting speeds are stated in feet per minute and the length of the cutting stroke in inches, it is necessary to change:
- feet by multiplying by 1/12
- divide the above calculation by 3/5 (since the cutting stroke takes up 3/5 of the cycle) or multiply by 5/3
- Instead of multiplying by 1/12 and then by 5/3, it is easier to use 0.14 or 1/7.
- Therefore, to calculate stokes per minute, divide the cutting speed of the metal by the length of the stroke times 0.14 or 1/7

Thus; N =CS/Lx 0.14 or N =CS/Lx1/7

Where, N=stroke per minute
CS=cutting speed
L=length of the stroke

Example:
Find the number of strokes per minute for a shaper to machine a piece of 11 in long cast iron at 60ft/minute.

$$N = \frac{CS \times 7}{L} = \frac{60 \times 7}{12} = 35 \text{ strokes/minute}$$

Note: L is always the length of the work plus 1 in for tool clearance

Metric speed calculations

$$N = \frac{CS\ (meters)}{L\ (meters)} \times 3/5 \quad or$$

$$N = \frac{CS\ (meters)}{L\ (meters)} \times 0.6$$

Note: When shaping a work piece with metric dimensions, set the length of stroke to 25mm longer than the work piece

Example:
How many strokes per minute will be required to machine a piece of tool steel 330 mm long (CS =15m/minute)

L= 330 mm+25 mm clearance= 355 mm =0.36 m

$$N = \frac{CS \times 0.6}{L} = \frac{5 \times 0.6}{0.36} = 25 \text{ stroke per minute}$$

2.5 Shaper feeds

- Shaper feed is defined as the distance the work is moved sideways toward the cutting tool for each forward, stroke of the ram.

When speeds and feeds are selected, the following factors must be considered:

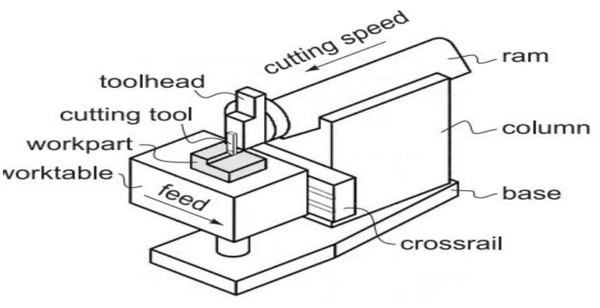
- The type of material being cut
- The type of cut (roughing or finishing)
- The type of cutting tool used
- The type of surface finish required
- Vertical feed on a shaper may be used for machining vertical or angular surfaces, by the down feed screw on the tool head or automatically by the power down feed.
- The power feed is attached to the down feed screw by a bevel gear drive, and on the return stoke of the ram, the feed mechanism indexes the amount of vertical feed per cut.
- Feeds used for flat or angular surfaces should always be operated on the return stroke

2.6 Setting the shaper stroke length

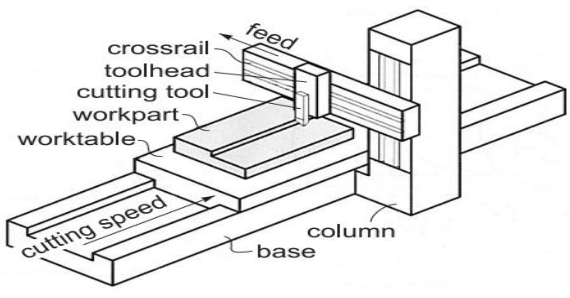
To set the shaper stroke length

- Measure the length of the work piece and add 25 mm to the length
- Start the shaper and stop it when the ram is at the back of the stroke
- Loosen the stroke regulator lock nut
- Adjust to the length required by turning the stroke regulator screw
- Tighten the stroke regulator lock cut

Shaping is performed on a machine tool called a *shaper*. The major components of a shaper are the *ram*, which has the *tool post* with cutting tool mounted on its face, and a *worktable*, which holds the part and accomplishes the feed motion.



Components of a shaper



Components of an open-side planer.

Planers

The machine tool for planning is a *planer*. Cutting speed is achieved by a reciprocating worktable that moves the part past the single-point cutting tool. Construction and motion capability of a planer permit much larger parts to be machined than on a shaper. Planers can be classified as either open side planers or double-column planers.

The *open side planer,* also known as a *single-column planer* has a single column supporting the cross rail on which a tool head is mounted. The configuration of the open side planer permits very wide work parts to be machined.

A *double-column planer* has two columns, one on either side of the bed and worktable. The columns support the cross rail on which one or more tool heads are mounted. The two columns provide a more rigid structure for the operation but limit the width of the work that can be handled.

2.6.1 POWER HACKSAW MACHINES

All power hacksaw machines are basically similar in design. Figure 6-1 shows a typical power hacksaw and identifies its main parts, which are discussed below.

Base

The base of the saw usually contains a coolant reservoir and a pump for conveying the coolant to the work. The reservoir contains baffles which cause the chips to settle to the bottom of the tank. A table which supports the vise and the metal being sawed is located on top of the base and is usually referred to as part of the base.

Vise

The vise is adjustable so that various sizes and shapes of metal may be held. On some machines the vise may be swiveled so that stock may be sawed at an angle. The size of a power hacksaw is determined by the largest piece of metal that can be held in the vise and sawed.

Frame

The frame of the saw supports and carries the hacksaw blade. The machine is designed so that the saw blade contacts the work only on the cutting stroke. This action prevents unnecessary wear on the saw blade. The cutting stroke is on the draw or back stroke.

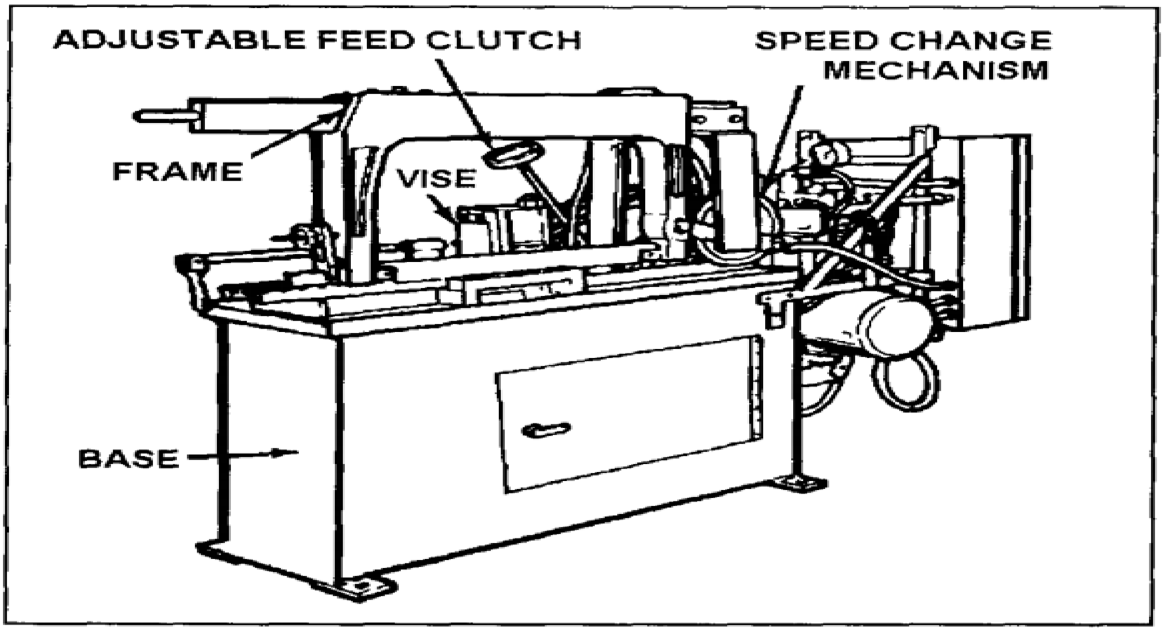


Figure 6-1. Power hacksaw.

a. ADJUSTABLE FEED CLUTCH

The adjustable feed clutch is a ratchet-and-pawl mechanism that is coupled to the feed screw. The feed clutch may be set to a desired amount of feed in thousandths of an inch. Because of the ratchet-and-pawl action, the feed takes place at the beginning of the cutting stroke. The clutch acts as a safety device and permits slippage if too much feed pressure is put on the saw blade. It may also slip because of a dull blade or if too large a cut is attempted. This slippage helps prevent excessive blade breakage.

b. SPEED-CHANGE MECHANISM

The shift lever allows the number of strokes per minute to be changed so that a variety of metals may be sawed at the proper speeds. Some saws have a diagram showing the number of strokes per minute when the shift lever is in different positions; others are merely marked “F,” M,” and “S” (fast, medium, and slow).

2.6.2 Cutting tools

Cutting tool is a wedge shaped and sharp edged device that is used to remove excess layer of material from the work piece by shearing during machining in order to obtain desired shape, size and accuracy. It is rigidly mounted on the machine tool. A relative velocity between work piece and cutting tool is also provided by various mechanical and other arrangements for cutting action.

Examples of cutting tools

Cutting tool is basically the cutter used in machining operation. Various machining operations utilize different cutters and thus various names are available for these cutters based on the application. A list of commonly used cutting tools is provided below.

1. **Single point turning tool**—cutter for turning operation performed in lathe
2. **Drill**—cutter for drilling operation performed on drilling machine or lathe or milling machine
3. **Milling cutter (or mill)**—cutter for milling operations performed on milling machine
4. **Fly cutter**—cutter for fly milling operation performed in milling machine
5. **Shaper**—cutter for shaping operation performed in shaping machine
6. **Planer**—cutter for planning operation performed in planning machine
7. **Boring bar**—cutter for boring operation performed in drilling or boring machine
8. **Reamer**—cutter for reaming operation performed in drilling machine
9. **Broach**—cutter for broaching operation performed in broaching machine
10. **Hob**—cutter for hobbling operation performed in hobbling machine
11. **Grinding wheel**—abrasive cutter for grinding operation performed in grinding machine

1. **Form tool: -** Form tool is used to produce a concave or convex form on the external or internal surface of the work piece. Lathe cutting tool edges are shaped differently depending on:-
    - The type of metal to be machined
    - The type of cut to be made.

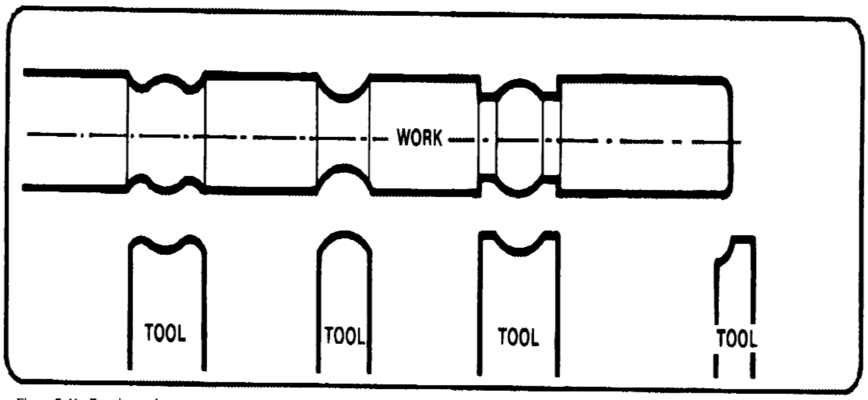


Figure 7-60. Forming tools.

Fig. Forming tools

a. Milling Cutters

1. Plain milling cutters

The most widely used milling cutter is the plain milling cutter which is a cylinder made of tool steel with teeth cut on the periphery. It is used to produce a flat surface. These cutters may be of several types

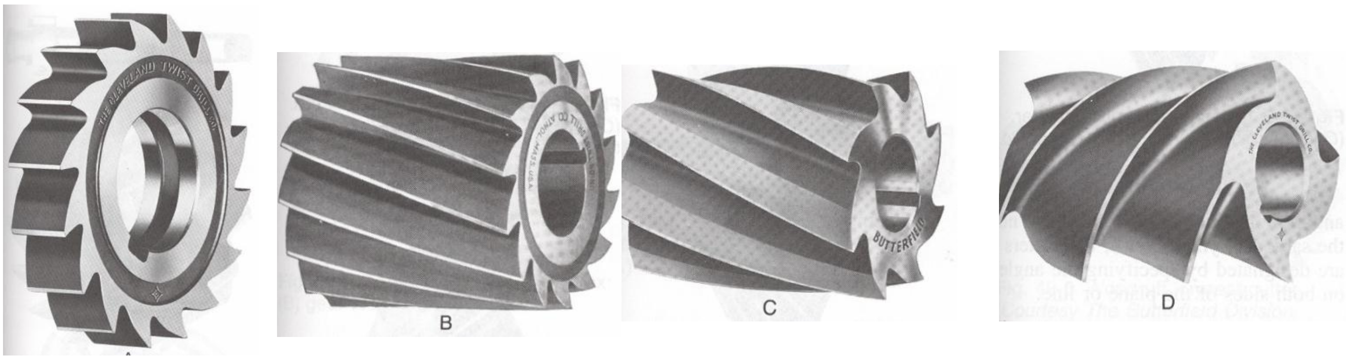


Fig: A & B Plan milling cutters A) Light duty B) light duty helical C) Heavy duty plain D) High helix

2. **Side milling cutters**

They are comparatively narrow cylindrical milling cutters with teeth on each side as well as on the periphery. Used for cutting slots face and straddle milling operations.

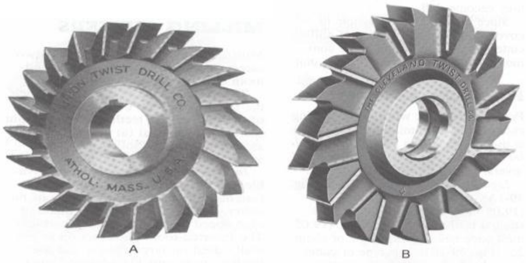
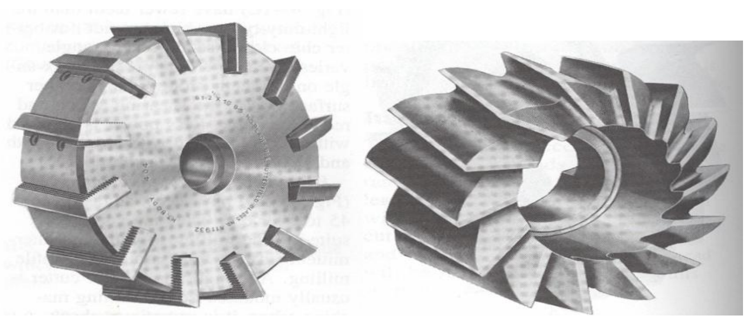


Fig: Side milling cutters A) plain B) staggered tooth

3. **Face milling cutters**

Face milling cutters are generally over 152.4 mm in diameter and have inserted teeth held in place by a wedging device. This type of cutter is often used as a combination cutter, making the roughing & finishing cut in one pass.



A B

Fig. Face milling cutters A) over ∅152.4 B) shell end mill & adapter

4. **Angular milling cutters**

Angular milling cutters have teeth that are neither parallel nor perpendicular to cutting axis. Used for milling angular surfaces, such as grooves, serration, chamfers, reamer teeth etc. They are divided in to two groups:

- **Single-angle mill cutters** – have teeth on the conical surface and may or may not have teeth on the flat side.

- **Double-angle milling cutters** – have two intersecting conical surface with cutting teeth on both

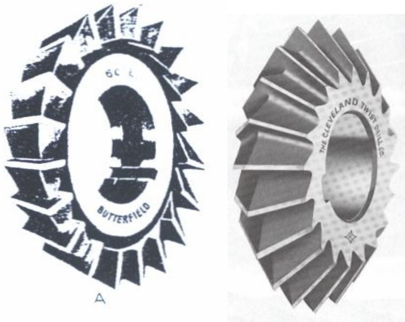


Fig: A) Single angle milling cutter B) Double-angle milling cutter

5. **End mills** are which have cutting teeth on the end as well as on the periphery & are fitted to the spindle by a suitable adapter.

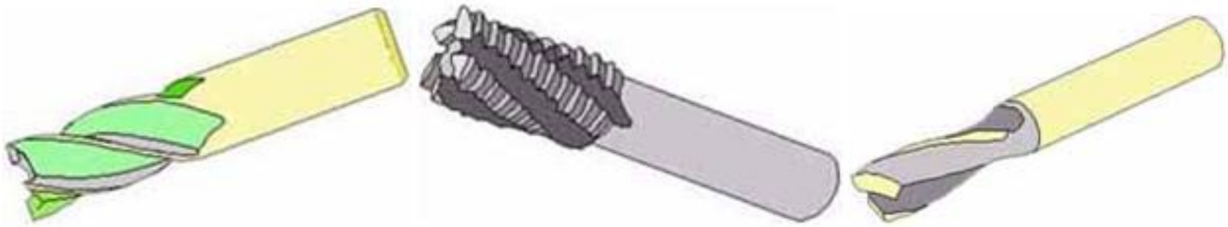
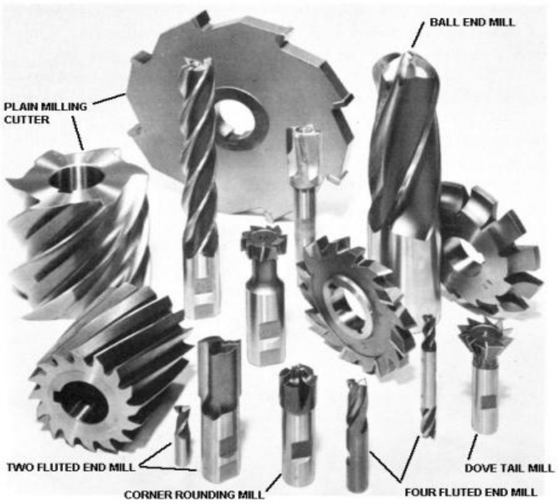


Fig: End mills

Milling cutters

Classification of milling cutters according to their design include the following:



❖ *HSS cutters.* Many cutters like end mills, slitting cutters, slab cutters, angular cutters, form cutters, etc., are made from high-speed steel (HSS).

Assortment of high-speed steel milling cutters. High-speed steel is the cutting tool material that is used to produce cutting tools of complex designs for low to medium cutting speeds.

Twist Drills/ Drill bits

Twist drills are end-cutting tools used to produce holes in most types of material. On standard drills, two helical flutes are cut lengthwise around its body. They provide cutting edges and space for the cuttings to escape during the drilling process. Since drills are among the most efficient cutting tools, it is necessary to know the main parts, how to sharpen the cutting edges, and how to calculate the correct speeds for drilling various materials in order to use them most efficiently and prolong their life.

2.6.3 Materials **of cutting tool**

CUTTING TOOL MATERIAL

- A machine tool is no more efficient than its cutting tool. 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 been, 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.

During machining, part of the cutting tool remains in physical contact with the work piece and thus experiences severe cutting temperature and insistent rubbing. The material of the cutting tool must have the capability to sustain such high cutting temperature as well as cutting force. Every tool material must possesses certain properties such as high hardness, high hot hardness, high strength, higher melting point and chemically inert even at high cutting temperature.

As a thumb rule, the hardness of the tool material should be at least 1.5 times of the hardness of the work piece for smooth cutting action.

Suitable coating can also be applied on the tool to improve various desired properties. However, a coated tool does not allow easy re-sharpening by grinding when the edges are worn out after prolonged use. Now-a-days, insert based tools are available where small interchangeable inserts can be attached or clamped on large shank. These inserts perform cutting action and thus worn out gradually. When wear exceeds the tolerable limit, the inserts can be replaced by a new one, while the shank can be used repeatedly. Some of the tool materials commonly available in todays' market are enlisted below.

- High Speed Steel (HSS)
- Tungsten carbide
- Ceramics
- Cubic Boron Nitride
- Diamond

1. Carbon Steel.

- Carbon steel, or tool steel is high in carbon content, hardens to a high degree of hardness when properly heated and quenched.
- The carbon-steel tool will give good results as long as constant care is taken to avoid overheating or "bluing," since the steel will lose its temper or hardness at a relatively low heat becoming ineffective as a cutting tool.
- For low-speed turning, high carbon steels give satisfactory results and are more economical than other materials.

2. 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.

3. **Satellite**
   - These cutter bits will withstand higher cutting speeds than high-speed steel cutter bits.
   - Satellite is a nonmagnetic alloy which is harder than common high-speed steel. The tool will not lose its temper, even though heated red hot from the friction that is generated by taking a cut.
   - Satellite is more brittle than high-speed steel.
   - To prevent breaking or chipping, it requires just enough clearance to permit the tool to cut freely.
   - Satellite is also used for machining hardened steel, cast iron, bronze, etc.
4. **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, and such abrasive nonmetallic materials as fiber, hard rubber.
   - They require special grinding wheels for sharpening, since tungsten carbide is too hard to be redressed on ordinary grinding abrasive wheels.
5. **Tantalum Carbide and Titanium Carbide**.

These cutting tools are similar to tungsten carbide tools but are used mostly for

- Machining steel where extreme heavy cuts are taken and
- Heat and pressure tend to deform the cutting edge of the other types of cutting tools.

2.6.4 Classification of cutting tool

Cutting tools can be classified in various ways; however the most common way is based on the number of main cutting edges that participates in cutting action at a time. On this basis, cutting tools can be classified into three groups as given below.

- **Single point cutting tool**: - such cutters have only one main cutting edge that participate in cutting action at a time. Examples include turning tool, boring tool, fly cutter, slotting tool, etc.
- **Double point cutting tool**:- such as the name implies, these tools contain two cutting edges that simultaneously participate in cutting action at a pass. Example includes drill (common metal cutting drill that has only two flutes).
- **Multi-point cutting tool**: - These tools contain more than two main cutting edges that can simultaneously remove material in a single pass. Examples include milling cutter, broach, gear hobbling cutter, grinding wheel, etc.

Self-Check -2	Written Test

Directions 1: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. The tool used in a lathe is a
   A. multipoint cutting tool B. Single point cutting tool
   C. saw tooth cutting tool D. drill
2. Knurling tool (knurl) is a
   A. single point cutting tool B. saw tooth cutting tool
   C. embossing tool D. parting tool
3. The assembly which consists of saddle, cross-slide, compound slide and tool post is
   A. headstock B. tailstock C. bed D. carriage
4. In a milling machine, cutters are mounted on
   A. column B. spindle C. overhanging arm D. arbor
5. One part of the milling machine it is fitted to the over arm and can be clamped at any location on the over arm
   A. Arbor support B. Spindle C. cross- slide D. Column
6. Which kinds of jaw are most important to hold square or rectangular work piece?
7. One is not parts of engine lathe
   A. Headstock, a tailstock,
   B. A carriage
   C. A bed upon which the tailstock and carriage move.
   D. Tool post
   E. Grinding wheel
8. One of the following is not properties of tail Stock
   A. The tail stock consists of the upper and lower tail stock castings can be adjusted for taper or parallel turning by two screws set in the base.
   B. The tail stock can be locked in any position along the bed of the lathe
   C. The tail stock spindle has an internal taper to receive the dead center, or live center
   D. The tail stock provides support for the right hand end of the work
   E. All of the above
9. One is not the main parts of carriage
   A. Compound rest
   B. The saddle
   C. Cross-slide,
   D. Apron
   E. Headstock
10. Popular types of tool posts are
    A. the standard tool posts
    B. castle tool posts
    C. The quick change tool posts
    D. All of the above
11. The sole purpose of the tool post is
    A. To provide a rigid support for the tool holder
    B. To provide cutting fluid
    C. To provide cutting action
    D. All of the above
12. One properties of lead screw.

A. The lead screw is used for thread cutting
B. Along its length are accurately cut acme (highest) threads which engage the threads of the half-nuts in the apron when the half-nuts are clamped over it.
C. When the lead screw turns inside the closed half-nuts, the carriage moves along the ways a distance equal to the lead of the thread
D. All of the above

13. Lathe chucks
A. Three-jaw universal chuck
B. Four- jaw independent chuck
C. Collect chuck
D. *Hollow Headstock Spindle Chuck*
E. *All of the above*

14. *One not Purpose of drill Chuck*
A. Used for holding straight-shank drills
B. Used for holding reamers
C. Used for holding taps
D. Used for holding Small-diameter work-pieces
E. Used for holding lathe check

15. *One is properties of faceplates*
A. A lathe faceplate is a flat, round plate that threads to the headstock spindle of the lathe.
B. The faceplate is used for clamping and machining irregularly-shaped work-pieces
C. The work-piece is either attached to the faceplate using angle plates or brackets, or is bolted directly to the plate.
D. All of the above

16. One of the following is not common types of centers
A. Male Center
B. Pipe Center
C. Female Center
D. Half-Male Center
E. None

17. One of the following is not lathe cutting tool materials
A. HSS
B. Carbide steel
C. Tungsten Carbide
D. Tantalum Carbide and Titanium Carbide.
E. None of the above

18. One of the following is used to support long work pieces
A. Tailstock
B. steady rest
C. follower rest
D. All of the above

19. Types of lathe cutting tools
A. Right-Hand Turning Cutter Bit.
B. Left-Hand Turning Cutter Bit.
C. Round-Nose Turning Cutter Bit.
D. All of the above

Directions 2: Match the term in column **A** with the term in column **B.**

Term A	Term B
_____ 1. Faceplate	a) chuck key
_____ 2. Four Jaw Chuck	b) lathe dog
_____ 3. Universal Lathe Chuck	c) collet chuck
_____ 4. Chuck partner	d) three (3) jaw chuck
_____ 5. Drill press type of lathe chuck	e) independent chuck

f) Three jaws

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

Answer Sheet

Score = ___________
Rating: ____________

Name: _________________________ Date: _______________

Short Answer Questions

Information Sheet-3	Determination of sequence of operations

3.1 Introduction to sequence of operations

Sequencing refers to the order in which activities occur in the operations process. But there are a lot of operations are performed by different machines such as;

1. **Operation on lathe machine** Facing, Turning, Chamfering, Grooving, Forming, Knurling, Undercutting, Eccentric turning, Taper turning, Thread cutting, Drilling, Reaming, Boring, Tapping and etc. these operation are depend on one another.

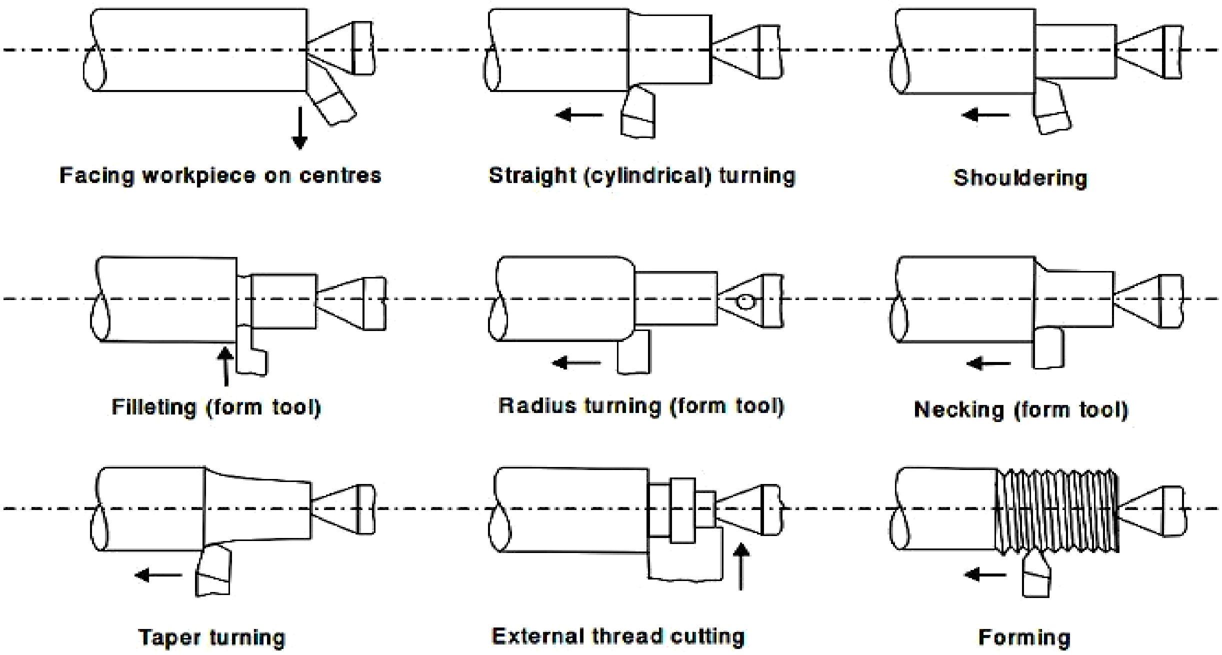


Fig. Lathe operation

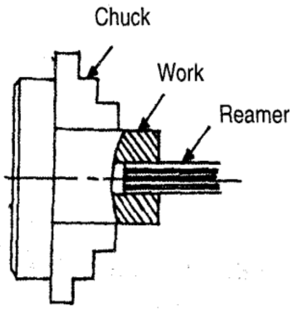
3.1.1 Common lathe operations:

1. Facing	2. Plain turning	3. Step turning
4. Drilling	5. Boring	6. Reaming
7. Under cutting or grooving	8. Threading	9. Knurling
10. Forming	11.Taper turning	

1. **Facing**

   "Facing" is an operation of machining the ends of a work piece.

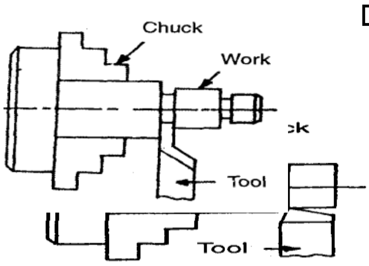
- ❑ used to produce a flat surface square with the axis;
- ☐ Used to cut the work to the required length.
- ☐ To obtain a reference surface to measure
- ☐ Involves feeding the tool perpendicular to the axis of rotation of the work piece.



- Rough facing operation is done by using a heavy cross feed of the order of 0.5 to 0.7 mm and a deeper cut up to 5 mm (maximum).
- a finer cross feed of 0.1 to 0.3mm and a smaller depth of cut of about 0.5 mm.

2. Plain turning

- It is an operation of removing excess material from the surface of the cylindrical work piece.



- The work is held either in the chuck or between centers and the longitudinal feed is given to the tool either by hand or power.

Fig.4.29 Plain turning

3. Step turning

- In this operation various steps of different diameters in the work piece are produced.
- Carried out in the similar way as plain turning.

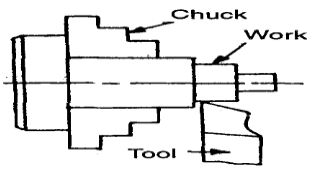
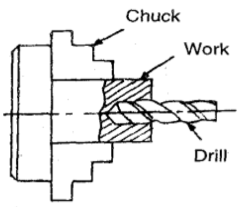


Fig.4.30 Step turning

4. Drilling on lathe machine

- It is an operation of producing a cylindrical hole in a rotating w/p by the cutting edge of a drill,
- The work is held in a suitable device, such as chuck or face plate and the drill is held in the sleeve or barrel of the tail stock.

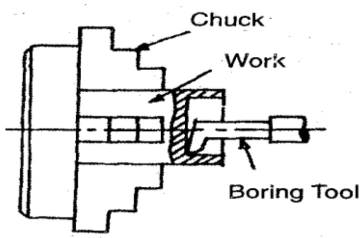


- The drill is fed by hand rotating the hand-wheel of the tailstock

Fig.4.31 Drilling

5. Boring

- Enlarging and turning a hole produced by drilling, punching, casting or forging.
- An equally important, and concurrent, purpose of boring may be to make the hole concentric with the axis of rotation of the work piece



- A boring tool or a bit mounted on a rigid bar held in the tool post is fed into the work by hand or power.
- Boring cannot originate a hole.

Fig.4.32 Boring

6. Reaming

- Reaming is the operation which usually follows the earlier drilling and boring.
- Reamer, which has multiple cutting edges, is used.
- Held on the tailstock spindle, while the work is revolved at very slow speed.
- Used when a very high grade of surface finish and dimensional accuracy is needed in a hole.
- The feed varies from 0.5 to 2 mm per revolution
- For reaming tapered holes, taper reamers are used

Fig.4.33 Reaming

7. Undercutting or grooving

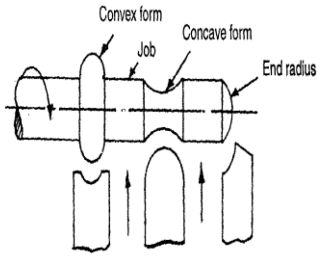
It is the process of reducing, the diameter of a w/p over a very narrow surface.

- Often done at the end of a thread or adjacent to a shoulder to leave a small margin.
- The work is revolved at half the speed of turning and a grooving.
- A tool of required shape is fed straight cross wise into the work.

Fig.4.34 Undercutting

8. Threading

It is an operation of cutting helical grooves on the external or internal cylindrical surface of the work piece.



Lathe provided the first method for cutting threads by machines. Although most threads are now produced by other methods, lathes still provide the most versatile and fundamentally simple method

- Work is held in a chuck or between centers and the threading tool is fed longitudinally to the revolving work.
- The longitudinal feed is equal to the pitch of the thread to be cut.

Fig 4.35 thread cutting

9. Knurling

- It is an operation of embossing a diamond or straight shaped pattern on the surface of a work piece by knurling tool (knurls).

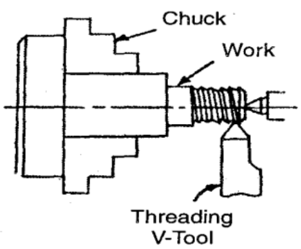
Knurls are classified (according to their pattern- diamond or straight):

-Course
-Medium and
-Fine

Purpose of knurling:

- to provide an effective gripping surface on a work piece (prevent slipping)
- to increase gripping diameter

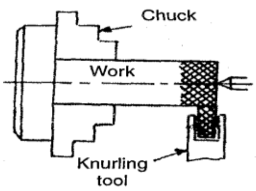
Procedure:-



- The tool is held rigidly on the tool post and the rollers are pressed against the revolving work piece to squeeze the metal against the multiple cutting edges, producing depressions in a regular pattern on the surface of the work piece
- Done at the slowest speed available in a lathe, and plenty of oil is flowed on the tool and work piece.

Fig4.36 knurling tool

10. Forming

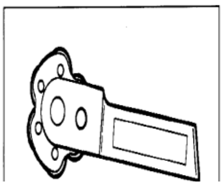


An operation of turning a convex, concave or any irregular shape.

Methods

(i) Using a forming tool,
(ii) Combining cross and longitudinal feed,
(iii) Tracing or copying a template.

Fig4.37 Forming



11. Taper turning

- A taper may be defined as a uniform increase or decrease in diameter of a w/p measured along its length.
- The amount of taper ("capacity") in a work piece (α) is the ratio of the difference in diameters of the taper to its length.

$\alpha = D - d / l$ or $\alpha = D - d / 2l$ –For short tapers

Where, D = large diameter of taper in mm,
d = small diameter of taper in mm,
L = length of tapered part in mm, and
α = half of taper angle.

Taper turning methods

1. By setting over the tail stock center.
2. By swiveling the compound rest.
3. By using a taper turning attachment.
4. By manipulating the transverse and longitudinal feeds of the slide tool simultaneously.
5. by using form tool

1. By setting over (s) the tail stock center

$S = L - D - d / 2l$ mm

= Total length x total taper / 2 x taper length

Where S = the required set over in mm,

D = large diameter in mm,
d small diameter in mm,
L = total length of work in mm, and
l = length of tapered portion in mm

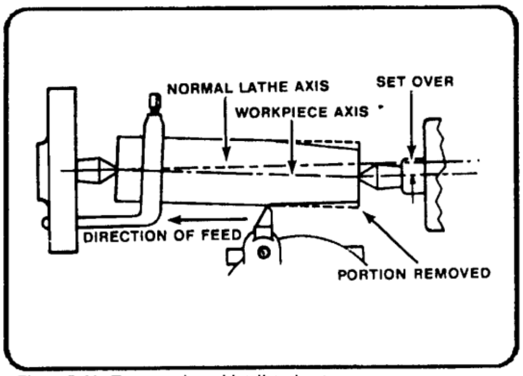


Figure 7-64. Taper turning with tailstock set over.

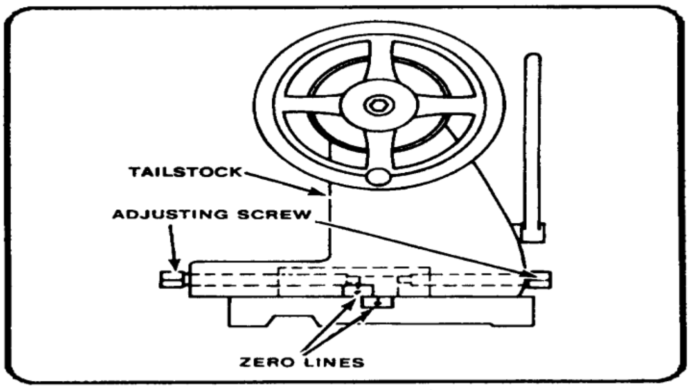


Figure 7-63. Tailstock offset for taper turning.

Fig 4.38 taper turning with tail stock set over

2. By swiveling the compound rest

- It is the best method as it does not affect the centering of the job or center

$$\tan \alpha \text{ or } \alpha = \tan^{-1}\left(\frac{D-d}{2L}\right)$$

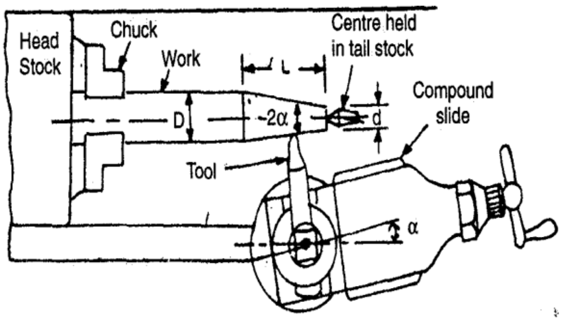


Fig 4.39 taper turning by swiveling compound rest

3. By using a taper turning attachment

- This method provides a very wide range of taper.

$$\tan \alpha = \frac{D-d}{2l} \text{ (all dimensions in mm)}$$

or

$$\alpha = \tan^{-1}\left(\frac{D-d}{2l}\right) \text{ degrees.}$$

where D = larger dia. in mm,
d = smaller dia. in mm, and
l = length of taper in mm.

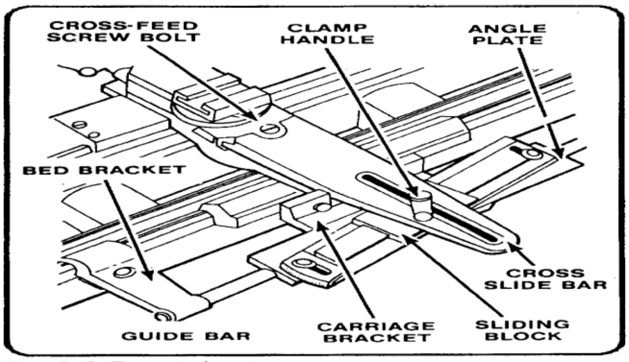


Figure 7-69. Taper attachment.

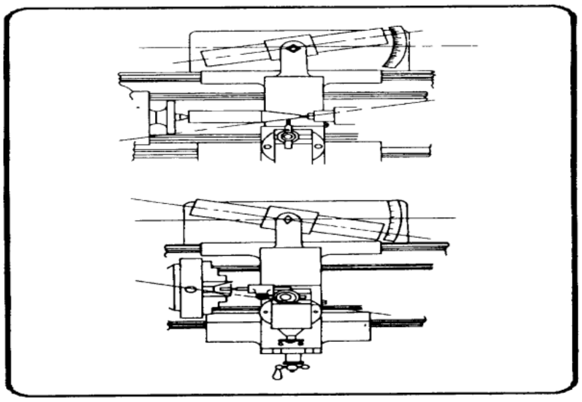


Figure 7-70. Taper turning and boring.

Fig 4.40 taper turning taper turning attachment

4. By manipulating the transverse and longitudinal feeds of the slide tool simultaneously:

- both feeds is inaccurate and requires skill
- used for sharp tapers only.

5. By using a broad nose form tool

- A broad nose tool having straight cutting edge is set on to the work at half taper angle and is fed straight into the work to generate a tapered surface.
- Only tapers of short length can be turned.

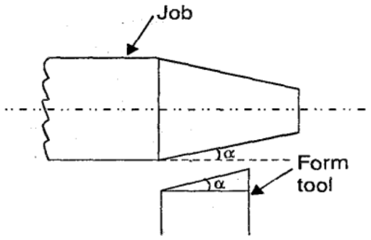


Fig 4.41 taper turning by using a broad nose form tool

Example on Taper problem

1. **By setting compound rest**

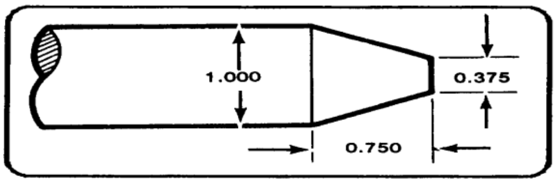


Figure 7-62. Taper problem

- The compound rest setting for the work piece shown in Figure would be calculated in the following manner

TPI = D – d/2L

Angle = Tan^{-1} (TPI)

Where TPI = taper per inch

D = large diameter,

d = small diameter,

L = length of taper

Angle = compound rest setting

The problem is actually worked out by substituting numerical values for the letter variables:

TPI = (1.000 - 0.375)/(2x0.750)=0.4165

Apply the formula to find the angle by substituting the numerical values for the letter variables:

Angle = Tan^{-1} (0.625/2x0.750)

Angle = Tan^{-1} (0.41650)

Using any source of trig charts, the Tan^{-1} of 0.41650 is found to be 22°37'.

This angle is referred to as 22 degrees and 37 minutes.

2. By using the offsetting Tailstock method

a) If the length of the work piece is 65 millimeter and has 3 millimeter diameter difference on tapper portion of 24 millimeter length.

So, the required set over in millimeter(S) calculated as:

$$\text{Tailstock offset} = \frac{D - d}{2 \times 1} \times L$$

Where S = required set over
D = large diameter
d = small diameter
I = length of taper
L = length of the work piece

S=65X (3/2X24) =4.0625

There for, the tail stock offset is 4 millimeter

b) Sometimes metric tapers are expressed as a ratio of 1 mm per unit of length.

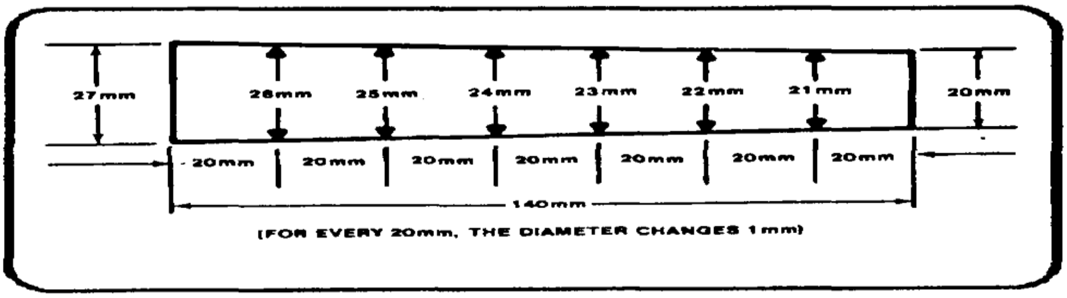


Figure 7-66. Metric taper, 1:200mm.

The work would taper 1 mm in a distance of 20 mm. This taper would then be given as a ratio of 1:20 and would be annotated on small diameter (d) will be 1 mm greater (d + 1).

Refer to the following formula for calculating the dimensions of a metric taper. If the small diameter (d), the unit length of taper (k), and the total length of taper (1) are known, then the large diameter (D) may be calculated. The large diameter (D) will be equal to the small diameter plus the amount of taper.

The amount of taper for the unit length (k) is (d + 1) - (d).
Therefore, the amount of taper per millimeter of unit length = (l/k). The total amount of taper will be the taper per millimeter (l/k) multiplied by the total length of taper (l).

$$\text{Total Taper} = \frac{1 \times 1}{k} \text{ or } \frac{1}{k}$$

$$D = d + \text{total amount of taper}$$

$$D = \frac{d}{k} + \frac{1}{k}$$

For example, to calculate for the large diameter D for a 1:30 taper having a small diameter of 10 mm and a length of 60 mm, do the following:
Since the taper is the ratio 1:30, then (k) = 30, since 30 is the unit of length.

$$D = \frac{d + 1}{k}$$

$$D = \frac{10 + 60}{30}$$

$$D = 10 + 2$$

$$D = 12 \text{ mm}$$

Thus, to determine the tailstock offset in millimeters for the taper in Figure blow, substitute the numbers and solve for the offset. Calculate the tailstock offset required to turn a 1:50 taper 200 mm long on a work piece 800 mm long. The small diameter of the tapered section is 49 mm.

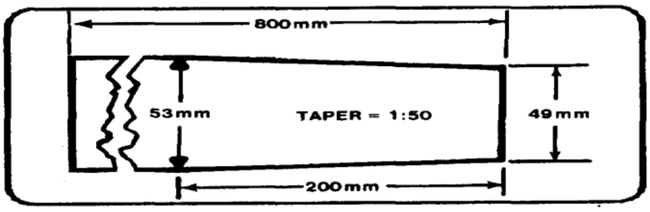


Figure 7-67. Metric taper problem.

$$D = d + \frac{1}{k}$$

$$D = 49 + \frac{200}{50}$$

$$D = 49 + 4 \text{ or } D = 53$$

$$(to) = \frac{53 - 49}{2 \times 200} \times 800$$

$$(to) = \frac{4}{400} \times 800$$

$$(to) = 0.01 \times 800 \text{ or } 8 \text{ mm}$$

So, the tailstock would be moved toward the operator 8 mm.

- Another important consideration in calculating offset is the distance the lathe centers enters the work piece. The length of the work piece (L) should be considered as the distance between the points of the centers for all offset computations.

Therefore, if the centers enter the work piece 1/8 inch on each end and the length of the work piece is 18 inches, subtract 1/4 inch from 18 inches and compute the tailstock offset using 17 3/4 inches as the work piece length (L).

- The amount of taper to be cut will govern the distance the top of the tailstock is offset from the centerline of the lathe.
- The tailstock is adjusted by loosening the clamp nuts, shifting the upper half of the tailstock with the adjusting screws, and then tightening them in place.

There are several methods the operator may use to measure the distance the tailstock has been offset depending upon the accuracy desired.

General operations on the lathe include straight and shoulder turning, facing, grooving, parting, turning tapers, Boring and cutting various screw threads

a) **Facing**: -Reducing the length of a shaft by lathe cutting tool.
b) **Turning;** It`s reducing the diameter of a shaft. It is two kinds:-
   - Straight turn or cylindrical and
   - Tapper turns: - when it increases or decreases in diameter at a uniform rate.

   There are four kinds of tapper making

I. By using form tool:-This technique is limited to the production of short tapers
II. Taper turning with compound rest:

This is given by:-

$\tan\Theta = \frac{D - d}{2l}$ 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
- Soul, $\tan\Theta = \frac{D-d}{2l} = \frac{80-40}{2x40} = 0.$

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

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

Solu.

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

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

III. Taper turning by offset tailstock method:

It 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).or Offset(S) must calculated for each job because the length of the piece plays an important part in the calculation. \

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

Example: - Calculate the <u>tailstock offset</u> for a project having 80mm large diameter and 40 mm small diameter with 40 taper lengths to make its taper using overall length 100mm.

IV. Turning a taper 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.

$\Theta = \frac{GL(D-d)}{2l}$ where . GL = guide bar length

D= diameter of large end

d= diameter of small end

l= length of taper

c) **Parting operations**: Parting is the operation of cutting off material after it has been machined.

d) **Grooving operations**:- it helps to make grooves on a machined part.

e) **KNURLING OPERATION**:

- Knurling 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 workspace 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.
- It is commonly called recessing, undercutting, or necking, is often done at the end of thread to permit full travel of the nut unto a shoulder, or at the edge of a shoulder to ensure a proper fit of mating.

f) Drilling, Boring, Reaming and Counter boring

- **Drilling:**_the operation of producing a hole by removing metal from a solid mass using a cutting tool called a twist drill.
- **Boring:** boring g 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.
- **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.
- .**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.

g) THREAD CUTTING

- Thread cutting on a lathe is a process that produces a helical ridge of uniform section on a work piece.
- This is performed by taking successive cut with a threading tool bit of the same shape as the thread form required.
- Work to be threaded may be held between centers or in a chuck.
- Types of thread: - v-thread, square thread Acme thread etc.

N.B:- Thread on a bolt is external whereas thread on a nut is internal thread.

3.1.2 **FACING OPERATION**

Materials:

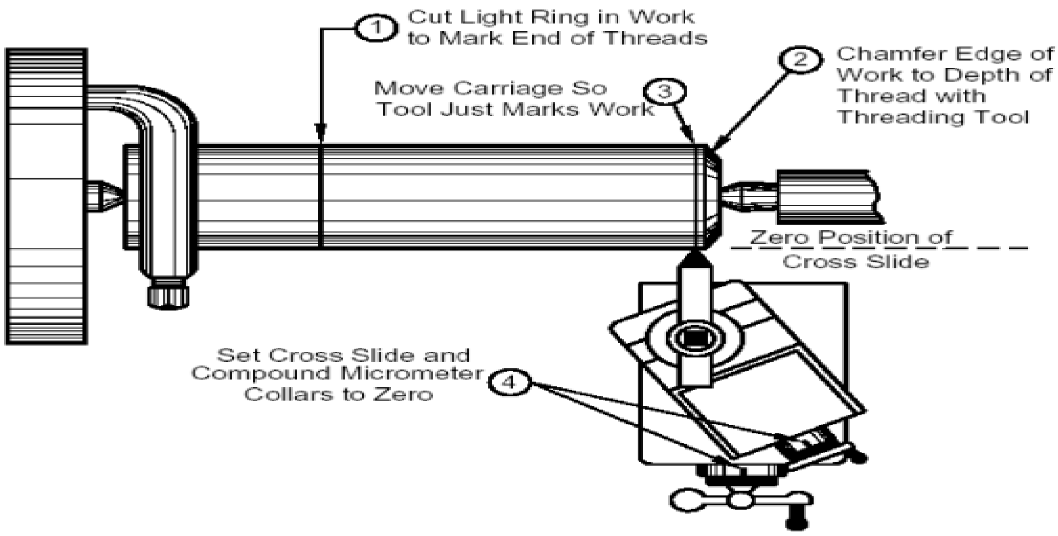
1. Cylindrical stock
2. Chuck key
3. Tool holder
4. Tool bit
5. Lathe machine

Procedure:

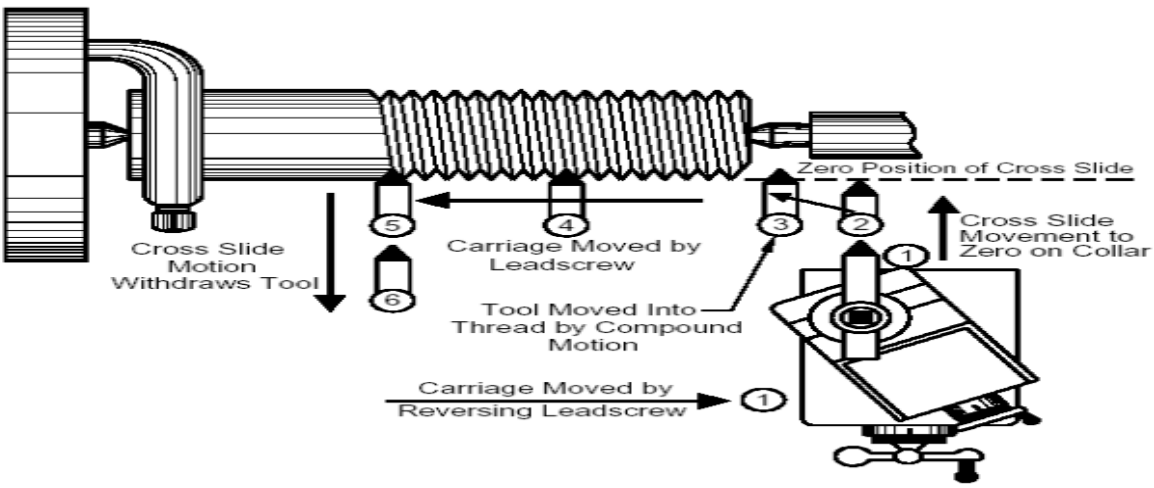
1. Prepare the tools and materials
Secure the tools and materials from the tool keeper
Check and clean them before using
2. Mount the work piece to the chuck.
3. Open the chuck jaws to the approximate size of the work piece diameter.
4. Position the work piece in the chuck and slightly tighten the chuck jaws.
5. Set the compound most at 90 degrees.
6. Position the tool holder, used right cut facing tool.
7. The point of the tool permits a slight clearance between the work faces the center.
8. Bring the cutting tool up until it touches the surface to be machined.
9. Observed safety precaution while performing facing operation used eye goggles to protect eyes from the chips.
10. Feed the center into the work with the compound rest.

CUTTING V-THREAD

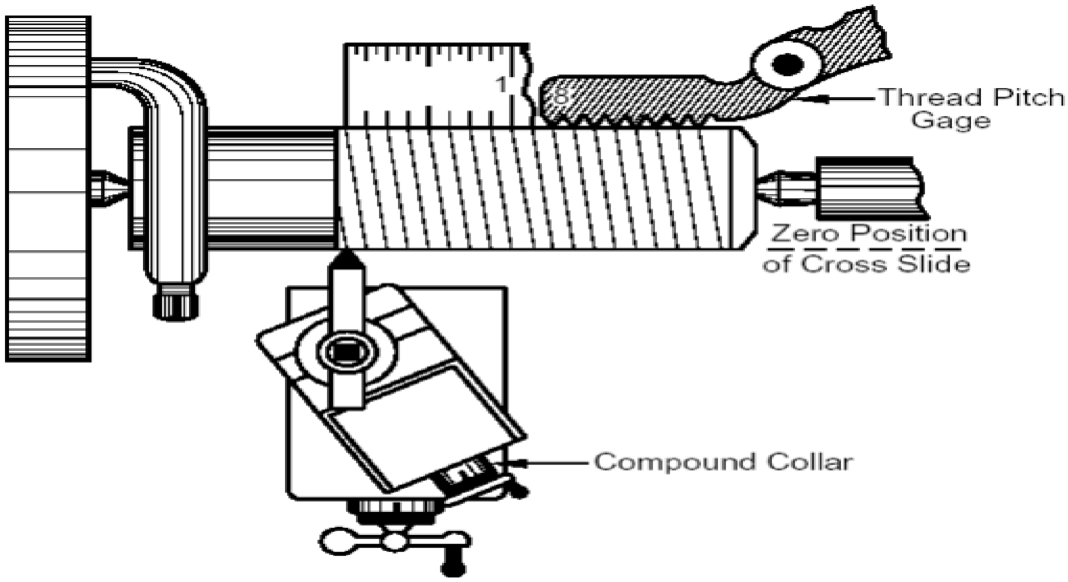
Procedures:



Preparation for threading.



Completing the threading process.



Checking thread pitch after initial threading pass.

3.1.3 Milling MACHINE operations

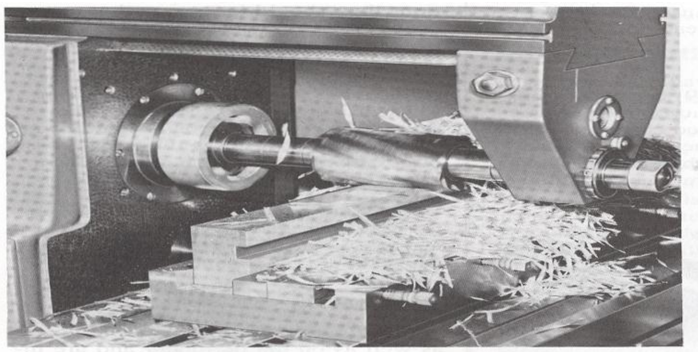
Milling operations may be classified under those general headings as follows:

(1) **Face Milling** - machining flat surfaces which are at right angles to the axis of the cutter.

Face milling cutters, end milling cutters, and side milling cutters are used for face milling operations. The size and nature of the work piece determines the type and size of the cutter required

(2) **Plain or Slab Milling** - machining flat surfaces which are parallel to the axis of the cutter.

Plain or slab milling: is the production of a horizontal flat surface parallel to the axis of the milling machine arbor



For plain milling, the work piece is generally clamped directly to the table or supported in a vise. The milling machine table should be checked for alignment before starting to make a cut. If the work piece surface that is to be milled is at an angle to the base plane of the piece, the work piece should be mounted in a universal vise or an adjustable angle plate. The holding device should be adjusted so that the work piece surface is parallel to the table of the milling machine.

(3) **Angular Milling** - machining flat surfaces which are at an inclination to the axis of the cutter.

Angular milling, or angle milling, is milling flat surfaces which are neither parallel nor perpendicular to the axis of the milling cutter. A single-angle milling cutter is used for this operation. Milling dovetails is a typical example of angular milling. When milling dovetails, the usual angle of the cutter is 45°, 50°, 55°, or 60°, based on common dovetail designs

(4) **Straddle Milling** - When two or more parallel vertical surfaces are machined at a single cut
(5) **Gang Milling** - which two or more milling cutters are used together on one arbor when cutting horizontal surfaces
Gang milling is the term applied to an operation in which two or more milling cutters are used together on one arbor when cutting horizontal surfaces. The usual method is to mount two or more milling cutters of different diameters, shapes and/or widths on an arbor. The possible cutter combinations are unlimited and are determined in each case by the nature of the job

(6) **Form Milling** - machining surfaces having an irregular outline.
Form milling is the process of machining special contours composed of curves and straight lines, or entirely of curves, at a single cut. This is done with formed milling cutters, shaped to the contour to be cut, or with a fly cutter ground for the job.

(7) **Woodruff Keyway Milling** -Keyways are machined grooves of different
(8) **Gear cutting**- Gear teeth are cut using formed milling cutters called involute gear cutters.

Gear cutting: gear cutting includes some specialized machining process which have been developed for the production of various kinds of gears and this specialized machining process is performed on a milling machine through different method

Gear teeth are cut on the milling machine using formed milling cutters called involute gear cutters. These cutters are manufactured in many pitch sizes and shapes for different numbers of teeth per gear.

If involute gear cutters are not available and the teeth must be restored on gears that cannot be replaced, a lathe cutter bit can be ground to the shape of the gear tooth spaces and mounted in a fly cutter for the operation. The gear is milled in the following manner:

NOTE This method of gear cutting is not as accurate as using an involute gear cutter and should be used only for emergency cutting of teeth.

- Fasten the indexing fixture to the milling machine table. Use a mandrel to mount the gear between the index head and the footstock centers. Adjust the indexing fixture on the milling machine table, or adjust the position of the cutter, to make the gear axis perpendicular to the milling machine spindle axis.
- Take the cutter bit that has been ground to the shape of the gear tooth spaces and fasten it in the fly cutter arbor. Adjust the cutter centrally with the axis of the gear. Rotate the milling machine spindle to position the cutter bit in the fly cutter so that its cutting edge is downward.
- Align the tooth space to be cut with the fly cutter arbor and cutter bit by turning the index crank on the index head. Proceed to mill the tooth or teeth in the same manner as you would when milling a keyway.

Table 6.8 Involute Gear milling cutters

Number of Cutter	Will Cut Gears From:	Number of Cutter	Will Cut Gears From:
1	135 teeth to a rack	5	21 to 25 teeth
2	55 to 134 teeth	6	17 to 20 teeth
3	35 to 54 teeth	7	14 to 16 teeth
4	26 to 34 teeth	8	12 to 13 teeth

(The regular cutters listed above are used ordinarily. The cutters listed below (an intermediate series having half-numbers) may be used when greater accuracy of tooth shape is essential in cases where the number of teeth is between the number for which the regular cutters are intended.)

Number of Cutter	Will Cut Gears From:	Number of Cutter	Will Cut Gears From:
1-1/2	80 to 134 teeth	5-1/2	19 to 20 teeth
2-1/2	42 to 54 teeth	6-1/2	15 to 16 teeth
3-1/2	30 to 34 teeth	7-1/2	13 teeth
4-1/2	23 to 25 teeth		

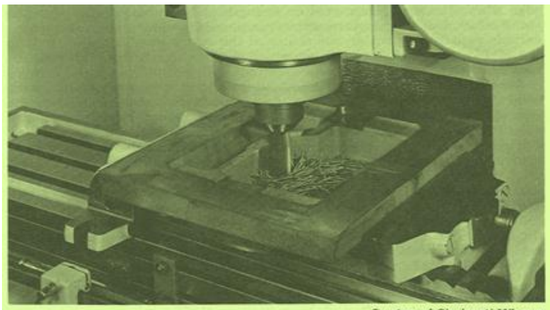
(9) **Drilling**

The milling machine may be used effectively for drilling, since the accurate location of the hole may be secured by means of the feed screw graduations. Spacing holes in a circular path, such as the holes in an indexing plate, may be accomplished by indexing the work piece with the indexing head that is positioned vertically.

(10) **Boring**

Various types of boring tool holders may be used for boring on the milling machine. The boring tool can either be a straight shank, held in chucks and holders, or tapered shanks to fit collets and adapters.

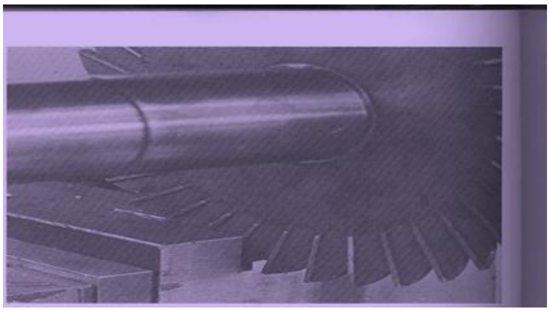
(11) **End milling**:is an operation similar toface milling but usig a much smaller cutter.cutting is done on the of the cutter aswell as on the periphery



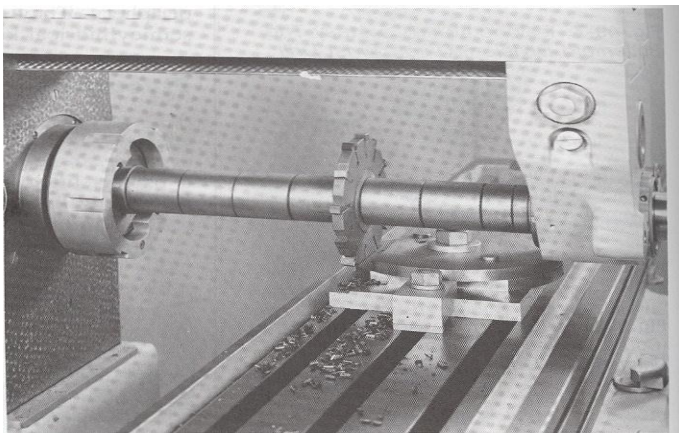
Courtesy of Cincinnati Milacron

(12) slotting : is the process of cutting grooves or slots in the work piece. A staggered-tooth side milling cutter or an end mill can be used for this operation.

(13) sawing: may performed on the milling machin e with a thin metal slitting saw



(14) Side milling: is the process of machining a vertical flat surface perpendicular to the axis of the milling machine arbour. This operation is performed by the combined action of the peripheral and side teeth on a side milling cutter.



Operation on drilling machines

Operation on Drilling machines may be used for performing a variety of operations besides drilling a round hole. A few of the more standard operations, cutting tools and work set-ups will be briefly discussed.

A. Drilling – may be defined as the operation of producing a hole by removing a metal from a solid mass using a cutting tool called a twist drill.

B. Countersinking – is the operation of producing a tapered or cone shaped enlargement to the end of the hole.

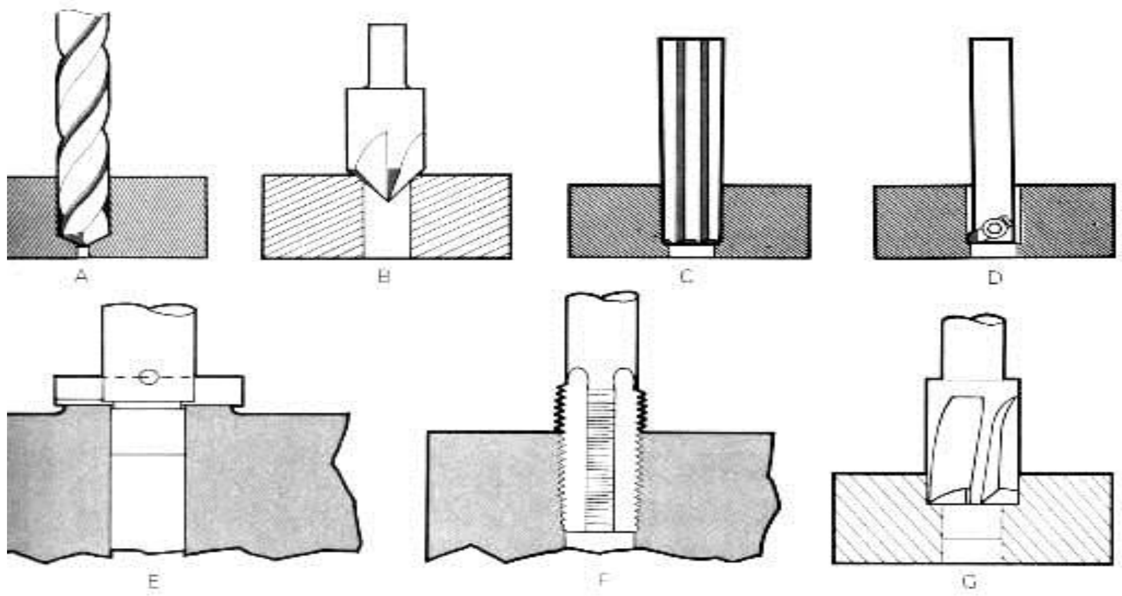
C. Reaming – is the operation of sizing and producing a smooth round hole from a previously drilled or bored hole with the use of a cutting tool having several cutting edges.

D. Boring – is the operation of enlarging and truing a hole by means of a single-point cutting tool which is usually held in a boring bar.

E. Spot-facing – is the operation of smoothing and squaring the surface around a hole to provide a seat for the head of a cap screw or a nut. A boring bar with a pilot section on the end to fit into the existing hole is generally fitted with a double-edged cutting tool. The pilot on the bar provides rigidity for the cutting tool and keeps it concentric with the hole. For the spot facing operation, the work being machined should be securely clamped and the machine set approximately ¼ of the drilling speed.

F. Tapping – is the operation of cutting internal threads in a hole with a cutting tool called a tap. Special machine or gun taps are used with a tapping attachment when this operation is performed by power in a machine.

G. Counter boring – is the operation of enlarging the top of a previously drilled hole to a given depth to provide a square shoulder for the head of a bolt or a cap screw.



Self-Check -3	Written Test

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

1. Which operation always performed fist?

   A. Tapering   B. Facing   C. Turning   D. forming

2. ______is defined as the operation of producing a hole by removing a metal from a solid mass using a cutting tool called a twist drill.

   A. Boring   B. Drilling   C. Countersinking   D. Reaming

3. Reaming is used for______

   A. Drilling work   B. Smoothing work   C. Tapping work   D. Countersinking

4. Name of four accepted methods for machining tapers?

5. Name the steps in tapering operations?

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

Answer Sheet

Score = ____________
Rating: _____________

Name: ____________________________   Date: _________________

Short Answer Questions

Operation Sheet 3	Perform sequence of operation

Direction: Perform procedures of Facing Operation?

State the 7 Steps in Facing Operation

1.
2.
3.
4.
5.
6.
7.

List of Reference Materials

1. Book: Machining and Machine Tools by A. B. Chattopadhyay.
2. Book: Metal Cutting: Theory And Practice by A. Bhattacharya.
3. Book: Geometry of Single-point Turning Tools and Drills – Fundamentals and Practical Applications by V. P. Astakhov.

BASIC METAL WORKS

NTQF Level - I

Learning Guide -41

Unit of Competence: -Operate Basic Workshop Machinery

Module Title:-Operating Basic Workshop Machinery

LG Code: IND BMW1 M12LO1-LG-41

TTLM Code: IND BMW1 TTLM 0919v1

LO2: Set-up machine

Instruction Sheet	Learning Guide 41

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

- **Shop Safety procedures and using personal protective devices**
- **Sharpening and inspecting of tools**
- **Mounting and positioning of tools**
- **Setting and adjusting guards and accessories**
- **Calculating cutting speeds, RPM, feeds and depth of cuts**
- **Performing setup operations** Specifically, upon completion of this Learning Guide, you will be able to –
- **using Shop Safety procedures and personal protective devices**
- **Inspect and sharpen tools if necessary according to the work requirements**
- **mount and position tools within machine specifications**
- **set and adjust guards and accessories as required**
- **Calculate cutting speeds RPM feeds and depth of cuts using appropriate techniques and reference material.**
- **setup operations**

Learning Instructions:

14. Read the specific objectives of this Learning Guide.
15. Follow the instructions described in number 3 to 20.
16. Read the information written in the "Information Sheets 1". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
17. Accomplish the "Self-check 1" in page 7.
18. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 1).
19. If you earned a satisfactory evaluation proceed to "Information Sheet 2". However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
20. Submit your accomplished Self-check. This will form part of your training portfolio.

21. Read the information written in the “Information Sheet 2”. Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
22. Accomplish the “Self-check 2” in page 10.
23. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 2).
24. Read the information written in the “Information Sheets 3 . Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
25. Accomplish the “Self-check 3” in page 19.
26. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 3).
27. Accomplish the “Self-check 4” in page 22 and “Self-check 5” in page24.
28. If you earned a satisfactory evaluation proceed to “Operation Sheet 1” in page 25. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
29. Read the “Operation Sheet 1” and try to understand the procedures discussed.
30. If you earned a satisfactory evaluation proceed to “Operation Sheet 2” in page 26. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
31. Read the “Operation Sheet 2” and try to understand the procedures discussed.
32. If you earned a satisfactory evaluation proceed to “Operation Sheet 3” in page 27. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
33. Do the “LAP test” in page 28 (if you are ready). Request your teacher to evaluate your performance and outputs. Your teacher will give you feedback and the evaluation will be either satisfactory or unsatisfactory. If unsatisfactory, your teacher shall advice you on additional work.

Information Sheet-1	Shop Safety procedures and using personal protective devices

1.1. Safety Measures in machine shop

As always we should be aware of safety requirements and attempt to observe safety rules in order to eliminate serious injury to ourselves or others.

Wear glasses, short sleeves, no tie, no rings, no trying to stop the work by hand. Stop the machine before trying to check the work. Don't know how it works? –"Don't run it." Don't use rags when the machine is running.

Unguarded moving parts of machines/equipment and the sudden or uncontrolled release of their power systems can result in serious injuries.

Personnel working with machines must be aware of the risks involved and follow safe work practices.

1.2. Causes of accidents while working with machinery

- Loose clothing, hair, jewelry being caught in moving parts.
- Materials ejected from the machine when it is operational.
- Unplanned starting of the machine.
- Slipping and falling into an unguarded nip.
- Contact with sharp edges, e.g., cutting blade.
- Making adjustments while the machine is operational.
- Unauthorized operation of machines.
- Lack of preventive maintenance.

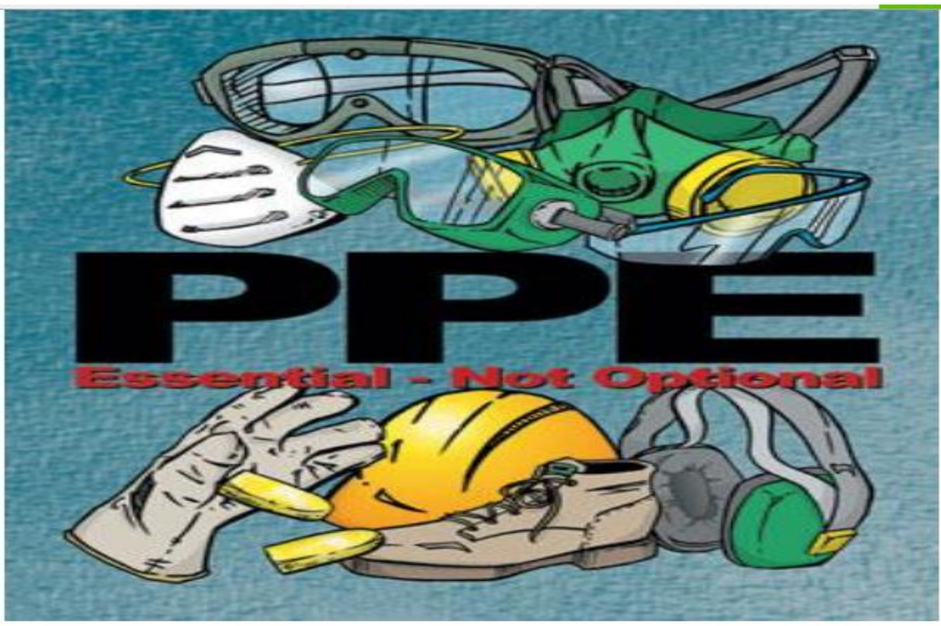
1.3 Safety when Machining Work Piece

- Do not operate any machine before understanding its mechanism.
- Always stop a machine before measuring, cleaning or making any adjustments. It is dangerous to do any type of work around moving parts of a machine.
- Never operate a machine unless all safety guards are in place.
- Keep hands away from moving parts. It is dangerous practice to "feel" the surface of the revolving work or to stop a machine by hand
- Never use a rag near the moving parts of a machine rag may be drawn into the machine, along with the hand that is holding it.
- Never have more than one person's operate a machine at the same time. Not knowing what the other person would or would not do has caused many accidents

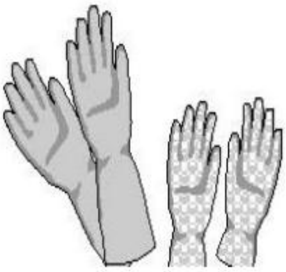
1.4 Personal safety

- Dress appropriately removes 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 you 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
- Keep hand tools in good conditions and store them in such away that peoples can not be injured when a tool is taken from a tool panel or storage rack.
- Avoid horse play



Eye and Face Protection Selection Chart

Hand protection saves you from the following hazards in the machine shop:

1. Skin absorption of harmful substances
2. Severe cuts or lacerations
3. Punctures
4. Chemical burns
5. Irritating materials
6. Thermal burns
7. Harmful temperature extremes

Safety Boots or Shoes

Earmuffs and Earpieces

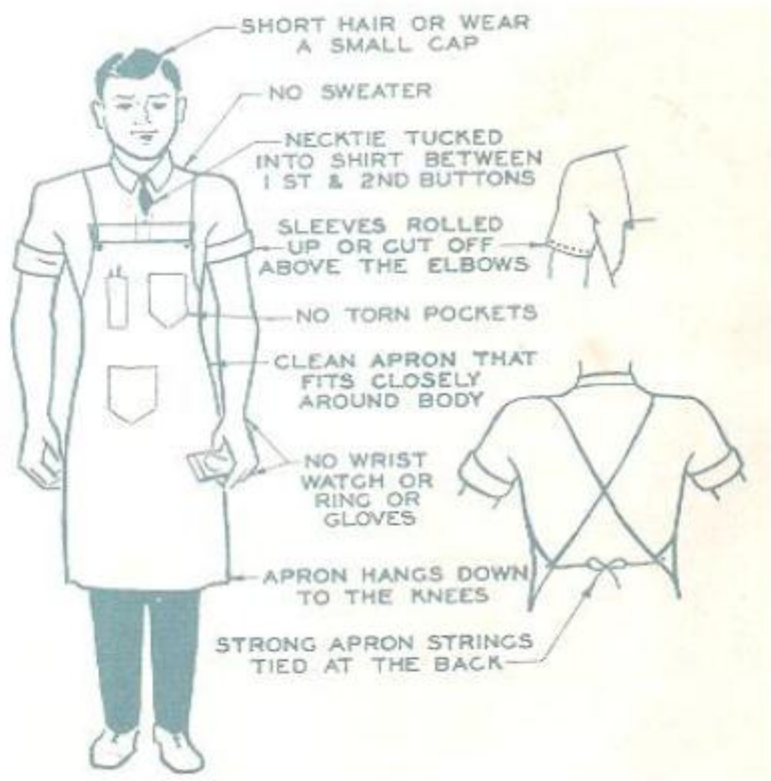


Fig.3-1-5

Important details to be inspected in a lathe machine

1.5 RULES IN MACHINE SHOP

1. Keep floors free of oil, grease, or any other liquid. Clean up spilled liquids immediately. They are slipping hazards.
2. Aisles should be clear at all times to avoid tripping or other accidents.
3. Store materials in such a way that they cannot become tripping hazards.
4. Do not leave tools or work on the table of machine even if the machine is not running. Tools or work may fall off and cause toe or foot injury.
5. Put tools away when not in use.
6. Place all scrap in scrap boxes.

1.6 Machine safety

a. 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.
b. Avoid using compressed air to remove chips and cutting oil from machine.
c. Keep the machine clear of tools
d. 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.

Self-Check -1	Written Test

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

1. Write five points that Causes of accidents while working with machinery?
2. Write three work shop safety rules?
3. Write three types of personal work shop safety rules?

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

Answer Sheet

Score = ____________
Rating: ____________

Name: ___________________________ Date: _______________

Short Answer Questions

Information Sheet-2	Sharpening and inspecting of tools

2.1 Sharpening HSS Lathe Tool Bits

When you purchase a new lathe tool bit, it might have an angle on the end, but it is not properly sharpened for turning. Grinding lathe tool bits is a bit of an art. It takes some practice to get good at it. You need to create a cutting edge that is sharp, extends out so that the cutting edge and not the side of the tool contacts the work, but that still has enough support to maintain sufficient strength to cut metal. Before diving in, there are some terms you need to understand. The illustration below shows these terms.

First, notice that there are two cutting edges on the tool bit. There is a cutting edge on the end of the tool bit called the **front cutting edge**. There is also a cutting edge on the side of the tool. Between these cutting edges is a rounded section of cutting edge called the nose?

There are typical rake and clearance (relief) angles for HSS tool bits.

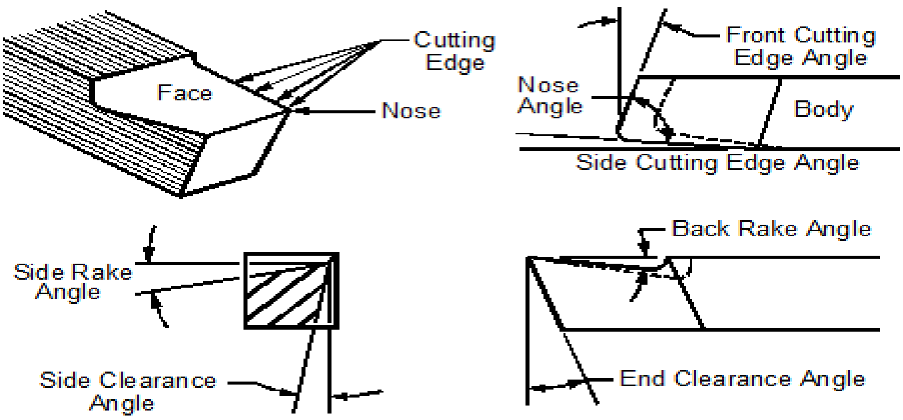


Figure 2.1. Tool bit nomenclature.

Sharpening cutting tools: Almost all milling cutters including straight and helical teeth are able to Be reconditioned using tool and cutter grinding machine.

Procedures for grinding a plain milling cutter:

A. selects the correct wheel for the job. True it with a diamond tool.

B. mounts the cutter on a suitable arbor and places the unit between centers. **Fig 13-48**

C. mounts the tooth rest to the wheel head. Position the edge about 6 mm above the center line of the grinding wheel, **fig. 13-49.** This will produce a 5 to 6 degree clearance angle on the cutting edge of a 1.5 mm diameter cutter. Adjust to suit the cutter being ground.

D. the setup should permit the wheel to grind away from the tooth cutting edge, while requiring more machining care than grinding into a cutting edge of the tooth there is Less chance of drawing the temper. Also, no burr is formed that must be oil stoned off to secure a sharp edge.

E. flaring cup wheels are also used for cutter and tool grinding. Since there is a greater area of contact when using flare cup wheels, ***lighter cuts*** should be taken than with straight grinding wheels.

F. start the machine and feed the cutter into the wheel. Take a light cut. A bit of thinned layout bluing should be applied to the back of the tooth. This will allow a visual check of how the grinding operation is progressing and whether the setup is producing the proper clearance angle.

g. When satisfied with the setup, bring the next tooth into position on the tooth rest and grind that tooth.

h. Repeat the operation until all of the teeth are sharpened. Make necessary adjustment to assure tooth concentricity (cutting surfaces of all teeth are some distance from arbor hole center line.)

After a cutter has been sharpened several times, the clearance angle ***flat*** (land) will become too wide. Then, it becomes necessary to grind in a secondary clearance. If it becomes apparent that more material is being removed from some teeth than others, a quick check must be made to determine the cause:

i) The grinding wheel may be too soft and wear down too rapidly. As the wheel wears, less material is removed from the cutter tooth.

ii) The tooth rest may NOT be mounted solidly and moves during the grinding operation.

iii) The arbor may NOT be running true on the centers. Test arbor run out with an indicator as it is rotated. When the trouble has been ***pin pointed,*** make the necessary corrections and continue the operation.

Grinding end mills

End mills are sharpened in much the same way as helical teeth cutters with the end mill mounted in a work head rather than between centers. The end teeth are sharpened with the same technique used to sharpen the side teeth on a side milling cutter.

Grinding form cutters

Form tooth cutters must be ground radially to preserve the tooth shape, An index disc may be employed or a special form cutter grinder may be utilized.

Grinding Taps

A universal tool and cutter grinder may also be used to re sharpen taps. Normally a tap becomes dull when the leading edges of the starting chamfer become worn. The chamfer can be reground by mounting the tap in a work head**.** Flutes are reground with a straight wheel with an edge that has been shaped to fit the flutes.

2.1.1 How to Grind Tool Bits

Use a bench grinder to sharpen your tool bits. Even an inexpensive bench grinder can do a good job grinding lathe tool bits. In some cases, you might want to purchase a higher quality fine grit wheel. Keep a small cup of water near your grinder. Grinding generates heat, which can cause two problems. The tool bit will become too hot to hold. Overheating can also affect the heat treatment of the tool bit, leaving the cutting edge soft.

Use a protractor to measure the angles. They are not super-critical, but you should try to stay within one degree of the recommendations.

a. Grind the Front Relief

The first step in creating a tool bit is to grind the front relief. For most work, a relief angle of 10° works well.

While you are grinding the front relief, you are also creating the front cutting edge angle. Make this angle about 10° also, so that the corner formed by the front cutting edge and the side cutting edge is less than 90°.

b. Grind the Left Side Relief

Form the left side relief next. Again, create about a 10° angle. You don't need to form a side cutting angle. The side cutting edge can be parallel to the side of the tool blank.

c. Grind the Top Rake

The top of the tool bit is ground at an angle that combines the back rake and the side rake. The side rake is most important, because the side cutting edge does most of the work. For cutting steel and aluminum, the side rake should be about 12° and the back rake should be about 8°. For cutting brass, the rake angles should be much less, or even 0°.

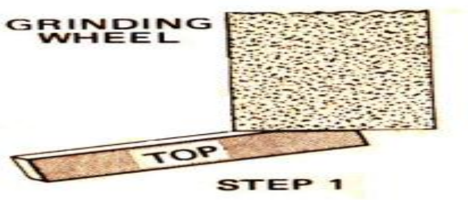
d. Round the Nose

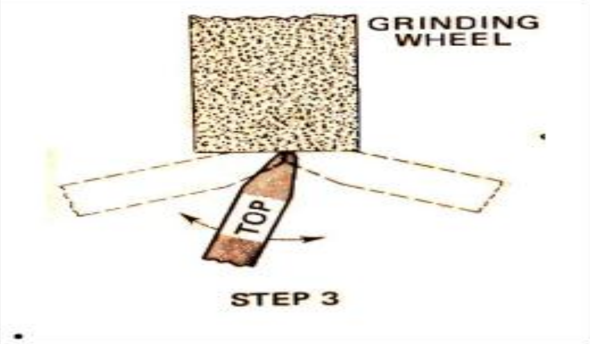
A small nose radius allows you to turn into tight corners. A large nose radius produces better surface finishes. Create a nose radius that is appropriate for the tool bit you are creating.

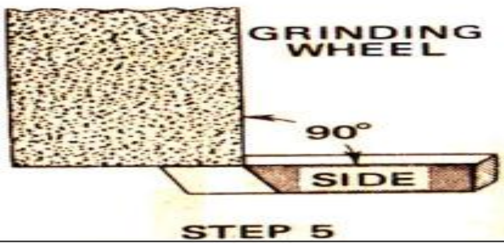
Relief and Rake Angles for Cutting Common Metals

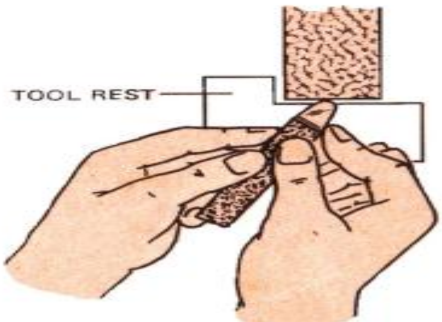
Material	Side Relief	Front Relief	Side Rake	Back Rake
Aluminum	12°	8°	15°	35°
Brass	10°	8°	5° to -4°	0°
Bronze	10°	8°	5° to -4°	0°
Cast iron	10°	8°	12°	5°
Copper	12°	10°	20°	16°
Machine Steel	10° to 12°	8°	12° to 18°	8° to 15°
Tool Steel	10°	8°	12°	8°
Stainless Steel	10°	8°	15° to 20°	8°

Suggested grinding procedures for round nose side cutting tool bit

Step 1. Grind the left relief angle. 8° to 15° depending on the diameter of the workpiece	
Step 2. Grind the right side – relief angle the same as Step 1	

Step 3. **Grinding a round nose**	

	**Step 4.** **Hold the toolbit at an angle 8° – 15° in order to get the end relief angle.**
	**Step 5.** **Grind the rake angle for right hand tool bit 6° to 15°**

	**NOTE:** **This is the correct way to grip a toolbit when grinding.**

CAUTION; ALWAYS WEAR GOGGLES FOR EYE PROTECTION WHEN PERFORMING GRINDING

2.1.2 Cutting Tool Terminology

There are many different tools that can be used for turning, facing, and parting operations on the lathe. Each tool is usually composed of carbide as a base material, but can include other compounds. This section covers the different appearances and uses of lathe cutting tools.

Figure A: shows a standard turning tool to create a semi-square shoulder. If there is enough material behind the cutting edge, the tool can also be used for roughing.

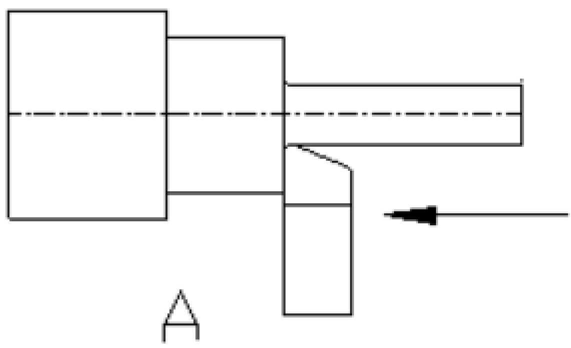


Figure B: shows a standard turning tool with a lead angle. This angle enables for heavy roughing cuts. It also possible to turn the tool to create a semi-square shoulder.

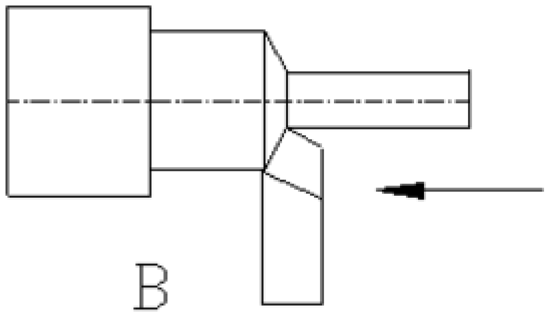


Figure C: nose has a very large radius, which helps with fine finishes on both light and heavy cuts. The tool can also be used to form a corner radius.

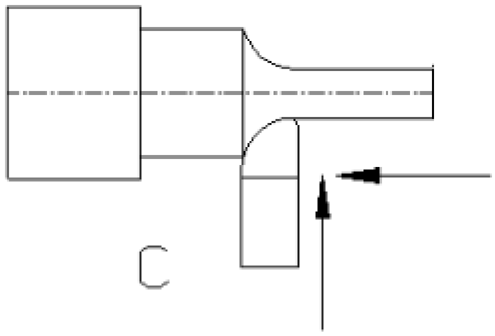


Figure D: shows a rotated standard turning tool. Its nose leads the cutting edge to create light finishing cuts on the outside diameter and face of the shoulder.

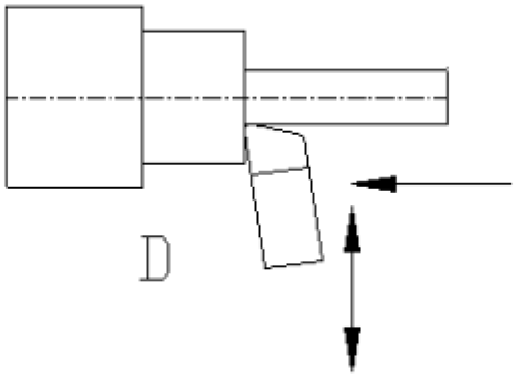


Figure E: shows a form tool. Different forms can be ground into the tool, which will be reproduced onto the part.

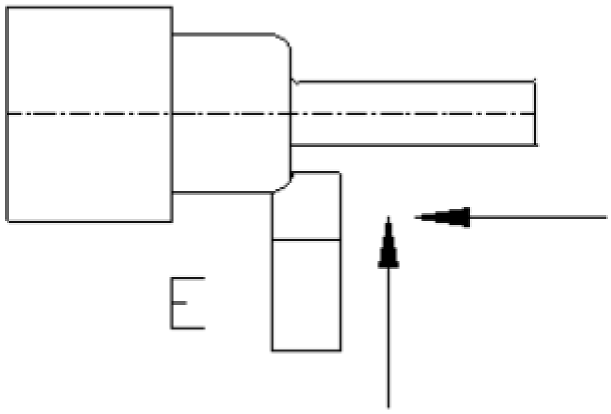


Figure F: shows a facing tool. This cutter is used to face the end of a work piece to provide for a smooth, flat finish. If the stock has a hole in the center, utilize a half-center to stabilize and support the work piece.

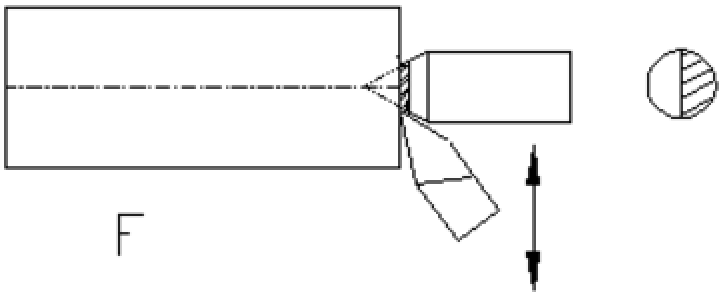
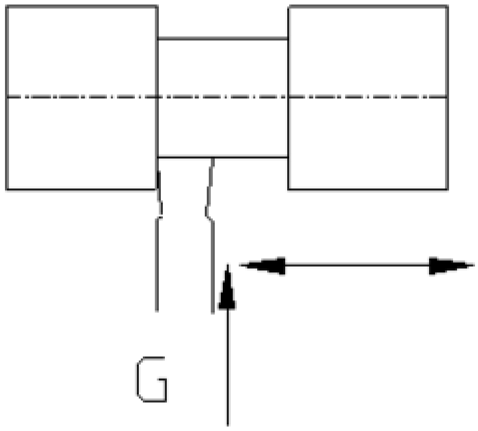


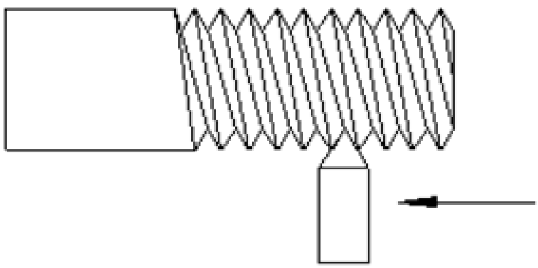
Figure G: shows a grooving or under-cutting tool. As shown, it is used to cut grooves into the work piece. When there are proper clearances, the tool can cut deeply, or cut to the left or right.



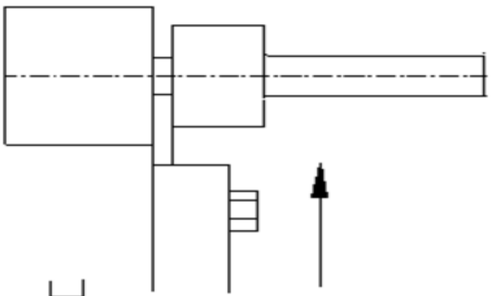
G

Figure H: shows a parting tool. Parting tools cut off the stock at a certain length. This tool requires a preformed blade and holder.

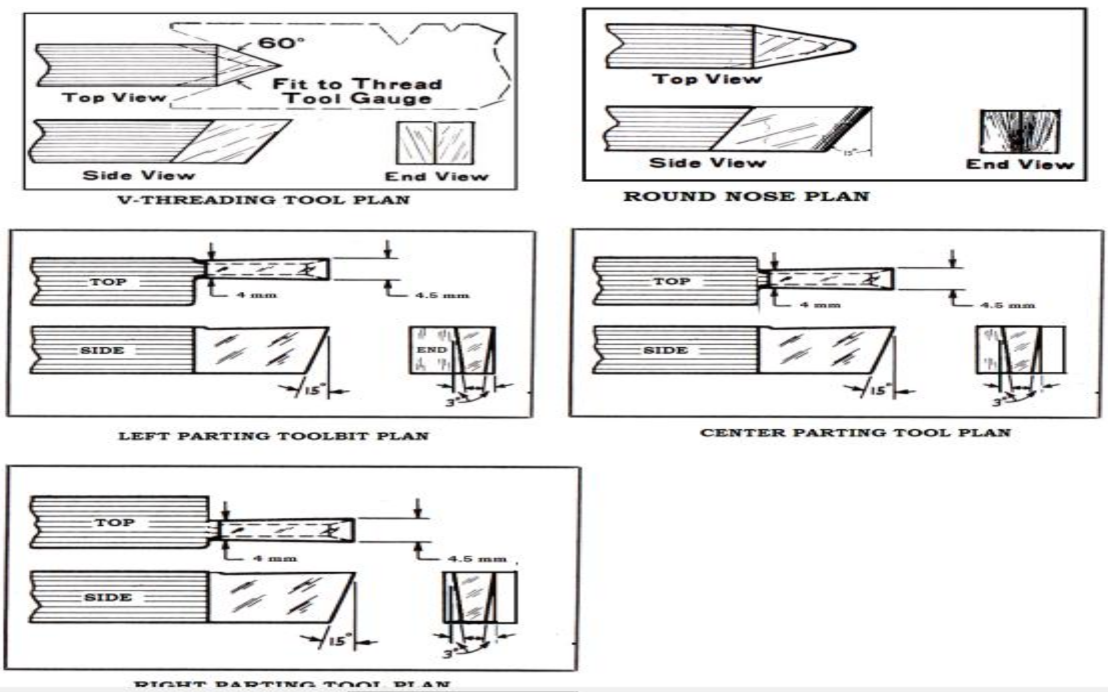
Figure I: shows a 60° threading tool used to thread stock.



I



H



Self-Check -2	Choose

Directions: Answer all the questions listed below.

1. Write the two cutting edges on the end of the tool bit? (2 Points)
2. A large nose radius produces better for _____?(1point)

   A. Cutting large amount B. Good surface finish C. Rough cutting D. All
3. List the six angles of turning tools? (6pints)
4. A 60° angle cutting tool used for___? (1point)

A. Parting B. Threading C. Grooving D. Turning

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

Answer Sheet

Score = ___________

Rating: ____________

Name: _________________________ Date: _______________

Short Answer Questions

Information Sheet-3	**Mounting and positioning of tools**

3.1 Mounting the cutting tools in machine tools

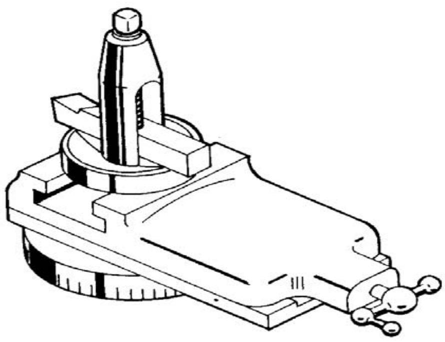
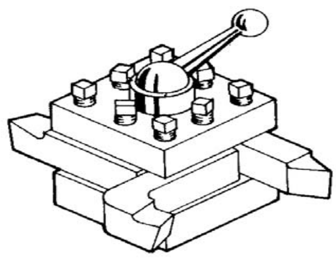
1. Appropriate selection of tool holder and the method of mounting
2. Proper positioning and orientation of the tool depending upon its
    - type
    - size and shape
    - geometry and it should also;

- Proper alignment in respect of coaxially, concentricity and machine tool configuration
- Accurate and quick locating, strong support and rigid clamping
- Minimization of run out and deflection during cutting operation easy and quick mounting and change
- Unobstructed (free) chip flow and cutting fluid action.

3.2 Mounting of tools in lathes

Different types of tools which are used in lathes are usually mounted in the following ways.

2. HSS tools (Shank type) in the tool post.
3. HSS form tools and threading tools in tool post
4. Carbide and ceramic inserts in standard tool-holders
5. Drills and reamers, if required, in the tailstock
6. Boring tools in tool post

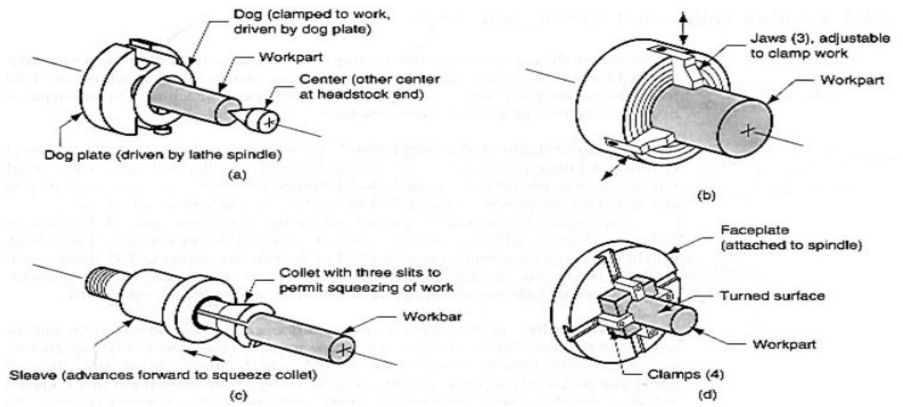



3.3 Mounting of work in lathes

Work is held or mounted in the lathe with a number of methods,

- **Between two *centers***. The work piece is driven by a device called a *dog*; the method is suitable for parts with high *length-to-diameter ratio*.
- **A *3 jaw self-centering chuck*** is used for most operations on cylindrical work parts. For parts with high *length-to-diameter ratio* the part is supported by center on the other end.

- ***Collet*** **c**onsists of tubular bushing with longitudinal slits. Collets are used to grasp and hold bar stock. A collet of exact diameter is required to match any bar stock diameter.
- **A *face plate*** is a device used to grasp parts with irregular shapes:



Four work holding methods used in lathes: (a) mounting the work between centers using a dog, (b) three-jaw chuck, (c) collet, and (d) face plate for noncylindrical workparts.

3.4 Mounting a Work on milling machine

A. General.

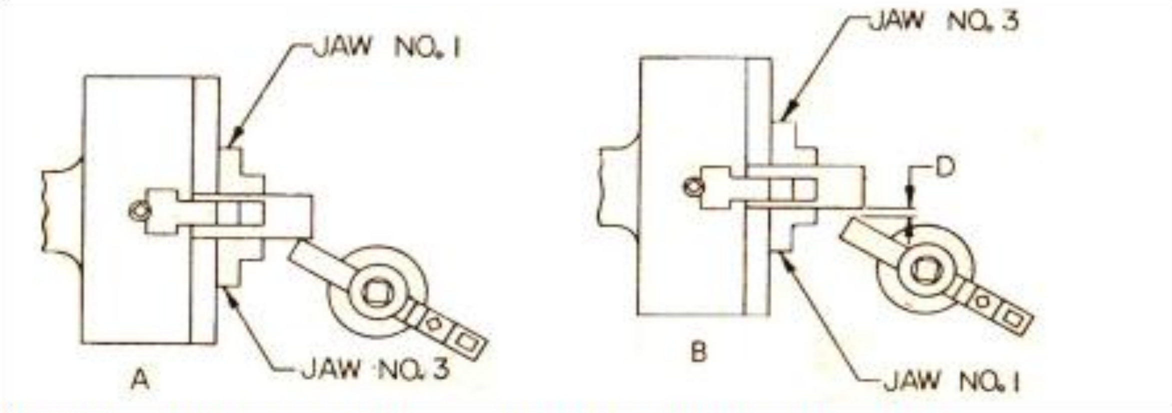
- An efficient and positive method of holding work pieces to the milling machine table is essential if the machine tool is to be used to advantage. Regardless of the method used in holding, there are certain factors that should be observed in every case. The work piece must not be sprung in clamping; it must be secured to prevent it from springing or moving away from the cutter; and it must be so aligned that it may be correctly machined.
- Milling machine worktables are provided with several T-slots, used either for clamping and locating the work piece itself or for mounting various holding devices and attachments. These T-slots extend the length of the table and are parallel to its line of travel. Most milling machine attachments, such as vises and index fixtures, have keys or tongues on the underside of their bases so that they may be located correctly in relation to the T-slots.

b. Methods of Mounting Work pieces.

(1) Clamping a Work piece To the Table.
(2) Clasping a Work piece to the Angle Plate.
(3) Clamping Work pieces in Fixtures.
(4) Holding Work pieces Between Centers.
(5) Holding Work pieces in a Chuck.
(6) Holding Work pieces in the Vise.

MOUNTING AND CENTERING OF WORKPIECE ON THE LATHE

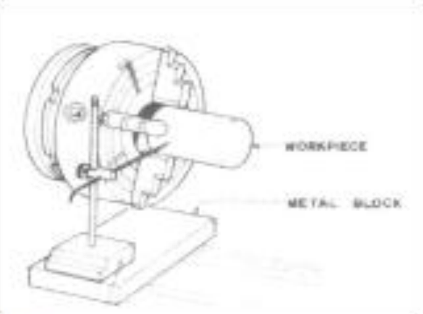
Mounting Work in a Four Jaw Independent Chuck
Visual Inspection Approach
Using High Spot – Low Spot Technique

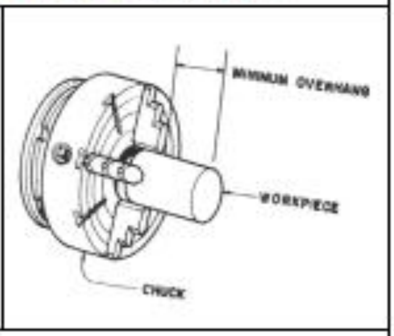


(A) High Spot is the highest point that touches the point of any centering devices.
(B) Low Spot is the gap created as it moves away from the workpiece
Illustration show the back of a loose tool holder as a means to determine equidistance

Visual Estimate Inspection Method

1. Chalk method /Surface gage /wriggler method/Alignment of Chuck's jaw





NOTE: Do visual inspection by;

1. Loosen up the jaw opposite of the high spot and screw tight the high spot's jaw to push the workpiece
2. Having plenty of light while using scratch white paper as background distinguisher between workpiece gap and the point of a centering devices
3. Repeating the procedures until an equal gap was visible in four front of the chuck's jaw. Tighten the high spot area if gap in the low spot equal 0.5mm.
4. Checking for wobble spin and remedied it by light tapping

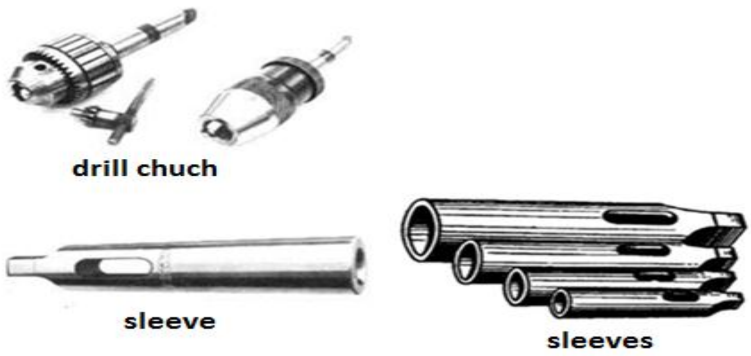
3.5. Tool holding devices

The drill press spindle provides the means for holding and driving the cutting tool. It may have tapered hole to accommodate tapered shank tools or its end may be tapered or threaded for mounting a drill chuck. Although there is a variety of tool holding devices and accessories, the most commonly found in a machine shop are drill chucks, drill sleeves and drill sockets.

- **Drill chucks** are the most common devices used on a drill press for holding straight –shank cutting tools.
- ***Drill sleeves*** are used to adapt the cutting tool shank to the machine spindle and if the taper on the cutting tool is smaller than the tapered hole in the spindle.
- A ***drill socket***s used when the hole in the spindle of the drill press is too small for the taper shank of the drill.

a. Mounting of tools in Drilling machines

Small straight shank type solid HSS and carbide drill are held in a drill chuck which is fitted in the drill spindle at its taper bore.



Cutter holding devices in milling machines are:

- **Arbors -** used for mounting the milling cutter, are inserted and held in the main spindle by a draw bolter a special quick - change adapter.
- **Shell-end mill arbors** - It may fit in to the main spindle or the spindle of the vertical attachment which permit face milling to be done either horizontally or vertically.
- **Collet adapters** - are used for mounting drills or other tapered - shank tools in the main spindle of the machine or the vertical milling attachment.

Arbor, Adapter,shell end mill arbor, sleeve

- **A quick-change adapter** - mounted in the spindle, & permits drilling, boring, and milling operations without a change in the setup of the work piece.

3.6 WORK HOLDING DEVICES

Many different devices, such as chucks, collets, faceplates, drive plates, mandrels, and lathe centers, are used to hold and drive the work while it is being machined on a lathe. The size and type of work to be machined and the particular operation that needs to be done will determine which work holding device is best for any particular job. Another consideration is how much accuracy is needed for a job, since some work holding devices are more accurate than others. Operational details for some of the more common work holding devices the universal scroll chuck, Figure 7-19, usually has three jaws which move in unison as an adjusting pinion is rotated. The advantage of the universal scroll chuck is its ease of operation in centering work for concentric turning. This chuck is not as accurate as the independent chuck, but when in good condition it will center the jaws are moved simultaneously within the chuck by a scroll or spiral-threaded plate. The jaws are threaded to the scroll and move an equal distance inward or outward as the scroll is rotated by the adjusting pinion. Since the jaws are individually aligned on the scroll, the jaws cannot usually be reversed. Some manufactures supply two sets of jaws, one for internal work and one for external work. Other manufactures make the jaws in two pieces so the outside or gripping surface may be reversed. Which can be interchanged? r work within 0.002 The universal scroll chuck can be used to hold and automatically center round or hexagonal workplaces. Having only three jaws, the chuck cannot be used effectively to hold square, octagonal, or irregular shapes. to The independent chuck, Figure 7-19, generally has four jaws which are adjusted individually on the chuck face by means of adjusting screws. The chuck face is scribed with concentric circles which are used for rough alignment of the jaws when chucking round workplaces. The final adjustment is made by turning the work piece slowly by hand and using a dial indicator to determine its concentricity. The jaws are then readjust The jaws of the independent chuck may be used as illustrated or may be reversed so that the steps face in the

Opposite direction; thus workplaces can be gripped either externally or internally. The independent chuck can be used to hold square, round, octagonal, or irregularly shaped workplaces in either a concentric or eccentric position due to the independent operation of each jaw.

Because of its versatility and capacity for fine adjustment, the independent chuck is commonly used for mounting odd shaped workplaces which must be held with extreme accuracy.

The drill chuck, Figure 7-19, is a small universal chuck which can be used in either the headstock spindle or the Tail stock for holding straight-shank drills, reamers, taps, or small diameter workplaces. The drill chuck has three or four hardened steel jaws which are moved together or apart by adjusting a tapered sleeve within which they are contained. The drill chuck is capable of centering tools and small diameter workplaces to within 0.002 or 0.003 inch when firmly tightened.

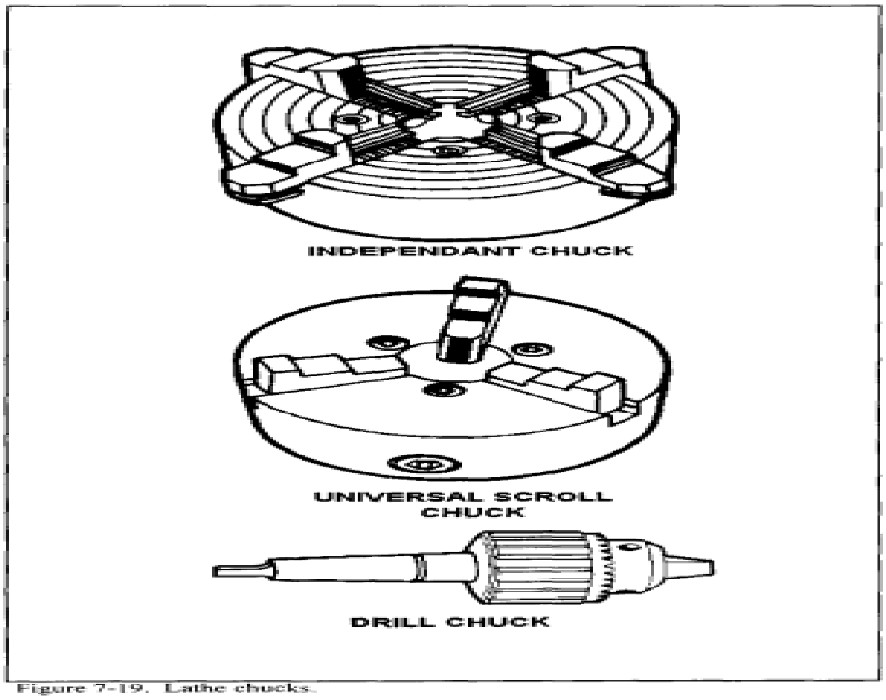


Figure 7-19. Lathe chucks.

3.7 THE COLLET CHUCK IS THE MOST ACCURATE MEANS OF HOLDING SMALL WORKPIECES IN THE LATHE

The collet chuck is the most accurate means of holding small workplaces in the lathe. The collet chuck consists of a spring machine collet (Figure 7-20) and a collet attachment which secures and regulates the collet on the headstock spindle of the lathe.

The spring machine collet is a thin metal bushing with an accurately machined bore and a tapered exterior. The collet has three lengthwise slots to permit its sides being sprung slightly inward to grip the work piece. To grip the work piece accurately, the collet must be no more than 0.005 inch larger or smaller than the diameter of the piece to be chucked. For this reason, spring machine collets are available in increments of 1/64 inch. For general purposes, the spring machine collets are limited in capacity to 1 1/8 inch in diameter.

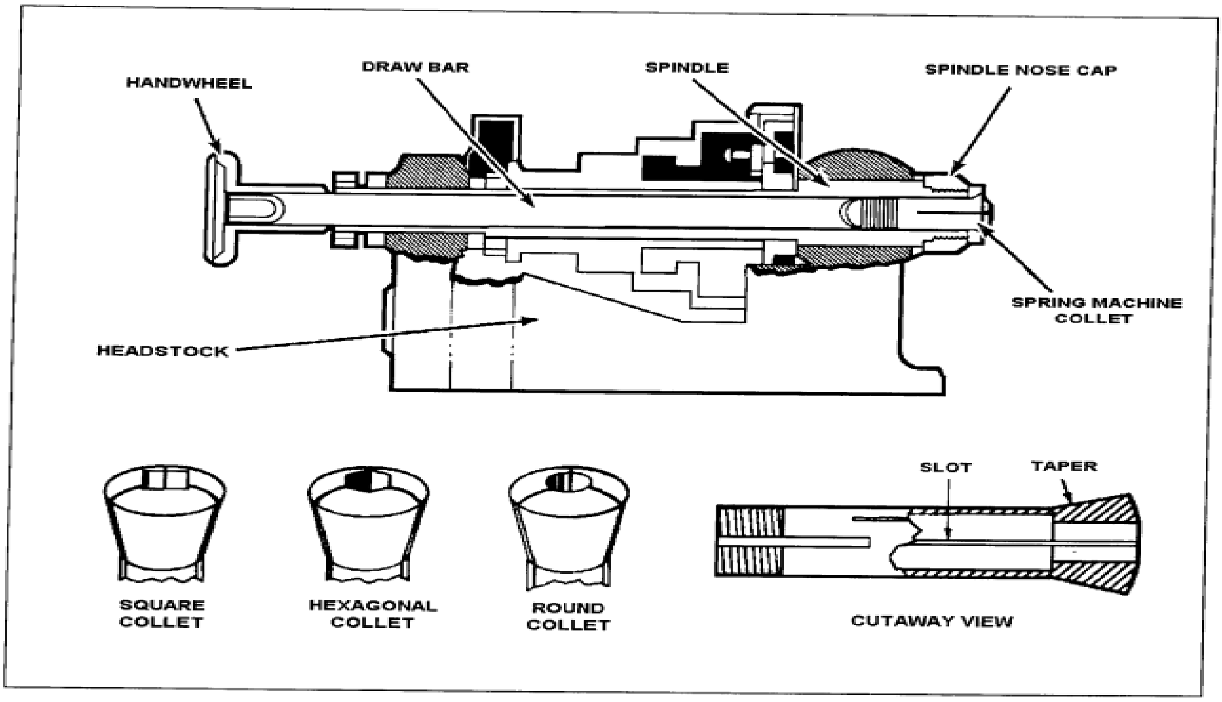


Figure 7-20. Spring machine collet chucks and installation method.

3.8 LAYING OUT AND MOUNTING WORK

There is relatively little layout work to be done for most lathe work because of the lathe's ability to guide the cutting tool accurately to the work piece. If center holes must be located and drilled into the end of a work piece for turning lay out and center-punch the work piece using other methods. Some suggested methods are to use a bell-type center punch between centers and this cannot be accomplished on the lathe, (Figure 7-32), use hermaphrodite calipers to scribe intersecting arcs, use the centering head of the combination square, or use dividers (Figure 7-33).

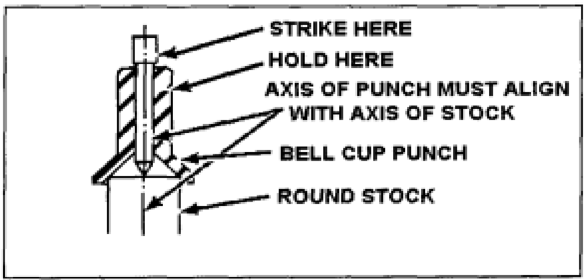


Figure 7-32. Bell-type center punch.

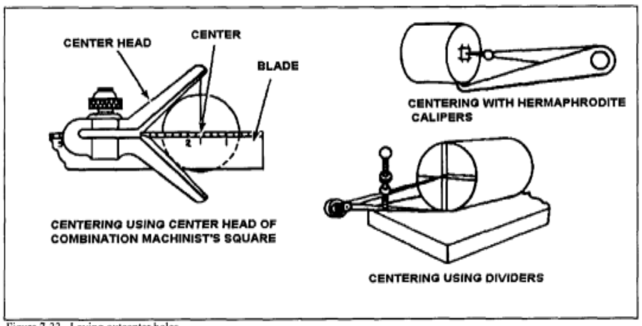


Figure 7-33. Laying outcenter holes.

3.9 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 (Figure 7-34).

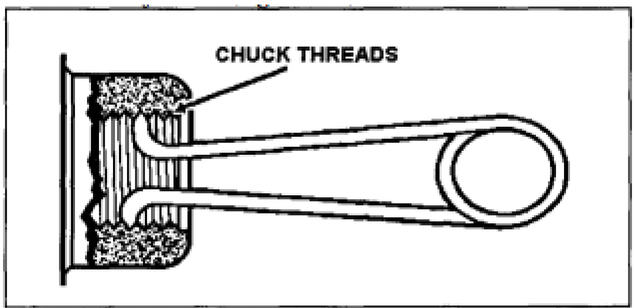


Figure 7-34. Spring thread cleaner.

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, collet 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 (Figure 7-35). 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.

Loosen the jaw opposite and tighten the jaw where the chalk marks are found. Repeat the process until the work piece is satisfactorily aligned.

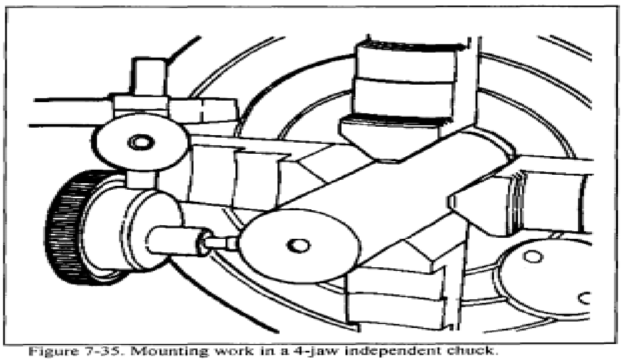


Figure 7-35. Mounting work in a 4-jaw independent chuck.

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.

3.9.1 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 7-36). 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

3.9.2 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 7-37 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.

Mount the work in a universal or independent chuck and mount the center drill in the lathe tailstock (Figure 7-38). 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 (Figure 7-39). 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 speed the work must be removed from the chuck and the point and feed them into the work slowly to avoid breaking off the drill point inside the work. If this happens, extracted. This is a time-consuming job and could ruin the work piece.

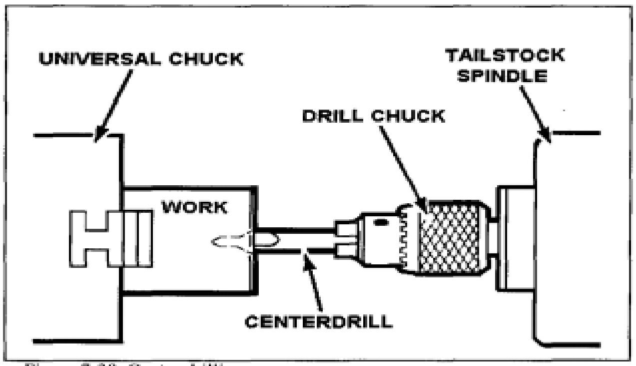


Figure 7-38. Center drilling.

To mount work between centers, the operator must know how to insert and remove lathe centers. The quality of workmanship depends as much on the condition of the lathe centers as on the proper drilling of the center holes. Install the lathe center in the tailstock spindle with a light twisting motion to ensure a clean fit. Install the center sleeve into the headstock spindle and install the lathe center into the center sleeve with a light twisting motion. To remove the center from the headstock spindle, hold the pointed end with a cloth or rag in one hand and give the center a sharp tap with a rod or knockout bar inserted through the hollow headstock spindle. To remove the center from the tailstock, turn the tailstock hand wheel to draw the tailstock spindle into the tailstock. The center will contact the tailstock screw and will be bumped loose from its socket

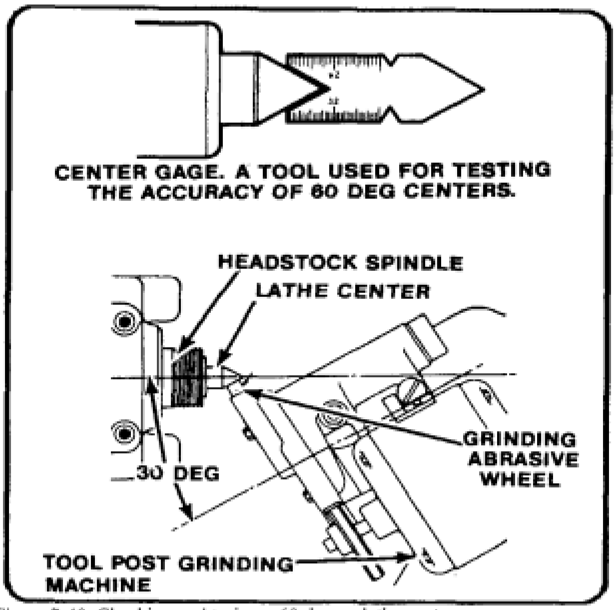


Figure 7-40. Checking and truing a 60 degree lathe center.

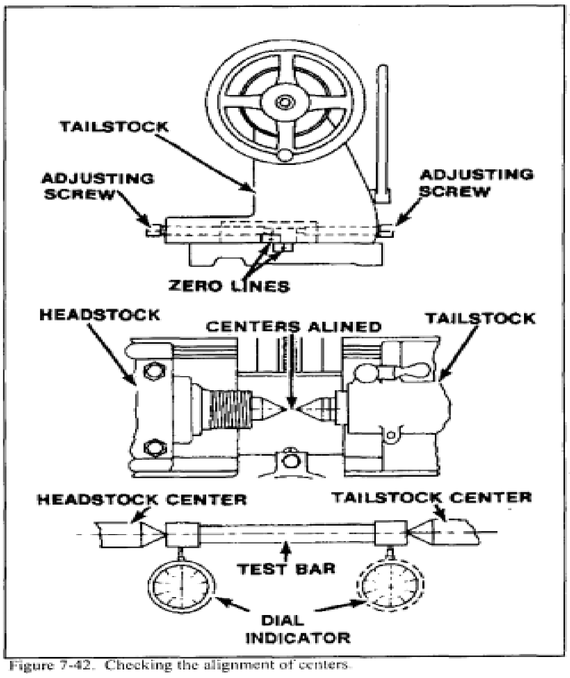


Figure 7-42. Checking the alignment of centers.

Lathe centers must be parallel with the ways of the lathe in order to turn workplaces straight and true. Before beginning each turning operation, the center alignment should be checked. The tailstock may be moved laterally to accomplish this alignment by means of adjusting screws after it has been released from the ways. Two zero lines are located at the rear of the tailstock and the centers are approximately aligned when these lines coincide (Figure 7-42). This alignment may be checked by moving the tailstock up close to the headstock so that the centers almost touch, and observing their relative positions (Figure 7-42).

After mounting the headstock and tailstock centers, the accuracy of the 60° point should be checked using a center gage or a dial indicator. If the center in the headstock is not at 60°, or is scarred and burred, it must be trued while inserted in the lathe headstock spindle. If the headstock center is a soft center (a center that is not heat-treated and hardened), it can be turned true with the lathe tool bit. If the center in the headstock is hardened, it must be ground with a tool post grinding machine to get a true surface (Figure 7-40).

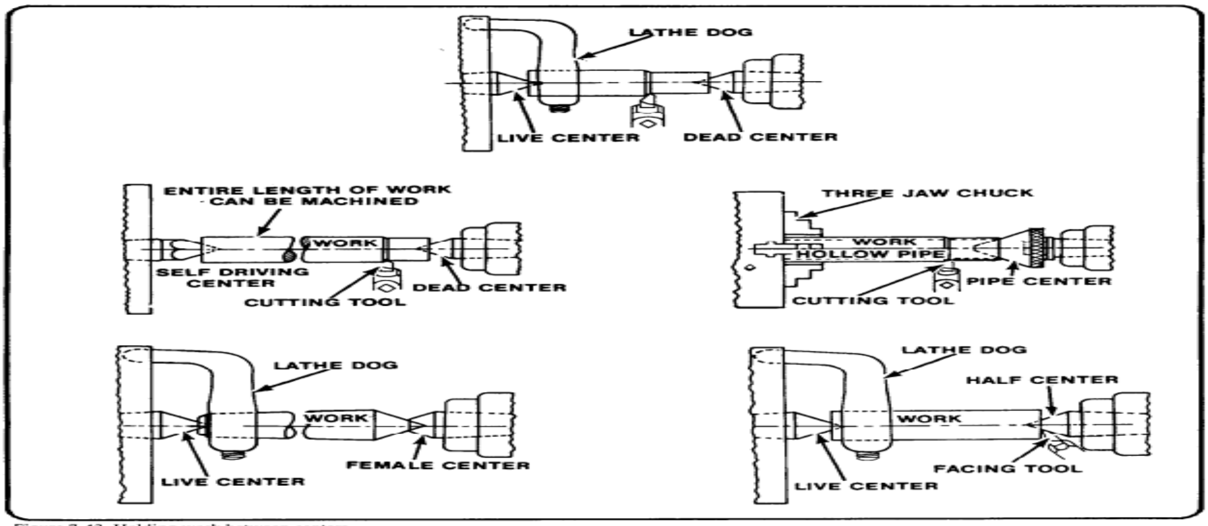


Figure 7-43. Holding work between centers.

3.9.4 **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 7-44). 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.

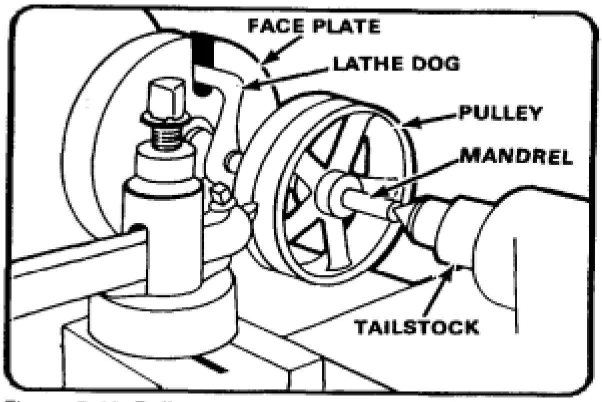


Figure 7-44. Pulley mounted on a mandrel.

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.

4. MOUNTING AND INDEXING WORK

An efficient and positive method of holding workplaces to the milling machine table is important if the machine tool is to be used to its fullest advantage. The most common methods of holding are clamping a work piece to the table, clamping a work piece to the angle plate, clamping the work piece in fixtures, holding a work piece between centers, holding the work piece in a chuck, and holding the work piece in a vise.

Regardless of the method used in holding, there are certain factors that should be observed in every case. The work piece must not be sprung in clamping, it must be secured to prevent it from springing or moving away from the cutter, and it must be so aligned that it may be correctly machined T-slots, Milling machine worktables are provided with several T-slots which are used either for clamping and locating the work piece itself or for mounting the various holding devices and attachments. These T-slots extend the length of the table and are parallel to its line of travel. Most milling machine attachments, such as vises and index fixtures, have keys or tongues on the underside of their bases so that they may be located correctly in relation to the T-slots.

4.1 Clamping Work pieces to the Table

When clamping a work piece to the worktable of the milling machine, the table and the work piece should be free from dirt and burrs. Work pieces having smooth machined surfaces may be camped directly to the table, provided the cutter does not come in contact with the table surface during milling. When clamping workplaces with unfinished surfaces in this way, the table face should be protected from damage by using a shim under the work piece. Paper, plywood, and sheet metal are shim materials. Clamps should be located on both sides of the work piece if possible to give a full bearing surface. These clamps are held by T-slot bolts inserted in the T-slots of the table. Clamp supports must be the same height as the work piece. Never use clamp supports that are lower than the work piece. Adjustable step blocks are extremely useful to raise the clamps, as the height of the clamp bar may be adjusted to ensure maximum clamping pressure. Clamping bolts should be placed as near to the work piece as possible so that the full advantage of the fulcrum principle may be obtained. When it is necessary to place a clamp on an overhanging part, a support should be provided between the overhang and the table to prevent springing or possible breakage. A stop should be placed at the end of the work piece where it will receive the thrust of the cutter when heavy cuts are being taken.

4.2 Clamping a Work piece to the Angle Plate

Work pieces clamped to the angle plate may be machined with surfaces parallel, perpendicular, or at an angle to a given surface. When using this method of holding a work piece, precautions should be taken similar to those mentioned for clamping work directly to the table. Angle plates are either adjustable or nonadjustable and are generally held in alignment by keys or tongues that fit into the table T-slots

4.3 Clamping Work pieces in Fixtures

Fixtures are generally used in production work where a number of identical pieces are to be machined. The design of the fixture depends upon the shape of the piece and the operations to be performed. Fixtures are always constructed to secure maximum clamping surfaces and are built to use a minimum number of clamps or bolts in order to reduce the setup time required. Fixtures should always be provided with

Keys to assure positive alignment with the table T-slots.

4.4 Holding Work pieces in a Chuck

Before screwing the chuck to the index head spindle, it should be cleaned and any burrs on the spindle or chuck removed. Burrs may be removed with a smooth-cut, three cornered file or scraper, while cleaning should be accomplished with a piece of spring steel wire bent and formed to fit the angle of the threads. The chuck should not be tightened on the spindle so tightly that a wrench or bar is required to remove it. Cylindrical workplaces held in the universal chuck may be checked for trueness by using a test indicator mounted upon a base resting upon the milling machine table. The indicator point should contact the

Circumference of small diameter work pieces or the circumference and exposed face of large diameter pieces. While checking, the work piece should be resolved by rotating the index head spindle.

4.5 Holding Work pieces in the Vise

AS previously mentioned, five types of vises are manufactured in various sizes for holding milling machine workplaces. These vises have locating keys or tongues on the underside of their bases so they may be located correctly in relation to the T-slots on the milling machine table (Figure 8-22).

The plain vise similar to the machine table vise is fastened to the milling machine table. Alignment with the milling machine table is provided by two slots at right angles to each other on the underside of the vise.

These slots are fitted with removable keys that align the vise with the table T-slots either parallel to the machine arbor or perpendicular to the arbor. The swivel vise can be rotated and contains a scale graduated in degrees at its base which is fastened to the milling machine table and located by means of keys placed in the T-slots. By loosening the bolts which clamp the vise to its graduated base, the vise may be moved to hold the work piece at any angle in a horizontal plane. To set a swivel vise accurately with the machine spindle, a test indicator should be clamped to the machine arbor and a check made to determine the setting by moving either the transverse or the longitudinal feeds, depending upon the position of the vise jaws. Any deviation as shown by the test indicator should be corrected by swiveling the vise on its base.

The universal vise is used for work involving compound angles, either horizontally or vertically. The base of the vise contains a scale graduated in degrees and can rotate 360° in the horizontal plane and 90° in the vertical plane. Due to the flexibility of this vise, it is not adaptable for heavy milling.

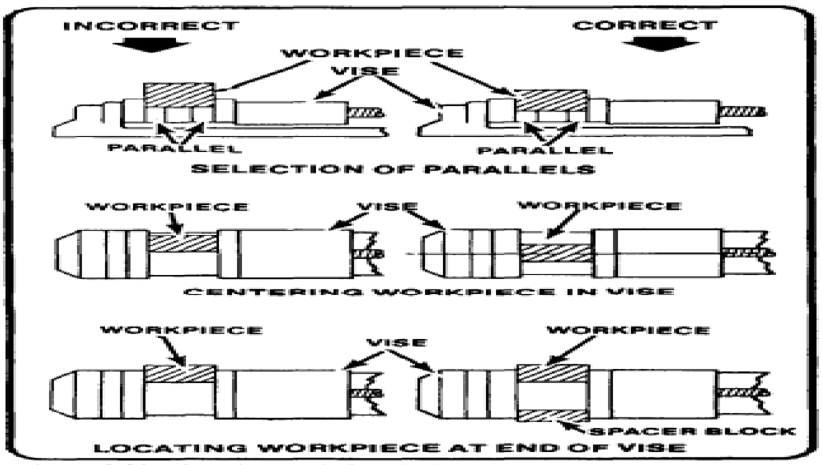


Figure 8-23. Mounting workpiece in the vise.

4.6 Mounting the Work piece

When face milling, the work piece may be clamped to the table or angle plate or supported in a vise, fixture, or jig. Large surfaces are generally face milled on a vertical milling machine with the work piece clamped directly to the milling machine table to simplify handling and clamping operations.

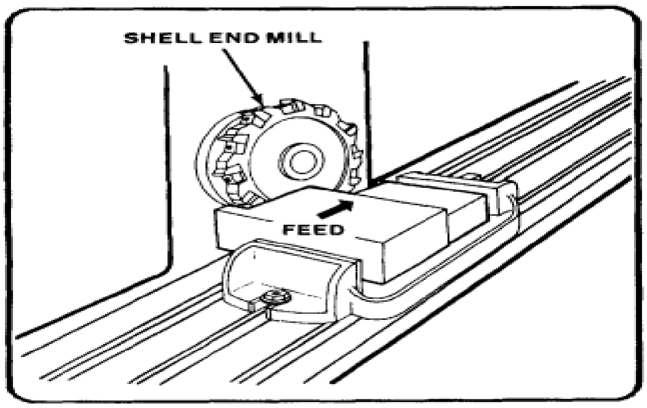


Figure 8-30. Face milling.

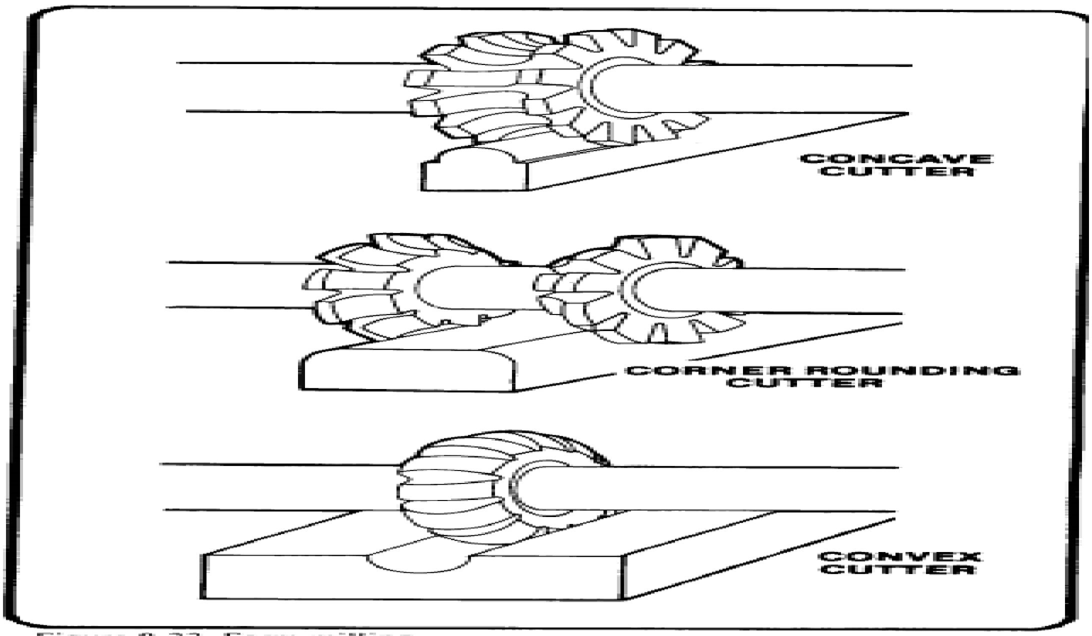


Figure 8-33. Form milling.

Self-Check -3	Choose

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

1. In lathe machine boring tool is mounted on____?
   A. Cross slide B. Tool posts C. Box D. None
2. In drilling machine straight shank drill bit is mounted on?
   A. Drill chuck B. sleeve D. Arbor D. socket
3. It is used when the hole in the spindle of the drill press is too small for the taper shank of the drill.
   A. Drill chuck B. sleeve D. Arbor D. socket
4. It is used for mounting drills or other tapered - shank tools in the main spindle of the machine or the vertical milling attachment.
   A. Arbor B. Collet adapter C. Shell end mill arbor D. Chuck

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

Answer Sheet

Score = ____________
Rating: ______________

Name: ___________________________ Date: ________________

Short Answer Questions

Information Sheet-4	Setting and adjusting guards and accessories

4.1 Accessories with Their Functions Used For Lathe Machine

The lathe accessories are used for holding and supporting the work or for holding the cutting the various lathe accessories are discussed as follows

A general purpose machine tool is basically comprised of power drive and kinematic system for the essential formative and auxiliary tool – work motions and a rigid body or structure to accommodate all of the above. But several additional elements or devices called accessories are also essentially required for that machines' general functioning, mainly for properly holding and supporting the work piece and the cutting tool depending upon the type and size of the tool – work and the machining requirements.

1. Adjusting Centers.

a) There are two types of centers i.e., **live center and dead center.**

b) A center which fits into the headstock spindle and revolves with the work is called live center.

c) The center which is used in a tailstock spindle and does not revolve is called dead center.

— Live and Dead Centers

2. Adjusting Chucks.

a) It is an important device used for holding and rotating the work piece in lathes.

b) The work pieces which are too short to be held between centers are clamped in a chuck.

c) It is attached to the lathe spindle by means of two bolts with the back plate screwed on to the spindle nose.

d) There are many types of the chuck, but the following two are commonly used.

i) Three jaw universal chuck.

The three jaw universal chuck, as shown in Fig. is also called self-centering chuck or scroll chuck. Thus chuck is used for holding round and hexagonal work.

3 jaw chuck lathe machine accessories

ii) Four jaw independent chuck.

4 jaw chuck lathe machine accessories

1. The four jaw independent chuck, as shown in Fig. has four reversible jaws, each of which may be independently adjusted to accommodate the work it supports.

2. This type of chuck can hold square, round and irregular shape of work in either a concentric or eccentric position.

a. The other types of the chucks are

iii) combination chucks, IV) magnetic chuck, v) collect chuck, vi) drill chuck, and vii) air or hydraulic chuck

3. Adjusting Lathe dog or carrier

a) The work placed on a mandrel or held between centers is rotated positively by clamping the dog or carrier to the end of the work.

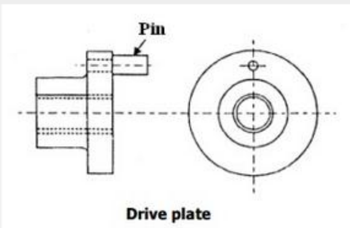
b) This is engaged with a pin attached to the drive plate or face plate.

c) The lathe dog or carrier may be of straight type or bent type as shown in Fig.

— lathe dogs carriers Lathe Accesories

4. Adjusting Drive plate

a) The drive plate, as shown in Fig. is a circular plate which is bored out and threaded so that it can be attached to the spindle nose.



— lathe accessories drive plate

b) It also carries a hole for the pin which is used only when the work is held in a lathe dog having straight tail. When bent-tail dog is used, this pin is taken out and the bent portion of the tail is inserted into the hole.

5) **Adjusting Face plate**.

— lathe accessories face plate

a) The face plate, as shown in Fig. is similar to drive plate except that it is larger in diameter.
b) It contains more open slots or T-slots so that bolts may be used to clamp the work piece to the face of the plate.

c) The face plate is used for holding work pieces which cannot be conveniently held in a chuck.

6. Adjusting Angle plate

a) An angle plate is simply a cast iron plate with to faces planed at right angles to each other and having slots in various positions for the clamping bolts.

b) It. is always used with the face plate for holding such parts which cannot be clamped against the vertical surface of the face plate.

Figure: Angle Plates

7. Adjusting Mandrels

a) The lathe mandrel is a cylindrical bar with center hole at each end. It is used to hold hollow work pieces to machine their external surface.

b) The work revolves with the mandrel which is mounted between the centers of the lathe. The various types of mandrels used for different classes of work are shown in Fig.

— Lathe Mandrels

8. Adjusting Steady, follower and other rests

Long work pieces often need to be supported in the middle, as cutting tools can push (bend) the work piece away from where the centers can support them, because cutting metal produces tremendous forces that tend to vibrate or even bend the work piece. This extra support can be provided by a s**teady rest (also called a steady, a fixed steady, a center rest, or sometimes, confusingly, a center)**. It stands stationary from a rigid mounting on the bed, and it supports the work piece at the rest's center, typically with three contact points 120° apart.

— a steady rest

A follower rest (also called a follower or a travelling steady) is similar, but it is mounted to the carriage rather than the bed, which means that as the tool bit moves, the follower rest "follows along" (because they are both rigidly connected to the same moving carriage)

4.2 Adjusting Machine vices

The machine vise is used on several machines such as, the milling and drilling machines and it is the most common type of work holding devise used on the machines.

Machine Vise

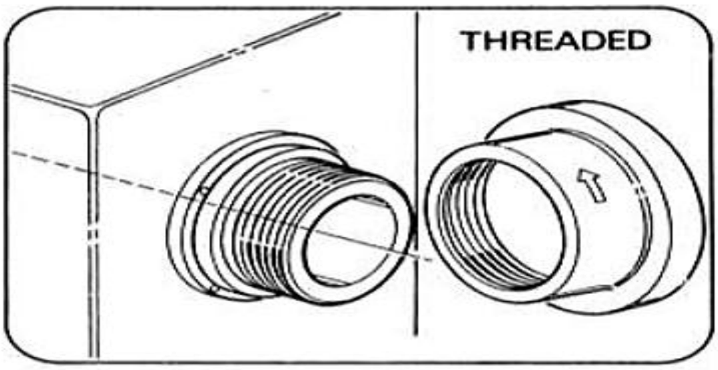
4.3 Types of spindle noses

There are four types of headstock spindle noses to which accessories are fitted. They are:

- **Threaded spindle**
- **Taper and locking ring**
- **Cam lock**
- **Bolted**

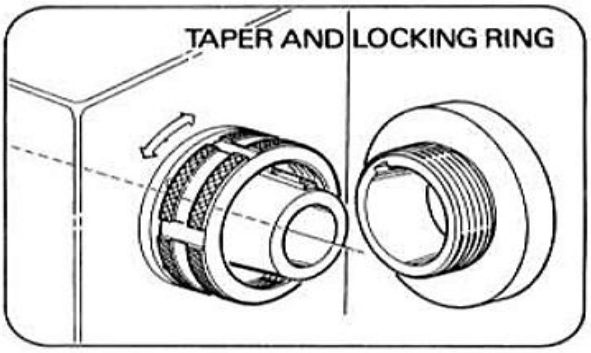
a. Threaded spindle

Accessories are screwed directly on to the spindle. The fit of the threaded holds the accessory true to the axis of the spindle. A shoulder bears against the necessary to hold it square to the spindle.



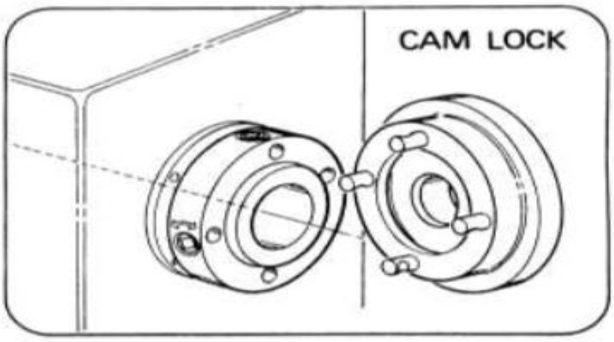
b. Taper and locking ring

When a threaded locking ring is tightened the accessory is firmly held on the taper.



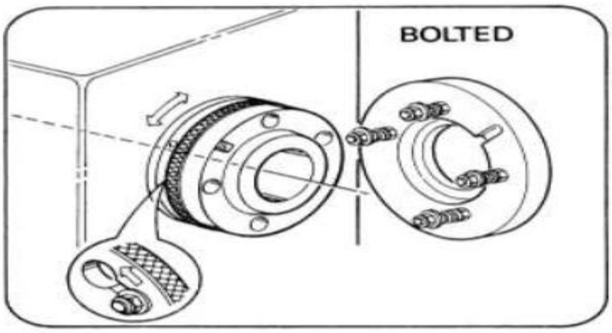
c. Cam lock

When the cam locks are tightened, the accessory is drawn firmly on to a short taper and against a flange.



d. Bolted

The accessory is bolted to a flange on the spindle by a number of studs that protrude from the rear face of the accessory. A short taper locates the accessory centrally. Drive is provided by a locating key.



4.4 Machine Guarding

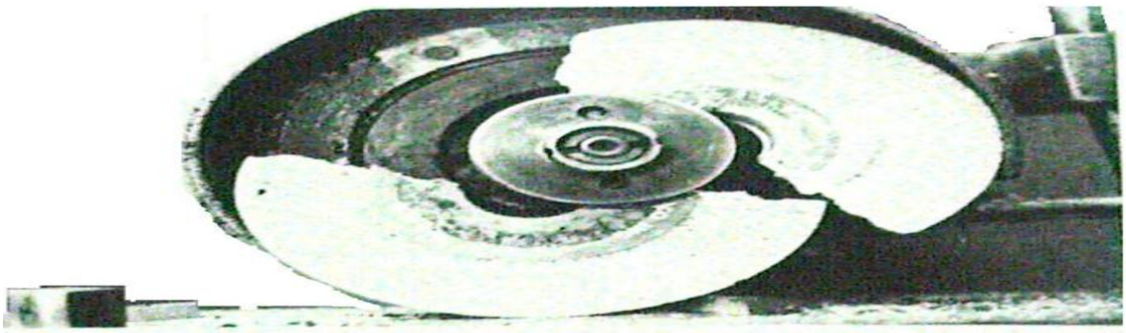
Any machine part which can cause injury must be guarded. Machine guards help to eliminate personnel hazards created by points of operation, ingoing nip points, rotating parts and flying chips.

Types of guards commonly used machine guards are:-

1. **Fixed guard**-is kept in place permanently by fasteners that can only be released by the use of a tool.
2. **Interlocked guard**-shuts off or disengages power to the machine and prevents it from starting when the guard is removed/ opened.

3. **Adjustable guard-**provides a barrier which can be adjusted to suit the varying sizes of the input stock.
4. **Self-adjusting guard**-provides a barrier which moves according to the size of the stock entering the danger area.
5. **Two hand controls** -concurrent use of both hands is required to operate the machine, preventing the operator from reaching the danger area.
6. **Pull back** -the device is attached to the wrist of the operator which pulls the operator's hands away from the point of operation or other hazardous areas when the machine operates.

Machine guards are surface grinding wheels main protective barrier against a deadly result of a broken grinding wheel.



4.4 Miscellaneous safeguarding aids

Shields can be used to provide protection from flying particles, splashing metal working fluids or coolants. Holding tools can be used to place and remove stock. Example, reaching into the danger area of a power press. Holding tools must not be used as a replacement of machine guards.

4.5 Safety precautions while working with machinery

- Ensure that the guards are in position and in good working condition before operating.
- Know the location of emergency stop switch.
- Do not wear loose clothing or jewelry that can be caught in the rotating parts.
- Confine long hair.

Self-Check -4	Written Test

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

1. One of the following is not holding work pieces in lathe machine
   A. Chucks B. Rests C. Mandrels D. Face plate E. None
2. ----------- Centre is fitted on the headstock spindle and rotates with the work
   A. Live B. Dead C. Rests D. Face plate

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

Answer Sheet

Score = ___________

Rating: ____________

Name: _________________________ Date: _______________

Short Answer Questions

Information Sheet-5	Calculating cutting speeds, RPM, feeds and depth of cuts

5.1 Cutting speed, feed and Depth of cut

When a cutting tool is applied to a work piece, its motion relative to the work piece has two components:-

1. The motion resulting from the primary motion of the machine tool, which is called the cutting speed.
2. The motion resulting from the feed motion of the machine tool, which is called, feed speed. The resultant of these two motions is called the resulting cutting motion.

5.1.1 Cutting speed (V)

The cutting speed is the distance traveled by the work surface in a unit of time in relation to the cutting edge of the tool. In other words, it is the speed at which the circumference of the work passes the tool bit.

5.1.2 Feed

Feed refers the ratio of the distance that the cutting tool is distanced in the direction of feed motion to full rotation of the work. Generally, feed is the distance the cutter moves during one revolution of the work. The cutting tool may be feed manually or automatically along the work while machining.
A longitudinal feed is when the tool travels in a direction parallel to the work axis. A cross feed is when the tool travels in a direction perpendicular to the work axis.

5.1.3 Depth of cut (t)

The depth of cut is the thickness of the layout of metal removed in one cut, or pass, measured in a direction perpendicular to the direction of the feed motion. In turning with axial (longitudinal) feed, depth of cut is measured the decrease in diameter. In radial (facing) feed, depth of cut is measured the decrease in length.

In turning cylindrical surfaces depth of cut (t) can be found by: - $t = \frac{D-d}{2}$

Where D- diameter of the work piece
d- Diameter of machined surface.

The cutting speed (V) can be found by:-

$$V = \frac{\pi D N}{1000} \text{ m/min}$$

Where: - D-work diameter in mm for lathe machine and diameter of cutting tools for drilling and milling machines.
N- Rotational frequency of the work in rev/min

Example:-

1. Calculate the speed of rotation of the machine if the diameter of the work piece is 50 mm and the cutting speed of the tool is 45 m/min

$$N = \frac{1000V}{\pi D} = \frac{1000 \times 45}{3.14 \times 50mm} \rightarrow \frac{45000}{157} \rightarrow 286.6$$

N= <u>286 r / min</u>

2. As the above speed of rotation of the machine and cutting speed of the tool the Diameter of the work decrease in to 46mm. Calculate the depth of cut.

$$t = \frac{D-d}{2} \rightarrow \frac{50mm - 46mm}{2} \rightarrow \frac{4mm}{2} = 2mm \qquad t=2mm$$

3. According to question number two the final diameter of the work piece is 44mm and r/min of the machine is the same, calculate the cutting speed of the tool.

$$V = \frac{\pi DN \ m/min}{1000} \rightarrow \frac{3.13 \times 44mm \times 286 \ r/min}{1000mm} \rightarrow \frac{39515}{1000}$$

V= <u>39.50 m/min</u>

For milling machine

Spindle Speed

Spindle speed in revolution per minute (R.P.M.) for the cutter can be calculated from the equation:-

$$N = \frac{CS \times 1000}{\pi d}$$

where -- N = R.P.M. of the cutter

CS = Linear Cutting Speed of the material in m/min. (see table 1)

d = Diameter of cutter in mm

Feed Rate calculation

Feed rate (F) is defined as the rate of travel of the workpiece in mm/min. But most tool suppliers recommend it as the movement per tooth of the cutter (f). Thus,

F = f .u . N

Where---- F = table feed in mm/min

f = movement per tooth of cutter in mm (see table 1)

u = number of teeth of cutter

N = R.P.M. of the cutter

Table: C.S. and feed rate for some common material

Tool Material	High Speed Steel		Carbide	
Material	Cutting Speed	Feed (f)	Cutting Speed	Feed (f)
Mild Steel	25	0.08	100	0.15
Aluminum	100	0.15	500	0.3
Hardened Steel	---	---	50	0.1

5.2 CUTTING VARIABLES

Cutting variables are the most important factors which affect the efficiency if the milling machine. These are:-

1) Cutting speed: The speed, in surface feet per minute (sf/min) or meters per minute (m /min) of which the metal may be machined efficiently.

The formula used to find the work speed in mm

$$Rpm = \frac{cs \times 320}{D}$$ Where, cs = meter per minute

D= diameter of cutter in millimeter

The speed used on a milling machine depends:-

- The type of work material
- The cutter material
- The diameter of the cutter'
- The surface finish required

- The depth of cut being taken

Example:1. calculater the r/min required for a 75 mm diameter high speed milling cutter when cutting machine steel (cs= 30 m/min)

solution: rpm= $\frac{30\ m/min\ x320}{7575mm} = \underline{9600} = \underline{128}$

2)Feed: it is the distance in inches/ or (mm/min) that the work moves

The formula used to calculate the feed in mm/min. Is:-

Feed(rev/min) = N x cpt x r/min

Where, N = number of teeth on the milling cutter

Cpt = chip per tooth for a particular cutter find from table

r/min = revolution per minute of milling cutter

The feed rate used on milling machine depends up on a variety of factors, such as

- The depth and width of cut
- The design or type of cutter
- The sharpness of the cutter
- The work piece material
- The strength and the uniformity of the work piece
- The type of finish and accuracy required.

- Example: Find the feed in millimeter per minute (mm/min) for a 75 mm diameter, six-tooth helical milling cutter when machining cast iron work piece.

Solution: first calculate the rpm of the cutter

$$Rpm = \frac{cs\ x\ 320}{D}$$

$$= \frac{60\ x\ 320}{75} = 256\ mm/rev$$

Thus, feed (mm/ min) = N x cpt x rpm

= 6 x o.18 x 256

= 276 mm/ min

a. **DEPTH OF CUT**: it is the advancement of the tool in the job in a direction perpendicular to the surface being machined.

5.3 DETERMINING CUTTING VARIABLES LATHS

Feed: The amount of tool advancement per revolution of the job parallel to the surface being machined.

- it is given in mm/rev of the job that which the tool is feed
- it depends upon :- finished required, depth of cut and the rigidity of the machine.
- Mathematically, (f) = feed per rev. X rpm

Cutting speed: It means the number of meters measured on the circumference of a job that passes the cutting edge of the tool in one minute.

- mathematically,

Cutting speed, (v) =$\frac{\prod DN}{1000}$ inm/min. Or N=$\underline{VX1000}$

Where, D= is diameter of a job in m N= is spindle or job speed in rpm

- For example: - calculate the rpm of a lathe machine spindle to turn a work having cutting speed 55 m/ min. Assume the diameter of the work is 80 mm.

Depth of cut It is the advancement of the tool in the job in a direction perpendicular to the surface being machined.

Depth of cut depends up on cutting speed, rigidity of machine and tool material.

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

Where, D = is diameter of the work piece

D = is diameter of the work required

5.4 Cutting speeds, feeds, and depth of cut of a milling

The speed of a milling cutter is the distance in feet (meter) per minute that each tooth travels as it cuts its chips. The number of spindle revolutions per minute necessary to give a desired peripheral speed on the size of the milling cutter. The best speed is determined by the type of material being cut and the size and type of cutter used. The smoothness of the finish desired, the depth of cut being taken and the rigidity of the machine (power available) and work set up are other factors relating to the cutter speed.

1. Speed Computation

$$r/\min = \frac{CS(ft)}{circumference(in)} \quad (for\ inch\ calculation)$$

$$r/\min = \frac{12\,x\,CS}{\Pi D}$$

$D-diameter\ of\ the\ cutter$

$CS-cutting\ speed$

$$r/\min = \frac{CS(m)\,x1000}{\Pi\,xD(mm)} = \frac{CS\,x1000}{3.1416\,x\,D} \quad (for\ metric\ calculation)$$

$D-diameter\ of\ the\ cutter$

Example:

1. Calculate the speed required to revolve a 3 in diameter high-speed steel milling cutter when cutting machine steel (90 ft/min)

$$r/\min = \frac{CS(ft)}{\Pi D} = \frac{90}{3x3.1416}\ or\ \frac{CS\ x\,4}{D}$$

Since the CS is m ft and diameter is in inch, the feet must be multiplied by 12.

$$There\ for\ r/\min = \frac{90\ x12}{33.1416} = \_\_\_\_\_\_\_\_\_\_\_\ r/\min$$

2. at what speed should be a 2in. diameter carbide cutter to mill a piece of cast iron (CS150 fit/mm)?

$$r/\min = \frac{4\,x150}{D} = \frac{4\,x150}{2} = 300\,r/\min$$

3. Calculate the r/min required for a 75 mm diameter high speed steel milling cutter when cutting machine steel (CS 30 m/min)

$$r/\min = \frac{1000V}{\Pi D} = \frac{320\,x\,30}{75} = 128$$

2. Selecting Proper Cutting Speed

- The approximate values given in table 5.3 on the following page may be used as a guide for selecting the proper cutting speed. If the operator finds that the machine, the milling cutter, or the work piece cannot be handled suitably at these speeds, immediate readjustment should be made.
- Table 6.3 lists speeds for high-speed steel milling cutters. If carbon steel cutters are used, the speed should be about one-half the speed recommended in the table. If carbide-tipped cutters are used, the speed can be doubled.
- If a plentiful supply of cutting oil is applied to the milling cutter and the work piece, the speeds can be increased from 50 to 100 percent.

- For roughing cuts, a moderate speed and coarse feed give best results; for finishing cuts, the best practice is to reverse these conditions, using a higher speed and a lighter cut.
- Table 6.4 is providing to facilitate spindle speed computations for standard cutting speeds and standard milling cutters.

a. Feed For Milling

-The rate of feed is the speed at which the work piece passes the cutter.
Milling machine feed may be defined as the distance in inches (millimeters) per minute that the work moves in to the cutter.
-It determines the time required for cutting a job.

Feed Computation

The milling machine feed is determined by multiplying the chip size (chip per tooth) desired, the number of teeth in the cutter, and the rotation per minute of the cutter.

Feed = No. of cutter teeth x chip /tooth x cutter r/min.

Inch calculation

Feed (in/min) = N x cpt x r/min

N- Number of teeth in the milling cutter
cpt - chip per tooth for a particular cutter and metal (from table)
r/min - revolutions per minute of the milling cutter

Example:

Find the feed per minute using a 3.5 in. diameter 12 teeth helical cutter to cut machine steel (CS 80)

$$Solution : r/\min = \frac{4\,x\,CS}{D} = \frac{4\,x\,80}{3.5} = 91$$

$$Feed\,(in/\min) = N\;x\,Cpt\;x\,r/\min$$
$$= 12\,x\,0.010\,x\,91 = 10.9 = 11$$

From table for helical cutter to cut machine steel = 0.010

Metric calculations

Example:

Calculate the feed in mill meters per minute for a 75 mm diameter, six-tooth helical milling cutter (HSS) when machining a cast iron work piece (CS 60)

$Feed\,(mm/\min) =$ N x cpt x r/min

cpt= cut per teeth; from table 6.6, 0.25 is found for helical cutter to machine cast iron.

$$r/\min = \frac{CS\;x\,320}{D} = \frac{60\,x\,320}{75} = 256$$

Feed (mm/min) = 6 x 0.25 x 256 = 384 mm/min

Table 6.6 Recommended feed per tooth (for high speed steel cutters)

Material	Face mills		Helical Mills		Slotting and side Mills		End mills		Form relived Cutters	
	In	Mm	In	Mm	In	Mm	In	Mm	In	mm
Aluminum	0.022	0.55	0.018	0.45	0.013	0.33	0.011	0.28	0.007	0.18
Brass & Bronze	0.014	0.35	0.011	0.28	0.008	0.20	0.007	0.18	0.004	0.10
Machine steel	0.012	0.30	0.010	0.25	0.007	0.18	0.006	0.15	0.004	0.10
Tool steel (medium)	0.010	0.25	0.008	0.20	0.006	0.15	0.005	0.13	0.003	0.08
Stainless steel	0.006	0.15	0.005	0.13	0.004	0.10	0.003	0.08	0.002	0.05
Cast iron	0.013	0.33	0.010	0.25	0.007	0.18	0.007	0.18	0.004	0.10

3. factors which should be considered in selecting the feed

These factors are:

(a) Forces are exerted against the work piece, the cutter, and their holding devices during the cutting process. The force exerted varies directly with the amount of metal removed and can be regulated by the feed and the depth of cut.
Therefore, the correct amount of feed and depth of cut are interrelated, and in turn are dependent upon the rigidity and power of the machine. Milling machines are limited by the power that they can develop to turn the cutter and the amount of vibration they can resist when using coarse feeds and deep cuts.

(b) The feed and depth of cut also depend upon the type of milling cutter being used. For example, deep cuts or coarse feeds should not be attempted when using a small diameter end milling cutter, as such an attempt would spring or break the cutter. Coarse cutters with strong cutting teeth can be fed at a faster rate because the chips may be washed out more easily by the cutting oil.

(c) Coarse feeds and deep cuts should not be used on a frail work piece, or on a piece that is mounted in such a way that its holding device is not able to prevent springing or bending.

(d) The degree of finish required often determines the amount of feed. Using a coarse feed, the metal is removed more rapidly but the appearance and accuracy of the surface produced may not reach the standard desired for the finished product. Because of this, finer feeds and increased speeds are used for finer, more accurate finishes. Most mistakes are made through over speeding, under speeding, and overfeeding. Over speeding may be detected by the occurrence of a squeaking, scraping sound. If vibration (referred to as "chattering") occurs in the milling machine during the cutting process, the speed should be reduced and the feed increased. Too much cutter clearance, a poorly supported work piece, or badly worn machine gears are common causes of "chattering."

4. Typical Feeds

- Feed for milling cutters will generally run from 0.002 to 0.250 inch per cutter revolution, depending upon the diameter of the cutter, the kind of material, the width and depth of the cut, the size of the work piece, and the condition of the machine.
- Good finishes may be obtained using a 3-inch plain milling cutter at a 40 feet per minute speed, with a feed of 0.040-inch per cutter revolution.

5. Direction of Feed

- It is usually regarded as standard practice to feed the work piece against the milling cutter (“called conventional milling”). When the piece is fed against the milling cutter, the teeth cut under any scale on the work piece surface and any backlash in the feed screw is taken up by the force of cut.

- As an exception to this recommendation, it is advisable to feed with the milling cutter (called “climb milling”) when cutting off stock, or when milling comparatively deep or long slots.
- The direction of cutter rotation is related to the manner in which the work piece is held. The cutter should rotate so that the piece springs away from the cutter; then there will be no tendency for the force of the cut to loosen the work piece. No milling cutter should be rotated backward as this will break the teeth. If it is necessary to stop the machine during a finishing cut, the power feed should never be thrown out, nor should the work piece be feedback under the cutter, unless the cutter is stopped or the work piece lowered. Never change feeds while the cutter is rotating.

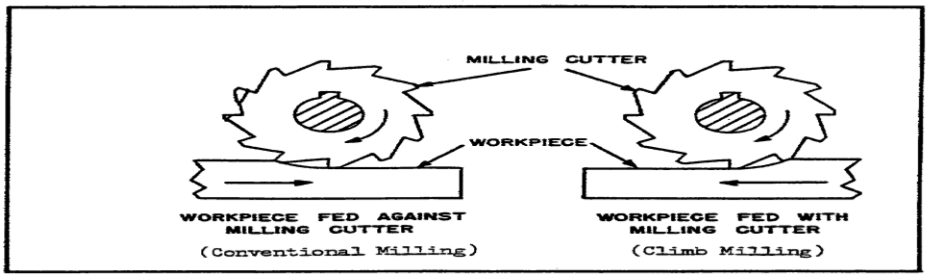


Fig milling machine direction of feed

Self-Check -5	Written Test

Directions 1: Answer all the questions listed below. Use the Answer sheet provided in the next page:

1. A cutting tool in a lathe work is the distance the tool advances for each revolution of the work is called-------.

   A. Feed B. cutting tool C. depth of cut D. all are correct

2. The distance travelled by a point on a milling cutter in one minute is known as

   A. cutting speed B. depth of cut C. spindle speed D. feed

3. The most important factors that affecting the efficiency of a milling operation are?

   A. cutting speed B. depth of cut C. Feed D. All of the above

4. ---------- The amount of tool advancement per revolution of the job parallel to the surface being machined.

   a) Feed b) cutting speed c) depth of cut d) all

5. --------- The feed rate used on milling machine depends up on

   a) The depth and width of cut b)The design or type of cutter
   c) The sharpness of the cutter d) all

6.------------ It means the number of meters measured on the circumference of a job that passes the cutting edge of the tool in one minute.

   a) Cutting speed b) feed C) depth of cut d) none

7. -------- It is the distance in inches/ or (mm/min) that the work moves

   a) Cutting speed b) feed C) depth of cut d) none

Directions II Short Answer Questions

1. a shaft of diameter 85 mm is assumed to be turn on a lathe machine, The RPM of the spindle is adjusted to have 850rev/min for turning the project of the given specimen Calculate the cutting speed of the metal.

2. Calculate the rpm of a lathe machine spindle to turn a work having cutting speed 55 m/ min. Assume the diameter of the work is 80 mm.

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

Answer Sheet

Score = ____________

Rating: _____________

Name: ___________________________ Date: ________________

Information Sheet-6	Performing setup operations

6.1 INTRODUCTION

Before starting a lathe machining operation, always ensure that the machine is set up for the job that is to be accomplished. After selecting and preparing proper cutting tool you have to adjust the center distance The cutting edge of the tool bit should be set to the center height of the lathe spindle. There are several methods for checking the height of the tool bit. Perhaps the simplest way is to place a thin strip of metal, such as a steel rule or feeler gage, between the work piece and the point of the tool bit. If the height is correct, the strip of metal will be held vertical. If the top is leaning toward you, the tool bit is too low. If the top is leaning away from you, the tool bit is too high. Using the standard tool post, you adjust the tool bit height using shims under the tool bit. You can get an economical set of shims, about the right size, at any auto parts store. Purchase a set of feeler gages and remove the pivot pin. The easy way to adjust the tool bit height is to get a quick change tool post. Virtually all quick change tool posts incorporate a mechanism for easily adjusting the tool bit height.

6.2 LATHE MACHINE OPERATION

General operations on the lathe include straight and shoulder turning, facing, grooving, parting, turning tapers, Boring and cutting various screw threads

A) Facing: -Reducing the length of a shaft by lathe cutting tool.
B) Turning; It`s reducing the diameter of a shaft. It is two kinds:-

- Straight turn or cylindrical and
- Tapper turns: - when it increases or decreases in diameter at a uniform rate.
  There are four kinds of tapper making

1. By using form tool:-This technique is limited to the production of short tapers
2. Taper turning with compound rest:

This is given by:-

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

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

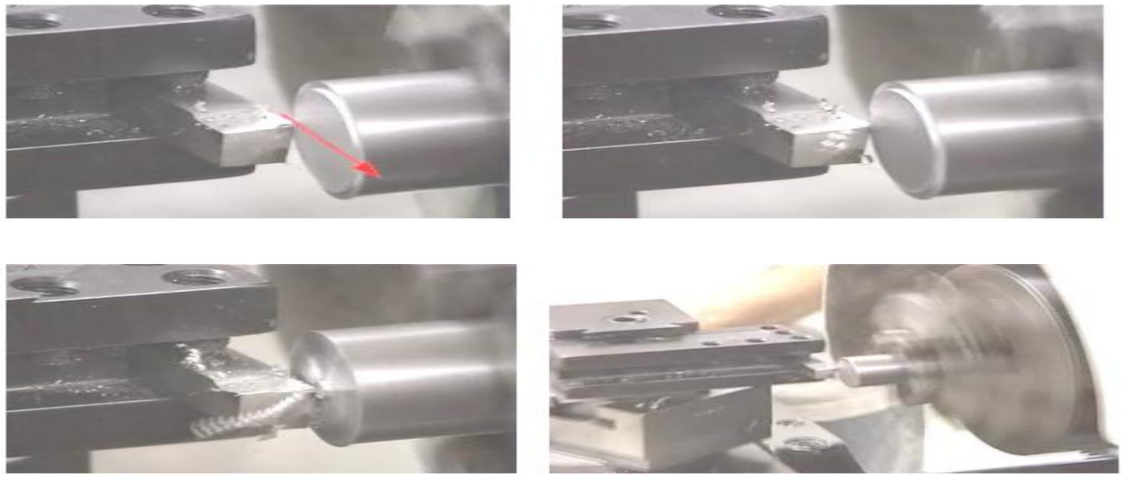
6.3 To face a work piece:

1. Put a tool bit in the tool holder and adjust the cutting edge to center height.
2. Angle the tool so that the side cutting edge forms an acute angle with the face of the work piece.
3. Move the carriage to the right so that the tool bit is past the right end of the work piece.
4. Ensure that the power feed forward/neutral/reverse lever is in the neutral position.
5. Push down on the power feed lever until the half nuts engage. You might have to move the carriage slightly so the half nuts will engage.
6. Turn the lathe on. Adjust the speed to an appropriate speed for the Material and diameter you are working on. Machine Shop
7. Using the compound rest feed handle, slowly advance the tool bit into the work until it just touches the surface of the work piece.
8. Move the cross slide back so that the tool bit is clear of the diameter of the work piece.
9. Using the compound rest feed handle, advance the tool bit about 0.005".
10. Using the cross slide feed handle, advance the cross slide slowly. As the tool bit meets the work piece, it starts cutting.
11. Continue advancing the cross slide until the tool bit reaches the center.

Use this meted for street tiring operation also

6.3.1 STEPS IN FACING OPERATION

1. Prepare the tools and materials.
2. Observe safety precautions while operating the lathe.
3. Mount the work piece to the chuck.
4. Center the work piece.
5. Set the facing tool used right cut facing tool.
6. Feed the cutting tool into the work with the compound rest.
7. Repeat procedure no. 5 until work piece meet the specified measurement.



FACING OPERATION

6.4 Turning

Turning is a form of machining, a material removal process, which is used to create rotational parts by cutting away unwanted material. The turning process requires a turning machine or lathe, work piece, fixture, and cutting tool. The work piece is a piece of pre-shaped material that is secured to the fixture, which itself is attached to the turning machine, and allowed to rotate at high speeds. The cutter is typically a single-point cutting tool that is also secured in the machine, although some operations make use of multi-point tools. The cutting tool feeds into the rotating work piece and cuts away material in the form of small chips to create the desired shape

There are several methods of turning angles or tapers.

For large angles of short length, such as a chamfer, turn the compound rest to the angle you want. Advance the tool across the work with the compound rest, and advance the tool into the work with the cross slide or the carriage.

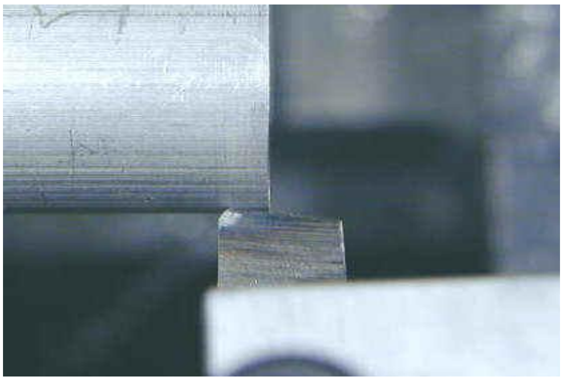
You can use the same method for small angles (usually called tapers) of a length less than the compound rest travel.

For longer tapers, the work is usually placed between centers with the tail center offset from the centerline of the lathe.

Adjusting the Tool Bit

Choose a tool bit with a slightly rounded tip, this type of tool should produce a nice smooth finish. For more aggressive cutting, if you need to remove a lot of metal, you might choose a tool with a sharper tip. Make sure that the tool is tightly clamped in the tool holder.

Adjust the angle of the tool holder so the tool is approximately perpendicular to the side of the work piece. Because the front edge of the tool is ground at an angle, the left side of the tip should engage the work, but not the entire front edge of the tool. The angle of the compound is not critical; I usually keep mine at 90 degrees**.**



6.5 Standard Operating Procedure (SOP) - Lathe

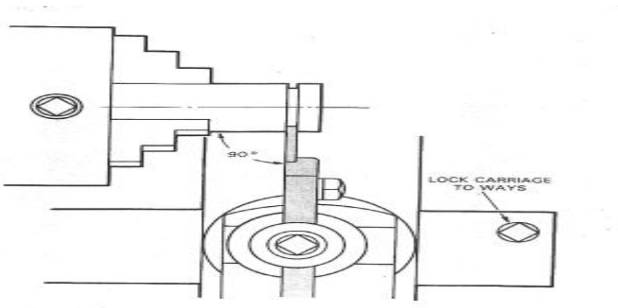
All stock must be properly secured in the lathe chuck or mounted prior to the machining process taking place. Use the correct sized clamp or vise for the stock being machined.

- Turn the chuck or faceplate by hand to ensure there is no binding or danger of the work striking any part of the lathe.
- Check to ensure the cutting tool will not run into the chuck or lathe dog. If possible, feed away from the chuck or dogs.
- Before starting the lathe, ensure the spindle work has the cup center imbedded; tail, stock and tool rests are securely clamped; and there is proper clearance for the rotating stock.
- Prior to starting the lathe, ensure that small diameter stock does not project too far from the chuck without support from the tail stock center.
- When using wood, do not mount a split work piece or one containing knots.
- When roughing stock, do not force the tool in the work piece or take too big a cut.
- The operator must always be aware of the direction and speed of the carriage or cross-feed prior to engaging the automatic feed.
- Never leave the key in the chuck. Do not let go of the key until it is free of the chuck and secured in its proper holding place.
- Select turning speed carefully. Large diameter stock must be turned at a very low speed. Always use the lowest speed to rough out the stock prior to final machining.
- The correct speed and feed for the specific material and cutting tool must be used. Stop the machine before making adjustments or measurements.
- Do not remove metal or wood chips from the table or stock by hand. Use a brush or other tool to properly remove chips or shavings from the table or stock.
- Never attempt to run the chuck on or off the spindle head by engaging the power.
- Do not stop the rotation of the chuck by reversing the power to the lathe unless tapping holes.
- Do not leave tools, bits or excess pieces of stock on the lathe bed.
- All belts and pulleys must be guarded. If frayed belts or pulleys are observed, the lathe must be taken out of service and the belts or pulleys replaced.
- Stop the machine immediately if odd noise or excessive vibration occurs.
- Only properly sharpened drill bits and cutting tools in good condition should be used. Dull drill bits and chipped or broken cutting tools must be removed from service.
- Disconnect the lathe from power source and follow OSEH Guideline IHS011, Lock-out/Tag-out - Control of Hazardous Energy Sources if making repairs or servicing.
- When an operator has finished working on the lathe, and before leaving the lathe for any reason, the power must be shut off and the machine must come to a complete stop.
- When an operator observes an unsafe condition with the lathe or stock being worked, the operator must report it immediately to the designated MSSA and the lathe shall be taken out of service until the problem has been corrected.

6.6 PARTING OPERATIONS

Parting is the operation of cutting off material after machining. This is one of the most difficult jobs performed on a lathe.

The cutting tool must be ground with the correct clearance (front, side, and end).



Work is held close in the chuck for the parting operation, the parting took blade is set a 90° angle to the cut, and the cartridge is locked to the ways.

6.7 KNURLING OPERATION

Knurling

It is the process that forms horizontal or diamond shaped serrations on the circumference of the work to provide a gripping surface. It is done with knurling tool that is mounted in the lathe tool post.

The Knurl pattern is raised by rolling the Knurl pattern against the metal to raise the surface STRAIGHT and DIAMOND PATTERN KNURLS.

6.7.1 How to Use a Knurling Tool

The knurling tool must be set up correctly. Otherwise, the knurls will not track properly and will dull rapidly. Use this procedure:

1. Mark off the section to be knurled.
2. Adjust the lathe to a slow back gear speed and a fairly rapid feed.
3. Place the knurling tool in the tool post and set it up to the work. Both wheels must bear evenly with the wheel faces parallel with the work surface.
4. Start the lathe and force the knurls slowly into the work surface until a pattern begins to form. Engage the automatic feed and let the tool move across the work. When it reaches the proper position, stop the machine but do not disengage the feed. Reverse spindle rotation and permit the tool to move back to the starting point. Repeat the operation until a satisfactory knurl has been formed. Flood the surface with cutting fluid during the operation.

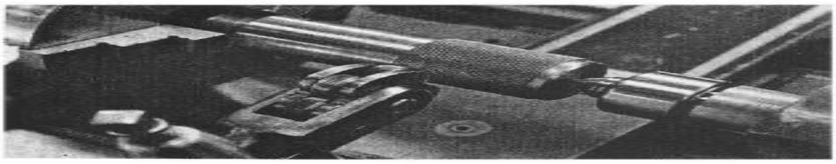


Fig. 9-3C2

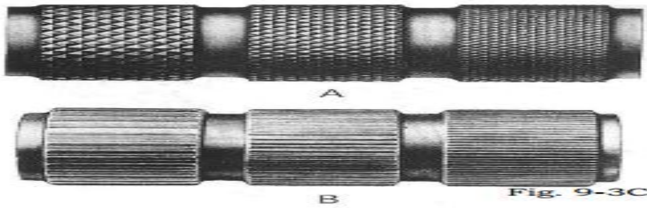


Fig. 9-3C3

6.8 SCREW THREADS OPERATIONS

Screw Threads

A screw thread is a spiral ridge formed on a straight cylinder or tapered cylindrical surface.

The thread may be on an external on internal surface.

Parts of a Screw Tread

1. ***Major Diameter***. It is the largest diameter of a straight external or internal thread.
2. ***Pitch Diameter***. It is the distance between diametrically opposite points on straight thread; it is equal to the major diameter minus one fourth.
3. ***Minor Diameter***. It is the smallest diameter of straight external or internal threads.
4. ***Pitch.*** It is the distance from a point on one thread to a corresponding point on the next thread measured parallel to the axes.
5. ***Flank***. It is the surface connecting crest and root.
6. ***Root***. It is the bottom surface joining the flanks of adjacent threads. The root diameter is the same as the minor diameter.
7. ***Crest*** .It is the top surface of the thread bounded by the flanks.
8. ***Thread Angle***. It is the included angle between the flanks of adjacent turns of the thread.
9. ***Depth.*** It is the perpendicular distance between the root and crest of the thread.
10. ***Lead Angle***. It is the angle which the spiral makes which its axis. The lead angle is determined by the lead and pitch diameter of the thread.
11. ***Lead***. It is the distance a thread surface moves parallel to the axis of the thread during one complete revolution.
12. ***Hand***. It is the direction in which the thread is turned to advance. A right hand is turned clockwise to advance. A left hand thread is turned anticlockwise to advance.
13. ***Clearance***. It is a space left between the mating external and internal threads of facilitate easy rotation of threaded parts.

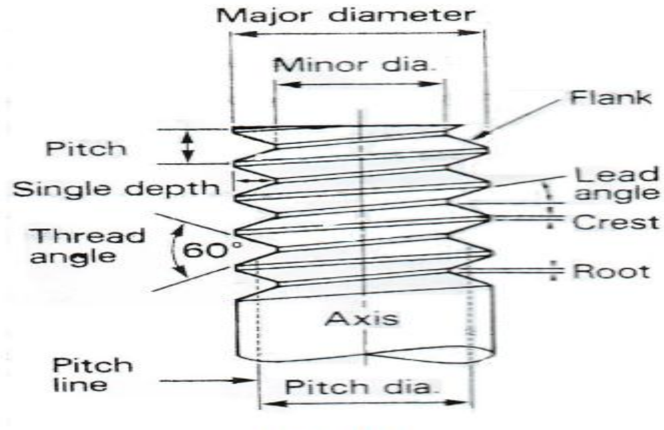


Fig. 9-3G1

PARTS OF A SCREW THREAD

6.9 MILLI NG MACHINE SETUP

Milling machine setups is necessary

- To prolong the life of a milling machine and its accessories and
- To produce accurat work,

This includes

- ✓ Prior to mounting any accessory , check that machine surface are free Of dirty and chips.
- ✓ Do not palce the tools cutters or parts on the milling machine table.
- ✓ When mounting cutters be sure to use keys on all but slitting saws.
- ✓ Check that the arbor spacers and bushings are clean and free of burrs.
- ✓ When tightening the arbor nut take care to make it only hand tight with a
- ✓ When work is mounted in a vise, tighten the vise securely by hand and tap it into place with a lead or soft faced hammer.

Self-Check -6	Written Test

Direction: Write **T** if the statement is ***true*** and **F** if the statement is ***False.*** Pass the test with at least 80% of this test.

1. Parting is the operation of cutting off material after machining.
2. The cutting tool must be ground with the wrong clearance.
3. If the blade is too far out of the holder, the cutting edge will vibrate.
4. If the blade is not square with the work the cutting edge will bind.
5. Speed should be much higher than in the usual turning operation.
6. The largest diameter of a straight external or internal thread is pitch.
7. A bolt has an internal thread.
8. A nut has an external thread.
9. Crest is the top surface of the thread bounded by the flanks.
10. Flank is the connecting crest and root.

Operation Sheet 1	Sharpening of HSS cutter for turning operation

Procedures for sharpening HSS for turning or facing?

Step 1- Dressing the grinding wheel

Step 2- Look up the typical angles for the work piece material

Step 3- Dip the tool in coolant frequently to keep it from overheating and annealing

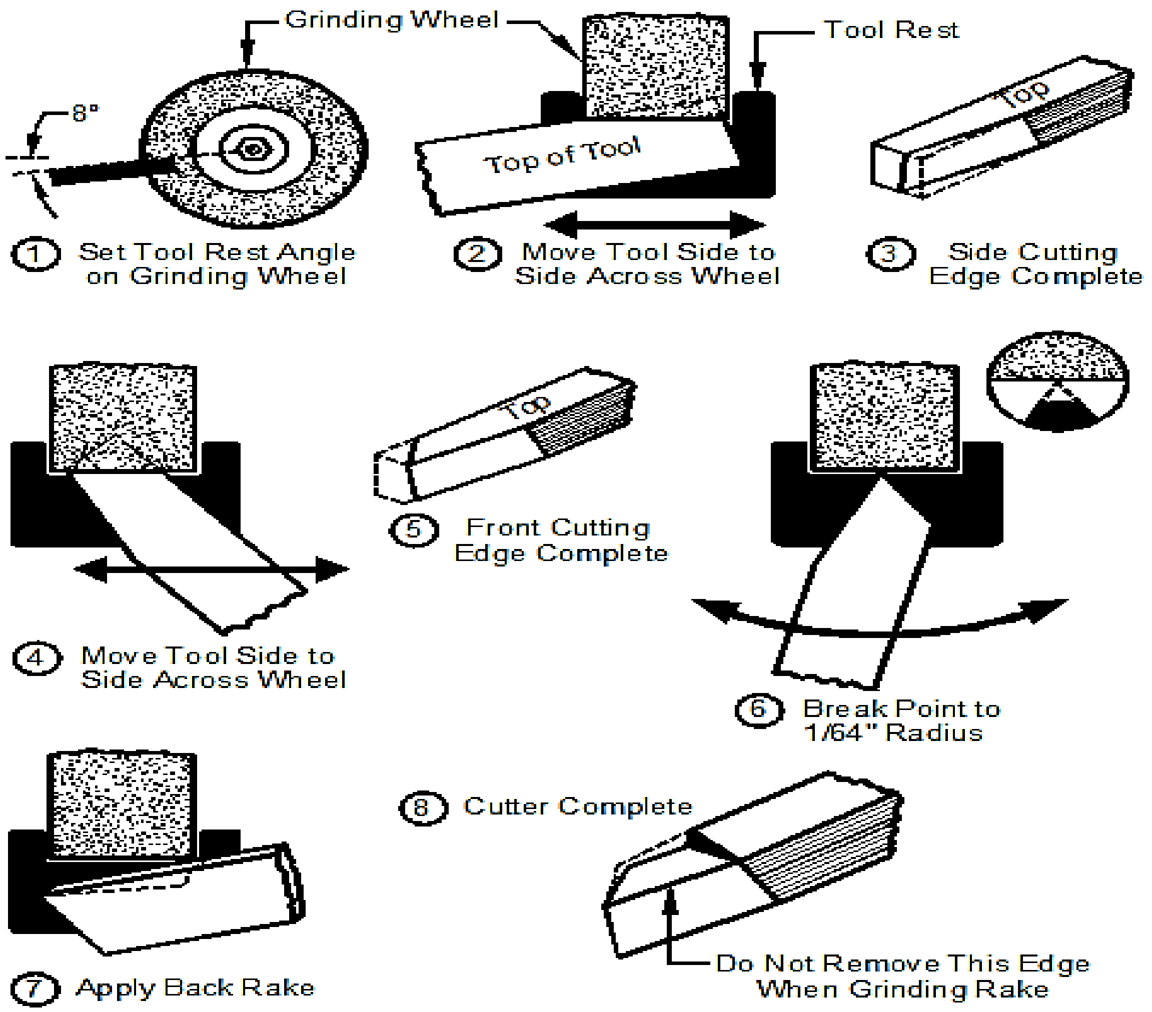
Step 4- set tool rest angle on grinding wheel

Step 5- Move tool size to side across wheel

Step 6- Complete grinding of side cutting edge

Step 7- Complete grinding of front cutting edge

Step 8- Grind or make rake angle



.

Operation Sheet 2	Adjusting the cutter and the work pieces on Milling machine

Procedures for adjusting the cutter and the work pieces on milling machine

Step 1- check the functionally of the milling machine

Step 2- Mount the cutter in the center of arbor.

Step 3- Hold the work piece on universal vise than tighten

Step 4- Move the knee upward the cutter and to touch the work piece

Step 5- Than align the cutter and the work pieces

Step 6- Check the work pieces straight from the tip point of cutter

Operation Sheet-3	Adjusting the cutter and the work pieces on lathe machine

Techniques for adjusting the cutter and the work pieces on lathe machine

Step 1- check all parameters found in lathe machine

Step 2- Insert the dead center into lathe tailstock of hollow spindle

Step 3- Mount the cutter in tool post

Step 4- Align the tip of cutter with the dead center

Step 5- Hold the work piece in the lathe machine chuck and check the alignment by centered cutter

Step 6- Check the work pieces out of center from the center by touching the tip of your cutter

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 2:00 hours.

Task 1. Grind or sharpen HSS Cutting tools for turning and facing operation

Task 2. Center the cutter and the work pieces on lathe machine.

Task 3. Adjust the cutters and the work pieces on milling machines

List of Reference Materials

1. Book: Machining and Machine Tools by A. B. Chattopadhyay.
2. Book: Metal Cutting: Theory And Practice by A. Bhattacharya.
3. Book: Manufacturing Process for Engineering Materials by S. Kalpakjain and S. Schmid.
4. Book: Geometry of Single-point Turning Tools and Drills – Fundamentals and Practical Applications by V. P. Astakhov.

BASIC METAL WORKS

NTQF Level - I

Learning Guide -42

Unit of Competence: -Operate Basic Workshop Machinery

Module Title: - Operating Basic Work shop Machinery

LG Code: IND BMW1 M12LO3-LG-42

TTLM Code: IND BMW1 TTLM 0919v1

LO3: Perform machine operations

Instruction Sheet	Learning Guide 42

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

- **Clamping devices used for machining**
- **Mounting and securing Materials in clamping devices**
- **Operating machines**

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 –**

- **use Clamping devices for machining**
- **Mount and secure Materials to be machined**
- **Operate machine correctly.**

Learning Instructions:

34. Read the specific objectives of this Learning Guide.
35. Follow the instructions described in number 3 to 19.
36. Read the information written in the "Information Sheets 1". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
37. Accomplish the "Self-check 1" in page 6.
38. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 1).
39. If you earned a satisfactory evaluation proceed to "Information Sheet 2". However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
40. Submit your accomplished Self-check. This will form part of your training portfolio.
41. Read the information written in the "Information Sheet 2". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
42. Accomplish the "Self-check 2" in page 9.
43. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 2).

44. Read the information written in the “Information Sheets 3 . Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
45. Accomplish the “Self-check 3” in page 17.
46. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 3).
47. If you earned a satisfactory evaluation proceed to “Operation Sheet 1” in page 19. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
48. Read the “Operation Sheet 1” and try to understand the procedures discussed.
49. If you earned a satisfactory evaluation proceed to “Operation Sheet 2” in page 20. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
50. Read the “Operation Sheet 2” and try to understand the procedures discussed.
51. If you earned a satisfactory evaluation proceed to “Operation Sheet 3,and 4” in page 21. And operation sheet 5 in page 22. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
52. Do the “LAP test” in page 23 (if you are ready). Request your teacher to evaluate your performance and outputs. Your teacher will give you feedback and the evaluation will be either satisfactory or unsatisfactory. If unsatisfactory, your teacher shall advice you on additional work.

Information Sheet-1	Clamping devices used for machining

1.1. Introduction

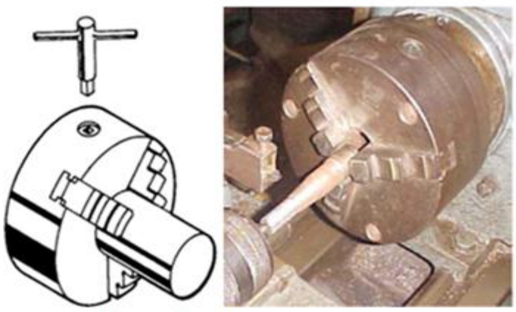
The work is held in the various types of work holding devices like three jaw chuck, four jaw chuck, combination chuck, magnetic chuck, hydraulic chuck, face plate, driving plate, angle plate and lathe carriers. The following are the various work holding devices used on lathe machine:

Work holding devices are used to hold the work steady for an accurate hole to be drilled, and so a safe drilling operation can be accomplished.

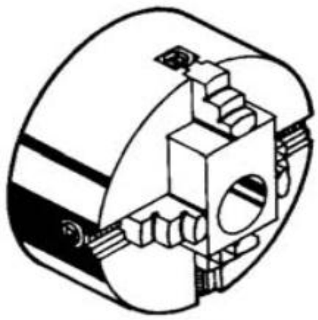
Drilling support devices are used to keep the work piece above the worktable or vise surface and to keep the work piece aligned for drilling.

1.2. Clamping devices or work holding devices used in lathe machine operations

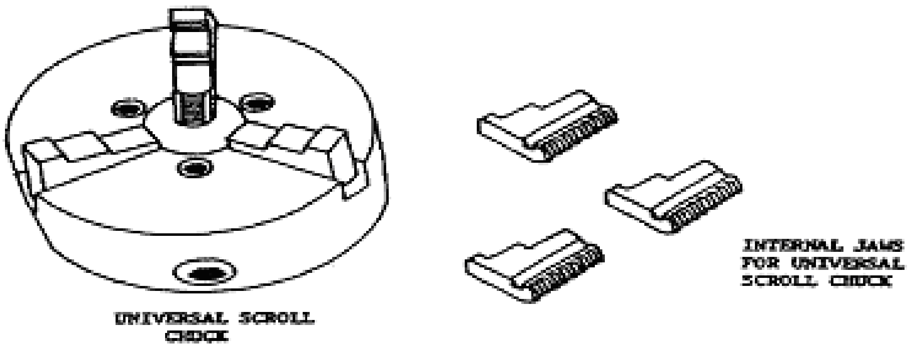
1. **Chucks**
    - 3 – jaw self-centering chuck
    - 4 – jaw independent chuck



3 Jaw chuck



4 Jaw chuck



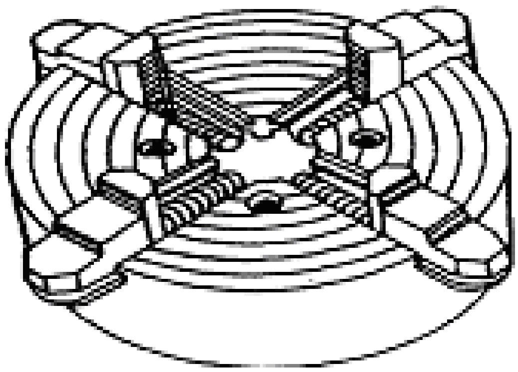
Three-jaw universal chuck

- **Three-jaw universal chuck has** three jaws which move in unison (agreement) as an adjusting pinion is rotated.
- The advantage of the universal scroll chuck is its ease of operation in centering the work for concentric turning. This chuck is not as accurate as the independent chuck but, when in good condition, it will center the work automatically within 0.003 of an inch of complete accuracy.

- The jaws are moved simultaneously within the chuck by means of a scroll or spiral threaded plate.
- The jaws are threaded to the plate and move an equal distance inward or outward as the scroll is rotated by means of the adjusting pinion. Since the jaws are individually aligned on the scroll, the jaws cannot be reversed.
- However, the chuck is usually supplied with two sets of jaws which can be interchanged. The universal scroll chuck can be used to hold and automatically center round or hexagonal work-pieces.
- Having only three jaws, the chuck cannot be used effectively to hold square, octagonal, or irregular shapes.

Four-jaw independent chuck

- **G**enerally have four jaws which are adjusted individually on the chuck face by means of adjusting screws. The final adjustment is made by turning the work-pieces slowly and using gages to determine its concentricity. The jaws are then readjusted as necessary to align the work-piece to desired tolerances.
- The jaws of the independent chuck may be used that the steps face in the opposite direction; thus, work pieces can be gripped either externally or internally.
- The independent chuck can be used to hold square, round, octagonal, or irregular shaped work pieces in either a concentric or eccentric position due to the independent operation of each jaw.
- Because of its versatility and capacity for fine adjustment, the independent chuck is commonly used for mounting work pieces that require extreme accuracy.



2. Face plate

For turning, facing, boring, threading and similar operations, jobs of odd shape and size are usually mounted on large face plate (instead of chuck) being fitted on the spindle nose.

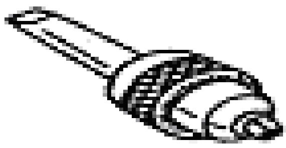
Face Plate

- A lathe faceplate is a flat, round plate that threads to the headstock spindle of the lathe.

- The faceplate is used for clamping and machining irregularly-shaped work-pieces that cannot be successfully held by chucks or mounted between centers.
- The work-piece is either attached to the faceplate using angle plates or brackets, or is bolted directly to the plate.
- Radial T-slots in the faceplate surface facilitate mounting work-pieces. The faceplate is valuable for mounting work-pieces in which an eccentric hole or projection is to be machined.
- The number of applications of the faceplate depends upon the ingenuity of the machinist. A small faceplate, known as a driving faceplate, is used to drive the lathe dog for work-pieces mounted between centers. The driving faceplate usually has fewer T-slots than the larger faceplates.
- When the work-piece is supported between centers, a lathe dog is fastened to the work-piece and engaged in a slot of the driving faceplate circular plate that fits to head stock spindle and drives or carries work to be machined.

3. Drill Chuck.

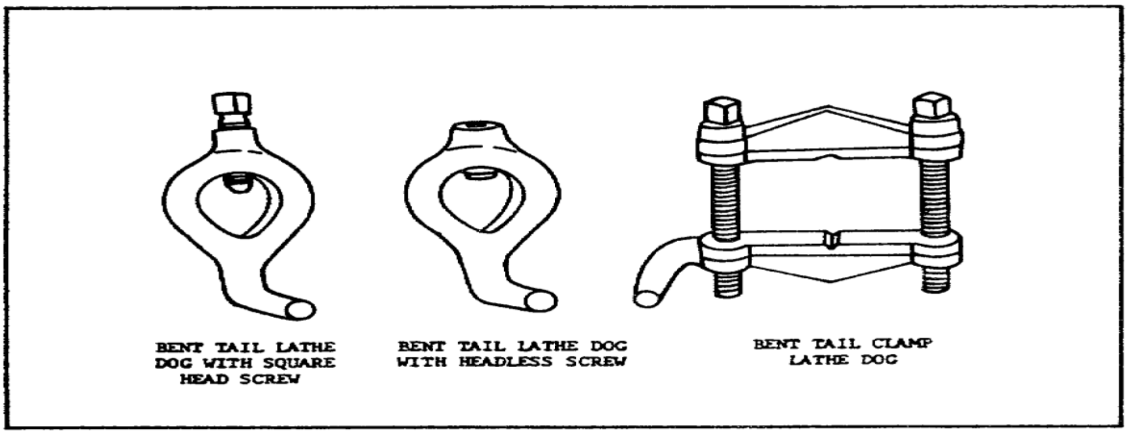
- The drill chuck is a small universal-type chuck which can be used in either the headstock spindle or in the tailstock for holding
  - ✓ Straight-shank drills,
  - ✓ reamers,
  - ✓ taps, or
  - ✓ Small-diameter work-pieces.
- The drill chuck has three or four hardened steel jaws which are moved together or apart by adjusting a tapered sleeve within which they are contained.
- The drill chuck is capable of centering tools and small-diameter work-pieces to within 0.002 or 0.003 of an inch when firmly tightened.



DRILL CHUCK

4. Lathe Dogs.

- Lathe dogs are cast metal devices used in conjunction with a driving plate or a faceplate to provide a firm connection between the headstock spindle and the work-piece that is mounted between centers. This firm connection permits the work-piece to be driven at the same speed as the spindle under the strain of cutting.
- Frictional contact alone, between the live center and the work-piece, is not sufficient to drive the work-piece.
- common types of lathe dogs are
  - ➢ Bent tails with head
  - ➢ Straight tails.
  - ➢ Bent tails without head
- When the bent tail dogs are used, the tail fits into a slot of the driving face plate.
- When straight tail dogs are used, the tail bears against a stud projecting from the faceplate.
- The bent tail lathe dog with a headless setscrew is considered safer than the dog with the square head screw because the headless setscrew reduces the danger of the dog catching in the operator's clothing and causing an accident.
- The bent tail clamp lathe dog is used primarily for holding rectangular work-pieces.



5. **Steady and follow rests:**

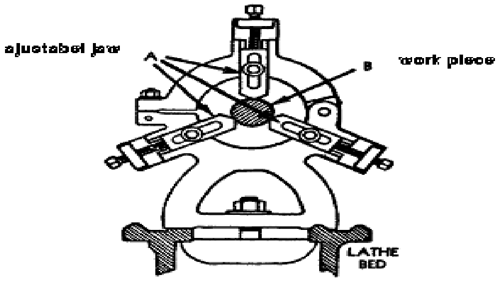
- Work-pieces often need extra support, especially long, thin work-pieces that tend to spring away from the cutter bit.
- Two common supports or rests are
  - ✓ The steady rest and
  - ✓ The follower rest.

1. Steady Rest.

- The steady rest or center rest is used to support long work-pieces or shafts being machined between centers or for boring operations.
- It is also used for internal threading operations where the work-piece projects a considerable distance from the chuck or faceplate.
- The steady rest is clamped to the lathe bed at the desired location and supports the work-piece within three adjustable jaws.
- The rest prevents the work-piece from springing under cut. The work-piece must be machined with a concentric bearing surface at the point where the steady rest is to be applied.
- The jaws must be carefully adjusted for proper alignment and locked in position. The area of contact must be lubricated frequently. The top section of the steady rest swings away from the bottom section to permit removal of the work-piece without disturbing the jaw setting.

2. **Follower Rest**

- The follower rest is used to back up a work-piece of small diameter to keep it from springing under the stress of the cutting operation.
- The follower rest gets its name because it follows the cutting tool along the work-piece. The follower rest has one or two jaws that bear directly on the finished diameter of the work-piece opposite and above the cutting tool.
- The rest is bolted to the lathe carriage so that it will follow the cutter bit and bear upon that portion of the work-piece that has just been turned. The cut must be started and continued for a short longitudinal distance before the follower rest is applied. The rest is generally used only for straight turning or threading long, thin work-pieces.



6. CUTTING TOOL HOLDING DEVICE

1. 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

2. 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.

3. 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.

4. A threading tool holder

A threading tool holder is designed to hold a special form-relieved thread cutting tool

5. 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.

6. 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.

7 Boring tool holder

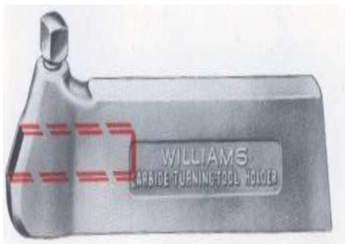
Boring tool holder: are made in several styles. Alight duty tool holder, medium boring tool holder and a heavy-duty boring bar holder.

.A

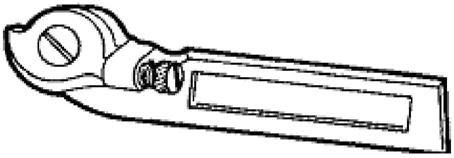
Left hand tool holder

Straight tool holders

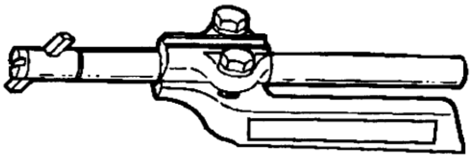
Right hand tool holder



carbide tool holder



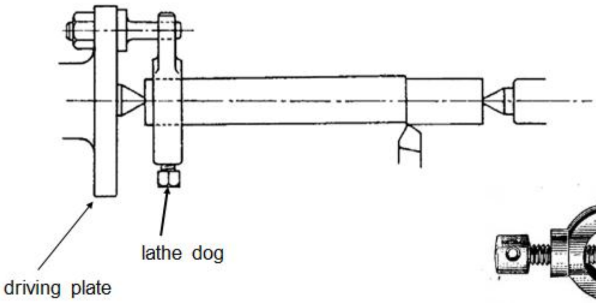
Thread cutting tool holder

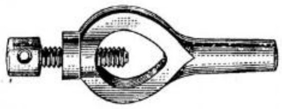


Boring tool holder

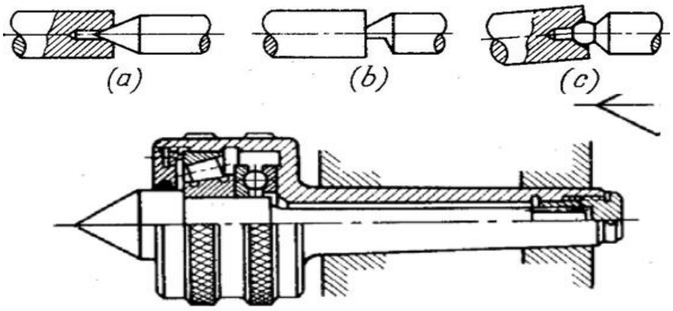
7. **Jobs requiring support from the tailstock are usually mounted**

a. In-between the centers



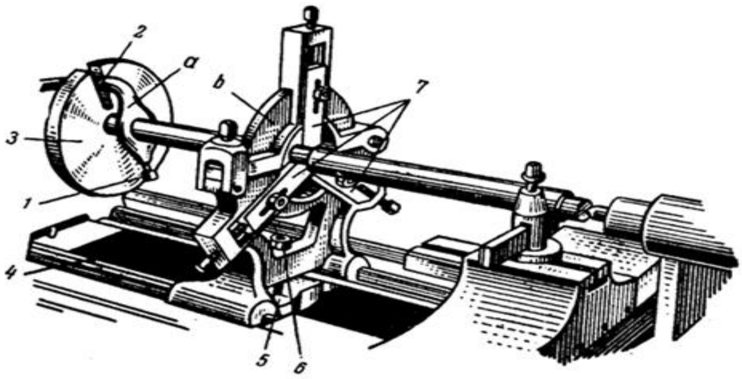


Depending upon the situation or requirement, different types of centers are used at the tailstock end.



b. In-between the chuck and center: Heavy and reasonably long jobs which have large diameters and required heavy cuts are essentially held strongly and rigidly in the chuck at headstock with support from the tailstock through a revolving center.

c. In-between headstock and tailstock with additional support of rest: To prevent deflection of the long slender jobs like feed rod, lead screw etc. due to sagging and cutting forces during machining, and some additional support are provided.

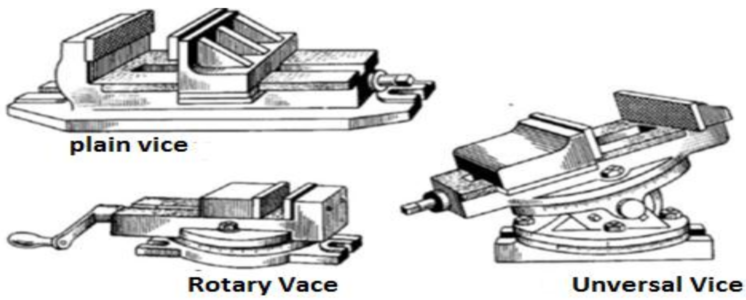


Work held between headstock and tailstock with steady rest

1.3. Mounting of Jobs in Drilling and milling machines

Mounting of Jobs are considered as:

1. By directly clamp when job is heavy and/or of odd shape and size
2. In a vice which is clamped on the bed.



Fixtures - is a work - holding device fastened to the table of a machine or to a machine accessory, such as a rotary table. Fixtures may be constructed to hold one or several parts at one time and should permit the changing of work pieces. The work may be positioned by stops, such as pins, strips, or set screws, clean the chips and cuttings from a fixture before mounting a new work piece to produce uniform work pieces.

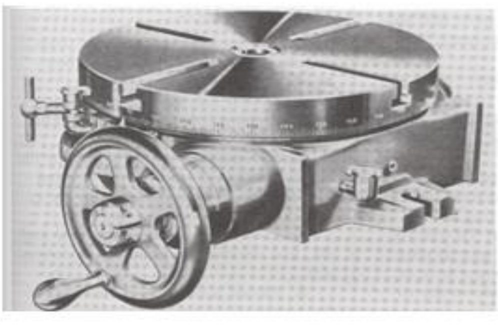


Fig. A hand-feed rotary table

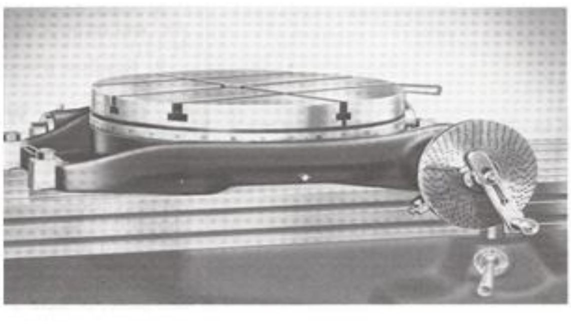


Fig. A rotary table with an indexing attachment

Self-Check -1	Written Test

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

1. One of the following is not holding work pieces in lathe machine
 A. Chucks B. Rests C. Mandrels D. Face plate E. None
2. ------------ Centre is fitted on the headstock spindle and rotates with the work
 A. Live B. Dead C. Rests D. Face plate
3. Name three work holding devices in a lathe machine? (3point)

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

Answer Sheet

Score = ____________

Rating: _____________

Name: ___________________________ Date: ________________

Short Answer Questions

Information Sheet-2	Mounting and securing Materials in clamping devices

2.1. Methods of mounting of jobs on machine tools

The job or work piece and the cutting tools essentially need to be properly mounted in the machine tool for achieving desired performance of the machining system.

2.1.1 Mounting the job or work piece in the machine tool

- appropriate selection of work holding device or system from the available resources depending upon;
  - ✓ Configuration of the machine tool
  - ✓ Shape, size and weight of the blank
  - ✓ Kind of machining work to be done
  - ✓ Order of dimensional accuracy desired
  - ✓ Volume (number of same job) of production
- Correct location, strong support and rigid clamping of the work piece against the cutting and other forces.
- easy and quick loading and unloading to and from the machine tool or the holding device
- proper alignment like coaxially, concentricity etc. of rotating jobs

Finally free flow of chips and cutting fluid.

2.2. Mounting work piece on Milling and Drilling machine

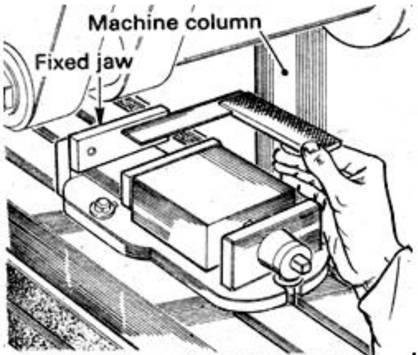
a. Aligning the vise

Two common methods of aligning the vise with the machines are:

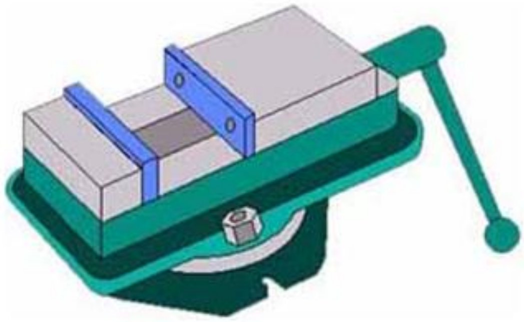
- using a dial indicator
- using a square

Using the dial indicator is the more accurate method of aligning the vise. The dial indicator is clamped to the machine arbor and passed across the face of the fixed jaw by operating the hand cross-feed.

A faster but less accurate method is to use the try square placed against the vertical column of the machine to align the fixed jaw of the vise



Aligning vice using a try square

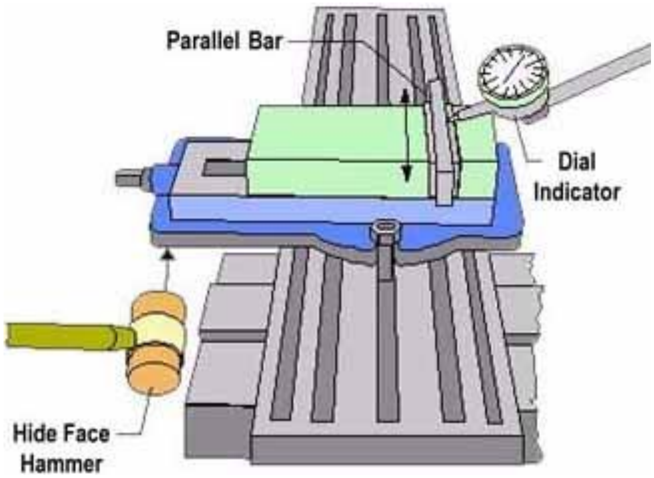


b. Milling Set Up

Correct use of holding device and a good set up are of crucial importance in achieving a safe, accurate, and efficient operation of the machine. Large work piece can be mounted directly onto the machine table by means of screws while small work pieces are usually held by machine vice as shown in figure bellow. In either case, a dial indicator is used for alignment checking.

C.**Vice Alignment**

In the setting up of the vice onto the machine table, the fix jaw of the vice must be set parallel to the machine table using a Parallel Bar and a Dial Indicator as illustrated in figure. Adjustments can only be made by using a hide face hammer to correct its position such that a near zero indicator movement is achieved at all positions along the parallel bar.



d. Work Holding Method

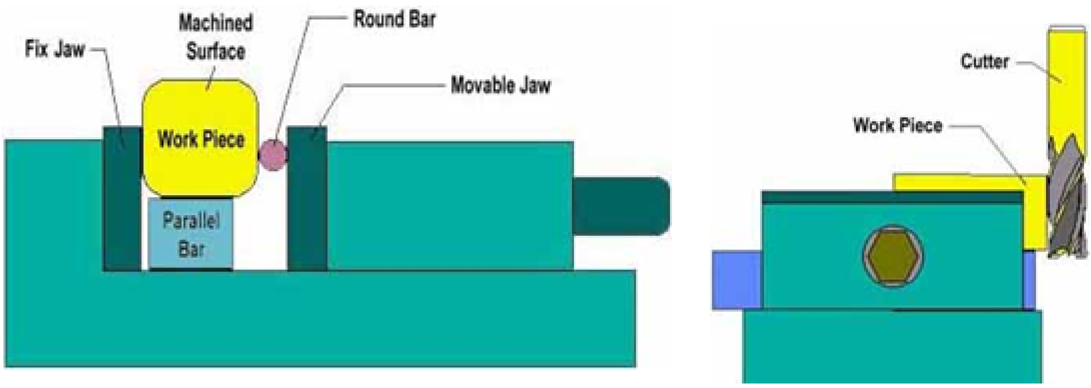
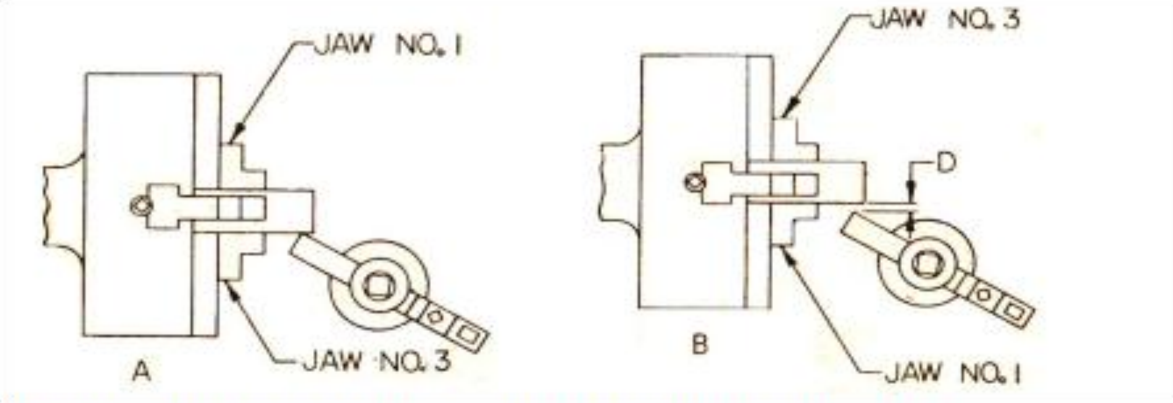


Figure: Holding Method by Using a Machine Vice

MOUNTING AND CENTERING OF WORKPIECE ON THE LATHE

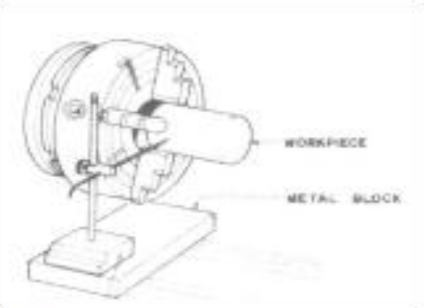
Mounting Work in a Four Jaw Independent Chuck
Visual Inspection Approach
Using High Spot – Low Spot Technique

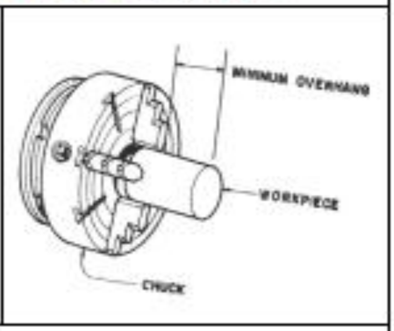


(A) High Spot is the highest point that touches the point of any centering devices.
(B) Low Spot is the gap created as it moves away from the workpiece
Illustration show the back of a loose tool holder as a means to determine equidistance

Visual Estimate Inspection Method

1. Chalk method /Surface gage /wriggler method/Alignment of Chuck's jaw





NOTE: Do visual inspection by;

1. Loosen up the jaw opposite of the high spot and screw tight the high spot's jaw to push the workpiece
2. Having plenty of light while using scratch white paper as background distinguisher between workpiece gap and the point of a centering devices
3. Repeating the procedures until an equal gap was visible in four front of the chuck's jaw. Tighten the high spot area if gap in the low spot equal 0.5mm.
4. Checking for wobble spin and remedied it by light tapping

2.3 Mounting a Work on milling machine

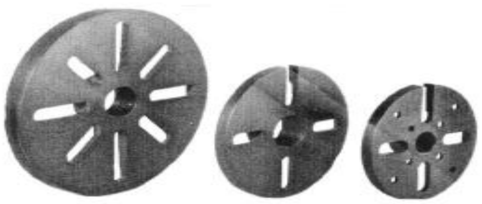
An efficient and positive method of holding work pieces to the milling machine table is essential if the machine tool is to be used to advantage. Regardless of the method used in holding, there are certain factors that should be observed in every case. The work piece must not be sprung in clamping; it must be secured to prevent it from springing or moving away from the cutter; and it must be so aligned that it may be correctly machined.

- Milling machine worktables are provided with several T-slots, used either for clamping and locating the work piece itself or for mounting various holding devices and attachments. These T-slots extend the length of the table and are parallel to its line of travel. Most milling machine attachments, such as vises and index fixtures, have keys or tongues on the underside of their bases so that they may be located correctly in relation to the T-slots.

2.4 Methods of mounting and securing work on clamping devices

(1) Clamping a Work piece To the Table.
(2) Clasping a Work piece to the Angle Plate.
(3) Clamping Work pieces in Fixtures.
(4) Holding Work pieces Between Centers.
(5) Holding Work pieces in a Chuck.

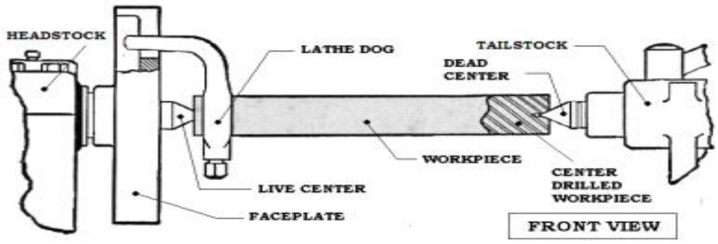
(6) Holding Work pieces in the Vise.

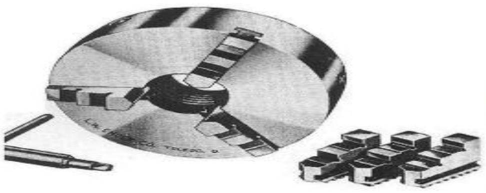


FACEPLATE

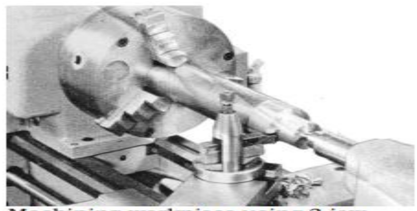
Machining work mounted between centers.

LATHE DOG (bent tail)



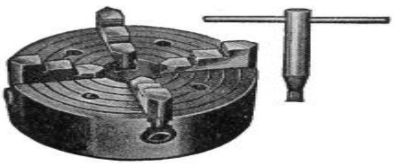


3 JAW UNIVERSAL CHUCK



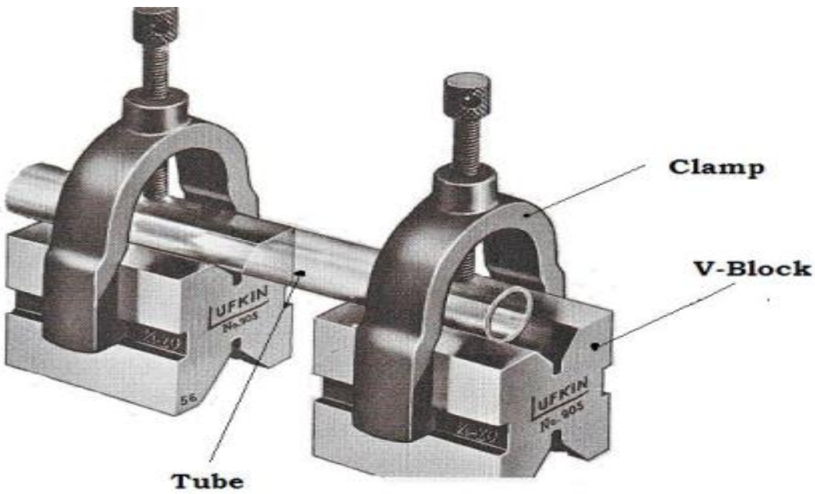
Machining workpiece using 3 jaw chuck

- Permits all jaws to operate simultaneously. This automatically centers round or hexagonal shaped stock



4 JAW INDEPENDENT CHUCK

Machining workpiece using 4 jaw chuck



Faceplates

- A lathe faceplate is a flat, round plate that threads to the headstock spindle of the lathe.
- The faceplate is used for clamping and machining irregularly-shaped work-pieces that cannot be successfully held by chucks or mounted between centers.
- The work-piece is either attached to the faceplate using angle plates or brackets, or is bolted directly to the plate.

- Radial T-slots in the faceplate surface facilitate mounting work-pieces. The faceplate is valuable for mounting work-pieces in which an eccentric hole or projection is to be machined.
- The number of applications of the faceplate depends upon the ingenuity of the machinist. A small faceplate, known as a driving faceplate, is used to drive the lathe dog for work-pieces mounted between centers. The driving faceplate usually has fewer T-slots than the larger faceplates.
- When the work-piece is supported between centers, a lathe dog is fastened to the work-piece and engaged in a slot of the driving face plate a circular plate that fits to head stock spindle and drives or carries work to be machined.

Lathe centers

- Lathe centers are the most common devices for supporting work-pieces in a lathe.
- Most lathe centers have a tapered point with a 60^0 included angle to fit the work-piece holes with the same angle.
- The work-piece is supported between two centers, one in the headstock spindle and one in the tailstock spindle. Centers for lathe work have standard tapered shanks that fit into the tailstock directly and into the headstock spindle, using a center sleeve to convert the larger bore of the spindle to the smaller taper size of the lathe center.

Self-Check -2	Written Test

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

1. Write the two common methods of aligning the vise with the machines?
2. Which one not the selection of work holding device or system from the available resources?
   A. Configuration of the machine tool
   B. Shape, size and weight of the blank
   C. Kind of machining work to be done
   D. Dimensional accuracy desired
   E. None of the above

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

Answer Sheet

Score = ____________
Rating: _____________

Name: ____________________________ Date: _________________

Short Answer Questions

Information Sheet-3	Operating machines

3.1 Lathe machine Operations

3.1.1 Facing

Facing is the operation of machining the ends of a piece of work to produce flat Surface Square with the axis. The operation involves feeding the tool perpendicular to the axis of rotation of the work. *Facing operation*

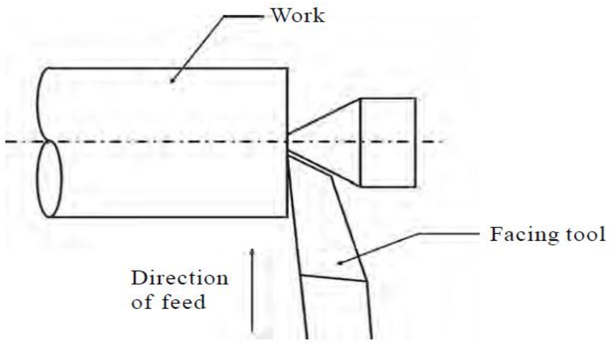


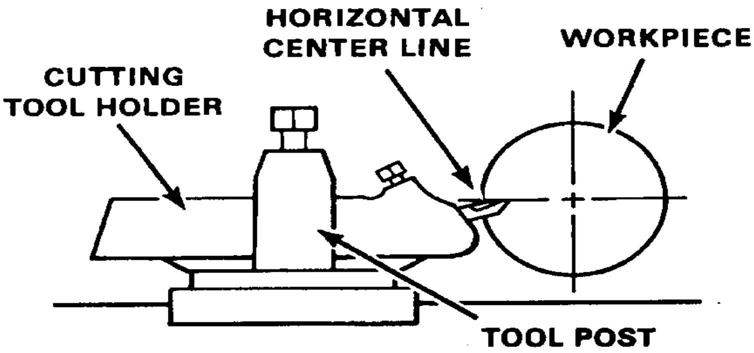
Fig Facing

FACING

- Facing is the square finishing of the ends of the work-piece and is often used to bring the piece to a specified length.
- In facing operations, the cutter bit does not traverse laterally (left or right) but cuts inward or outward from the axis of the piece.
- Facing of the ends is usually performed before turning operations.

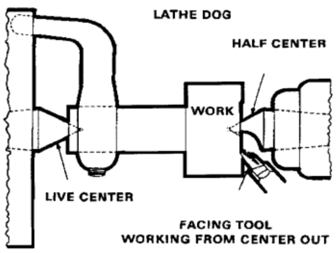
A. Facing Work in a Chuck

- Facing is usually performed with the work held in a chuck or collets.
- Allow the work piece to extend a distance no more than 1 1/2 times the work diameter from the chuck jaws.
- And use finishing speeds and feeds calculated using the largest diameter of the work piece.
- The tool bit may be fed from the outer edge to the center or from the center to the outer edge.
- Normal facing is done from the outer edge to the center since this method permits the operator to observe the tool bit and layout line while starting the cut. This method also eliminates the problem of feeding the tool bit into the solid center portion of the work piece to get a cut started.
- Use a left-hand finishing tool bit and a right-hand tool holder when facing from the outer edge toward the center.
- Work that has a drilled or bored hole in the center may be faced from the center out to the outer edge if a right-hand finishing tool bit is used.
- Avoid excessive tool holder and tool bit overhang when setting up the facing operation. Set the tool bit exactly on center to avoid leaving a center nub on the work piece.
- Use the tailstock center point as a reference point when setting the tool bit exactly on center.
- If no tailstock center is available, take a trial cut and readjust as needed. If using the cross slide power feed to move the tool bit (into the center), disengage power when the tool bit is within l/16 inch of the center and finish the facing cut using hand feed.



B. Facing Work between Centers

- Sometimes the work piece will not fit into a chuck or collets, so facing must be done between centers. To properly accomplish facing between centers, the work piece must be center-drilled before mounting into the lathe.
- A half male center (with the tip well lubricated with a white lead and oil mixture) must be used in the lathe tailstock to provide adequate clearance for the tool bit.
- The tool bit must be ground with a sharp angle to permit facing to the very edge of the center drilled hole Start the facing cut at the edge of the center-drilled hole after checking for tool bit clearance, and feed the cutting tool out to the edge.
- Use light cuts and finishing feeds, which will reduce the tension, put on the half male center.
- Replace the half male center with a standard center after the facing operation, since the half male center will not provide adequate support for general turning operations.
- Only a small amount of material can be removed while facing between centers.
- If too much material is removed, the center-drilled hole will become too small to support the work piece.



Turning

Turning in a lathe is to remove excess material from the work piece to produce cylindrical surface of required shape and size

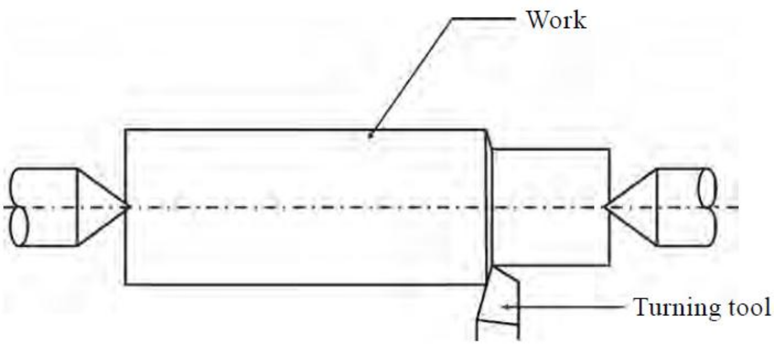


Fig Turning

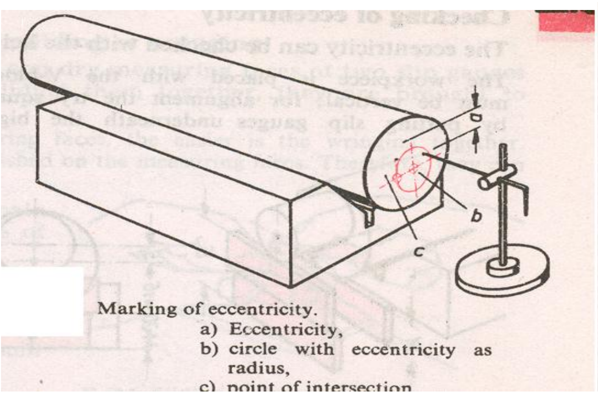
- Straight turning may he performed upon a work-piece supported in a chuck, but the majority of work pieces turned on an engine lathe are turned between centers.
- Turning is the removal of metal from the external or internal surface of cylindrical work pieces using various types of cutter tool bits.

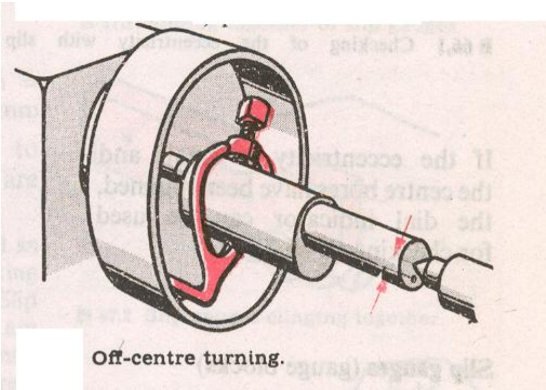
3.1.2 ECCENTRIC TURNING

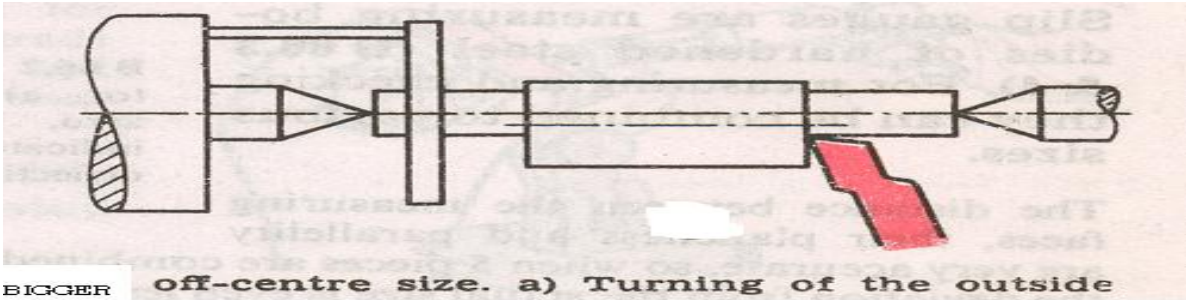
- If the off-center size is beg enough both the required centers can be drilled.
- First the big diameter and then the off-center journal have to be turned. If the off-center size is small then the bigger diameter finished first.

Eccentric turning is done

- ➢ Using lathe dogs and face plates by drilling the two centers.
- ➢ On four jaws chuck by off-setting the independent jaw equal to the off-set center.

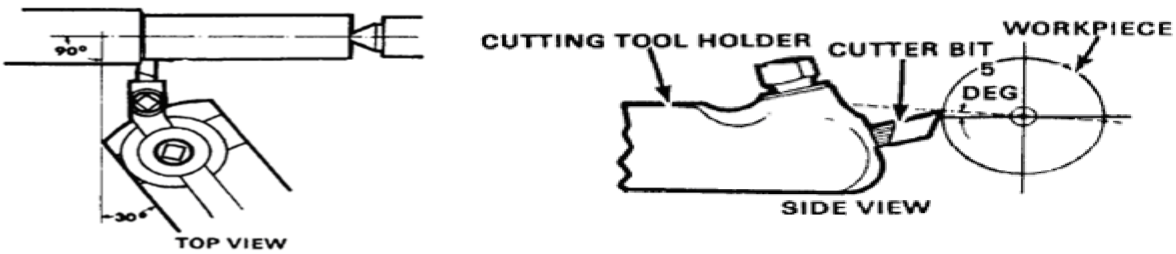

Marking of eccentricity.
a) Eccentricity,
b) circle with eccentricity as radius,
c) point of intersection


Off-centre turning.


BIGGER off-centre size. a) Turning of the outside

3.1.3 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.



Taper may be defined as a uniform increase or decrease in diameter of a piece of work measured along its length.

Taper turning methods

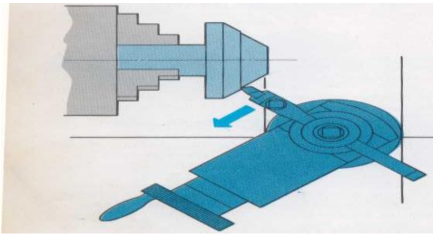
1. Form tool method
2. Compound rest method
3. Tailstock set over method
4. Taper turning attachment method

3.1.4 Taper turning

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

3.1.4 Taper turning with compound rest:

- The compound rest is generally used for turning or boring short steep tapers, but it can also be used for longer, gradual tapers, providing the length of the taper does not exceed the distance the compound rest will move upon its slide.
- This method can be used with a high degree of accuracy, but is somewhat limited due to the lack of an automatic feed and the length of the taper being restricted to the movement of the slide.



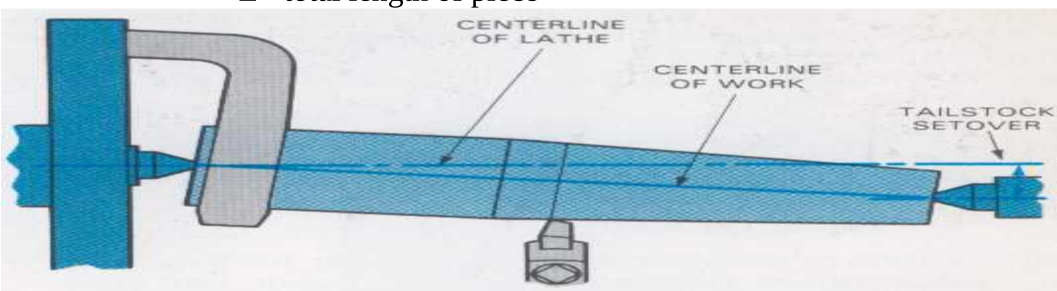
When turning a taper using compound rest, note cut is made from small diameter to large diameter.

3.1.5 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 vary, different will be produced with the same tail stock offset.

☞ Formula used ; Offset, $S = \frac{LX(D-d)}{2Lo}$

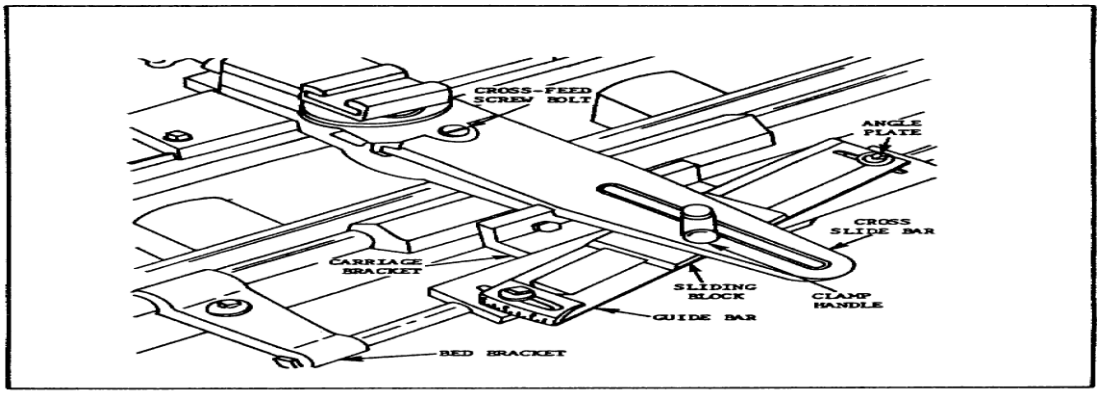
Where D= diameter of large end
d= diameter of small end
Lo= length of taper
L= total length of piece



Study how to machine a taper by offset tailstock technique.

3.1.6 Turning a taper with taper turning attachment:

- It is an accurate way to cut tapers and offers advantages over other methods of machining tapers.
  - **a)** Internal and external tapers can be cut and accurate fit is assured for mating parts.
  - **b)** Work can be held by any conventional means.
  - **c)** 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.



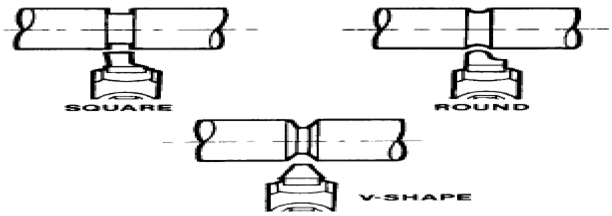
Note lathe fitted with telescopic taper attachment.

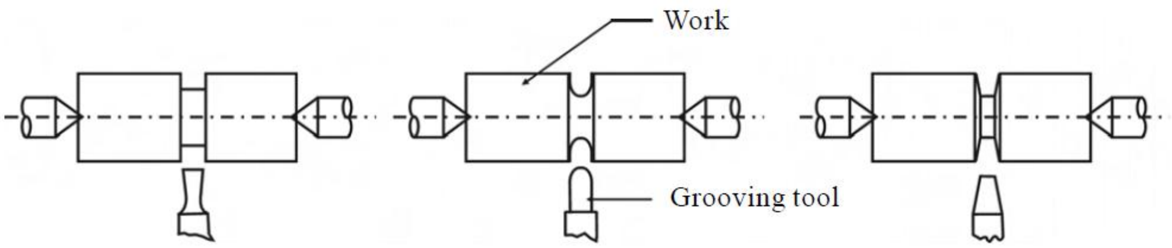
3.1.7 Grooving and parting operations

Grooving:

Grooving is the process of cutting a narrow groove on the cylindrical surface of the work piece. It is often done at end of a thread or adjacent to a shoulder to leave a small margin. The groove may be square, radial or beveled in shape

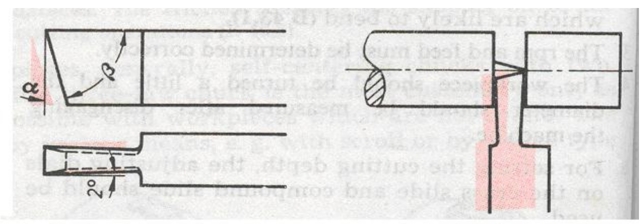
- It is commonly called
  - recessing,
  - undercutting, or
  - Necking is often done at the end of thread to permit full travel of the nut up to a shoulder, or at the edge of a shoulder to ensure a proper fit of mating parts.
    Grooves are generally
    - square,
    - round, or
    - V-shaped.





3.1.9 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.

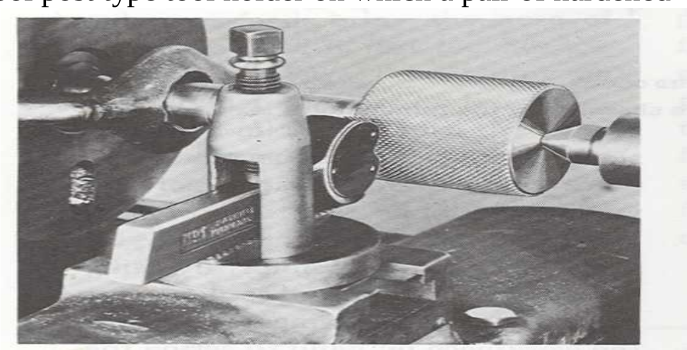


3.2 KNURLING:

Knurling is the process of embossing a diamond shaped pattern on the surface of the work piece. The knurling tool holder has one or two hardened steel rollers with edges of required pattern. The tool holder is pressed against the rotating work. The rollers emboss the required pattern. The tool holder is fed automatically to the required length. Knurls are available in coarse, medium and fine pitches. The patterns may be straight, inclined or diamond shaped.

Knurls are available in coarse, medium and fine pitches. The patterns may be straight, inclined or diamond shaped.

- Knurling 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 are mounted.



Diamond –point knurling of a cylindrical work piece

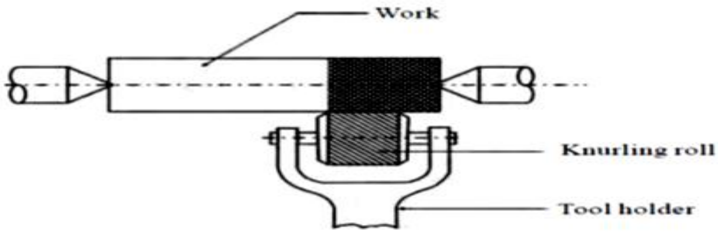


Fig **Knurling**

The purpose of knurling is:

1. To provide an effective gripping surface
2. To provide better appearance to the work
3. To slightly increase the diameter of the work

3.3 Chamfering

Chamfering is the operation of beveling the extreme end of the work piece. The form tool used for taper turning may be used for this purpose. Chamfering is an essential operation after thread cutting so that the nut may pass freely on the threaded work piece.

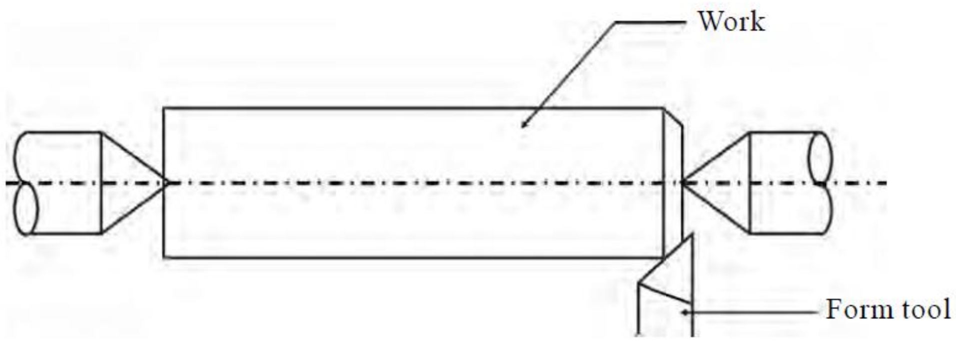
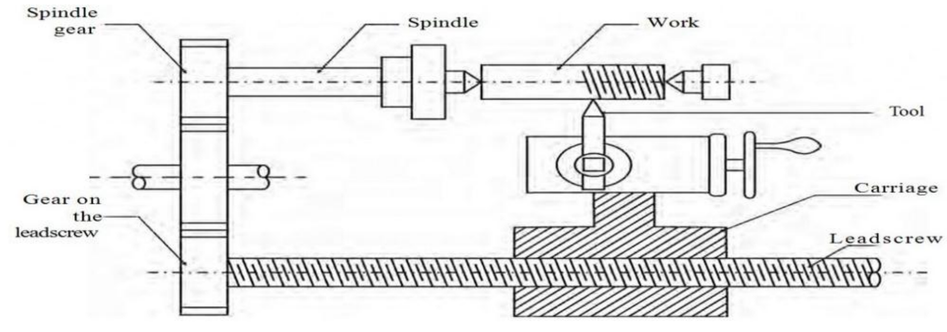


Fig: Chamfering

Thread cutting

Thread cutting is one of the most important operations performed in a lathe. The process of thread cutting is to produce a helical groove on a cylindrical surface by feeding the tool longitudinally.



Thread cutting

3.4 THREAD CUTTING

- A thread is a uniform helical groove cut on or in a cylinder or cone. Thread cutting on a lathe is one of the most exacting lathe operations.
- It requires a thorough knowledge of the principles and procedures of thread cutting.
- It ties together a number of operations and dimensions in such a way that accuracy must be maintained to achieve a proper working thread.
- Before attempting such operations, the operator should have knowledge of the fundamental principles of threads and the types in general use.

A. Screw Thread Terminology

1. *External or Male Thread*. A thread on the outside of a cylinder or cone.
2. *Internal or Female Thread*. A thread on the inside of a hollow cylinder or bore.
3. *Pitch (P)*. The distance from a given point on one thread to a similar point on a thread next to it, measured parallel to the axis of the cylinder. The pitch in inches is equal to one divided by the number of threads per inch.

1. *Lead*.
The distance a screw thread advances axially in one complete revolution. On a single-thread screw, the lead is equal to the pitch. On a double-thread screw, the lead is equal to twice the pitch, and on a triple thread screw, the lead is equal to three, times the pitch.
2. *Crest (also called "flat")*. The top or the outer surface of the thread joining the two sides.
3. *Root*. The bottom or inner surface joining the sides of two adjacent threads.
4. *Side*. The side of a thread is the surface which connects the crest and the Root.
5. *Thread Angle*. The angle between the sides of the adjacent threads, measured in an axial plane. For most V-threads, the angle is fixed at 600.

6. *Depth*. The depth of the thread is the distance between the crest and the root of a thread, measured perpendicular to the axis.
7. *Major Diameter (D).* The major diameter is the largest diameter of a screw thread.
8. *Minor Diameter (K).* The minor diameter is the smallest diameter of a screw thread.
9. *Pitch Diameter*. The pitch diameter is the diameter of an imaginary cylinder formed where the width of the groove is equal to one-half of the pitch. This is the critical dimension of threading as the fit of the thread is determined by the pitch diameter.
10. *Number of Threads per Inch*/mm. The number of threads per inch may be counted by placing a rule against the threaded parts and counting the number of pitches in 1 inch. A second method is to use the screw pitch gage. This method is especially suitable for checking the finer pitches of the screw threads.

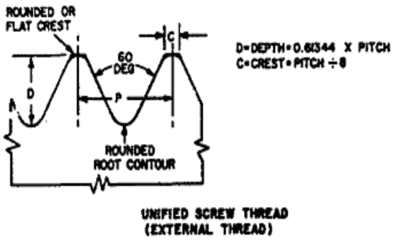
B. *Screw Thread Forms.*

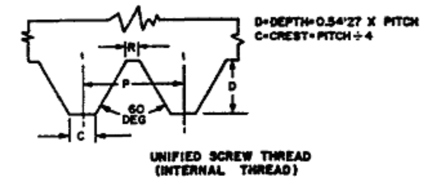
The most commonly used thread forms are discussed below.

- The Acme,
- Square, threads are the most common used for devices which function to transmit motion.

1. V- threads

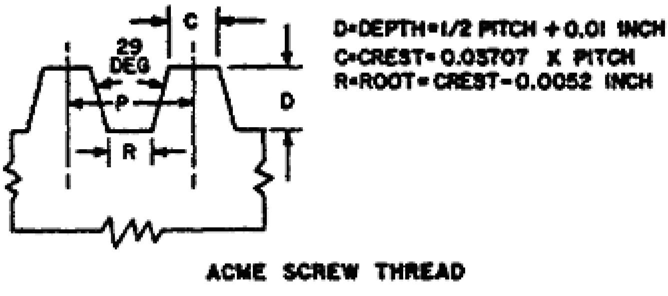
V- Threads with a 60° thread angle. (National) Standard thread, a flat should be carefully ground at the point of the bit, perpendicular to the center line of the 60° thread angle.





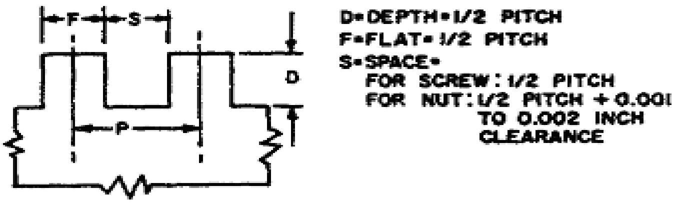
2. ***Acme Screw Thread*.**

- The Acme screw thread form classified as a power-transmitting type of thread. This is because the 29° included threaded angle at which its sides are established reduces the amount of friction when matching parts are under load. Because the root and crest are wide, this thread form is strong and capable of carrying a heavy load. The Acme thread form is specially suitable



2. *Square Screw Thread*.

Because of their design and strength, square screw threads are used for vise screws, jack screws, and other devices where maximum transmission of power is needed. All surfaces of the square thread are square with each other and the sides are perpendicular to the center axis of the threaded part. Because the contact areas are relatively small and do not wedge, friction between the matching threads is reduced to a minimum under heavy pressure.

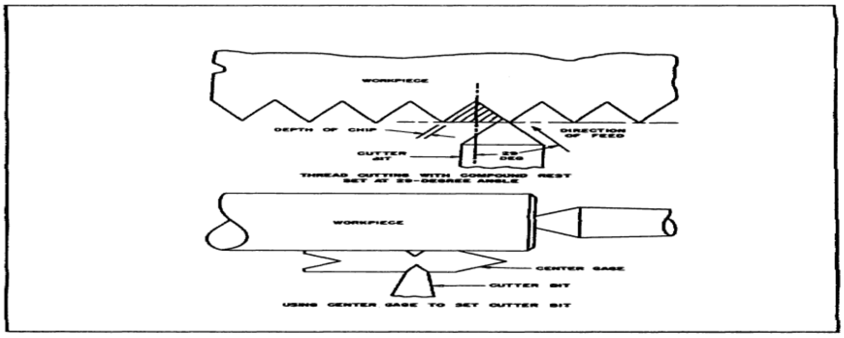


SQUARE SCREW THREAD

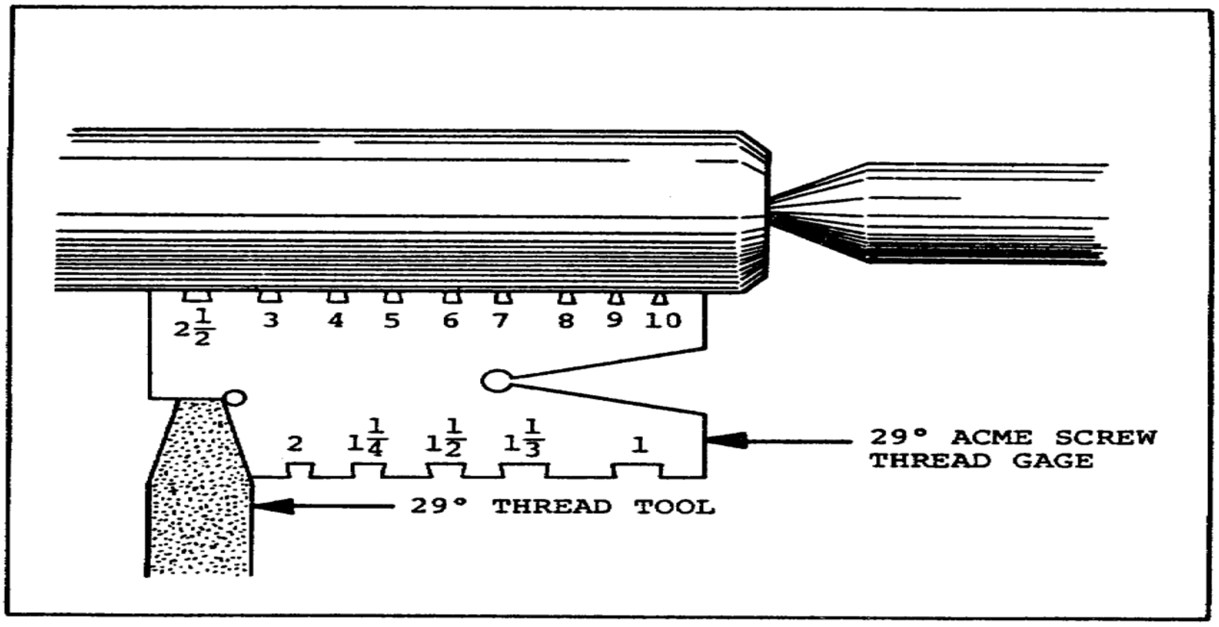
3.5 Setting Lathe for Thread Cutting

- In cutting threads on a lathe, the pitch of the thread or the number of threads per inch obtained is determined by the speed ratio of the headstock spindle and the lead screw which drives the carriage.
- Lathes that are equipped for thread cutting have gear arrangements for varying the speed of the lead screw.
- Most modern lathes have a quick-change gearbox for varying the lead screw to the spindle ratio, but many older lathes, modern inexpensive lathes, and special lathes come equipped with standard change gears which must be arranged by computation to achieve the desired speed ratio.

☞ *Quick-Change Gearbox* for lathes equipped with quick-change gearboxes, the operator need only follow the instructions on the direction plates of the lathe to set the proper feed to produce the desired number of threads per inch. Once set to a specific number of threads per inch, the spindle speed can be varied depending upon the material being cut and the size of the work piece, without affecting the threads per inch.

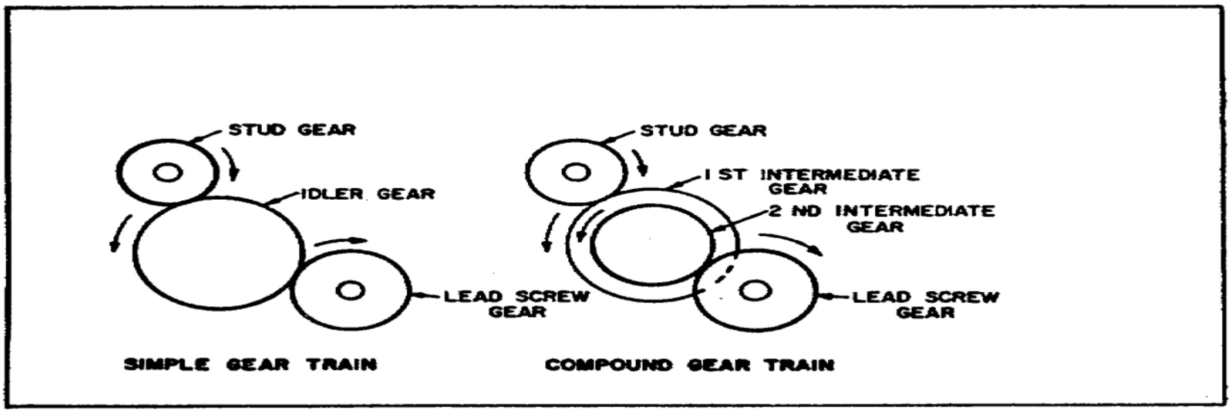


SETTING UP CUTTER BIT FORTHREAD CUTTING OPERATIONS



Standard Change Gears.

- Lathes equipped with standard-change gears require that the operator be familiar with the methods of selecting the proper gears to produce the desired thread pitch in case the manufacturer supplied gear tables are missing.
- On most lathes with standard change gears, the gears may be arranged in a simple gear train or in a compound gear train.



- ❖ For example, to cut 80 threads per inch with a lathe having a lead screw of 8 threads per inch, the ratio would be 8:80 (8 units to 80 units).

Factoring each term, 8=2X4 (factors), and 80=8X10 (factors). Then multiplying 2, 4, 8, and 10 each by a convenient number, say 12, the result is the ratio, 24X48: 96X120. The gearing then must be:

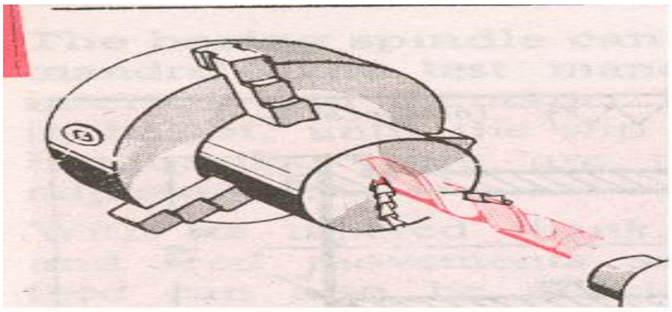
Stud gear-------------------------24 teeth
First intermediate gear-----------96 teeth
Second intermediate gear----------48 teeth
Lead screw gear------------------120 teeth

3.7 Drilling, Boring, Reaming, and counter boring

3.7.1 DRILLING

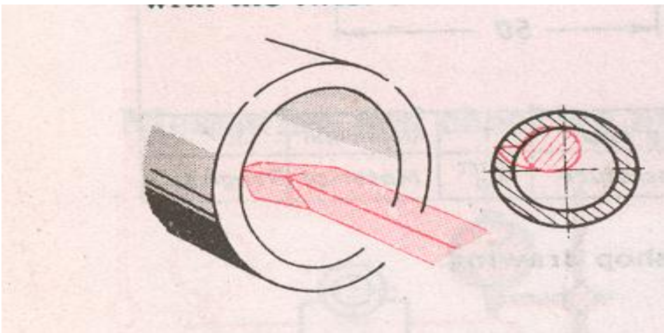
- Work held in a chuck can be drilled quickly and accurately in a lathe.

- The drill held in a drill chuck or in the tailstock spindle, brought against the revolving work by turning the tailstock handle wheel. Various methods can be used to hold drills in a lathe depending on the size of the drill being used.
- the most common methods are
  - I- Straight shank drills in a drill chuck.
  - II- Taper shank drills in the tailstock spindle taper.
  - III- Large taper shank drills in a drill holes.



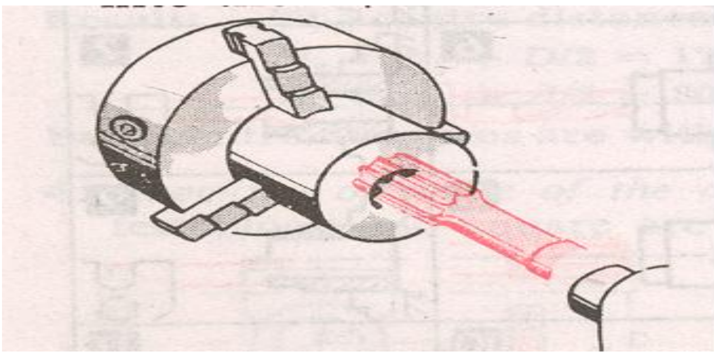
3.7.2 Boring:

- 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.



3.7.3 Reaming

- It 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.

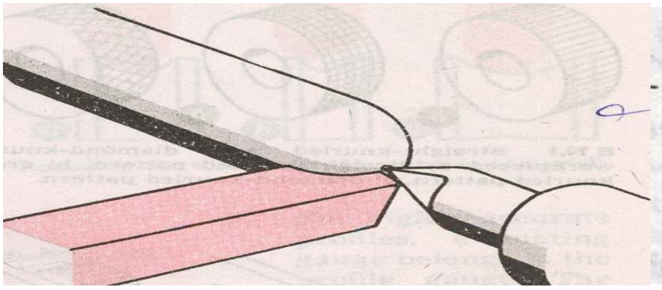


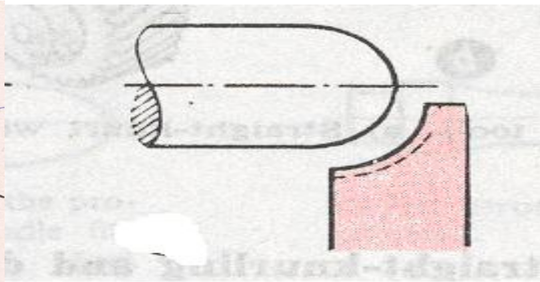
3.7.4 Counter boring:

- It 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.
- Counter bores are supplied in a variety of styles, each having a pilot in the end to keep the tool in line with the hole being counter bored.

3.8 PROFILE TURNING

Radii and other shapes on a work piece are produced by profile turning. Generally, profile tools corresponding to the desired shape of the work-piece are used.





3.9 Milling Machine operations

Although the majority of operations performed on a peripheral milling machine are either plain milling or side milling, several other operations or combinations of operations may be performed.

3.9.1 Plain or slab milling: the production of a horizontal flat surface parallel to the axis of the milling machine arbor. The work piece may held in a vise or fixture or fasten directly to the table and the cutter width extends beyond the work piece on both sides.

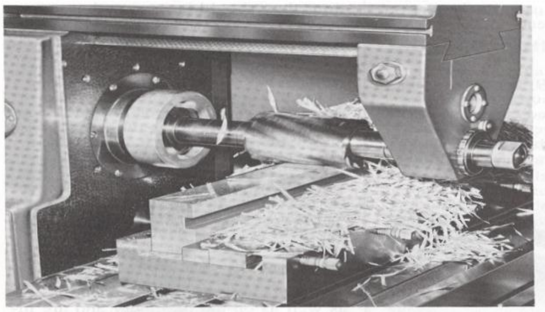


Fig Milling the surface of a work piece held in a fixture.

3.9.2 Side milling: is the process of machining a vertical flat surface perpendicular to the axis of the milling machine arbor. This operation is performed by the combined action of the peripheral and side teeth on a side milling cutter.

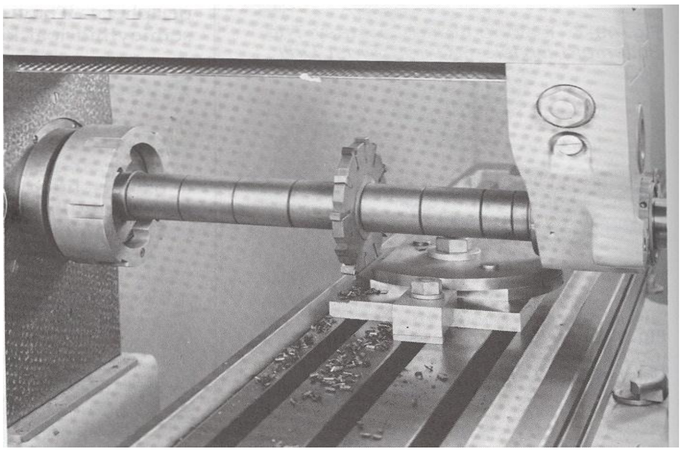


Fig: Side milling a vertical surface.

3.9.3 Face milling: is used to produce a flat surface parallel to the machine. This is done by means of a face milling cutter mounted in the milling-machine spindle. Face milling may also be done using a vertical milling attachment to produce horizontal flat surfaces. Both the periphery and the end of the teeth do the cutting.

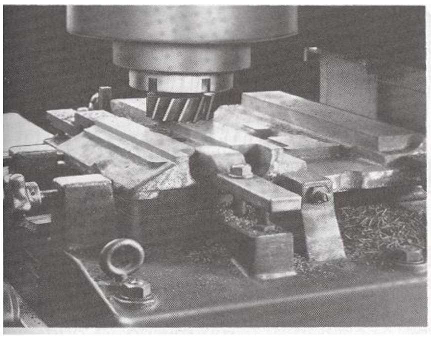


Fig: Face milling a flat surface with a shell end mill.

3.9.4 Slotting: is the process of cutting grooves or slots in the work piece. A staggered-tooth side milling cutter or an end mill can be used for this operation

The two methods of milling

1. ***Conventional and Climb Milling:*** The direction in which the work piece is fed into the cutter indicates whether conventional (up) milling or climb (down) milling is being used. In climb (down) milling method, work is fed in the same direction to the direction of rotation of the cutter.
2. ***Conventional (Up) Milling:*** In Conventional milling method, work is fed against (in opposite) the direction of rotation of the cutter. It is the most commonly used method of milling.

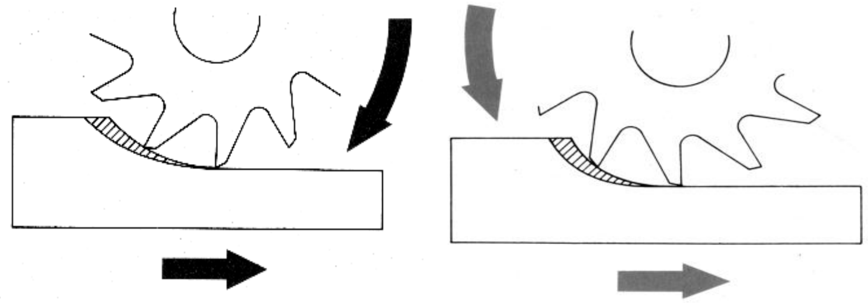


Fig. Conventional (Up) milling Fig. Climb (Down) milling

Certain types of work can best be milled by climb milling, if the machine is equipped with a backlash eliminator. Although climb milling is not as widely used as conventional milling, it has certain advantages and disadvantages.

Advantages

1. It is particularly suited to the machining of thin and hard-to-hold parts, since the work piece is forced against the table or holding device by the cutter.
2. Work need not be clamped as tightly.
3. Consistent parallelism and size may be maintained particularly on thin parts.
4. It may be used where breakout at the edge of the work piece could not be tolerated.
5. It requires up to 20 present less power to cut by this method.
6. It may be used when cutting off stock or when milling deep, thin slots.

Disadvantages

1. It cannot be used unless the machine has a backlash eliminator and the table gibs have been tightened.
2. It cannot be used for machining castings or hot rolled steel, since the hard outer scale will damage the cutter.

Self-Check -3	Written Test

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

1. ---------------- is to remove excess material from the work piece to produce a cylindrical surface of required shape and size

A. Tuning B. Facing C. Taper **D**. Knurling

2. Operations which is not on the lathe include

A. straight and shoulder turning,
B. facing,
C. grooving,
D. parting,
E. turning tapers,
F. None

3. which working condition of the lathe will determine

A. speed,
B. feed,
C. Depth of cut
D. All of the above

4. Depth of cut depends up on

A. cutting speed,
B. rigidity of machine and
C. Tool material.
D. All of the above

5. Safety rules and procedures in lathe machine operation are

A. Personal safety
B. Be sure all guards are in place before attempting to operate the machine
C. Do not operate machines while taking medication(medicine)
D. All of the above

6. One of the following is not types of turning

A. Cylindrical turning
B. Tapper turning
C. Eccentric turning
D. Decreasing the diameter of the work-pieces
E. Off-set turning
F. None of the above

7. One of the following is not types of tapper turning

A. Taper turning with compound rest
B. Taper turning by offset tailstock method
C. Turning a taper with taper turning attachment
D. Taper turning with forming tools
E. None

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

Answer Sheet

Score = ___________
Rating: ____________

Name: _________________________ Date: _______________

Short Answer Questions

Operation Sheet 1	Facing work piece on lathe machine

Procedure of facing operation on lathe machine

Step 1- Mount a three jawed chuck onto the head stock

Step 2- Install a Drill chuck into your tailstock

Step 3- Secure your stock in the three jawed chuck with about 1 inch – 2 inch extending out beyond the jaws

Step 4- Using your “right handed cutting tool” square or face off the end of the stock. You

Step 5- want to take off just enough material to clean up the end so make sure that you are not taking too aggressive of a pass.

Step 6- Insert a #4 (stamped on to the drill) center drill into the drill chuck on your tailstock Center drill the end about ¼ inches deep

Step 7- Take your part out of the three jawed chuck, flip it over, and secure it back in the chuck

Step 8- Face off the second end. Take off the smallest amount of material possible Center drill the second end about ¼ inches

Step 9- When done; remove the 3 jaw chuck from the headstock and the drill chuck from the tailstock.

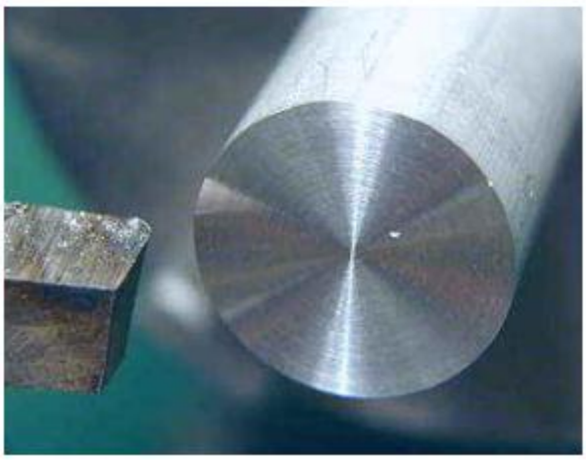


Fig: Facing work on lathe machine

Operation Sheet 2	Turning Between center by using face plate and lathe dog

Procedures for Turning Between center by using face plate and lathe dog

Step 1- Install a dead center into your headstock

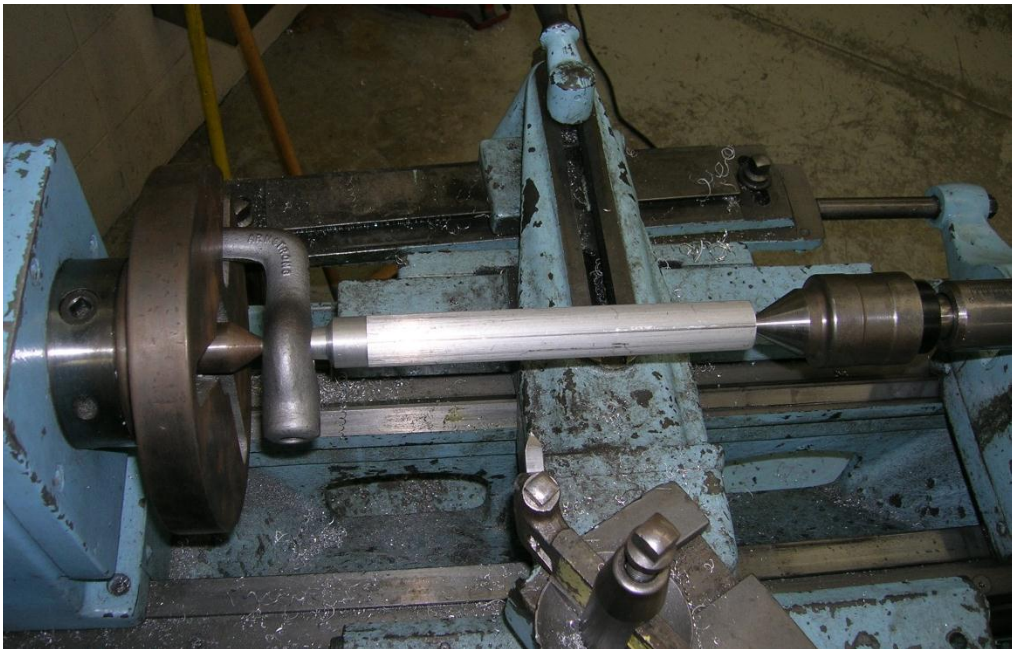
Step 2- Mount a faceplate onto your headstock

Step 3- Install a live center into your tailstock

Step 4- Secure a vise dog onto your part, securing it as close to the end as possible.

Step 5- Insert your part between centers and lock down the tailstock. Make certain that any play has been removed and that the setup is correct and secured before turning on the machine.

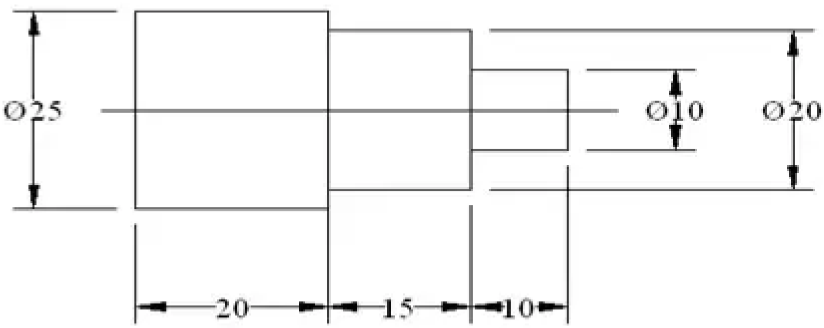
Step 6- Set up the tool on center and square to the part.



Operation Sheet-3	Machining step shaft from aluminum ingot

Procedures for machining of step shaft from aluminum ingot

Step 1- Cutting the work piece by using power hacksaw
Step 2- Adjust the machine with proper parameters (speed or revolution per minute)
Step 3- Mount the cutter and the work pieces in proper way
Step 4- Face the two ends
Step 5- Turn the first step according to drawing
Step 6- Turn the second step according to drawing
Step 7- Turn the third step according to drawing
Step 8- Polishing the work pieces by using sand paper



.

Operation Sheet-4	Machining of taper shaft from aluminum ingot

Procedures for machining of taper shaft from aluminum ingot

Step1- Cutting the work piece by using power hacksaw

Step2- Adjust the machine with proper parameters

Step3- Mount the cutter and the work pieces in proper way

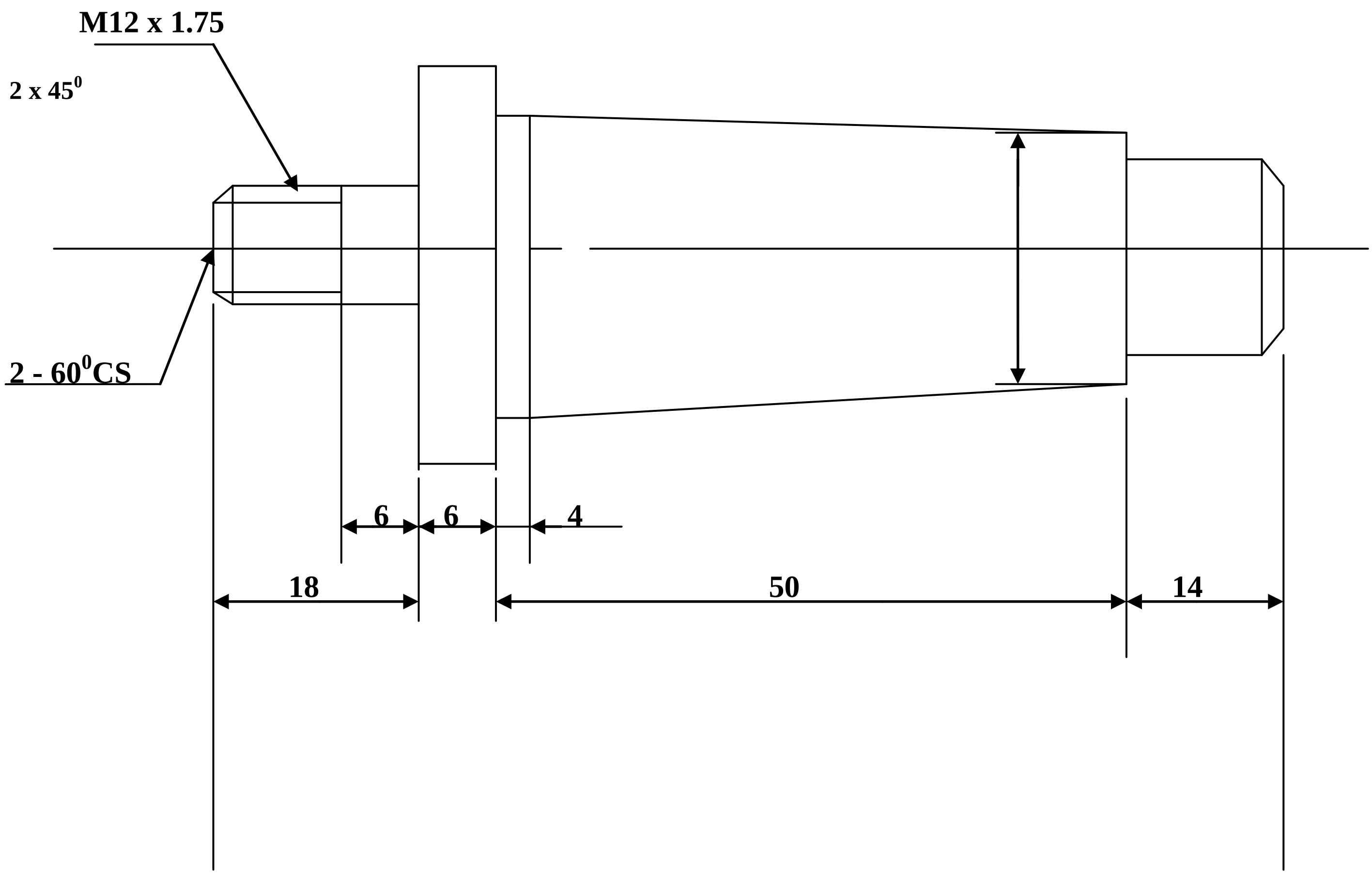
Step4- Face the two ends

Step5-calculate taper angle

Step6-Taper by using of Compound rest method

Step7-Turn the work pikes according to drawing

Step8- Polishing the work pieces by using sand paper



LAP Test	Practical Demonstration

Name: ______________________________ Date: ________________

Time started: _________________________ Time finished: ________________

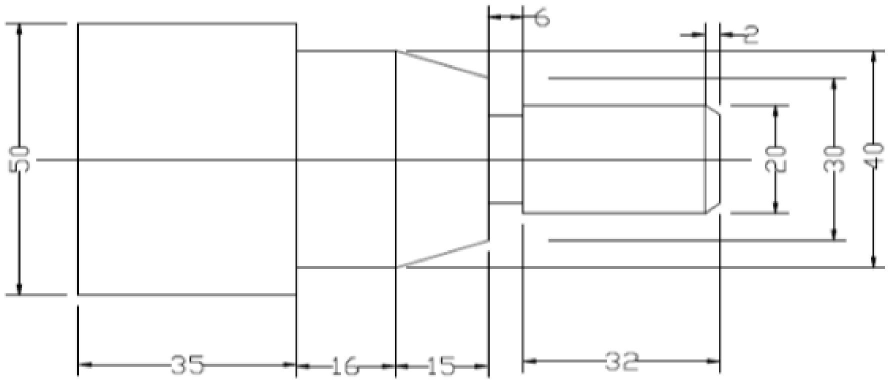
Instructions: Given necessary templates, tools and materials you are required to perform the following tasks within 4 hours.

Task 1.Machine step shaft on lathe machine

Task 2.Machine Tapper turning by using compound rest method

Task 3.Mill Hexagonal shape by using milling machine

Project 1: Facing, Straight Turning, Taper turning and grooving



All dimensions are in millimeters

Materials: Aluminum ingot diameter 60mmm

List of Reference Materials

5. Book: Machining and Machine Tools by A. B. Chattopadhyay.
6. Book: Metal Cutting: Theory And Practice by A. Bhattacharya.
7. Book: Manufacturing Process for Engineering Materials by S. Kalpakjain and S. Schmid.

BASIC METAL WORKS

NTQF Level - I

Learning Guide -43

Unit of Competence: -Operate Basic Workshop Machinery

Module Title:-Operating Basic Work shop Machinery

LG Code: IND BMW1 M12LO4-LG-43

TTLM Code: IND BMW1 TTLM 0919v2

LO4: Assure quality of finished component

Instruction Sheet	Learning Guide 43

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

- **Checking component for conformance**
- **Using techniques, measuring tools and equipment**
- **Handling deviations**
- **Carrying out routine maintenance and adjustments**

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 –**

- **Check component for conformance**
- **Use appropriate techniques, measuring tools and equipment**
- **Handle deviations**
- **Carry out Routine maintenance and adjustments.**

Learning Instructions:

53. Read the specific objectives of this Learning Guide.
54. Follow the instructions described in number 3 to 19.
55. Read the information written in the "Information Sheets 1". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
56. Accomplish the "Self-check 1" in page 4.
57. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 1).
58. If you earned a satisfactory evaluation proceed to "Information Sheet 2". However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
59. Submit your accomplished Self-check. This will form part of your training portfolio.
60. Read the information written in the "Information Sheet 2". Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
61. Accomplish the "Self-check 2" in page 9.
62. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 2).

63. Read the information written in the “Information Sheets 3 . Try to understand what are being discussed. Ask you teacher for assistance if you have hard time understanding them.
64. Accomplish the “Self-check 3” in page 11.
65. Ask from your teacher the key to correction (key answers) or you can request your teacher to correct your work. (You are to get the key answer only after you finished answering the Self-check 3).
66. If you earned a satisfactory evaluation proceed to “Operation Sheet 1” in page 13. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
67. Read the “Operation Sheet 1” and try to understand the procedures discussed.
68. If you earned a satisfactory evaluation proceed to “Operation Sheet 2” in page 13. However, if your rating is unsatisfactory, see your teacher for further instructions or go back to Learning Activity #1.
69. Read the “Operation Sheet 2” and try to understand the procedures discussed.
70. Read the “Operation Sheet 3” and try to understand the procedures discussed.
71. Do the “LAP test” in page 14 (if you are ready). Request your teacher to evaluate your performance and outputs. Your teacher will give you feedback and the evaluation will be either satisfactory or unsatisfactory. If unsatisfactory, your teacher shall advice you on additional work.

Information Sheet-1	Checking component for conformance

1.1. Definitions of conformance

Conformance is usually defined as testing to see if an implementation successfully meets the requirements of a standard or specification. There are many types of testing including testing for performance, strength, surface finish, shape and dimensions. Although conformance testing may include some of these kinds of tests, it has one fundamental difference of the requirements or criteria for conformance must be specified in the standard or specification. This is usually in a conformance clause or conformance statement, but sometimes some of the criteria can be found in the body of the specification. Some standards have subsequent documentation for the test methodology and assertions to be tested. If the criteria or requirements for conformance are not specified, there can be no conformance testing.

1.1.1 Consequences of incorrect sharpening

- Reduce tool life
- Complete failure of cutting edge
- Bend easily the cutting tool

1.1.2 Effects of incorrect speeds and feeds

- **Too Fast**: Too much spindle speed will generate excess heat which softens the tool and dulls it faster. There are exceptions and mitigating circumstances we'll talk about in more advanced installments.
  - ✓ Best Tool Life: Slowing down the spindle a bit and feeding at slightly less than appropriate for maximum MRR gives the best tool life. We'll talk more below about Taylor's equations for tool life, but suffice it to say that reducing the spindle rpm is more important than reducing the federate, but both will help.
  - ✓ Surface Finish: Reducing your federates while keeping the spindle speed up lightens the chip load and leads to a nicer surface finish. There are limits, the biggest of which is that you'll eventually lighten the federate too much, your tools will start to rub, and tool life will go way down due to the excess heat generated by the rubbing.
- **Feeding Too Slow**: As discussed, feeding too slow leads to rubbing instead of cutting, which can radically shorten your tool life and is to be avoided. Now that you know how the sweet spots break down, you'll have a better idea how to steer your feeds and speeds to the desired results.

1.2. Non-Conformance Report Items

A non-conformance report must include at a minimum of the following information:

- What is the main reason for the Non-Conformance Report or what went wrong
- Why the work doesn't meet the requirement
- What can be done to prevent the problem from happening again?
- Explanation of corrective action taken/to be taken

1.3. Product Inspection

Product Inspections conducted at various stages of the manufacturing process help you secure your production, safeguard the quality of your product and protect your brand image.

Product Inspections allow you to verify product quality on operation at different stages of the production process and prior to its dispatch. Inspecting your product before it leaves the working drawing requirement is an effective way of preventing quality problems.

An inspector checks your product against your chosen specifications to meet a range of requirements including passing the given standards of your destination market. With the use of inspection checklists that you can select online and tailor to your needs, your quality control process can be standardized and key quality concerns communicated to all parties involved in the inspection.

Self-Check -1	Written Test

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

1. Which one types of testing or testing for conformance of the machined component?

   A. conformance   B. Surface finish   C. Shapes and dimensions   D. None

2. Which is one must be included in a non-conformance report?

   A. Main reason error   B. The solution to prevent the problem

   C. Explanation of corrective action to be taken   D. All

3. Component inspections allow on operation at final stages of the production process.

   A. True   B. False

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

Answer Sheet

Score = ___________

Rating: ____________

Name: _________________________   Date: _______________

Short Answer Questions

Information Sheet-2	Using techniques, measuring tools and equipment

2.1. Techniques of checking conformance

Some common **methods** are visual; using measuring tools and equipment, industrial computed tomography scanning, microscopy, dye penetrant **inspection**, magnetic-particle **inspection**, X-ray or radiographic testing, ultrasonic testing, eddy-current testing, acoustic emission testing, and thermography **inspection**. For example, steel **ruler** is generally used by students for measuring length in few centimeters or millimeters.

Inspection in manufacturing includes measuring, examining, testing, or gauging one or more characteristics of a product or process and comparing the results with specified requirements to determine whether is the requirements are met for each characteristic. Common examples of inspection by measurement or gauging include using a caliper or micrometer to determine if a dimension of a manufactured part is within the dimensional tolerance specified in a drawing for that part, and is thus acceptable for use. Measurement instruments used to certify manufacturing conformity should be considered early in the design of products.

2.2. Measuring tools and equipment in checking conformance.

1. Vernier caliper:-You can obtain better accuracy with the vernier caliper.

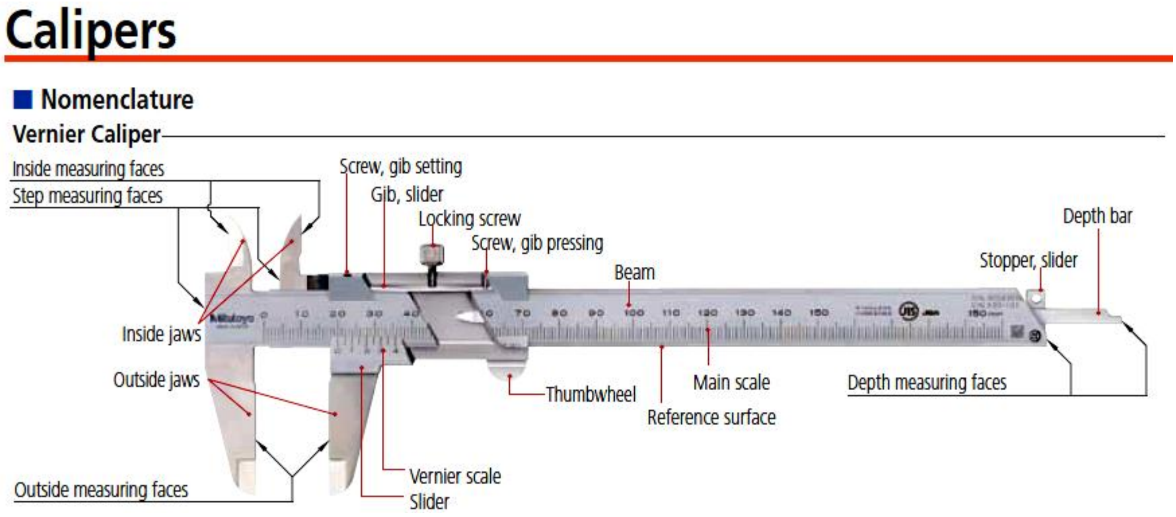
Vernier caliper consists principally of:-

- A main scale (the fixed scale);
- A fixed jaw (part of the rule scale);
- A vernier scale (a moving scale);
- A sliding jaw (attached to the moving scale).

The rule scale is graduated in millimeters. The moving scale moves on the rule scale, attached to the sliding jaw in a clamp. The vernier scale is graduated to read up to 49 mm. There are 50 divisions, which mean that there is a difference of 0.02 mm between the vernier scale and the main scale.

You need regular practice in using the vernier caliper to ensure that you understand its principle and use:

- Move the sliding jaw to be in contact with the face being measured.
- Tighten the locking screw on the clamp.
- Make fine adjustment using the fine-setting screw.
- Move the jaws so that they just touch the work; do not apply any force.
- Tighten the head lock.



To read (for example) 25.44 mm from the caliper, look for the number of the millimeter division below the vernier zero: for example, it is 25. Next find the line on the vernier scale that coincides with a line on the main scale: in this case 22. To calculate the total measurement, multiply 20 by 0.02 and add to 25:

That is:

Main scale reading = 25.00 mm

Vernier scale reading = 22 X 0.02 (0.44 mm) Final reading = 25.44 mm

The vernier caliper is a useful tool for taking external and internal measurements. Add the widths of the jaws (which are always stated on the caliper when taking internal measurements.

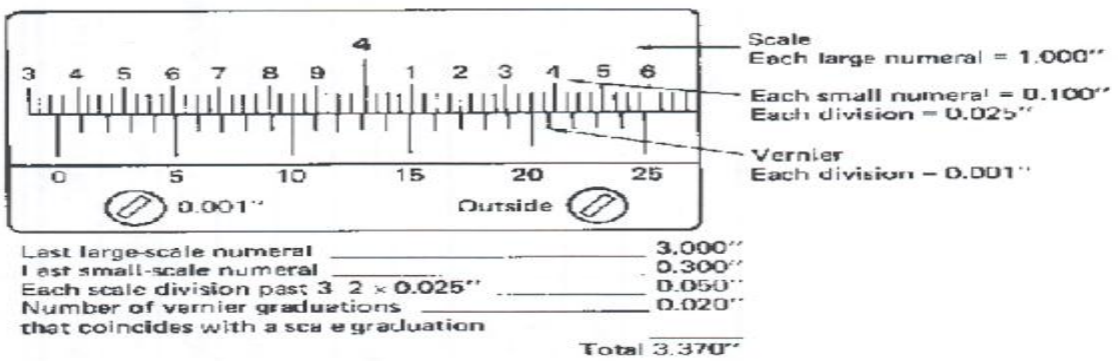


Fig. 6.1.7 Vernier Caliper

Principle of Vernier Caliper

• Rule consists of the slide that fits over the rule. A distance on the slide of six-hundred thousandths (0.600) of an inch is graduated into 25 equal parts so that each division measures twenty four thousandths (0.024) of an in. Fig. 6.1.13. The graduations on the rule itself are twenty-five thousandth (0.025) of an inch, so in a distance of 600 there are 24 divisions on the rule. The difference in the size of the division on the rule and those on the vernier scale is one thousandth (.001) of an inch.

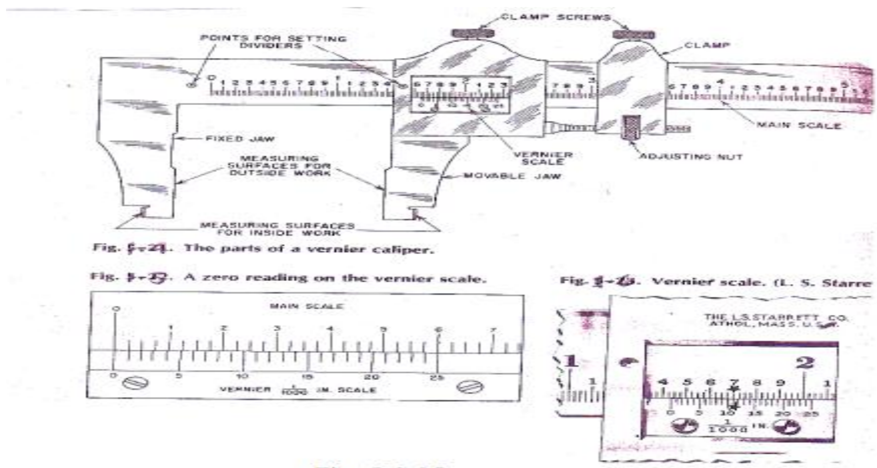
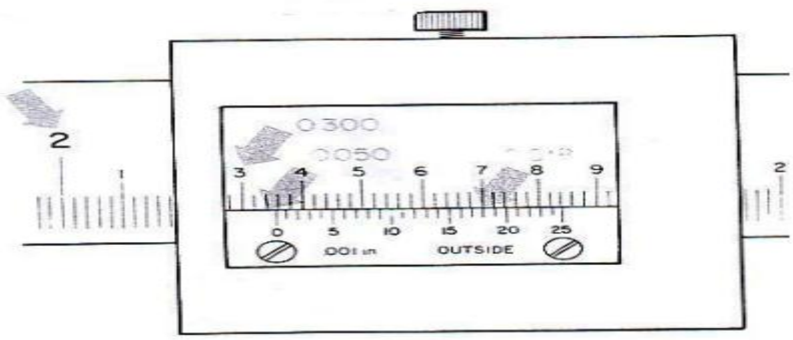


Fig. 6.1.13

How to read Vernier Caliper (Fig. 6.1.14)

• The zero mark on the vernier scale indicates the measurement to be read on the rule in Fig. 6.1.14. This is seen to be 1.425 in. and a little more. The exact amount over 1.425 is found by examining the division lines of the vernier scale to see which one exactly coincides with one of the lines on the rules. In this case it is line 11, so the full measurement is 10425 plus 0.011 which equals 1.436.



EXAMPLE the reading is composed of: The "0" line on the Vernier plate is between 2 and 3 on the beam = 2,000 Plus two 0.100 (1/10) Graduations = 0.200 Plus one 0.050 (1/20) Graduations = 0.050 Plus fifteen 0.001 (1/1000) Graduations = 0.015 Total reading = 2.368 in.

2. Micro meter

A micrometer is a very useful instrument. It enables you to take measurements to within one hundredth of a millimeter (0.01 mm). The metric micrometer is able to measure ranges of 25 mm (that is, for 0-25 mm, 25-50 mm, and so on). A common type is shown in Figure 4.51.

The micrometer has a thread with pitch 0.5 mm.

This means that the spindle advances by 0.5 mm for each turn. However, there are 50 graduations on the thimble. So the movement advanced for each graduation of the thimble is 0.5/50 = 0.01 mm.

The procedure for using the micrometer is as follows.

1. Hold the plastic insert to prevent thermal expansion.

2. Keep the measuring faces square with the surfaces that you are measuring, to ensure an accurate measurement.

3. Turn the thimble until the faces touch the work.

4. 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.

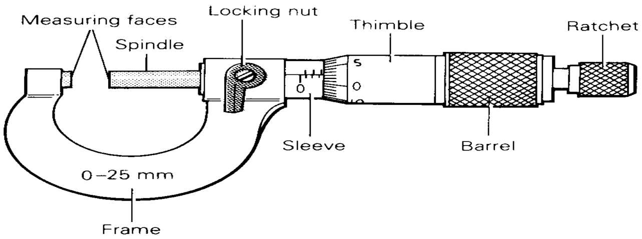


Figure 4.51 Outside micrometer.

- The reading in Figure 4.52 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

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.

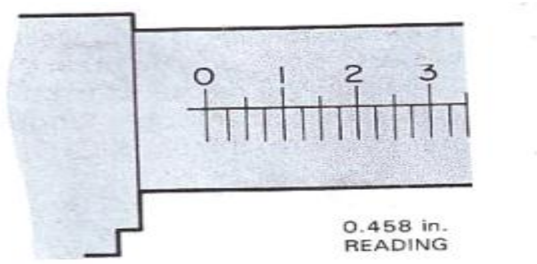
• Oil the micrometer regularly to avoid rust.

• Pack the micrometer in its box when not in use.

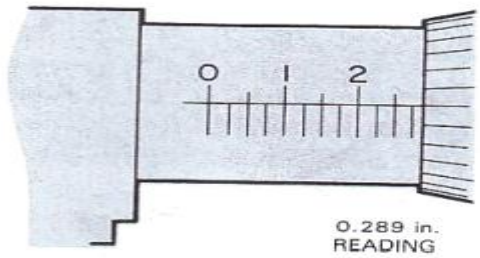
• Do not use force on the thimble or ratchet.

How to read micrometer?

• To read micrometer, we multiply the number of vertical position visible on the sleeve by 25 and add the number of divisions on the bevel of the thimble from 0 to the line that coincides with the horizontal line of the sleeve.



EXAMPLE 1
Reading is composed of:
4 large graduation of 4 X 0.100 = 0.400
2 small graduation or 2 X 0.025 = 0.050
and 8 graduation on the
thimble or 8 X 0.001 = <u>0.008</u>
Total reading = 0.458 in.



EXAMPLE 2
Reading is composed of:
2 large graduation of 2 X 0.100 = 0.200
3 small graduation or 3 X 0.025 = 0.075
and 8 graduation on the
thimble or 8 X 0.001 = <u>0.014</u>
Total reading = 0.289 in.

Fig. 6.1.12

8. Vernier Height Gauge

The vernier height gauge is used in conjunction with the surface plate, which has a smooth surface. The gauge has a heavy base. This supports the main scale, which is graduated in a similar way to the calipers. It has a means for of adjustment (Figure 4.53). It is used for accurate measurement of the depths of holes, slots, keyways and the like.

You can use the vernier protractor to measure angles to within 5 min of arc (Figure 4.54). The tool makes use of the vernier scale discussed above. It has the following parts: the main scale, with angular divisions; the vernier scale, divided into 12 divisions on each side of a zero mark; the rotating arm, which controls the movement of the vernier scale.

The procedure for using the vernier protractor is as follows

1. Observe the value of degrees on the main scale to each side of the zero mark on the vernier scale: for example, in Figure 4.54(b) it is 12°.

2. Find the mark on the vernier scale that coincides with a mark in the main scale. This gives the number of minutes (15 min in this case).

3. Add the two values together to give the reading: 12° 15'

For accurate readings take care when setting and reading the vernier protractor.

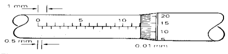


Figure 4.52 Micrometer reading.

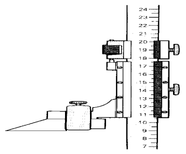


Figure 4.53 Vernier height gauge.

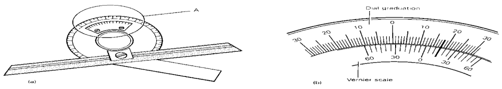


Figure 4.54 (a) The vernier protractor; (b) detail at A, showing reading of 12° 15'.

9. Dial Gauge

The dial gauge, or dial test indicator (Figure 4.55), is a comparator. You use it to compare measurements. The tool magnifies minute movements for easy reading. It converts the linear movement of the plunger into rotary movement of an indicator, which moves over a circular scale. The scale is divided into equal parts, each representing 0.01 mm.

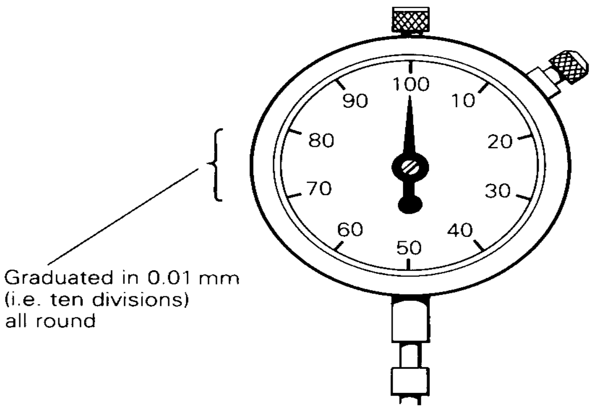


Figure 4.55 The dial gauge.

10. **Bevel Protractor (Fig. 6.1.8).** It is an instrument having a dial graduated in degree and sliding blade which is usually about 1/16 thick and it is used to measure angles and degree. Show fig. 6.1.9

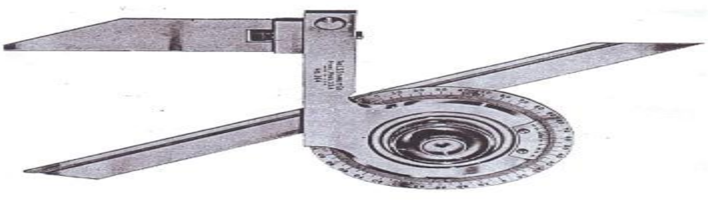


Fig. 6.1.8. Vernier Bevel Protractor

Fig. 6.1.9 Reading the size of an angle of vernier bevel protractor

Fig. 6.1.8. Vernier Bevel Protractor

Apply techniques Using Precision Measuring Tools

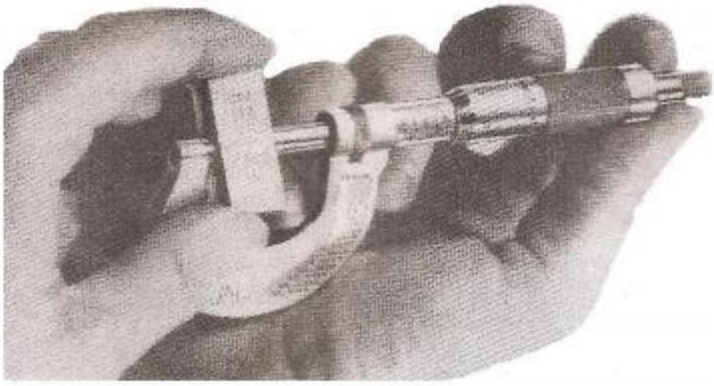


Fig. 2.8 A micrometer being checked using a gage block

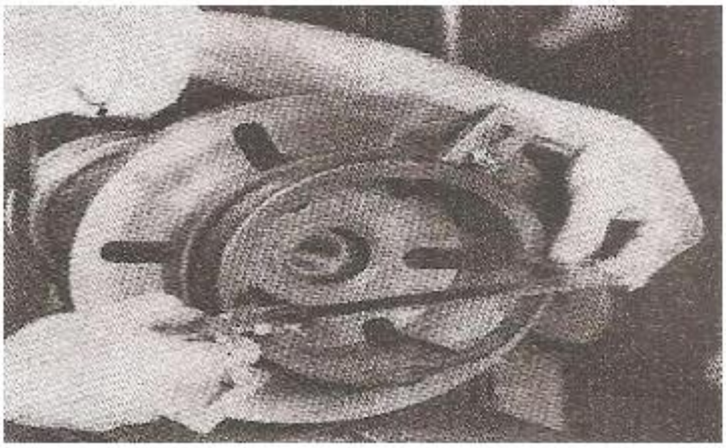


Fig. 2.9 An inside Vernier Caliper is used

Fig. 3.1 This is how to hold the micrometer correctly when piece is held in the hand.

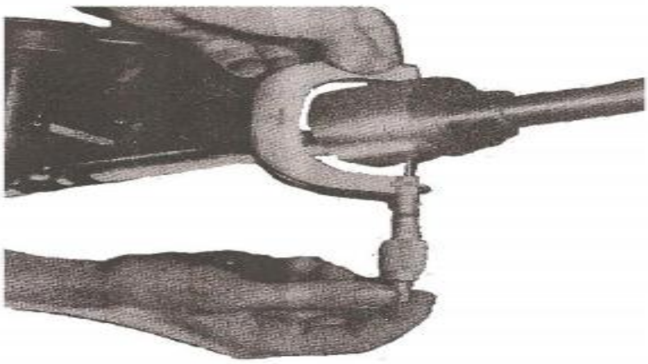


Fig. 3.2 This is how to hold the micrometer correctly when piece is not held in the hand.

Self-Check -2	Written Test

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

a. List the parts of Vernier caliper?

b. List the parts of Vernier Micrometer

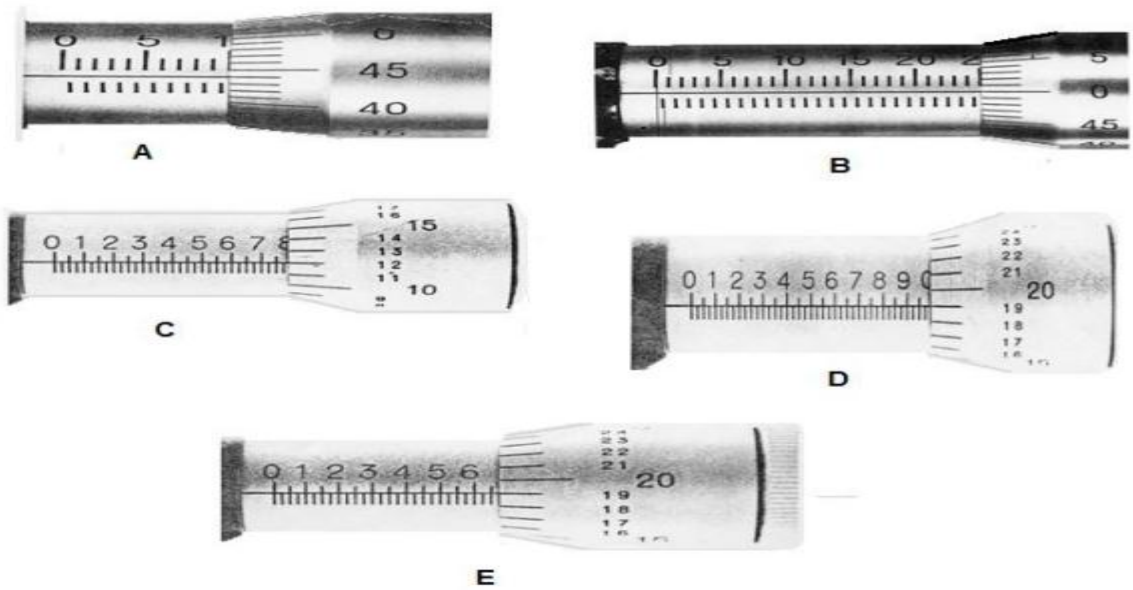
3. It is used to measure squarness of the edge of the work piece.

a. try square b. steel rule c. caliper d. micrometer

4. Is a precision tool used to measure diameter of round stock.

a. micrometer b. rule c. divider d. try square

Directions: Identify the different reading of micrometer and vernier caliper. Use separate sheets of paper.



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

Answer Sheet

Score = ____________
Rating: _____________

Name: ___________________________ Date: ________________

Short Answer Questions

Information Sheet-3	Handling deviations

Introduction

Quality Risk Management was mainly designed to be used prospectively when manufacturing operations are defined and validated. Therefore, potential deviations are identified and avoided by implementing risk control measures and preventive actions. QRM is based on the identification of product attributes and operational parameters which are critical to manufacturing operations in order to identify in advance their associated risks. This guidance document describes how this information may be used as criteria for the categorization and treatment of events, and eventually, deviations.

Under this approach, a sequence of steps may be identified when handling events and possible deviations:

- Event Detection
- Decision Making Process / Deviation Categorization
- Deviation Treatment
- Root cause investigation
- CAPA

Concept of deviations

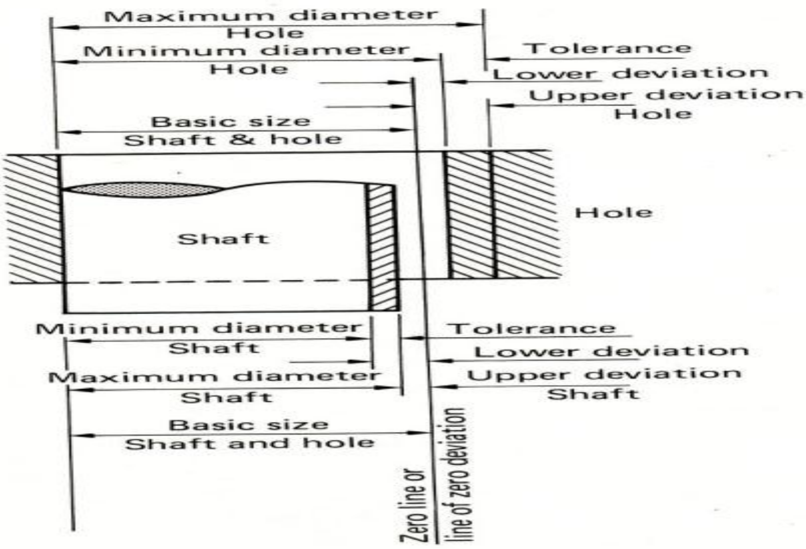
The differences of measurement from the given tolerances and the differences of measurement between each value in from working drawing's dimensions.

Actual deviation. This difference between a particular size and the basic. On the clearance fit diagram below the:-

Lower the deviation on the hole = the minimum diameter hole – basic size.

The upper deviation on the shaft = basic size – maximum

Diameter shaft.



Clearance fit

Fig. 11-01A2 **System of Limit and Fit**

3.2. Errors in Measurement System

An error may be defined as the difference between the measured value and the actual value. For example, if the two operators use the same device or instrument for finding the errors in measurement, it is not necessary that they may get similar results. There may be a difference between both measurements. The differences that occurs between both the measurements. Systematic Error / Random Error

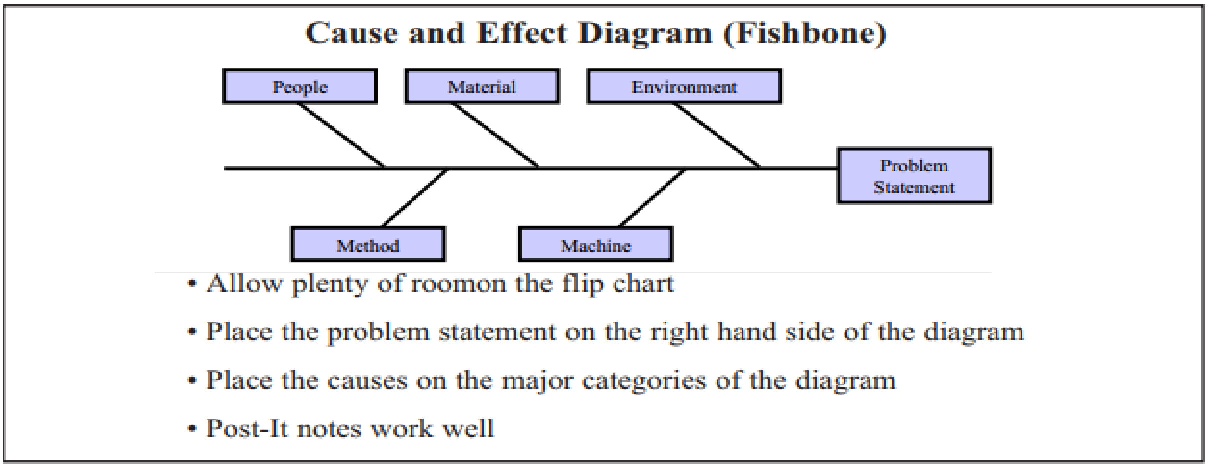
Examples

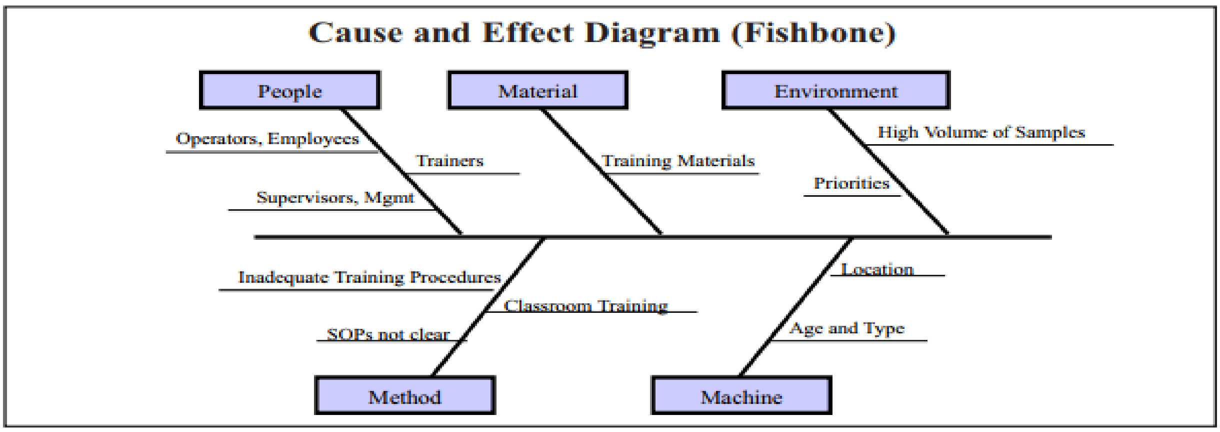
- A worn out instrument: For example, a plastic tape measure becomes slightly stretched over the years, resulting in measurements that are slightly too high,
- An incorrectly calibrated or tarred instrument, like a scale that doesn't read zero when nothing is on it,
- A person consistently takes an incorrect measurement.

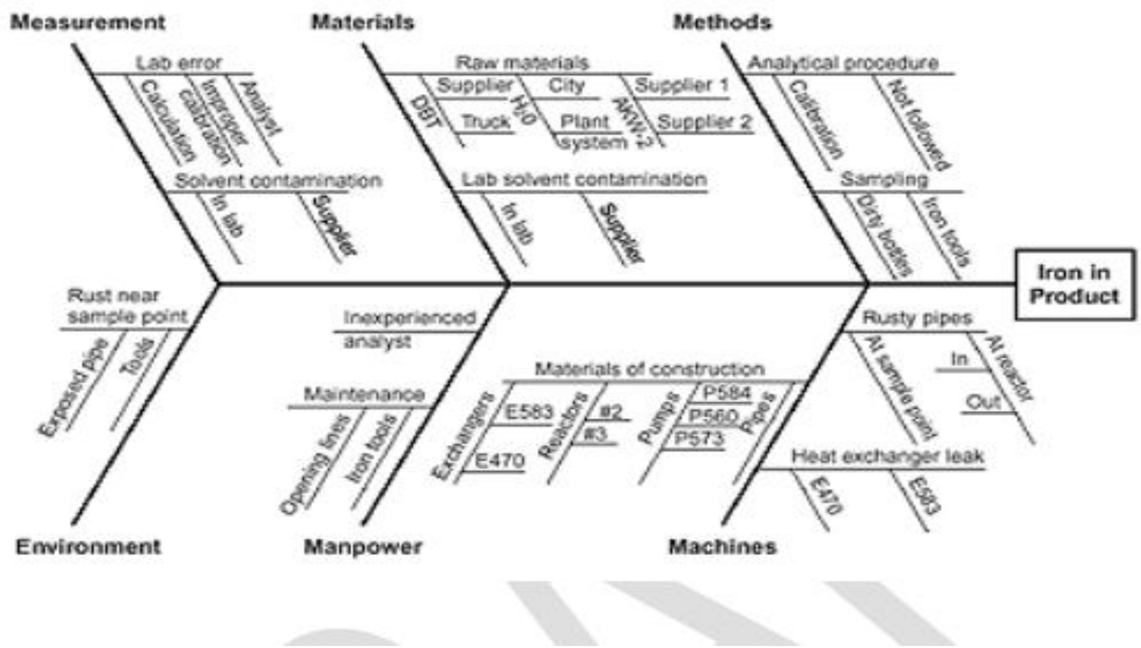
3.2 Organization procedures and standard for handling deviations

Policies and procedures are designed to influence and determine all major decisions and actions, and all activities take place within the boundaries set by them. Procedures are the specific methods employed to express policies in action in day-to-day operations of the organization.

Causes of deviation







- Using the wrong tool
- In accurate taking care of operation
- Lack of operator skill
- Selecting wrong material

Corrective and Preventive Action

Corrective and Preventive Action (CAPA) focuses on the investigation of deviations. It does so in an attempt to either prevent their recurrence or their occurrence in the first place. To ensure the effectiveness of any corrective and preventive actions, organizations should continue monitoring them after the completion of the RCA and overall investigation. The most common CAPA-related audit observations include "inadequate—did not sufficiently address root cause;" "inappropriate, did not address root cause;" "corrective and preventive were not clearly defined;" and "not completed in the timeline identified." One of the biggest pitfalls associated with CAPA occurs when someone assigns corrective and preventive actions without regard for resource requirement, capacity, ownership or timeline—in other words, without a plan. When it comes to CAPA, regulatory authorities expect organizations to ensure:

• The identified CAPA addresses the root cause;

• The solution can be implemented;

• There is clear understanding of the overall impact of the CAPA;

• Timelines and responsibilities (for implementation) have been reviewed and agreed to;

• There is a plan; and

• There is a monitoring phase. If an organization makes it through the investigation and determines the root cause, that forms just part of the equation. In the case of an inappropriate CAPA, further problems may ensue. Thus, the appropriate CAPA should be applied and monitored to ensure its effectiveness.

Self-Check -3	Written Test

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

2. Write three examples for Errors in Measurement System?
3. Define deviations?

Note: **Satisfactory rating - 5 points** **Unsatisfactory - below 5 points**

Answer Sheet

Score = ____________

Rating: _____________

Name: ___________________________ Date: ________________

Short Answer Questions

Information Sheet-4	Carrying out routine maintenance and adjustments

4.1 Introduction

Maintenance is all about preserving inherent reliability or built –in capacity of any asset. When we maintain an asset, the state which we wish to preserve must be one in which it continues to do whatever its users want it to do.

4.2 Purpose and objective of maintenance

The main purpose of maintenance in all industrial perspective is to reduce the business risks. In general, operation and maintenance is synonymous with high level of availability, reliability and assets operability linking directly with production capacity, productivity and business profit. The main objectives of maintenance are:-

1. To obtain plants and equipment at its maximum operating efficiency, reducing downtimes and ensuring operational safety.
2. To safeguard instruments by minimizing rate of deterioration and achieving this at optimum cost through budgeting and controls.
3. To help management in taking decisions on replacements or new investments and actively participate in specification preparation, equipment selection, its correction commissioning etc.
4. Help in implementation of suitable procedures for procurement, storage and consumption of spares, tools and consumables etc.
5. Standardization of spares and consumables, in conformity with plant, national and international standards and help in adoption of this standard by all users in the plant.
6. Running of centralized sciences like steam generation and distribution, water supply, air supply and fuel supply etc.

Running of captive workshops for repairs and conditioning and also for making some new spares.

4.3 Routine maintenance

Routine maintenance is the simplest but very essential form of maintenance system. Earlier the routine maintenance was considered about preventing failures. Today routine maintenance is being considered about avoiding, reducing or eliminating the consequences of failures. It involves jobs such as **cleaning, lubrication, inspection and minor adjustments pressure, flow, tightness** etc. and tightening of loose parts etc. It also includes **inspection of bearings, V-belts, couplings, jointing, foundation bolts, earthlings and protective covers** etc. The small and critical defects, observed during such inspection, are rectified immediately and bigger jobs are planned for rectification during next available shutdown. Such maintenance is essential for effective scheduled and preventive maintenance.

Routine maintenance is not necessarily need-based. In a equipment !plant some motors may be running for 4 hours a day and some others may be running 20 hours a day but, in routine as maintenance, all may be inspected at the same frequency. This may lead to some amount of over maintenance in some components, but the system pays up handsomely in the long run. “Regularity”,

.e. carrying out planned jobs regularly in simple cyclic schedule, is very essential in routine maintenance. Such schedules are simple (like check, clean, lubricate, tighten, adjust etc) and repetitive. Routine maintenance may also be considered as a small portion of preventive maintenance. Frequency of routine maintenance is generally once every shift or every day (normally at the start). Of course in sophisticated and automatic working equipment or in equipment having enough condition monitoring gadgets to indicate failures or deviations, the period of routine maintenance may change. Again, if such jobs are more and time availability is less, one group of job may be planned for Monday, another group of jobs for Tuesday and so on.

Routine maintenance needs very little investment in time and money. The duration of routine maintenance is generally so small that it does not affect the output of machine appreciably. However, the cost of not-doing routine maintenance may be very high as a small defect may develop in big and catastrophic failure.

As one example of routine maintenance, in few railway suburban electric trains system, whenever the train stops at few bigger stations, a group of maintenance people immediately starts checking and doing minor jobs like identifying loose parts and tightening, cleaning moisture traps and checking brakes etc. The whole job may take 10 to 12 minutes by the time the train is due to start for onward journey. In industries, during shift change periods, a small group of maintenance personnel carry out necessary inspection, lubrication, adjustments and tightening etc., which may take about 15 minutes.

4.3.1 Routine Maintenance, also known as preventive maintenance and it is an essential part of the ongoing care and upkeep of any machine while on operations. For example, lubricating the machine parts, cleaning machine regularly, changing the coolant of the machine

Regular maintenance of equipment is an important and necessary activity. The term 'maintenance' covers many activities, including inspection, testing, measurement, replacement and adjustment, and is carried out in all sectors and workplaces.

4.4 Corrective maintenance where equipment is repaired or replaced after wear, malfunction or break down.

4.5 Predictive maintenance, which uses sensor data to monitor a system, then continuously evaluates it against historical trends to predict failure before it occurs.

Operation Sheet 1	**Checking for conformance of components**

Procedures for Checking for conformance of components

Step 1- Establishing Standards and Methods for Measuring Performance

Step 2- Measuring the Performance

Sep 3- Determination of Whether the Performance Matches the Standard,

Step 4- Taking Corrective Action

Operation Sheet 2	Undertaking routine maintenance of the Machine

Procedures for undertaking routine maintenance of the Machine

Step 1- check the oil, coolant and grease lubrication manuals

Step 2- check the performance of the machine based on the manuals

Sep 3-open the oil, coolant box and fill the oil or the coolant

Step 4- check the indicator and reclose the oil coolant boxes

Step 5- clean the machine the areas

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 **1**hour.

Task 1. Checking for conformance of the machined part

Task 2.Undertake routine maintenance of the machine

List of Reference Materials

1. Book: Machining and Machine Tools by A. B. Chattopadhyay.

2. Book: Metal Cutting: Theory And Practice by A. Bhattacharya.

Acknowledgement

The Federal TVET Agency (FTA) wishes to extend thanks and appreciation to the representatives of TVET teachers; facilitators form the regional TVET Bureaus and Agencies, Federal TVET Agencies who donated their time and expertise to the development of this Model Learning Guide/TTLM/ for the TVET Program of ***Basic Metal Works Level-1***

The trainers who developed this TTLM

No	Name of trainer	Qualification	Region	E-mail
1	TEFERA DEMISSIE	A-Level	SNNPR	teferad123@gmail.com
2	MULUKEN YENET	B-Level	AMHARA	mulukeynet@gmail.com
3	KABTAMU SHEWANGIZAW	A-Level	OROMIA	Kabtamushewaangizaw@gmail.com
4	YOHANNIS HAILE	B-Level	B.G.R	
5	AHMED HUSSEN	B-Level	SOMALI	abdirashidhusen@hotmail.com
6	AYSHA SHIBESHI	B-Level	AFAR	yasinmohammed459@gmail.com
7	TEKALIGN BEKELE	B-Level	SNNPR	
8	BOKA TEREFE	A-Level	OROMIA	bokaterefe@yahoo.com

This model TTLM was developed on September 04/01/2012 E.C at ADAMA