# MODULE-1

# MACHINE TOOLS

**LESSON CONTENTS:**

Introduction, Classification, construction and specifications of lathe, drilling machine, milling machine, boring machine, broaching machine, shaping machine, planning machine, grinding machine.

**OBJECTIVES:**

- To study the constructional details of various machine tools and their specifications.

## 1.0 Introduction

**Machining Process:** Machining is an essential process of finishing by which work pieces are produced to the desired dimensions and surface finish by gradually removing the excess material from the preformed blank in the form of chips with the help of cutting tool(s) moved past the work surface(s).

**Machine Tool:** A machine tool is a non-portable power operated and reasonably valued device or system of devices in which energy is expended to produce jobs of desired size, shape and surface finish by removing excess material from the preformed blanks in the form of chips with the help of cutting tools moved past the work surface(s).

## 1.1 Classification of Machine Tool:

Based on the principle of operation, the type of relative motion exists between the tool and the work surface etc., the machine tools are classified as,

1. **According to the direction of major axis:**
    a. Horizontal axis machine tools (Lathes, Horizontal milling machine, Boring machines, cylindrical grinding machines etc.,)
    b. Vertical axis machine Tools (Vertical Milling machines, Drilling machines etc.,)
    c. Inclined or multiple axis (CNC machine tools and Special purpose machines)

2. **According to the purpose of Use:**
    a. general purpose (center lathes, milling machines, drilling machines etc.)
    b. single purpose (facing lathe, roll turning lathe etc.)
    c. special purpose (for mass production.)
3. **According to the degree of automation:**
    a. non-automatic (center lathes, drilling machines etc.)
    b. semi-automatic (capstan lathe, turret lathe, hobbing machine etc.)
    c. automatic (single spindle automatic lathe, swiss type automatic lathe, CNC milling machine etc.)
4. **According to Size:**
    a. Heavy duty (heavy-duty lathes (e.g. ≥ 55 kW), boring mills, planning machine, horizontal boring machine etc.)
    b. Medium duty (lathes – 3.7 ~ 11 kW, column drilling machines, milling machines etc.)
    c. Small duty (table top lathes, drilling machines, milling machines.)
    d. Micro duty (micro-drilling machine etc)
5. **According to precision:**
    a. Ordinary (Conventional Machine tools)
    b. High Precision (CNC machines, Grinding machines, lapping machines)
6. **According to the type of automation:**
    a. Fixed automation (Single spindle and multi spindle)
    b. Flexible automation (CNC Milling Machines)

## 1.2 LATHE MACHINE TOOL

The lathe is a machine tool which holds the work piece between two rigid and strong supports called centers or in a chuck or face plate which revolves. The cutting tool is rigidly held and supported in a tool post which is fed against the revolving work. The normal cutting operations are performed with the cutting tool fed either parallel or at right angles to the axis of the work. The cutting tool may also be fed at an angle relative to the axis of work for machining tapers and angles.

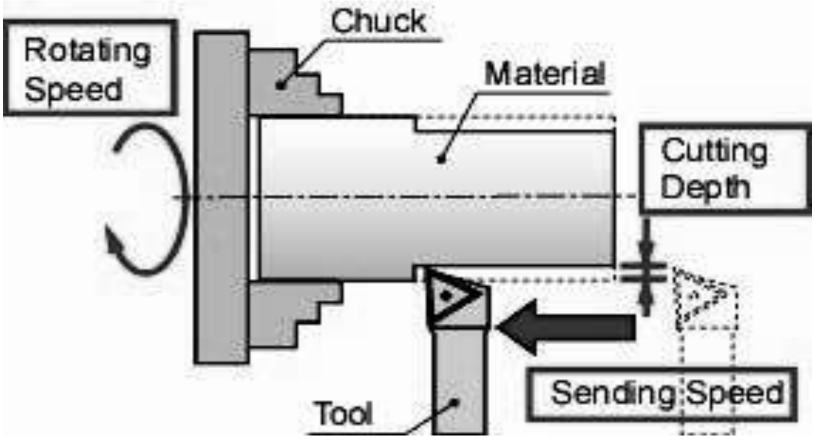


*Fig: Principle of working of a lathe*

**1.2.1 Construction of Centre Lathe:** The main parts of the lathe are the bed, headstock, quick changing gear box, carriage and tailstock.

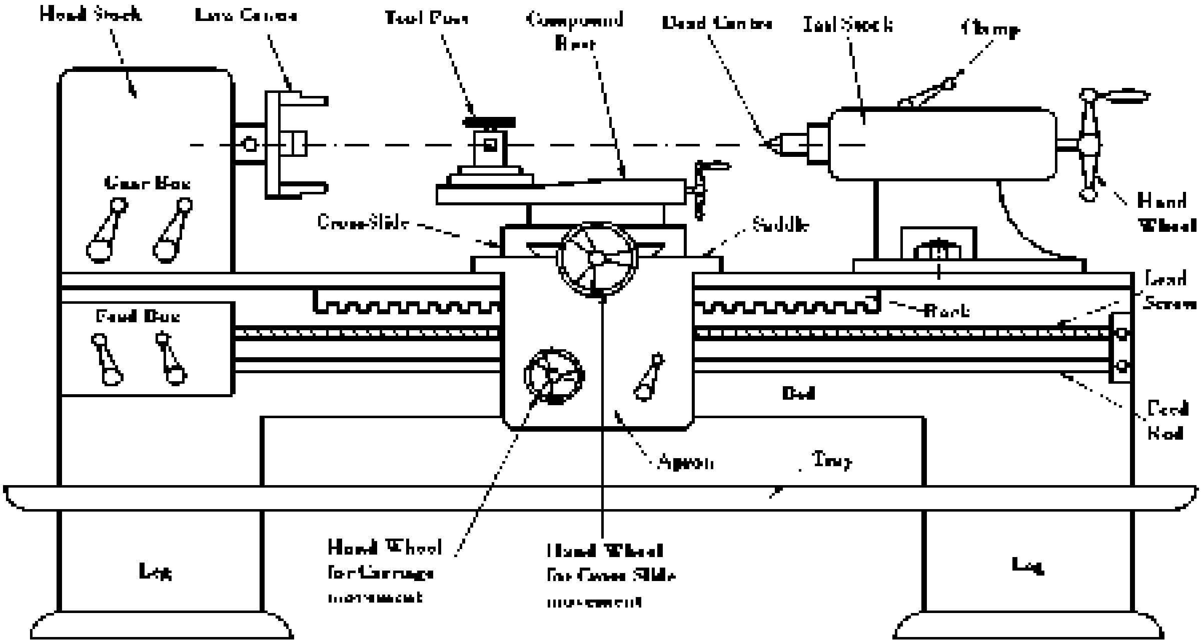


*Fig: Parts of Lathe*

- **Bed:** Usually made of cast iron. Provides a heavy rigid frame on which all the main components are mounted. It is the foundation part of a lathe and supports the remaining parts. The top of the bed is formed by precision-machined guide ways.
- **Guide Ways**: Inner and outer guide rails that are precision machined parallel to assure accuracy of movement.
- **Headstock:** mounted in a fixed position on the inner ways, usually at the left end. Using a chuck, it rotates the work. The housing comprising of the feed gearbox and the cone

pulley called headstock of the lathe. The main spindle projects out from the headstock. The motor drives the cone pulley drives the main spindle through belting. Spindle speeds can be further varied using beck gear mechanism.

- **Gearbox:** inside the headstock, providing multiple speeds with a geometric ratio by moving levers.
- **Spindle:** Hole through the headstock to which bar stock can be fed, which allows shafts that are up to 2 times the length between lathe centers to be worked on one end at a time.
- **Chuck:** allows the mounting of difficult work pieces that are not round, square or triangular. 3-jaw (self centering) or 4-jaw (independent) to clamp part being machined.
- **Tailstock:** Fits on the inner ways of the bed and can slide towards any position the headstock to fit the length of the work piece. Tail stock is the movable part of the lathe that carries the dead centre in it. The main function of the tailstock is to support the free end of the long work pieces. It is mounted loosely on the bed ways and can be moved in desired direction an optional taper turning attachment would be mounted to it.
- **Carriage Assembly**: Moves on the outer ways. Used for mounting and moving most the cutting tools. The carriage assembly consists of.
    - **Saddle**: is a H-shaped casting slides over the outer set of guide ways and serves as the base for the cross slide.
    - **Cross slide**: is mounted on the saddle and enables the movement of the cutting tool laterally across the lathe bed by means of cross-feed hand wheel.
    - **Compound Rest**: is mounted on the top of the cross slide and is swiveled to any angle in the horizontal plane to facilitate taper turning and thread cutting operations.
    - **Apron**: is mounted in front of the saddle beneath it and houses the carriage and cross slide mechanisms.
    - **Tool Post**: is mounted in the T-Slot of the compound rest and properly clamps the cutting tool.
- **Feed Rod:** Has a keyway, with two reversing pinion gears, either of which can be meshed with the mating bevel gear to forward or reverse the carriage using a clutch. is a stationary rod mounted in front of lathe bed and facilitates longitudinal movement of the carriage.
- **Lead Screw**: is the screw rod that runs longitudinally in front of the lathe bed. The gyration of the lead screw moves the carriage to and fro longitudinally during thread cutting operations.

### 1.2.2 Specification of Lathe:

The size of the lathe is specified in order to know the work holding capacity of the lathe. The specification of the lathe is shown in fig below.

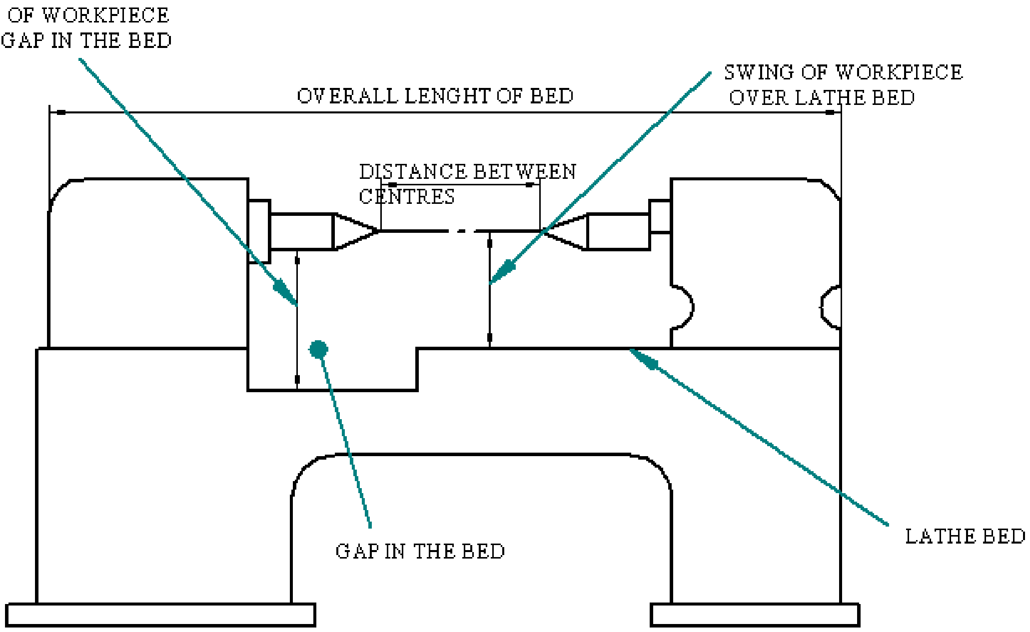


***Fig: Specification of Lathe***

1. **Distance between Centers:** The maximum length of the work pieces that can be held between the centers.
2. **Overall length of bed:** It is the total length of the lathe.
3. **Swing over lathe bed:** The Maximum diameter of the work piece that can be revolved over the lathe bed.
4. **Swing over the gap in bed:** The maximum diameter and width of the work piece that can be revolved over the lathe when the lathe has gap bed.

### 1.3 Drilling Machine Tool:

A power operated machine tool, which holds the drill in its spindle rotating at high speeds and when manually actuated to move linearly simultaneously against the work piece produces a hole is called drilling machine. In a drilling machine the holes can be produced to the sizes as small as thousandth of a centimetre and up to 7.5 cm diameter.

The different of types of drilling machines are:

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

### 1.3.1: Construction details of Portable/Upright Drilling Machine:

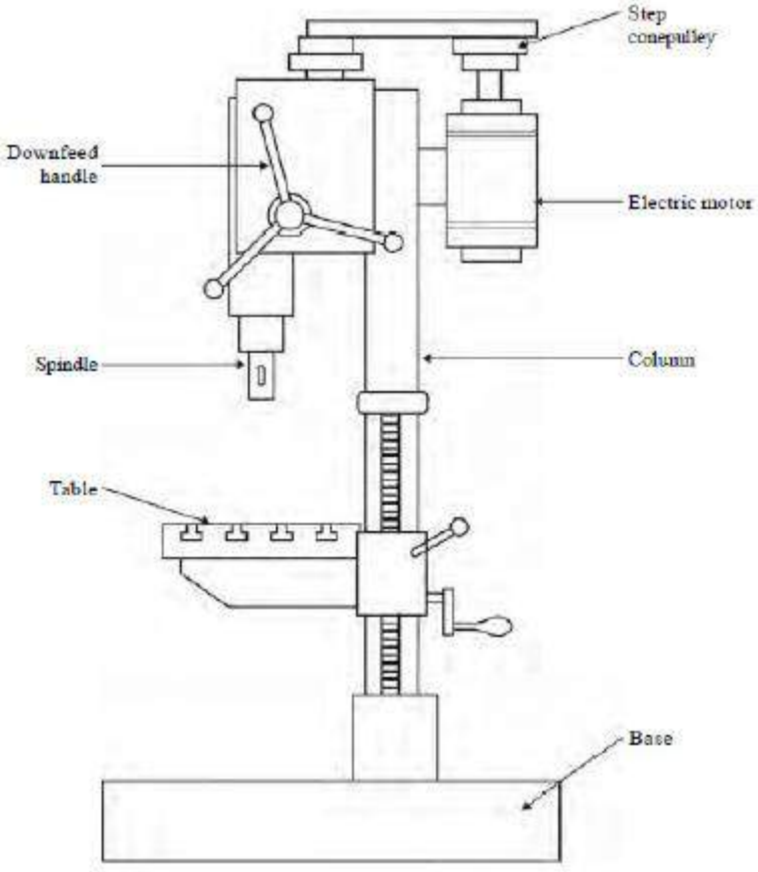


*Fig: Upright Drilling Machine*

The upright drilling machine is designed for handling medium sized workpieces. Though it looks like a sensitive drilling machine, it is larger and heavier than a sensitive drilling machine. Holes of diameter upto 50mm can be made with this type of machine. Besides, it is supplied with power feed arrangement. For drilling different types of work, the machine is provided with a number of spindle speeds and feed.

**Base:** The base is made of cast iron and so can withstand vibrations. It may be mounted on a bench or on the floor. It supports all the other parts of the machine on it.

**Column:** The column stands vertically on the base at one end. It supports the work table and the drill head. The drill head has drill spindle and the driving motor on either side of the column.

**Table:** The table is mounted on the vertical column and can be adjusted up and down on it. The table has 'T'-slots on it for holding the workpieces or to hold any other work holding device. The table can be adjusted vertically to accommodate workpieces of different heights and can be clamped at the required position.

**Drill head:** Drill head is mounted on the top side of the column. The drill spindle and the driving motor are connected by means of a V-belt and cone pulleys. The motion is transmitted to the spindle from the motor by the belt. The pinion attached to the handle meshes with the rack on the sleeve of the spindle for providing the drill the required down feed. There is no power feed arrangement in this machine. The spindle rotates at a speed ranging from 50 to 2000 r.p.m.

### 1.3.2: Radial Drilling Machine:

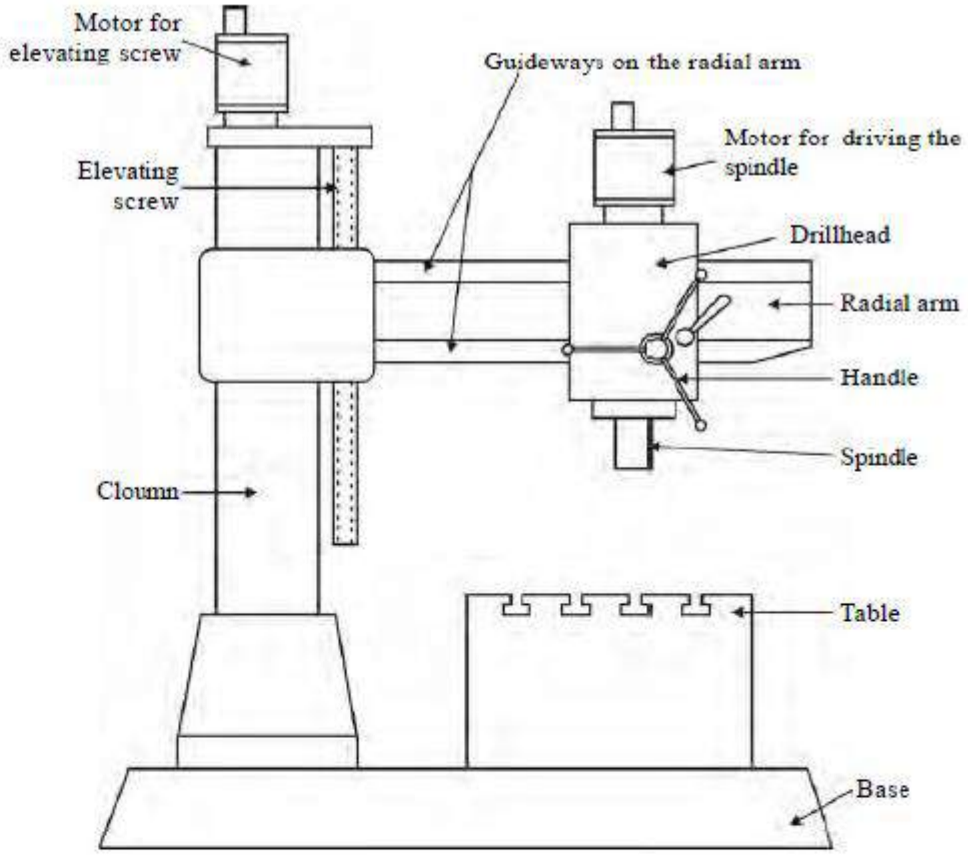


*Fig: Radial Drilling Machine*

The radial drilling machine is intended for drilling on medium to large and heavy workpieces. It has a heavy round column mounted on a large base. The column supports a radial arm, which can be raised or lowered to enable the table to accommodate workpieces of different heights. The arm, which has the drill head on it, can be swung around to any position. The drill head can be made toslide on the radial arm. The machine is named so because of this reason. It consists of parts like base, column, radial arm, drill head and driving mechanism.

### 1.3.3: Specification of Drilling Machine:

Drilling machines are specified according to their type.
To specify the machine completely the following factors are considered:

1. the maximum diameter of the drill that it can handle
2. the size of the largest workpiece that can be centred under the spindle
3. distance between the face of the column and the axis of the spindle
4. diameter of the table
5. maximum travel of the spindle
6. numbers and range of spindle speeds and feeds available
7. Morse taper number of the drill spindle
8. floor space required
9. weight of the machine
10. Power input is also needed to specify the machine completely.

## 1.4 Milling Machine Tool:

Milling is a metal cutting operation in which the cutting tool is a slow revolving cutter having cutting teeth formed on its periphery. The milling cutter is a multipoint cutting tool. The work piece is mounted on a movable worktable, which will be fed against the revolving milling cutter to perform the cutting operation.

### 1.4.1 Working Principle:

Figure shown above is the principle of cutting action of a milling cutter. The milling cutter is mounted on a rotating shaft known as arbor. The work piece which is mounted on the table can be fed either in the direction opposite to that of the rotating cutter as shown in above fig (a) or in the same direction to that of the cutter as shown in above fig (b).

*Fig (a): Up Milling* *(b): Down Milling*

### 1.4.2 Classification/ Types of Milling Machine:

Various types of milling machines are

1) Plain or horizontal type of milling machine.
2) Vertical Milling Machine
3) Universal Milling machine
4) Planer type milling machine
5) Profile cutting milling machine.

### 1.4.3 Horizontal/ Column & Knee type milling Machine:

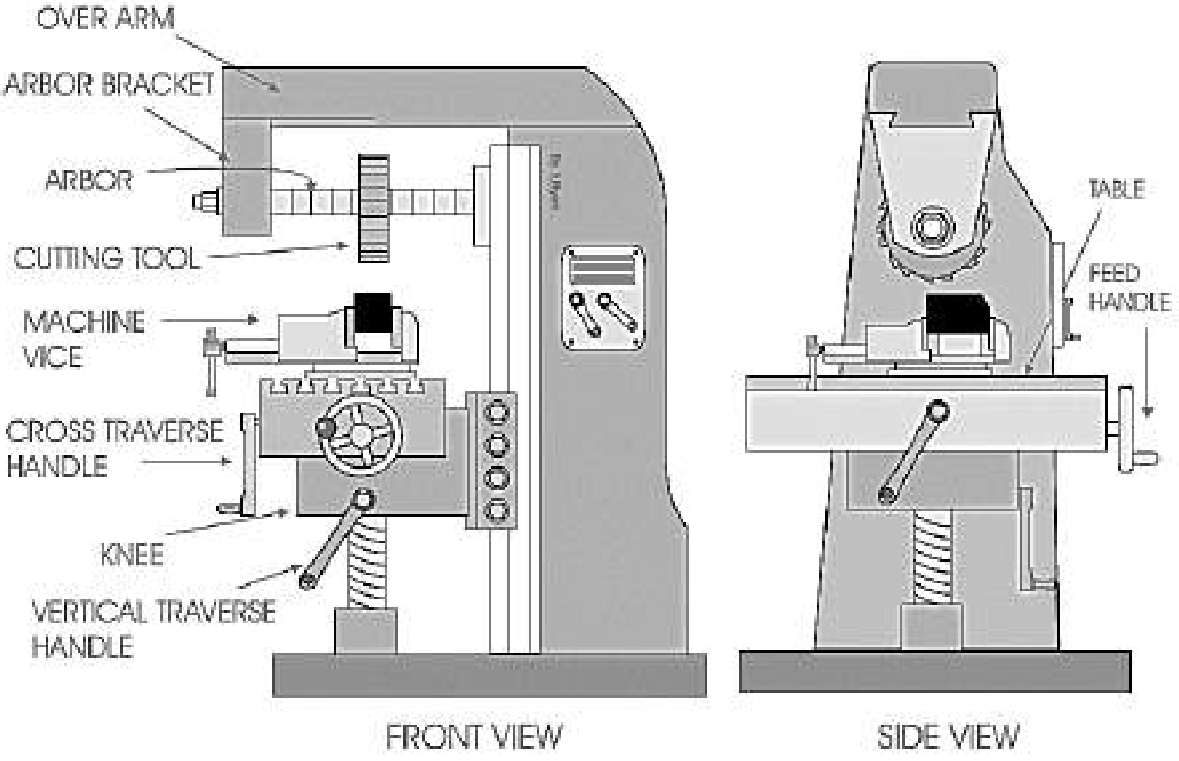


*Fig: Horizontal Milling Machine*

The main part of machine is base, Column, Knee, Saddle, Table, Overarm, Arbor Support and Elevating Screw.

**1. Base:** It gives support and rigidity to the machine and also acts as a reservoir for the cutting fluids.

**2. Column:** The column is the main supporting frame mounted vertically on the base. The column is box shaped, heavily ribbed inside and houses all the driving mechanisms for the spindle and table feed.

**3. Knee:** The knee is a rigid casting mounted on the front face of the column. The knee moves vertically along the guide ways and this movement enables to adjust the distance between the cutter and the job mounted on the table. The adjustment is obtained manually or automatically by operating the elevating screw provided below the knee.

**4. Saddle:** The saddle rests on the knee and constitutes the intermediate part between the knee and the table. The saddle moves transversely, i.e., crosswise (in or out) on guide ways provided on the knee.

**5. Table:** The table rests on guide ways in the saddle and provides support to the work. The table is made of cast iron, its top surface is accurately machined and carriers T-slots which accommodate the clamping bolt for fixing the work. The worktable and hence the job fitted on it is given motions in three directions:

a). Vertical (up and down) movement provided by raising or lowering the knee.

b). Cross (in or out) or transverse motion provided by moving the saddle in relation to knee.

c). Longitudinal (back and forth) motion provided by hand wheel fitted on the side of feed screw.
In addition to the above motions, the table of a universal milling machine can be swivelled 45° to either side of the centre line and thus fed at an angle to the spindle.

**6. Overarm:** The Overarm is mounted at the top of the column and is guided in perfect alignment by the machined surfaces. The Overarm is the support for the arbor.

**7. Arbor support:** The arbor support is fitted to the Overarm and can be clamped at any location on the Overarm. Its function is to align and support various arbors. The arbor is a machined shaft that holds and drives the cutters.

**8. Elevating screw:** The upward and downward movement to the knee and the table is given by the elevating screw that is operated by hand or an automatic feed.

### 1.4.4: Milling machine Specification:

a) Milling machines are usualy specified by the size of the table (lxb)
b) Along with the size of the table, milling machine is also specified by

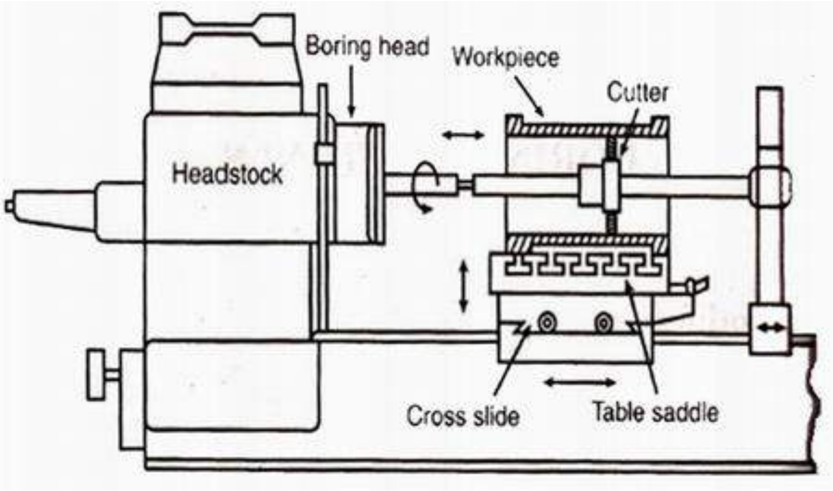
- Motor power required,
- Feed
- Floor space required
- Spindle speed
- Drives used

## 1.5 Boring Machine:

Boring is a process of producing circular internal profiles on a hole made by drilling or another process. It uses single point cutting tool called a boring bar. In boring, the boring bar can be rotated, or the work part can be rotated. Machine tools which rotate the boring bar against a stationary workpiece are called boring machines (also boring mills). Boring can be accomplished on a turning machine with a stationary boring bar positioned in the tool post and rotating workpiece held in the lathe chuck as illustrated in the figure. In this section, we will consider only boring on boring machines.

### 1.5.1 Boring Machine Tool:

Boring machines can be *horizontal* or *vertical* according to the orientation of the axis of rotation of the machine spindle. In *horizontal boring* operation, boring bar is mounted in a tool slide, which position is adjusted relative to the spindle face plate to machine different diameters. The boring bar must be supported on the other end when boring long and small-diameter holes.



A *vertical boring mill* is used for large, heavy workparts with diameters up to 12 m. The typical boring mill can position and feed several cutting tools simultaneously. The workpart is mounted on a rotating worktable.

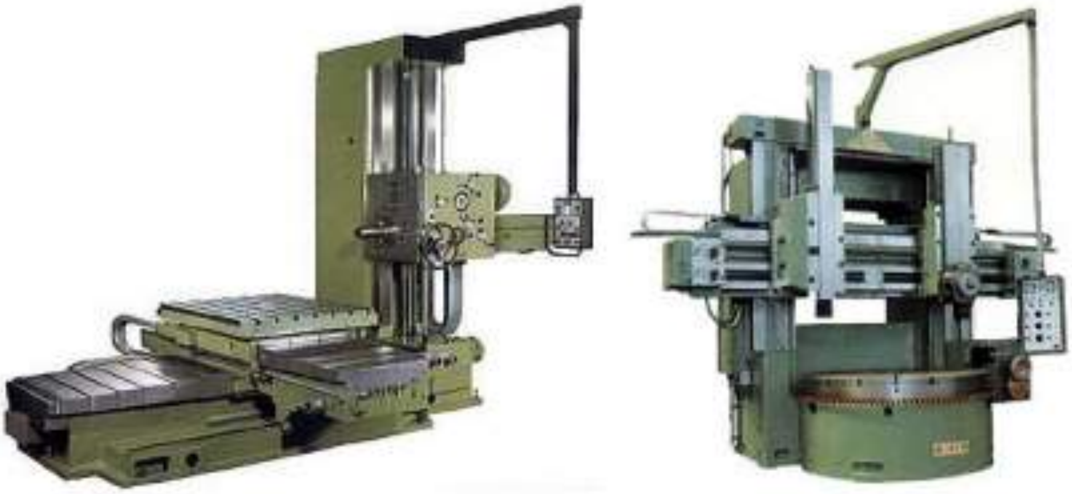
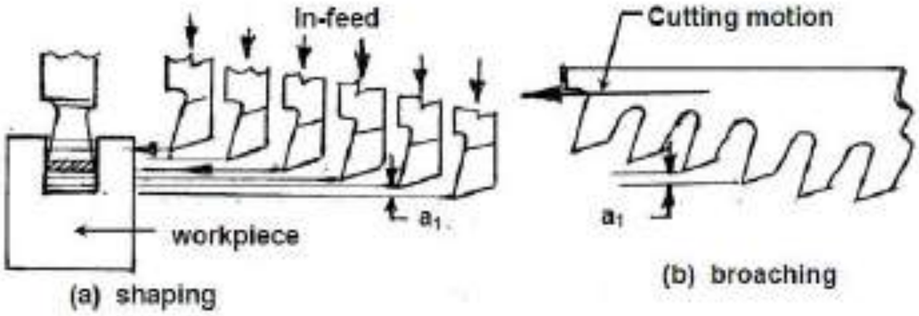


*Fig: Horizontal boring machine (Left) and vertical boring mill (Right).*

### 1.6 Broaching Machine:

Broaching is a machining process for removal of a layer of material of desired width and depth usually in one stroke by a slender rod or bar type cutter having a series of cutting edges with gradually increased protrusion as indicated in Figure. In shaping, attaining full depth requires a number of strokes to remove the material in thin layers step – by – step by gradually infeeding the single point tool. Whereas, broaching enables remove the whole material in one stroke only by the gradually rising teeth of the cutter called broach. The amount of tooth rise between the successive teeth of the broach is equivalent to the infeed given in shaping.



(a) shaping

(b) broaching

*Figure: principle of Broaching*

### 1.6.1 Broaching machine Tool:

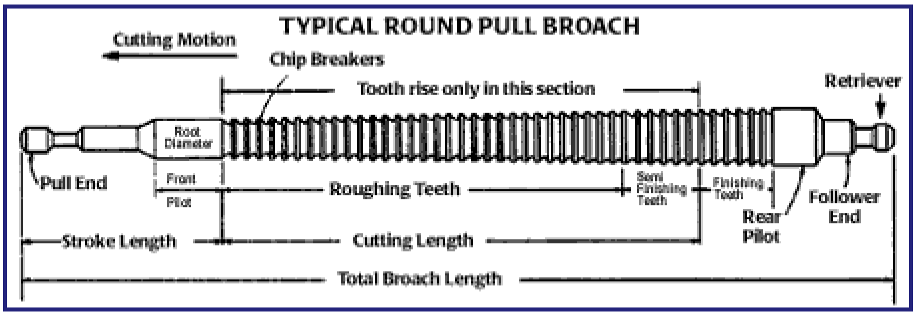
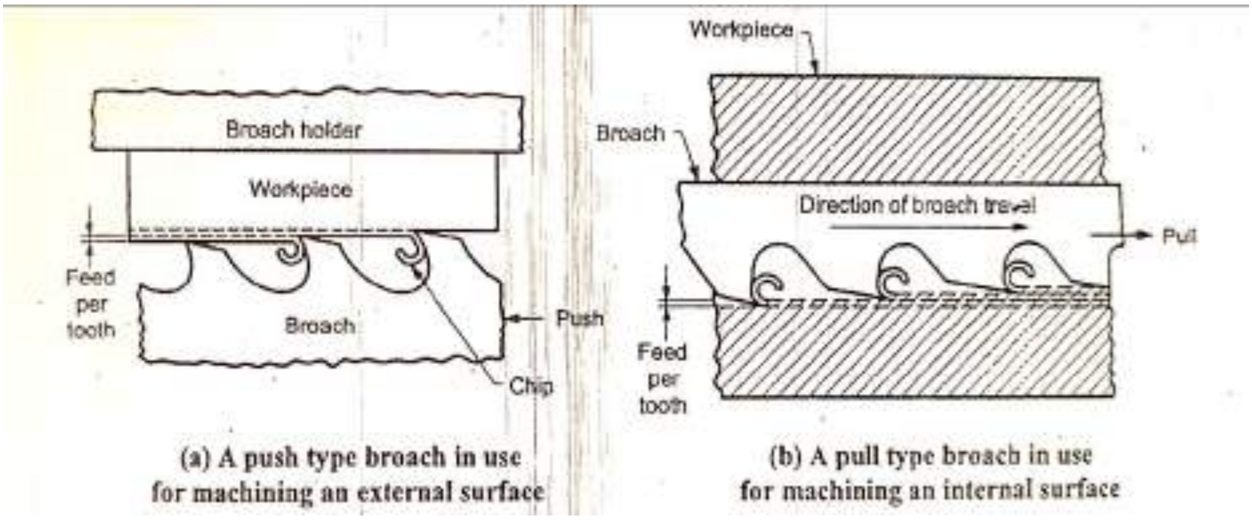


*Fig: Horizontal broaching tool*



(a) A push type broach in use for machining an external surface

(b) A pull type broach in use for machining an internal surface

*Fig: principle of broaching operation*

a) Horizontal broaching machines are applicable for machining of both internal and external surfaces. Figure shows the principle operation of broaching for internal and external machining.
b) In operation either workpiece is kept stationary and broach is fed past on the workpiece or broach is kept stationary and workpiece is fed past on the broach.
c) Horizontal broaching machines have a bed similar to the lathe machine and the broach is moves like a tailstock on the bed ways.
d) Horizontal internal broaching machines range from 2 to 60 tons and stroke upto 3m, whereas horizontal externa broaching machine are available upto 100 tons and stroke upto 9m.
e) Horizontal internal broaching is generally used for producing internal splines in the boss od a gear.

## 1.7 Shaper machine Tool:

The shaper is a reciprocating type of machine tool intended primarily to produce flat surfaces. These surfaces may be horizontal, vertical, or inclined. In general, the shaper can produce any surface composed of straight line elements. Modern shapers can generate contoured surface. The shaper is a machine tool used primarily for:

1. Producing a flat or plane surface which may be in a horizontal, a vertical or an angular plane.
2. Making slots, grooves and keyways
3. Producing contour of concave/convex or a combination of these

### 1.7.1 Working principle of Standard Shaper:

The job is rigidly fixed on the machine table. The single point cutting tool held properly in the tool post is mounted on a reciprocating ram. The reciprocating motion of the ram is obtained by a quick return motion mechanism. As the ram reciprocates, the tool cuts the material during its forward stroke. During return, there is no cutting action and this stroke is called the idle stroke. The forward and return strokes constitute one operating cycle of the shaper.

### 1.7.2 Construction details of Standard Shaper:

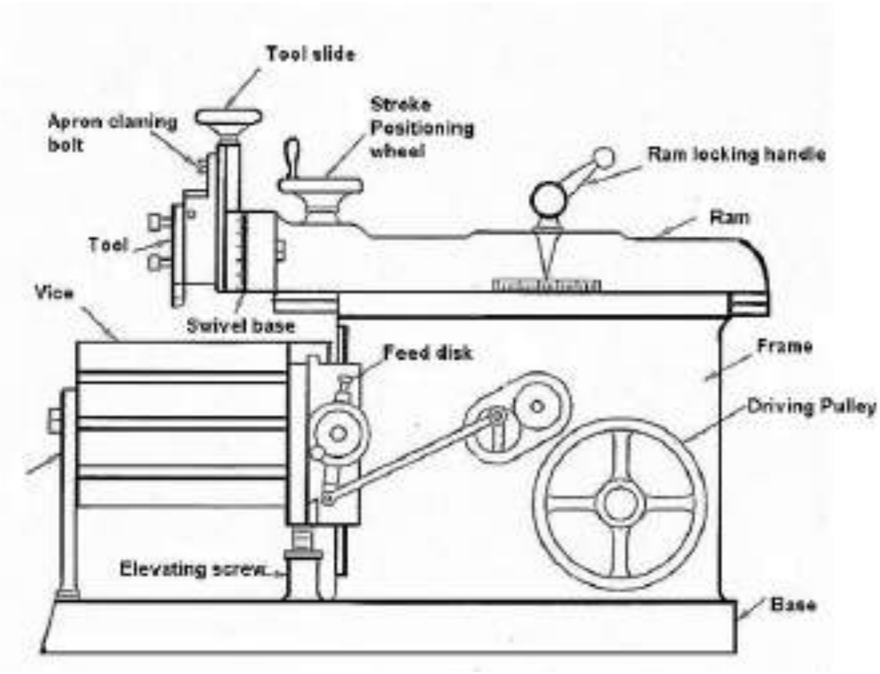


*Fig: Standard Horizontal Shaper*

**Construction**: The main parts of the Shaper machine is Base, Body (Pillar, Frame, Column), Cross rail, Ram and tool head (Tool Post, Tool Slide, Clamper Box Block).

**Base:** The base is a heavy cast iron casting which is fixed to the shop floor. It supports the body frame and the entire load of the machine. The base absorbs and withstands vibrations and other forces which are likely to be induced during the shaping operations.

**Body (Pillar, Frame, Column):** It is mounted on the base and houses the drive mechanism compressing the main drives, the gear box and the quick return mechanism for the ram movement. The top of the body provides guide ways for the ram and its front provides the guide ways for the cross rail.

**Cross rail:** The cross rail is mounted on the front of the body frame and can be moved up and down. The vertical movement of the cross rail permits jobs of different heights to be accommodated below the tool. Sliding along the cross rail is a saddle which carries the work table.

**Ram and tool head:** The ram is driven back and forth in its slides by the slotted link mechanism. The back and forth movement of ram is called stroke and it can be adjusted according to the length of the workpiece to be-machined.

### 1.7.3 Specification of Standard Shaper:

a) The size of the shaper is specified by the strike length or maximum length of cut

b) Shapers are made with wide variety of sizes depending upon their stroke length. It is uasually from 175mm to 900mm.

c) Along with the length the stroke number other details are required specify shaper

  - Type of drive
  - Type of speed reduction
  - Power in-put
  - Maximum momnet of tool, table
  - Cutting to return stroke ratio
  - Type of feed (manual or Automatic)
  - Total floor space required

### 1.8 Planer machine Tool:

The planer is a machine tool designed to produce plane and flat surface on a workpiece which is too large or too heavy. The workpiece is securely fixed on a table called platen, and it reciprocates horizontally against a single edged cutting tool. The surface machined may be horizontal, vertical or at an angle.

The planer is used for:

1. Planing flat horizontal, vertical and curved surfaces.

2. Planing at an angle and machining dovetails.

3. Planing slots and grooves.

The planer is available in different types for doing different types and sizes of job; the most common being the standard and double housing planer.

### 1.8.1 Types of Planner machines:

1 Standard or double housing planer.
2 Open side planer.
3 Pit planer.
4 Edge or plate planer.
**5** Divided or latching table planer.

### 1.8.2 standard Double Housing Planner:

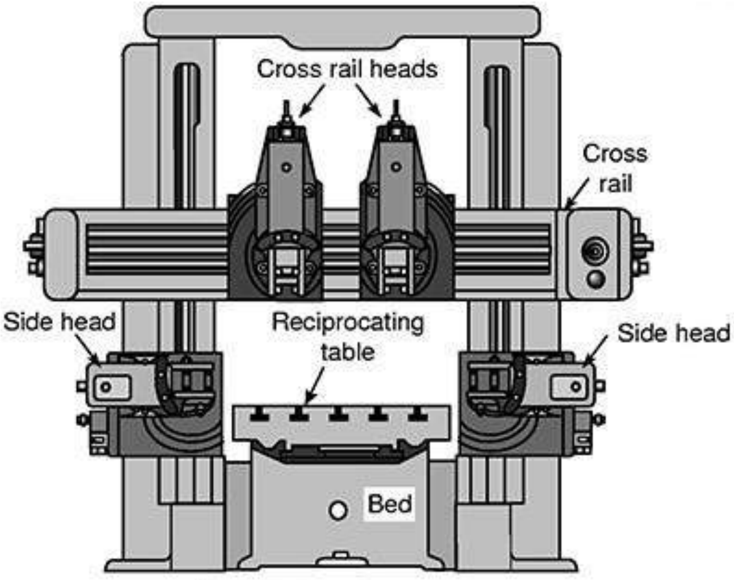


***Fig: Double housing Planner***

**Construction:** The main parts of the double Housing Planer machine is Bed and table, Housings, Cross rail, Tool heads, Driving and feed mechanism.

**Bed and table:** The bed is a long heavy base and table made of cast iron. Its top surface is flat and machined accurately. The flat top surface has slots in which the workpiece can be securely clamped. The workpiece needs rigid fixing so that it does not shift out of its position. The standard clamping devices used on planer machine are: Heavy duty vice, T-holders and clamps, angle plate, planer jack, step blocks and stop. The table movement may be actuated by a variable speed drive through a rack and pinion arrangement, or a hydraulic system.

**Housings:** The housings are the rigid and upright column like castings. These are located near the centre on each side of the base.

**Cross rail:** The cross rail is a horizontal member supported on the machined ways of the upright columns. Guide ways are provided on vertical face of each column and that enables up and vertical movement of the cross rail. The vertical movement of the cross rail allows to accommodate workpiece of different heights. Since the cross rail is supported at both the ends, this type of planer machine is rigid in construction.

**Tool heads:** Generally, two tool heads are mounted in the horizontal cross rail and one on each of the vertical housing. Tool heads may be swivelled so that angular cuts can be made.

**Driving and feed mechanism:** The tool heads may be fed either by hand or by power in crosswise or vertical direction. The motor drive is usually at one side of the planer near the centre and drive mechanism is located under the table.

The size of the planer is specified by the maximum length of the stroke, and also by the size of the largest rectangular solid that can be machined on it.

### 1.8.3 Specification of a planner machine:

The planer is specified by the following parameters:

- Radial distance between the top of the table and the bottom most position of the cross rail.
- Maximum length of the table and maximum stroke length of table.
- Power of the motor.
- Range of speeds and feeds available.
- Type of feed and type of drives required.

- Horizontal distance between two vertical housings. Net weight of machine and Floor area required.

### 1.9 Grinding Machine:

Grinding, also called abrasive machining, is a process in which the material is removed in form of fine chips, almost as dust particles by the abrasive action using some kind of abrasive materials. Generally, grinding is employed when a thick layer of material is to be removed in general classes of work, where the surface finish is not that important, and for finishing and polishing works on the parts already machined by some other machining processes.

### 1.9.1 Grinding Machines:

The different methods of grinding are:
1. Surface Grinding,
2. Cylindrical Grinding and
3. Centre less Grinding.

### 1.9.2 Surface Grinding Machine:

In the surface grinding process, the grinding wheel revolves on a spindle and the work-piece mounted on a reciprocating table as shown in Figure, is brought into contact with the grinding wheel. Flat, angular and irregular surfaces may be produced by surface grinding.

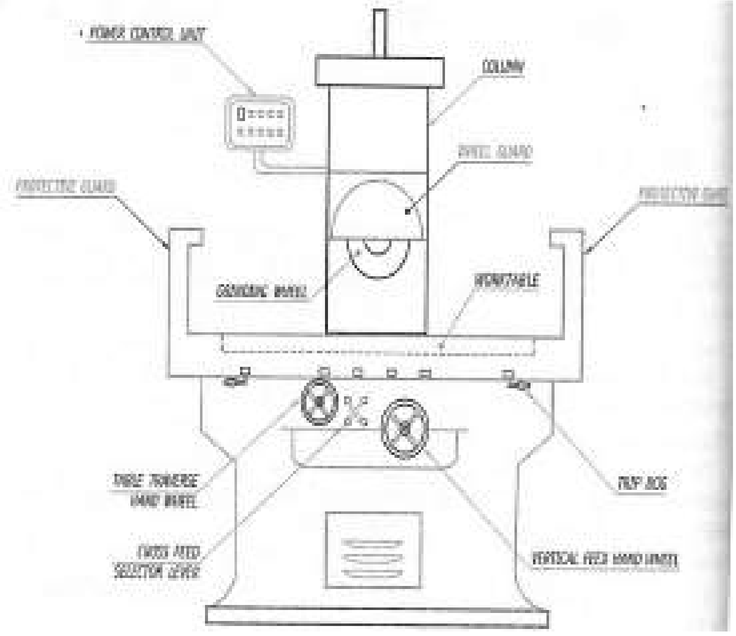


*Fig: Horizontal Surface grinding machine*

The Surface Grinding Machine is used to grind flat surfaces. Here, the job is mounted on a rectangular table which moves longitudinally as well as in the transverse direction below the rotating grinding wheel. The longitudinal and transverse feed movements can be accomplished either by manual feed or through power feed arrangement. The work-piece can be clamped in two ways; one is by clamping it to the work table by means of clamping elements; the other way is by using a magnetic chuck, which holds the work-piece through its strong magnetic field. There is an internal pump and a piping arrangement to take care of automatic application and recirculation of the coolant. There is a protective safety guard at the end of the table to prevent the wheel from hitting any person or object.

### 1.9.3 Cylindrical Grinding Machine:

Cylindrical grinding is the process of grinding the curved surfaces of cylindrical pieces. These surfaces may be straight, tapered or contoured. Fig. 8.3 shows the basic principle of the cylindrical grinding. Fig 8.3 shows a typical cylindrical grinding machine. The work-piece is mounted on the two centres, one is the tail stock centre and the other is the headstock centre. The tail stock centre is the dead centre and the headstock centre mayor may not revolve during grinding. When high accuracy is required the two supporting canters must remain stationary when the work-piece revolves.

When both canters are dead, precision sizes and good finish can be obtained, because there is no possibility of run out from the headstock spindle. As the work-piece revolves, the grinding wheel rotating much faster in the opposite direction is brought into contact with the work-piece. The work-piece and the table reciprocate while the grinding wheel in contact with the work-piece removes the material.

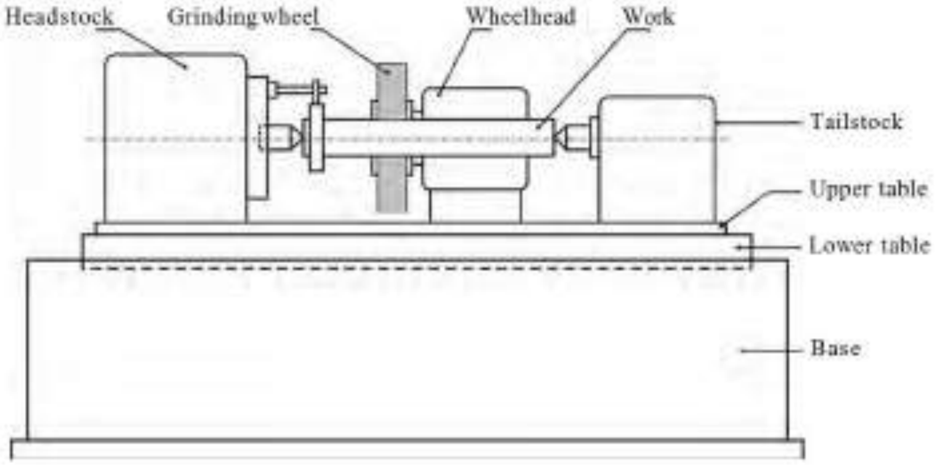


*Fig: Cylindrical Grinding Machine*

### 1.9.4 Centreless Grinding Machine:

Centre less grinding method also employed for grinding the curved surfaces of long slender rods which cannot be ground by cylindrical grinding due to the lateral thrust of the wheel on the work-piece. In the centre less grinding, shown in Figure the work-piece rests on a work-rest blade and is backed up by a second wheel, called the regulating wheel. The rotation of the grinding wheel pushes the work-piece down on the work-rest blade and against the regulating wheel. The regulating wheel, usually made up of a rubber bonded abrasive, rotates in the same direction as the grinding wheel and controls the longitudinal feed of the work-piece. A typical centre less grinding machine is shown in Figure.

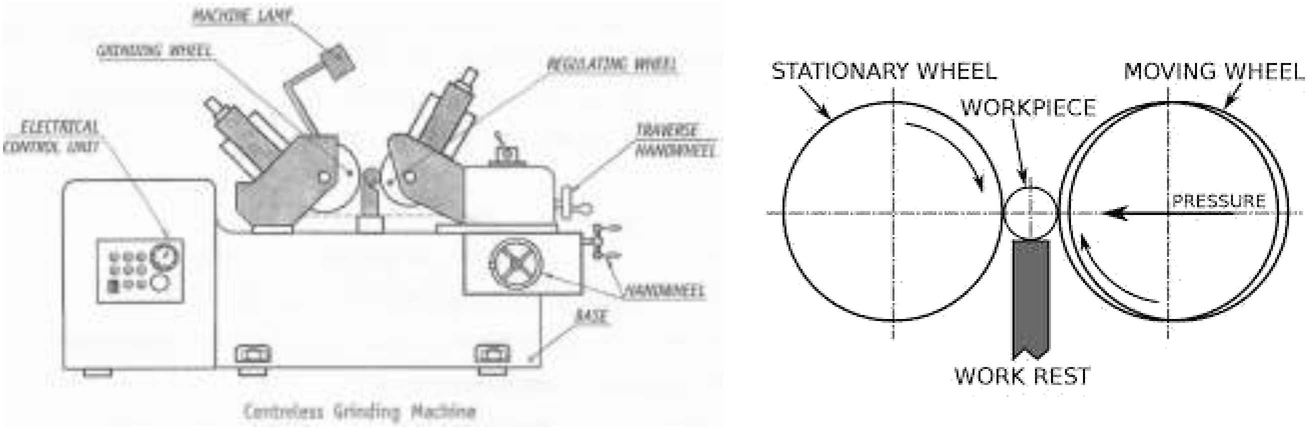


*Figure: (a) centreless Grinding machine (b) Principle of centreless grinding operations*

**OUTCOMES:**

- Students will bale to explain the construction & specification of various machine tools.

**QUESTIONS:**

1. Explain the principle of working of a lathe with the help of a neat sketch
2. What factors govern the specification of lathe?
3. What is meant by machine tool? Give the classification of machine tools.
4. What are the basic elements of drilling machine? Explain the construction of upright drilling machine.
5. Explain the principle of broaching.
6. With the help of a neat sketch explain column and knee type milling machine.
7. Give a constructional details of surface grinding machine.
8. Define centre less grinding. Briefly explain the construction of it.

Machine Tools & Operations,

## Further Reading

1. **“Metal cutting principles”**, Milton C. Shaw, Oxford University Press, Second Edition,2005.
2. **“Manufacturing Technology”,** Vol 2, P N Rao, McGraw Hill Education, 3rd Edition
3. **“Workshop Technology, Vol-II”,** by Hazara Chowdary

# MODULE-2

# MACHINING PROCESS

**LESSON CONTENTS:**

Introduction, Types of motions in machining, turning and Boring, Shaping, Planning and Slotting, Thread cutting, Drilling and reaming, Milling, Broaching, Gear cutting and Grinding, Machining parameters and related quantities.

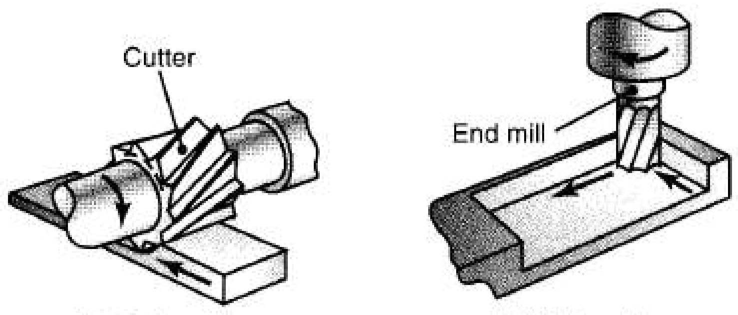
**OBJECTIVES:**

- To Study the Various Machining Process on various machine tools.
- To study the machining parameters and related quantities.

## 2.0 Machining Process:

Cutting processes remove material from the surface of a workpiece by producing chips. Some of the more common cutting processes, illustrated in Figure are as follows:

(a) Straight turning (b) Cutting off



(c) Slab milling (d) End milling

*Fig: Various machining process*

- Turning, in which the workpiece is rotated and a cutting tool removes a layer of material as the tool moves to the left.
- Cutting off, in which the cutting tool moves radially inward and separates the right piece from the bulk of the blank.
- Slab milling, in which a rotating cutting tool removes a layer of material from the

surface of the workpiece.

- End milling, in which a rotating cutter travels along a certain depth in the workpiece and produces a cavity.

## 2.1 Types of motions in Machining:

For obtaining the required shape on the workpiece, it is necessary that the cutting edge of the cutting tool should move in a particular manner with respect to the workpiece. The relative movement between the workpiece and cutting edge can be obtained either by the motion of the workpiece, the cutting tool, or by a combination of the motions of the workpiece and cutting tool.

These motions which are essential to impart the required shape to the workpiece are known as working motions. Working motions can further be classified as:

1. Drive motion or primary cutting motion
2. Feed motion

Working motions in machine tools are generally of two types: ***Rotary & Translatory***. Working motions of some important groups of machine tools are shown in figure:

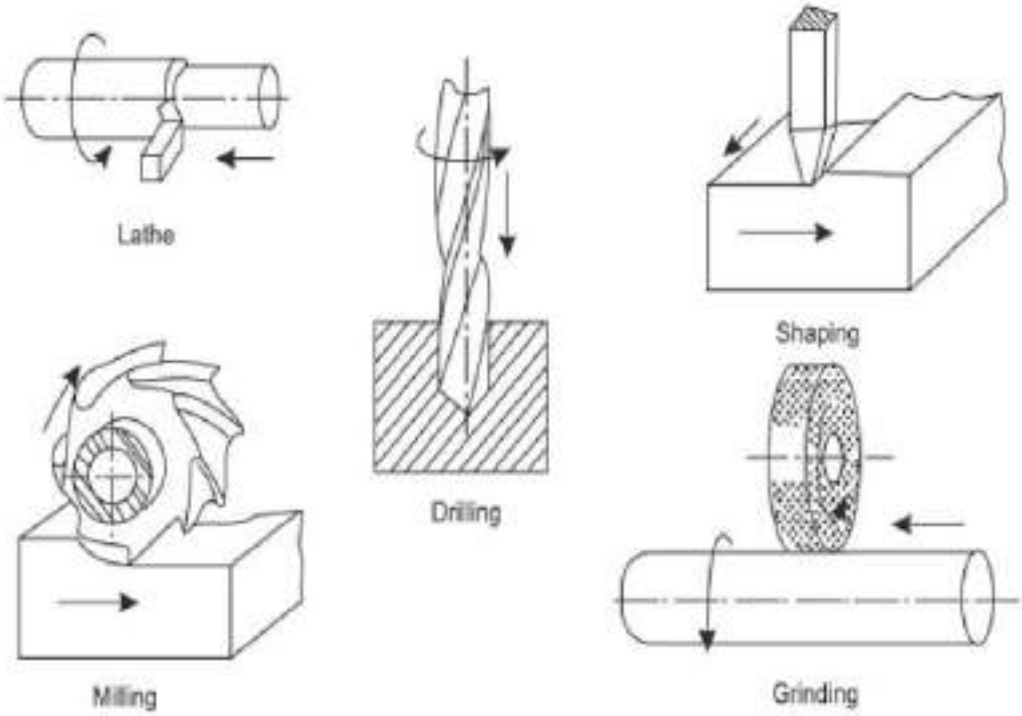


Fig: various machining process with the various relative motions

1. For lathes and boring machines:
    a. Drive motion - Rotary motion of workpiece
    b. Feed motion – translatory motion of cutting tool in the axial or radial directions
2. For Drilling machines:
    a. Drive motion - rotary motion of drill
    b. Feed motion - translatoey motion of drill
3. For milling machines:
    a. Drive motion – rotary motion of the cutter
    b. Feed motion – translatory motion of the workpiece
4. For shaping, planning and slotting machines:
    a. Drive motion – Reciprocating motion of the cutting tool
    b. Feed motion – intermittent translation motion of workpiece
5. For Grinding Machines:
    a. Drive motion – Rotary motion of the grinding wheel
    b. Feed motion – Rotory as well as translatory motion of the workpiece

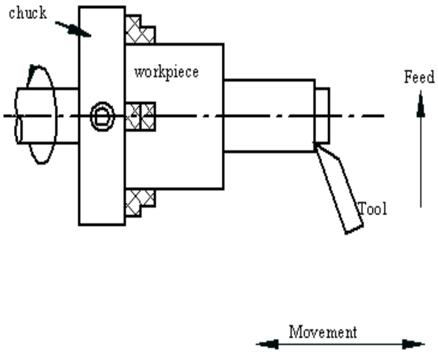
### 2.2 Lathe operations/ Turning operations:

The operations that can be performed on a lathe are

1. **By holding the job between centers or between chuck and dead center**
    a) Turning – plain, step, taper, etc
    b) Facing
    c) Chamfering
    d) Knurling
    e) Thread cutting
    f) Polishing
    g) Spinning
2. **By holding the job by a chuck alone**
    a) Turning and Facing of short length work piece.
    b) Drilling
    c) Reaming
    d) Boring
    e) Thread cutting, internal/external.
3. **By using special attachments.**
    a) Grinding
    b) Milling

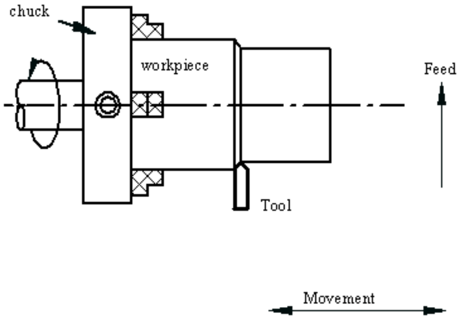
### 2.2.1 Facing

Is the operation of machining the ends of a piece of the work to produce a flat surface square with the axis. This is used to cut the work to the required length. The operation involves feeding the tool perpendicular to the axis of rotation of the work piece. A regular cutting tool may be used for facing a large work piece. The cutting edge should be set at the same height as the centre of the work piece. A properly ground facing tool is mounted in a tool holder in the tool post to accomplish facing operation.



### 2.2.2 Plain Turning

The process of metal removal from the cylindrical jobs is called straight or plain turning. Cross-slide and the carriage are used to perform turning operation and make the operation faster and economical. Plain turning operations are generally performed in two steps-rough and finish turning. Rough turning is usually done for rolled, cast or forged parts to remove the uneven or sandy or rough surface on the jobs. A roughing tool does roughing and used for excess stock removal. For finishing a tool with slightly round cutting edge is used. The depth of cut rate is at the range of 0.2 to 1 mm and the feed rate between 0.1 to 0.3 mm.

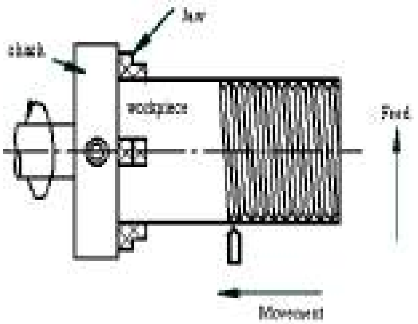


### 2.2.3 Step Turning

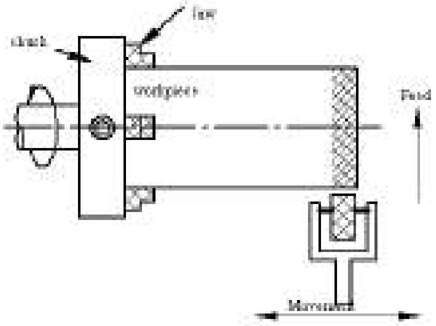
A step turning operation is performed using a step cutting tool, after the turning operation. The work is held in between the centers or with the chuck the tool is held at a height equal is the axis of the work. The depth to obtain the step on the cylinder is provided by cross slide movement and the carriage movement. These operations are performed manually/ automatically.

### 2.2.4 Thread Cutting:

A thread is a helical ridge formed on the cylindrical rod surface. By employing V-Shaped cutting tool it is possible to accomplish threads on the work piece. When the tool is moved longitudinally with linear uniform motion while the work piece is rotating with uniform speed. An appropriate gear ratio is maintained between the spindle on which the work piece is mounted and the lead screw has the ability to enable the tool to move longitudinally at the appropriate linear speed, the screw thread of the required pitch can be cut.



### 2.2.5 Knurling:



Knurling is an operation performed on the lathe to generate serrated surface on the work piece. This is used to produce a rough surface for griping like the barrel of the micrometer or screw gauge. This is done by a special tool called knurling tool which has a set of hardened roller with the desired serrations.

During knurling operation, the hardened rollers of the tool are pressed against the slowly rotating work pieces such that the impression of tool serrations are formed on the work pieces' surface. Usually, there are three different pattern of knurling produced as per requirements and is as shown.

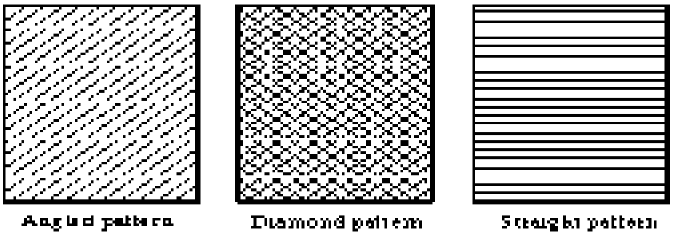
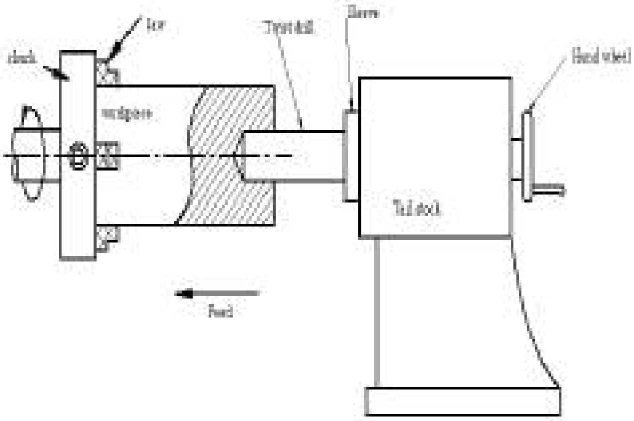

Angled pattern
Diamond pattern
Straight pattern

Fig: Patterns of Knurling

### 2.2.6 Drilling



Drilling is a metal cutting process carried out by a rotating cutting tool to make circular holes in solid materials. The tool which makes the hole is called a drill bit or twist drill. In operation the drill bit is fed against the revolving work piece by rotating the hand wheel of the tail stock as shown in fig. drilling on lathe is limited to produce hole through the axis of the work piece only. So drilling operation is best to perform on drilling machine.

### 2.2.7 Taper Turning

Taper turning is an operation to produce conical surface on the work piece. This can be machined by either work piece inclined to the axis of the lathe or tool moving inclined to the axis of the lathe.

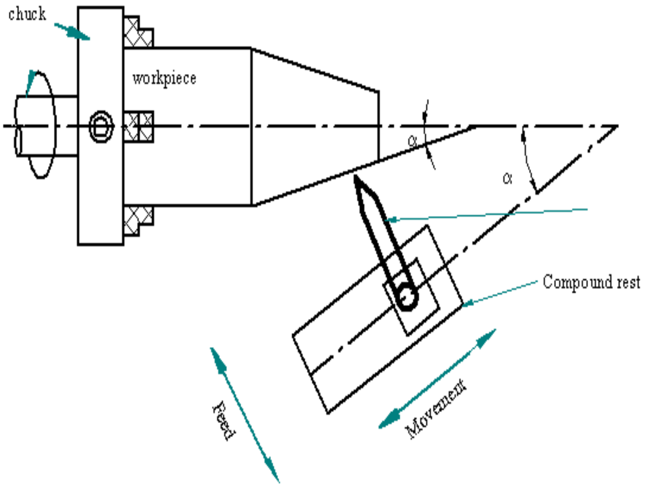
**There are many methods for taper turning**

a) By swiveling the compound rest.
b) By offsetting the tail stock.
c) By taper turning attachment.
d) By using form tool.

### a) Taper Turning by Swiveling the compound Tool Rest

$$\tan\alpha = \frac{(D-d)}{2l}$$

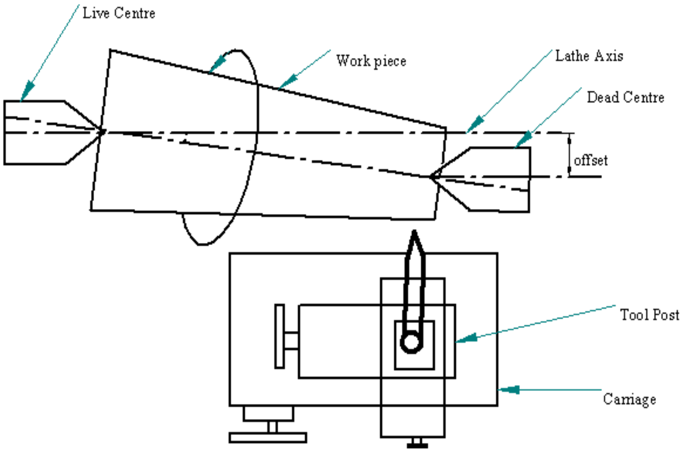
The work pieces which requires steep taper for short lengths the taper turning was done by swiveling the compound rest method. In this method the compound tool rest is swiveled to the required taper angle and then locked in the angular position. The carriage is also locked at that position. For taper turning the compound rest is moved linearly at an angle so that the cutting tool produces the tapered surface on the work piece. This method is limited to short taper lengths due to the limited movement of the compound tool rest.



The taper angle is calculated by,

Where, D= bigger diameter of the taper (mm)
d= smaller diameter of the taper (mm)
L= Length of the taper (mm)

### b) Taper Turning by Offsetting the Tailstock (or Tailstock set over method)



$$\mathbf{offset} = \frac{\mathbf{(D-d)L}}{\mathbf{2}\boldsymbol{l}}$$

In this method the work piece is inclined with respect to the lathe axis and tool movement is in line with the lathe axis to produce the required taper. Here the tail stock body is shifted by small distance (offset) laterally. This makes the work piece is shifted at one end and hence there will be an inclination with respect to the lathe axis. The tool is moved parallel to the lathe axis and fed against the revolving work piece which produces the required taper.

The offset is calculated by,

Where, D= bigger diameter of the taper (mm)
d= smaller diameter of the taper (mm)
L= Length of the taper (mm)
l= Total length of the work piece

This method is used to produce only external taper and suitable for small taper on long work pieces.

### c) Taper Turning by Taper Turning Attachment

A taper tuning attachment is used to cut both internal and external tapers. The taper turning attachment is as shown in fig, consists of a bracket which will be connected to the rear side of the lathe bed. A guide bar which can be swiveled in the horizontal plane and locked in position is mounted over the bracket. A guide block pivoted to a draw-link will slide in the longitudinal slot in the guide bar. The draw-link is connected firmly to the cross slide. The tool is mounted on the tool post slide. The cross slide is allowed to move freely on its ways by loosening the cross feed screw and the engaging nut.

## 2.3 Boring Operations:

In machining, boring is the process of enlarging a hole that has already been drilled (or cast) by means of a single-point cutting tool. Boring is used to achieve greater accuracy of the diameter of a hole, and can be used to cut a tapered hole. Boring can be viewed as the internal-diameter counterpart to turning, which cuts external diameters.

Boring operations may be carried out on various machine tools, such as lathe, milling or boring machine itself.

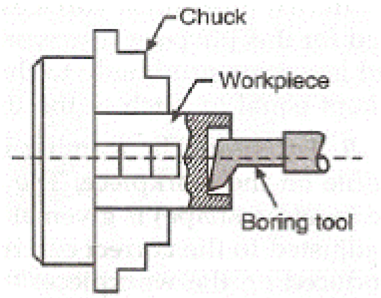


*Fig: Boring operation*

In boring operations, the single point cutting tool is set to the diameter of the hole to be turn and fed against the rotary workpiece. The diameter of the hole to be bore can be increased or decreased by moving the cutting tool in and out of the tool holder.

## 2.4 Shaping operations:

The shaper machine tool makes use of a single-point cutting tool that traverses the work and feeds over at the end of each stroke. The types of surfaces it is best able to produce are shown in figure.

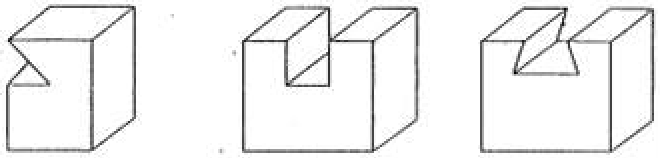


*Fig: Grooves machined on Shaper*

Contour work may also be done by coordinating the tool hand feed to a layout line, or by duplicating attachment places on the machine. Thus, contours can be formed with inexpensive single point cutting tools for short runs where the cost of from tools for milling work is prohibitive.

The different operations, which a shaper can perform, are as follows:

1. Machining horizontal surface
2. Machining vertical surface
3. Machining angular surface
4. Slot cutting
5. Key ways cutting
6. Machining irregular surface
7. Machining splines and cutting gears

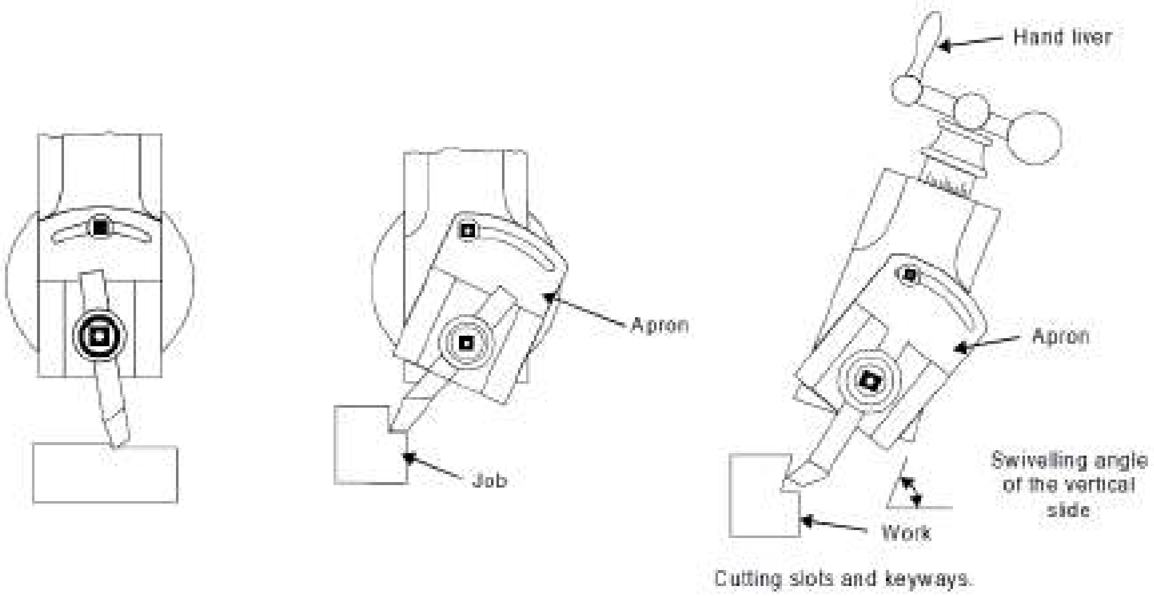


***Fig: (a) machining horizontal Surfaces (b) vertical Surfaces (c) angular surfaces***

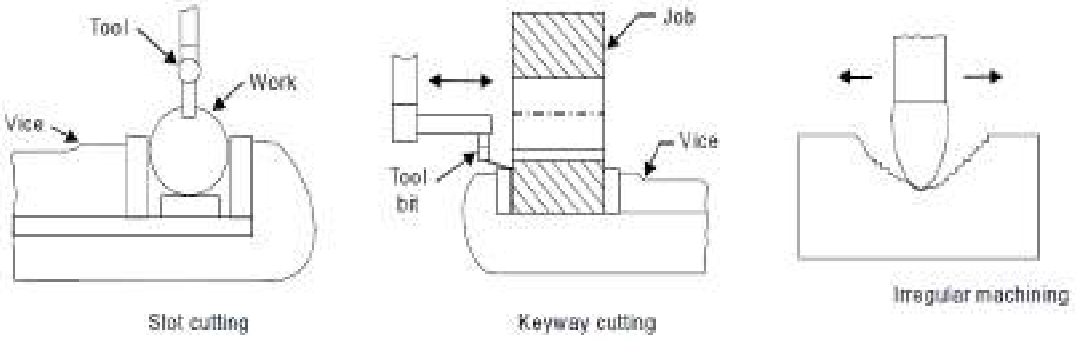


***Fig: (d) slot cutting (e)key way cutting (f) Shaping irregular surfaces***

## 2.5 Planning operations:

Figure depicts the working principle of a planer. In a planer, the work which is supported on the table reciprocates past the stationary cutting tool and the feed is imparted by the lateral movement of the tool. The tool is clamped in the tool holder and work on the table. Like shaper, the planner is equipped with clapper box to raise the tool in idle stroke.

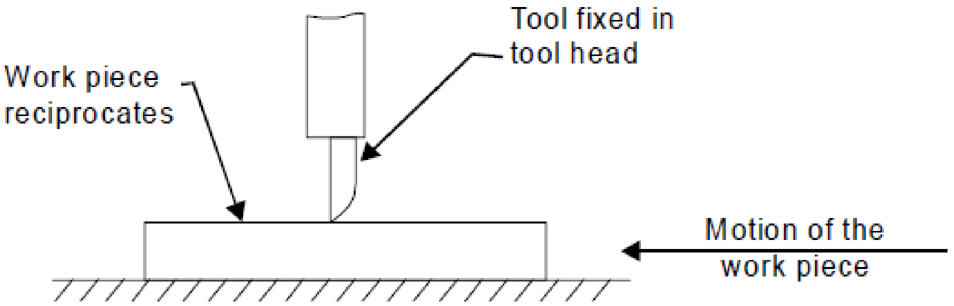


***Fig.: principle of planing operation***

### 2.5.1: Differences between Shaper and planer

| Sl. No | Shaper | Planer |
|---|---|---|
| 1 | The work is held stationary and the cutting tool on the ram is moved back and forth across the work | In a planer, the tool is stationary and the work piece travels back and forth under the tool |
| 2 | It is used for shaping much smaller jobs | A planer is meant for much larger jobs than can be undertaken on a shaper. |
| 3 | A shaper is a light machine | It is heavy duty machine |
| 4 | Shaper can employ light cuts and finer feeds | Planer can employ heavier cuts and coarse feed |
| 5 | Shaper uses one cutting tool at a time | Several tools can cut simultaneously on a planer |
| 6 | The shaper is driven by quick return motion mechanism | The planer table is driven either by gear drives or hydraulic means |

## 2.7 Drilling operations:

Though drilling is the primary operation performed in a drilling machine, a number of similar operations are also performed on holes using different tools. The different operations that can be performed in a drilling machine are:

1. Drilling
2. Reaming
3. Boring
4. Counter boring
5. Countersinking
6. Spot facing
7. Tapping

### 2.7.1 Drilling:

This is the operation of making a circular hole by removing a volume of metal from the job by a rotating cutting tool called drill as shown in Figure. Drilling removes solid metal from the job to produce a circular hole. Before drilling, the hole is located by drawing two lines at right angle and a center punch is used to make an indentation for the drill point at the center to help the drill in getting started. A suitable drill is held in the drill machine and the drill machine is adjusted to operate at the correct cutting speed. The drill machine is started and the drill starts rotating. Cutting fluid is made to flow liberally and the cut is started. The rotating drill is made to feed into the job. The hole, depending upon its length, may be drilled in one or more steps. After the drilling operation is complete, the drill is removed from the hole and the power is turned off.

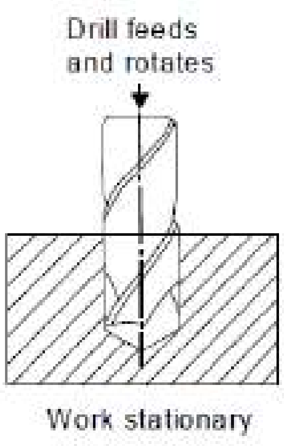


*Fig: Drilling operations*

### 2.7.2 Reaming:

This is the operation of sizing and finishing a hole already made by a drill. Reaming is performed by means of a cutting tool called reamer as shown in Figure. Reaming operation serves to make the hole smooth, straight and accurate in diameter. Reaming operation is performed by means of a multi-tooth tool called reamer. Reamer possesses several cutting edges on outer periphery and may be classified as solid reamer and adjustable reamer.

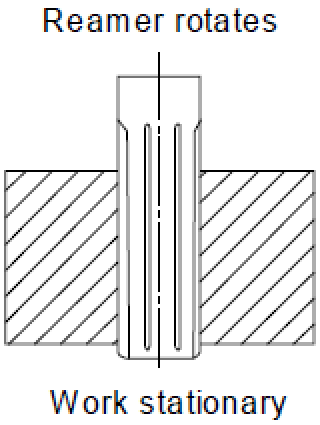


*Fig.: Reaming Operation*

### 2.7.3 Boring:

Figure shows the boring operation where enlarging a hole by means of adjustable cutting tools with only one cutting edge is accomplished. A boring tool is employed for this purpose.

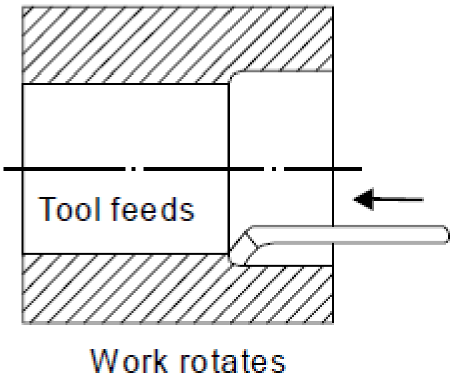


*Fig.: Boring Operations*

### 2.7.4 Counter boring:

Counter boring operation is shown in Figure. It is the operation of enlarging the end of a hole cylindrically, as for the recess for a counter-sunk rivet. The tool used is known as counter-bore.

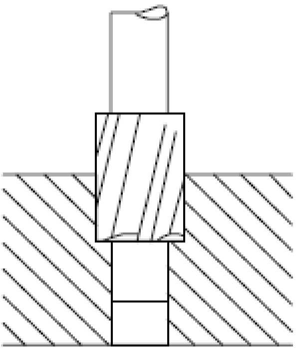
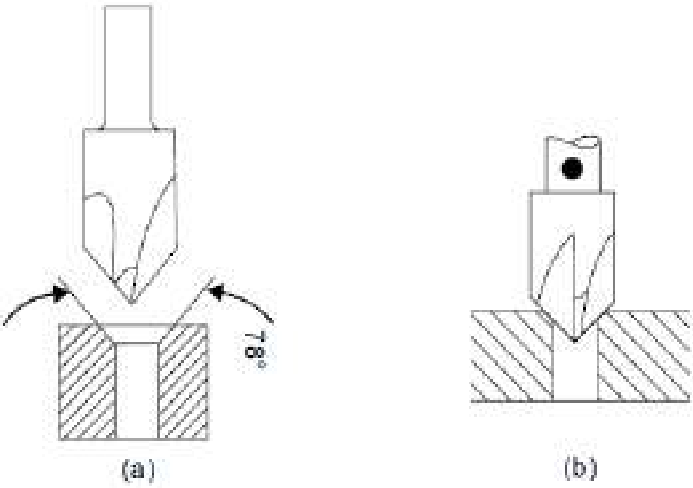


**Fig.: Counter Boring**

### 2.7.5: Counter Sinking:

Counter-sinking operation is shown in Figure. This is the operation of making a Cone shaped enlargement of the end of a hole, as for the recess for a flat head screw. This is done for providing a seat for counter sunk heads of the screws so that the latter may flush with the main surface of the work.



(a) (b)

***Fig.: Counter Sinking operation***

### 2.7.6: Spot Facing:

This is the operation of removing enough material to provide a flat surface around a hole to accommodate the head of a bolt or a nut. A spot-facing tool is very nearly similar to the counter-bore.

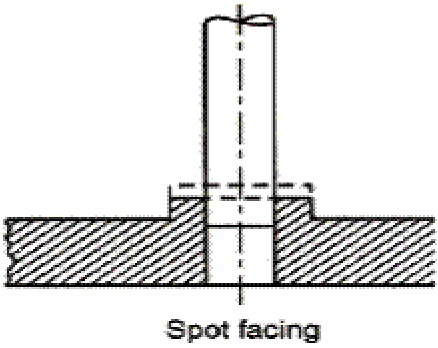


**Fig.: Spot Facing**

### 2.7.7 Tapping:

It is the operation of cutting internal threads by using a tool called a tap. A tap is similar to a bolt with accurate threads cut on it. To perform the tapping operation, a tap is screwed into the hole by hand or by machine. The tap removes metal and cuts internal threads, which will fit into external threads of the same size. For all materials except cast iron, a little lubricate oil is applied to improve the action. The tap is not turned continuously, but after every half turn, it should be reversed slightly to clear the threads. Tapping operation is shown in Figure.

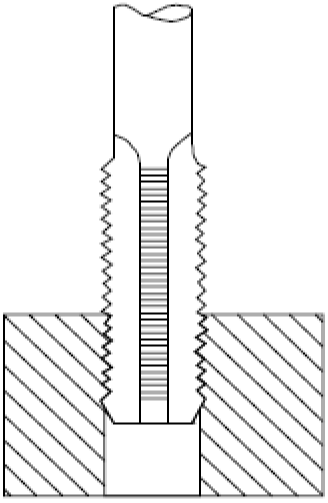


**Fig.: Tapping operations**

## 2.8 Milling operations:

Unlike a lathe, a milling cutter does not give a continuous cut, but begins with a sliding motion between the cutter and the work. Then follows a crushing movement, and then a cutting operation by which the chip is removed. Many different kinds of operations can be performed on a milling machine but a few of the more common operations will now be explained. These are:

### 2.8.1 Plain milling or slab milling

Figure (a) illustrates the plain and slab milling operation. It is a method of producing a plain, flat, horizontal surface parallel to the axis of rotation of the cutter.

### 2.8.2 Face milling

Figure (b) illustrates the face milling operation. It is a method of producing a flat surface at right angles to the axis of the cutter.

### 2.8.3 Side milling

Figure (c) illustrates the side milling operation. It is the operation of production of a flat vertical surface on the side of a work-piece by using a side milling cutter.

### 2.8.4 Angular milling

Figure (d) illustrates angular milling operation. It is a method of producing a flat surface making an angle to the axis of the cutter.

### 2.8.5 Gang-milling

Figure (e) illustrates the gang milling operation. It is a method of milling by means of two or more cutters simultaneously having same or different diameters mounted on the arbor of the milling machine.

### 2.8.6 Form milling

Figure (f) illustrates the form milling operation. It is, a method of producing a surface having an irregular outline.

### 2.8.7 End milling

Figure (g) illustrates end milling operation. It is a method of milling slots, flat surfaces, and profiles by end mills.

### 2.8.8 Profile milling

Figure (h) illustrates profile milling operation. It is the operation of reproduction of an outline of a template or complex shape of a master die on a workpiece.

### 2.8.9 Saw milling

Figure (i) illustrates saw milling operation. It is a method of producing deep slots and cutting materials into the required length by slitting saws.

### 2.8.10 T-slot milling

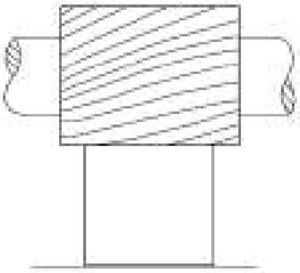
Figure (j) illustrates T-slot milling operation.

### 2.8.11 Keyway milling

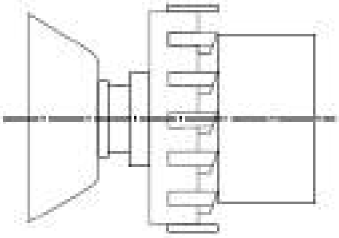
Figure (k) illustrates keyway milling operation.

### 2.8.12 Gear cutting milling

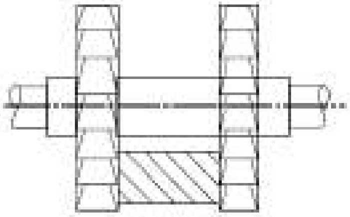
Figure (l) illustrates gear cutting milling operation.



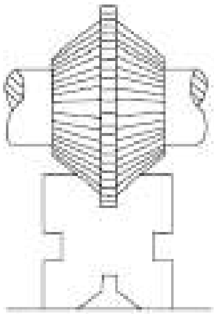
(a) Plane milling



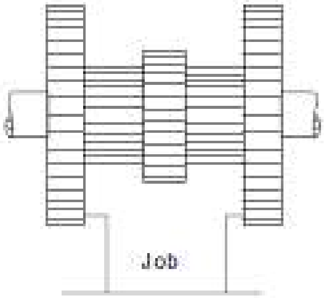
(b) Face milling



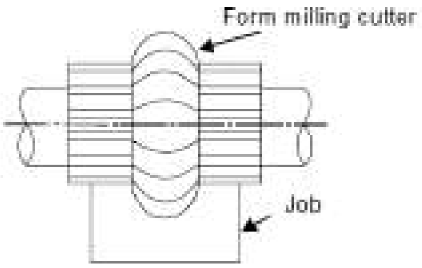
(c) Side milling



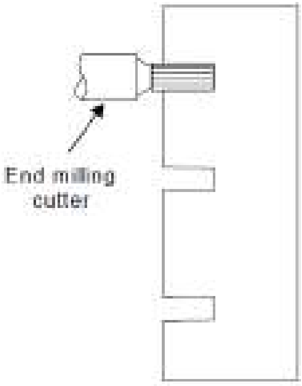
(d) Angular milling



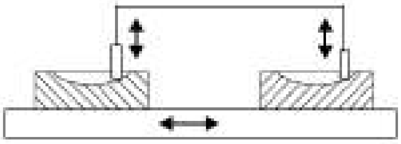
(e) Gang milling



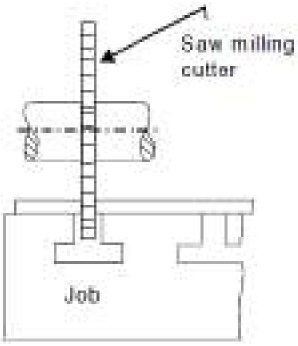
(f) Form Milling



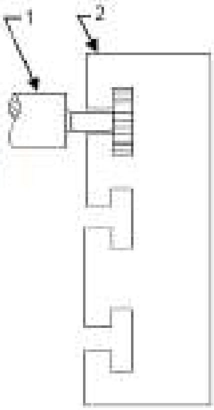
(g) End milling



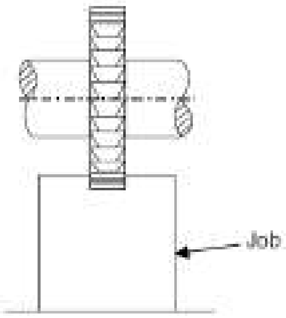
(h) Profile milling



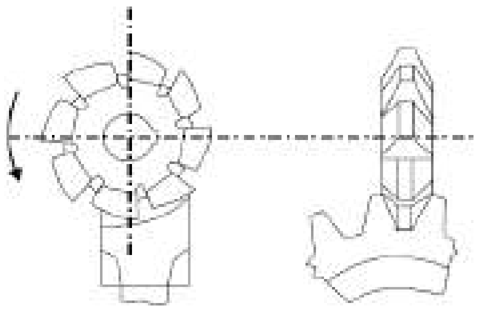
(i) Saw milling



(j)
T - Slot
Milling



(k) Key way milling



(l) Gear Cutting Milling

## 2.9 Grinding Operations

Grinding is a material removal process in which abrasive particles arc contained in a bonded grinding wheel that operates at very high surface speeds. The grinding wheel is usually disk shaped and is precisely balanced for high rotational speeds.

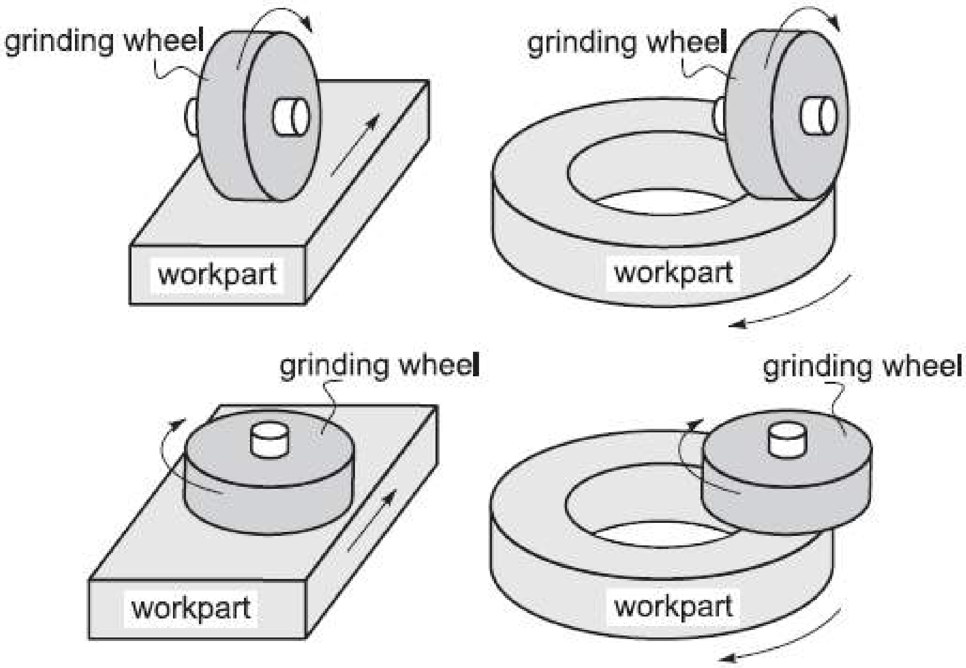
Grinding operations are carried out with a variety of wheel-work part configurations. The basic type of grinding are,

- surface grinding,
- cylindrical grinding, and
- centreless grinding.

### 2.9.1 Surface grinding:

Surface grinding is an abrasive machining process in which the grinding wheel removes material from the plain flat surfaces of the workpiece.

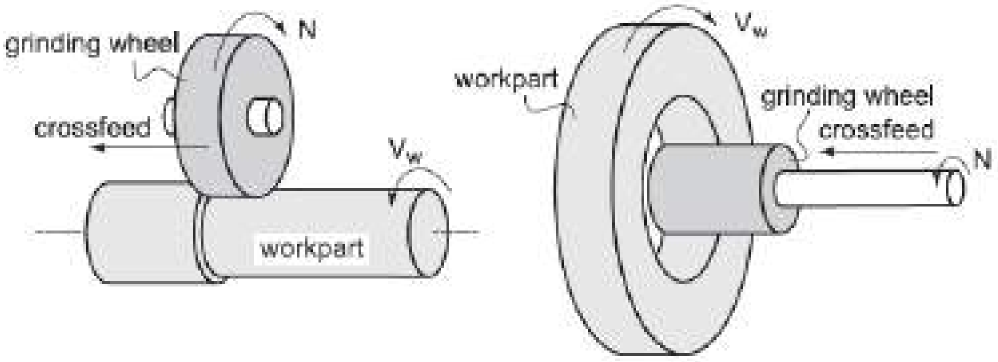
In surface grinding, the spindle position is either horizontal or vertical, and the relative motion of the workpiece is achieved either by reciprocating the workpiece past the wheel or by rotating it. The possible combinations of spindle orientations and workpiece motions yield four types of surface grinding processes illustrated in the figure:



Four types of surface grinding with horizontal or vertical spindles, and with reciprocating linear motion or rotating motion of the workpiece.

### 2.9.2 Cylindrical grinding

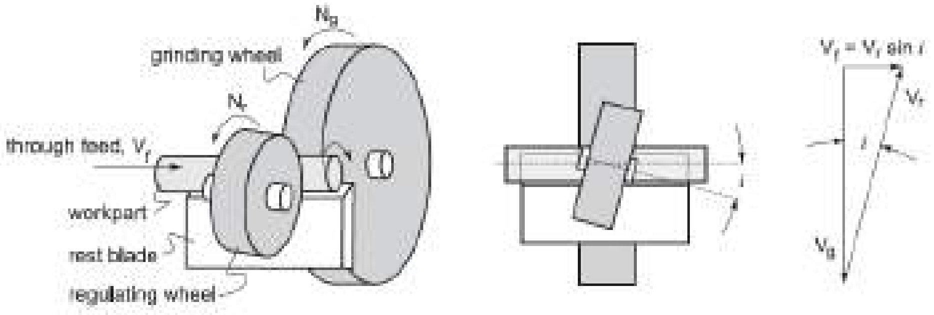
In this operation, the external or internal cylindrical surface of a workpiece are ground. In external cylindrical grinding (also center-type grinding) the workpiece rotates and reciprocates along its axis, although for large and long work parts the grinding wheel reciprocates. In internal cylindrical grinding, a small wheel grinds the inside diameter of the part. The workpiece is held in a rotating chuck in the headstock and the wheel rotates at very high rotational speed. In this operation, the workpiece rotates and the grinding wheel reciprocates.



Two types of surface grinding, (*Left*) external, and (*Right*) internal.

### 2.9.3 Centerless grinding

Centerless grinding is a process for continuously grinding cylindrical surfaces in which the workpiece is supported not by centers or chucks but by a rest blade. The workpiece is ground between two wheels. The larger grinding wheel does grinding, while the smaller regulating wheel, which is tilted at an angle i, regulates the velocity $V_f$ of the axial movement of the workpiece. Centerless grinding can also be external or internal, traverse feed or plunge grinding. The most common type of centerless grinding is the external traverse feed grinding, illustrated in the figure:



External traverse feed centerless grinding. The regulating wheel is tilted at an angle *i* to control the velocity of through feed.

## 2.10 Machining parameters:

**The parameters which effect on the machining process are**

1. **Speed:**

   It expresses with the number of rotations (rpm) of a spindle. When the rotating speed is high, processing speed becomes quick, and a processing surface is finely finished. However, since a little operation mistakes may lead to the serious accident, it is better to set low rotating speed at the first stage.

2. **Depth of Cut**

   The depth of cut affects to the processing speed and the roughness of surface. When the cutting depth is big, the processing speed becomes quick, but the surface temperature becomes high, and it has rough surface. Moreover, a life of byte also becomes short. If you do not know a suitable cutting depth, it is better to set to small value.

3. **Feed**

   The feed of the tool also affects to the processing speed and the roughness of surface. When the sending speed is high, the processing speed becomes quick. When the sending speed is low, the surface is finished beautiful. There are 'manual sending' which turns and operates a handle, and 'automatic sending' which advances a byte automatically. A beginner must use the manual sending. Because serious accidents may be caused, such as touching the rotating chuck around the byte in automatic sending.

## OUTCOMES:

- Describe various machining processes pertaining to relative motions between tool & work piece

## QUESTIONS:

1. Explain the working of auxillary cutting motions in machine tool.
2. What is machining? Give classification of machining process.
3. List and explain the operations carried over lathe machine.
4. What are the operations performed on Drilling machine?
5. Differentiate between shaping and planning
6. List and Explain various operations performed on milling machine.
7. Describe the parameters which affects the machining operations.

**FURTHER READING:**

**1. “Metal cutting principles”**, Milton C. Shaw, Oxford University Press, Second Edition,2005.
**2. “Manufacturing Technology”,** Vol 2, P N Rao, McGraw Hill Education, 3rd Edition

**3. “Workshop Technology, Vol-II”,** by Hazara Chowdary

# MODULE-3

# CUTTING TOOL MATERIALS, GEOMETRY AND SURFACE FINISH

## LESSON CONTENTS:

Introduction, desirable Properties and Characteristics of cutting tool materials, cutting tool geometry, cutting fluids and its applications, surface finish, effect of machining parameters on surface finish.

**Machining equations for cutting operations:** Turning, Shaping, Planing, slab milling, cylindrical grinding and internal grinding, Numerical problems

## OBJECTIVES:

- To develop the knowledge about Cutting tool materials, geometry and their characteristics.
- To study about the Cutting fluids, their application and surface finish
- Solving numerical on machining equations for various cutting operations

## 3.0 Introduction:

The selection of cutting-tool materials for a particular application is among the most important factors in machining operations. We will discuss throughout this chapter the relevant properties and performance characteristics of all major types of cutting-tool materials, which will help us in tool selection.

## 3.1 Characteristics of Cutting tool materials:

The cutting tool is subjected to (a) high temperatures, (b) high contact stresses, and (c) rubbing along the tool-chip interface and along the machined surface. Consequently, the cutting-tool material must possess the following characteristics:

- **Hot hardness:** Hot hardness is the ability of a material to retain its hardness at high temperatures. This is required because of the high-temperature environment in which the tool operates.
- **Toughness and impact Strength:** To avoid fracture failure, the tool material must possess high toughness. Toughness is the capacity of a material to absorb energy without failing. It is usually characterized by a combination of strength and ductility in the material.

- **Wear resistance.** Hardness is the single most important property needed to resist abrasive wear. All cutting-tool materials must be hard. However, wear resistance in metal cutting depends on more than just tool hardness, because of the other tool-wear mechanisms. Other characteristics affecting wear resistance include surface finish on the tool (a smoother surface means a lower coefficient of friction), chemistry of tool and work materials, and whether a cutting fluid is used.

- **Thermal Shock Resistance:** To withstand the rapid temperature cycling encountered in interrupted cutting.

- **Chemical stability and inertness** with respect to the material being machined, to avoid or minimize any adverse reactions, adhesion, and tool-chip diffusion that would contribute to tool wear.

### 3.2 Cutting Tool Materials:

The various cutting tool materials which are broadly used in machining of materials are:
High Speed Steels, Carbon Steels, Carbides, Coated tools, Cubic boran Nitrade, Diamond, Aluminium Oxides etc.,

**Carbon Steels:** Carbon steels are the oldest tool materials and have been used widely for drills, taps, broaches, and reamers since the 1880s. Low-alloy and medium-alloy steels were developed later for similar applications but with longer tool life. Although inexpensive and easily shaped and sharpened, these steels do not have sufficient hot hardness and wear resistance for cutting at high speeds when the temperature rises significantly. Their use is limited to very low speed cutting operations, particularly in woodworking; hence, they are not of any particular significance in modern machining operations.

**High Speed Steels:** High-speed steel (HSS) tools are so named because they were developed to machine at higher speeds than Carbon Steels. High-speed steel (HSS) is a highly alloyed tool steel capable of maintaining hardness at elevated temperatures better than high carbon and low alloy steels. Its good hot hardness permits tools made of HSS to be used at higher cutting speeds. Compared with the other tool materials at the time of its development, it was truly deserving of its name ‘‘high speed.’’ A wide variety of high-speed steels are available, but they can be divided into two basic types: (1) tungsten-type, designated T-grades by the American Iron and Steel Institute (AISI); and (2) molybdenum-type, designated M-grades by AISI.

**Tungsten-type HSS** contains tungsten (W) as its principal alloying ingredient. Additional alloying elements are chromium (Cr), and vanadium (V). One of the original and best known HSS grades is T1, or 18-4-1 high-speed steel, containing 18%W,4%Cr, and 1%V. **Molybdenum HSS** grades contain combinations of tungsten and molybdenum(Mo), plus the same additional alloying elements as in the T-grades. Cobalt (Co) is sometimes added to HSS to enhance hot hardness. Of course, high-speed steel contains carbon, the element common to all steels.

**Cemented Carbides:** Cemented carbides (also called sintered carbides) are a class of hard tool material formulated from tungsten carbide (WC) using powder metallurgy techniques with cobalt (Co) as the binder. There may be other carbide compounds in the mixture, such as titanium carbide (TiC) and/or tantalum carbide (TaC), in addition to WC.

Because of their high hardness over a wide range of temperatures high elastic modulus, high thermal Conductivity, and low thermal expansion, carbides are among the most important, versatile, and cost-effective tool and die materials for a wide range of applications. The two major groups of carbides used for machining are tungsten Carbide and titanium carbide.

**Ceramics:** ceramic cutting tools are composed primarily of fine-grained aluminium oxide (Al2O3), pressed and sintered at high pressures and temperatures with no binder into insert form (Section 17.2). The aluminum oxide is usually very pure (99% is typical), although some manufacturers add other oxides (such as zirconium oxide) in small amounts. In producing ceramic tools, it is important to use a very fine grain size in the alumina powder, and to maximize density of the mix through high-pressure compaction to improve the material’s low toughness.

Aluminum oxide cutting tools are most successful in high-speed turning of cast iron and steel. Applications also include finish turning of hardened steels using high cutting speeds, low feeds and depths, and a rigid work setup. Many premature fracture failures of ceramic tools are because of non-rigid machine tool setups, which subject the tools to mechanical shock. When properly applied, ceramic cutting tools can be used to obtain very good surface finish. Ceramics are not recommended for heavy interrupted cut operations (e.g., rough milling) because of their low toughness. In addition to its use as inserts in conventional machining operations, Al2O3 is widely used as an abrasive in grinding and other abrasive processes.

**Cubic Boron Nitride:** Next to diamond, cubic boron nitride (Section 7.3.3) is the hardest material known, and its fabrication into cutting tool inserts is basically the same as Synthetic polycrystalline Diamonds; that is, coatings on WC–Co inserts. Cubic boron nitride (symbolized CBN) does not react chemically with iron and nickel as SPD does; therefore, the applications of CBN-coated tools are for machining steel and nickel-based alloys. Both SPD and CBN tools are expensive, as one might expect, and the applications must justify the additional tooling cost.

**Diamonds:** Diamond is the hardest material known. By some measures of hardness, diamond is three to four times as hard as tungsten carbide or aluminum oxide. Since high hardness is one of the desirable properties of a cutting tool, it is natural to think of diamonds for machining and grinding applications. Synthetic diamond cutting tools are made of sintered polycrystalline diamond (SPD), which dates from the early 1970s. Sintered polycrystalline diamond is fabricated by sintering fine-grained diamond crystals under high temperatures and pressures into the desired shape. Little or no binder is used. The crystals have a random orientation and this adds considerable toughness to the SPD tools compared with single crystal diamonds. Tool inserts are typically made by depositing a layer of SPD about 0.5mm (0.020 in) thick on the surface of a cemented carbide base. Very small inserts have also been made of 100% SPD.

Applications of diamond cutting tools include high-speed machining of nonferrous metals and abrasive non-metals such as fiberglass, graphite, and wood. Machining of steel, other ferrous metals, and nickel-based alloys with SPD tools is not practical because of the chemical affinity that exists between these metals and carbon (a diamond, after all, is carbon.)

## 3.3 Cutting Tool Geometry:

A cutting tool must possess a shape that is suited to the machining operation. One important way to classify cutting tools is according to the machining process. cutting tools can be divided into single-point tools and multiple-cutting-edge tools. Single-point tools are used in turning, boring, shaping, and planing. Multiple-cutting-edge tools are used in drilling, reaming, tapping, milling, broaching, and sawing. Many of the principles that apply to single-point tools also apply to the other cutting-tool types, simply because the mechanism of chip formation is basically the same for all machining operations.

### 3.3.1 Single point Tool Geometry:

The general shape of a single-point cutting tool is illustrated in Figure 3.1.

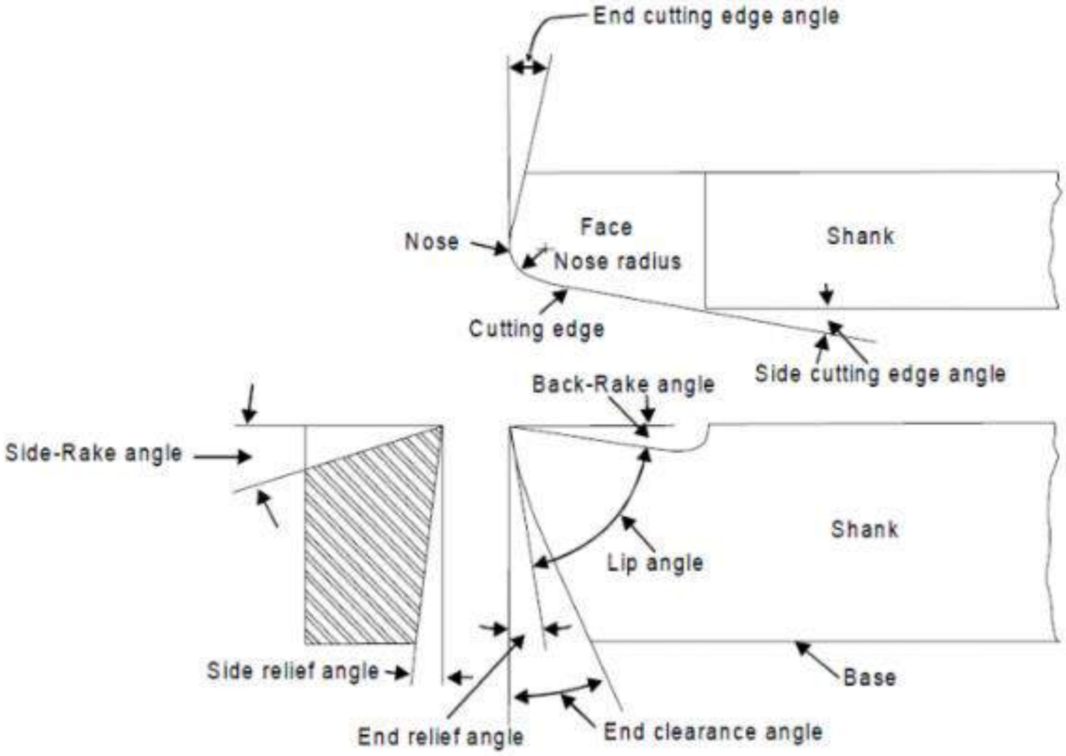


*Figure 3.1: Single point Cutting Tool nomenclature*

**i)** **Shank:** It is that portion of the tool which will be hold on the tool post.

**ii)** **Back Rake angle:** Back rake angle is the angle between the face of the single point cutting tool and a line parallel with base of the tool measured in a perpendicular plane through the side cutting edge. If the slope face is downward toward the nose, it is negative back rake angle and if it is upward toward nose, it is positive back rake angle. Back rake angle helps in removing the chips away from the workpiece.

**iii)** **Side rake angle:** Side rake angle is the angle by which the face of tool is inclined sideways. Side rake angle is the angle between the surface the flank immediately below the point and the line down from the point perpendicular to the base. Side rake angle of cutting tool determines the thickness of the tool behind the cutting edge. It is provided on tool to provide clearance between workpiece and tool so as to prevent the rubbing of workpiece with end flake of tool.

**iv)** **End relief angle:** End relief angle is defined as the angle between the portion of the end flank immediately below the cutting edge and a line perpendicular to the base of the tool, measured at right angles to the flank. End relief angle allows the tool to cut without rubbing on the workpiece.

v) **Side relief angle:** Side rake angle is the angle between the portion of the side flank immediately below the side edge and a line perpendicular to the base of the tool measured at right angles to the side. Side relief angle is the angle that prevents the interference as the tool enters the material. It is incorporated on the tool to provide relief between its flank and the workpiece surface.

vi) **End cutting edge angle:** End cutting edge angle is the angle between the end cutting edge and a line perpendicular to the shank of the tool. It provides clearance between tool cutting edge and workpiece

vii) **Side cutting edge angle:** Side cutting edge angle is the angle between straight cutting edge on the side of tool and the side of the shank. It is responsible for turning the chip away from the finished surface.

viii) **Nose Radius:** It is the fillet ground on the edge of the cutting point. This is done in order improve the surface finish on the workpiece while machining.

ix) **Shank:** It is that portion of the tool which will be hold on the tool post.

### 3.3.2 Twist Drill Tool Geometry:

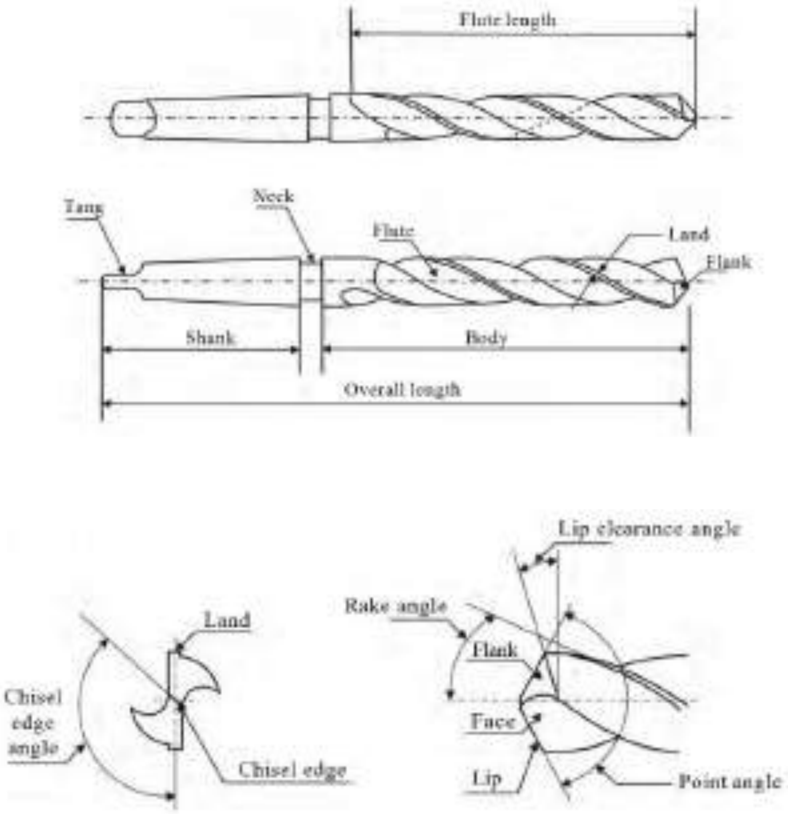


*Figure 3.2: Twist Drill Tool nomenclature*

**Axis:** It is the longitudinal center line of the drill running through the centres of the tang and the chisel edge.

**Body:** It is the part of the drill from its extreme point to the commencement of the neck, if present. Otherwise, it is the part extending up to the commencement of the shank. Helical grooves are cut on the body of the drill.

**Shank:** It is the part of the drill by which it is held and driven. It is found just above the body of the drill. The shank may be straight or taper. The shank of the drill can be fitted directly into the spindle or by a tool holding device.

**Tang:** The flattened end of the taper shank is known as tang. It is meant to fit into a slot in the spindle or socket. It ensures a positive drive of the drill.

**Neck:** It is the part of the drill, which is diametrically undercut between the body and the shank of the drill. The size of the drill is marked on the neck.

**Point:** It is the sharpened end of the drill. It is shaped to produce lips, faces, flanks and chisel edge.

**Lip:** It is the edge formed by the intersection of flank and face. There are two lips and both of them should be of equal length. Both lips should be at the same angle of inclination with the axis (59°).

**Land:** It is the cylindrically ground surface on the leading edges of the drill flutes adjacent to the body clearance surface. The alignment of the drill is maintained by the land. The hole is maintained straight and to the right size.

**Flutes:** The grooves in the body of the drill are known as flutes. Flutes form the cutting edges on the point. It allows the chips to escape and make them curl. It permits the cutting fluid to reach the cutting edges.

**Chisel edge angle:** The obtuse angle included between the chisel edge and the lip as viewed from the end of the drill. It usually ranges from 120° to 135°.

**Helix angle or rake angle:** The helix or rake angle is the angle formed by the leading edge of the land with a plane having the axis of the drill. If the flute is straight, parallel to the drill axis, then there would be no rake. If the flute is right handed, then it is positive rake and the rake is negative if it is left handed. The usual value of rake angle is 30° or 45°.

**Point angle:** This is the angle included between the two lips projected upon a plane parallel to the drill axis and parallel to the two cutting lips. The usual point angle is 118°. When hard alloys are drilled the value increases.

**Lip clearance angle:** The angle formed by the flank and a plane at right angles to the drill axis. The angle is normally measured at the periphery of the drill. The lip clearance angle ranges from 12° to 15°.

### 3.3.3 Milling Cutter Nomenclature:

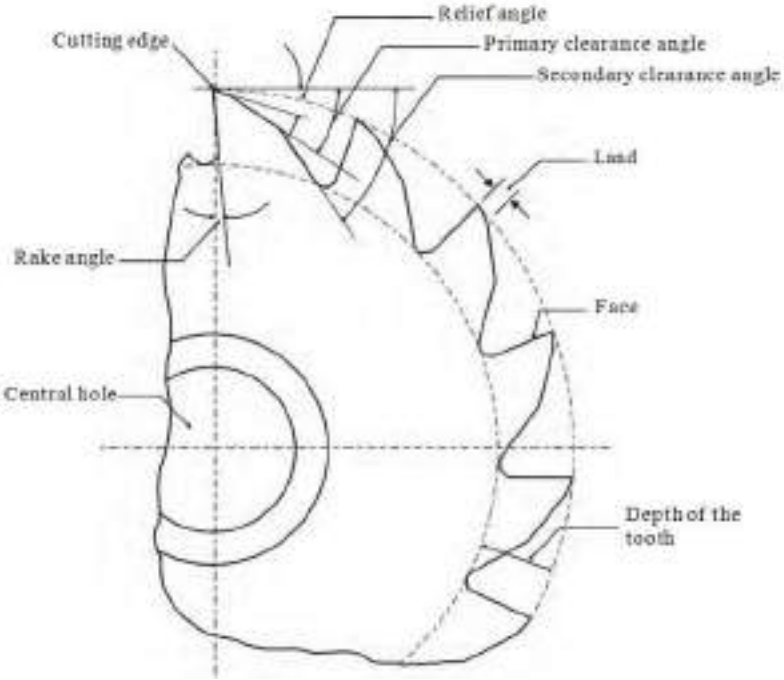


*Figure 3.3: Milling cutter Nomenclature*

**Body of cutter**: It is the part of the cutter left after exclusion of the teeth.

**Face**: The portion of the teeth next to the cutting edge is known as face.

**Land**: The relieved back portion of the tooth adjacent to the cutting edge. It is relieved to avoid interference between the surface being machined and the cutter.

**Outside diameter**: The diameter of the circle passing through the peripheral cutting edges.

**Relief angle**: It is angle the between the land of the tooth and the tangent to the outside diameter of the cutter at the cutting edge of the particular tooth. (approx. 7.5 °)

**Primary clearance angle**: It is the angle between the back of the tooth and the tangent drawn to the outside diameter of the cutter at the cutting edge. (approx. 15 °)

**Secondary clearance angle**: It is the angle formed by the secondary clearance surface and the tangent to the periphery of the cutter at the cutting edge.

**Rake angle**: The angle measured in the diametral plane between the face of the tooth and a radial line passing through the cutting edge of the tooth. The rake angles may be positive, negative or zero.

### 3.4 Cutting fluids & its applications:

A cutting fluid is any liquid or gas that is applied directly to the machining operation to improve cutting performance. Cutting fluids address two main problems: (1) heat generation at the shear zone and friction zone, and (2) friction at the tool–chip and tool–work interfaces.

In addition to removing heat and reducing friction, cutting fluids provide additional benefits, such as washing away chips (especially in grinding and milling), reducing the temperature of the work part for easier handling, reducing cutting forces and power requirements, improving dimensional stability of the work part, and improving surface finish.

### 3.4.1 Types of Cutting fluids:

There are four categories of cutting fluids
according to chemical formulation: (1) cutting oils, (2) emulsified oils, (3) semi chemical fluids, and (4) chemical fluids.

**Cutting oils** are based on oil derived from petroleum, animal, marine, or vegetable origin. Mineral oils (petroleum based) are the principal type because of their abundance and generally desirable lubricating characteristics. To achieve maximum lubricity, several types of oils are often combined in the same fluid. Chemical additives are also mixed with the oils to increase lubricating qualities. These additives contain compounds of sulphur, chlorine, and phosphorus, and are designed to react chemically with the chip and tool surfaces to form solid Films (extreme pressure lubrication) that help to avoid metal-to-metal contact between the two.

**Emulsified oils** consist of oil droplets suspended in water. The fluid is made by blending oil (usually mineral oil) in water using an emulsifying agent to promote blending and stability of the emulsion. A typical ratio of water to oil is 30:1. Chemical additives based on sulphur, chlorine, and phosphorus are often used to promote extreme pressure lubrication. Because they contain both oil and water, the emulsified oils combine cooling and lubricating qualities in one cutting fluid.

**Chemical fluids** are chemicals in a water solution rather than oils in emulsion. The dissolved chemicals include compounds of sulphur, chlorine, and phosphorus, plus wetting agents. The chemicals are intended to provide some degree of lubrication to the solution. Chemical fluids provide good coolant qualities but their lubricating qualities are less than the other cutting fluid types. Semi-chemical fluids have small amounts of emulsified oil added to increase the lubricating characteristics of the cutting fluid. In effect, they are a hybrid class between chemical fluids and emulsified oils.

### 3.4.2 Applications/ functions/ purpose of Cutting fluids:

The primary function of cutting fluid is temperature control through cooling and lubrication. Application of cutting fluid also improves the quality of the workpiece by continually removing metal fines and cuttings from the tool and cutting zone.

1. **To cool the tool:** Cooling the tool is necessary to prevent metallurgical damage and to assist in decreasing friction at the tool chip interface and at the tool workpiece interface.
2. **To cool the work piece:** The role of the cutting fluid in cooling the workpiece is to prevent its excessive thermal distortion.
3. **To lubricate and reduce the friction:** The energy or power consumption in removing metal is reduced. Absorption or wear on the cutting tool is reduced thereby increasing the life of the tool.
4. **To improve the surface finish.**
5. **To protect the finished surface from corrosion.**
6. **To cause the chips break up into small parts.**
7. **To wash away the chips away from the tool.**

### 3.4.3 Method of application of cutting Fluids:

Cutting fluid may be applied to a cutting tool/workpiece interface through manual, flood or mist application.

**Manual application:** Simply consists of an operator using a container, such as an oil can, to apply cutting fluid to the cutting tool/workpiece. Although this is the easiest and least costly method of fluid application, it has limited use in machining operations and is often hampered by inconsistencies in application.

**Flood application:** Delivers fluid to the cutting tool/workpiece interface by means of a pipe, hose or nozzle system. Fluid is directed under pressure to the tool/workpiece interface in a manner that produces maximum results. Pressure, direction and shape of the fluid stream must be regulated in order to achieve optimum performance.

**Mist Application:** Cutting fluids may also be atomized and blown onto the tool/workpiece interface via mist application. This application method requires adequate ventilation to protect the machine tool operator. The pressure and direction of the mist stream are also crucial to the success of the application.

### 3.5 Parameters affecting the surface finish:

The major machining parameters affecting the surface finish are Speed, feed and depth of cut. Other parameters are coolants, machining conditions, cutting tool geometry, tool materials etc.,

1. **Speed:** Speed at which the work piece is moved against the cutting tool or the tool against the workpiece plays a vital role. As the cutting speed is more at lower feed rate the machined surface will be smoother. But at higher feed rates the surface finish will be rough as there is a huge friction and heat generation.
2. **Feed:** In order to obtain better surface finish, the feed rate should be keep low. Also it is important to keep the cutting speed high at lower feed rates.
3. **Depth of cut:** Lower depth of cut is always preferred in order to obtain good surface finish. Higher the depth of cut larger will be the friction between the tool and work interface.
4. **Cutting tool Geometry**: The tool should have maintained the desired shape and angels. Wear is unavoidable in machining process. However, the change in the tool geometry results in excessive wear of the tool and also affecting poor surface finish.
5. **Cutting tool materials:** Selection of appropriate cutting tool materials is essential in machining process. Finishing operations requires softer cutting tools than machining rough surfaces. Also based the work piece to be machined, cutting tool is selected.

## 3.6 Machining equations for cutting operations:

**3.6.1 Turning:** Turning is a machining process in which a single-point tool removes material from the surface of a rotating workpiece. Turning is traditionally carried out on a machine tool called a lathe, which provides power to turn the part at a given rotational speed and to feed the tool at a specified rate and depth of cut.

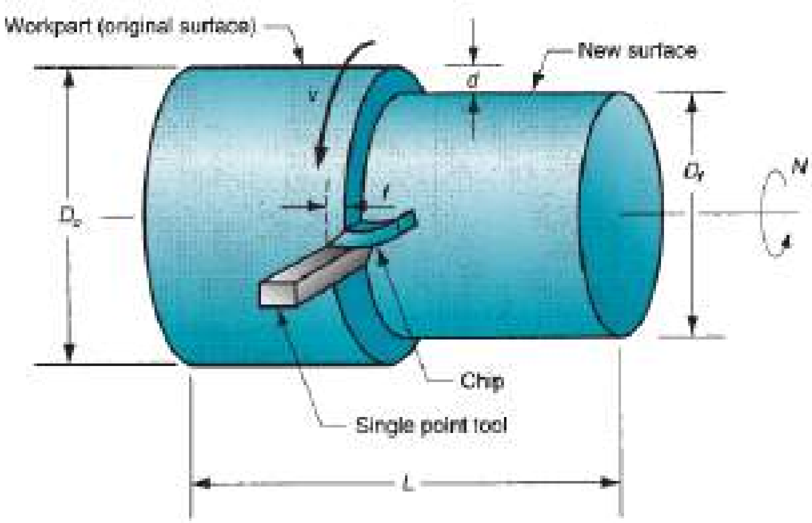


*Figure 3.4: Turning operations*

The rotational speed in turning is related to the desired cutting speed at the surface of the cylindrical workpiece by the equation

$$N = \frac{v}{\pi D_o}$$

where N=rotational speed, rev/min; v= cutting speed, m/min (ft/min); and Do = original diameter of the part, m (ft).

The turning operation reduces the diameter of the work from its original diameter Do to a final diameter Df, as determined by the depth of cut d:

$$D_f = D_o - 2d$$

The feed in turning is generally expressed in mm/rev (in/rev). This feed can be converted to a linear travel rate in mm/min (in/min) by the formula,

$$f_r = Nf$$

where fr = feed rate, mm/min (in/min); and f = feed, mm/rev (in/rev).

The time to machine from one end of a cylindrical work part to the other is given by,

$$T_m = \frac{L}{f_r}$$

where Tm = machining time, min; and L = length of the cylindrical work part, mm (in). A more direct computation of the machining time is provided by the following equation:

$$T_m = \frac{\pi D_o L}{fv}$$

Where Do = work diameter, mm(in); L = work part length, mm(in); f = feed, mm/rev (in/rev); and v = cutting speed, mm/min (in/min). As a practical matter, a small distance is usually added to the work part length at the beginning and end of the piece to allow for approach and over travel of the tool. Thus, the duration of the feed motion past the work will be longer than Tm. The volumetric rate of material removal can be most conveniently determined by the following equation:

$$R_{MR} = vfd$$

Where, RMR= material removal rate, mm3/min (in3/min). In using this equation, the units for f are expressed simply as mm(in), in effect neglecting the rotational character of turning. Also, care must be exercised to ensure that the units for speed are consistent with those for f and d.

### 3.6.2 Drilling:

Drilling is a machining operation used to create a round hole in a work part. This contrasts with boring, which can only be used to enlarge an existing hole. Drilling is usually performed with a rotating cylindrical tool that has two cutting edges on its working end. The rotating drill feeds into the stationary work part to form a hole whose diameter is equal to the drill diameter. Drilling is customarily performed on a drill press, although other machine tools also perform this operation.

The cutting speed in a drilling operation is the surface speed at the outside diameter of the drill. It is specified in this way for convenience, even though nearly all of the cutting is actually performed at lower speeds closer to the axis of rotation. To set the desired cutting speed in drilling, it is necessary to determine the rotational speed of the drill. Letting N represent the spindle rev/min,

$$N = \frac{v}{\pi D}$$

where v = cutting speed, mm/min (in/min); and D = the drill diameter, mm (in). In some drilling operations, the workpiece is rotated about a stationary tool, but the same formula applies. Feed f in drilling is specified in mm/rev (in/rev). Recommended feeds are roughly proportional to drill diameter; higher feeds are used with larger diameter drills. Since there are (usually) two cutting edges at the drill point, the uncut chip thickness (chip load) taken by each cutting edge is half the feed. Feed can be converted to feed rate using the same equation as for turning:

$$f_r = Nf$$

where fr = feed rate, mm/min (in/min). Drilled holes are either through holes or blind holes, Figure 3.5. In through holes, the drill exits the opposite side of the work; in blind holes, it does not. The machining time required to drill a through hole can be determined by the following formula:

$$T_m = \frac{t + A}{f_r}$$

where Tm = machining (drilling) time, min; t = work thickness, mm (in); fr = feed rate, mm/min (in/min); and A = an approach allowance that accounts for the drill point angle, representing the distance, the drill must feed into the work before reaching full diameter, Figure 22.10(a). This allowance is given by

$$A = 0.5\,D \tan\left(90 - \frac{\theta}{2}\right)$$

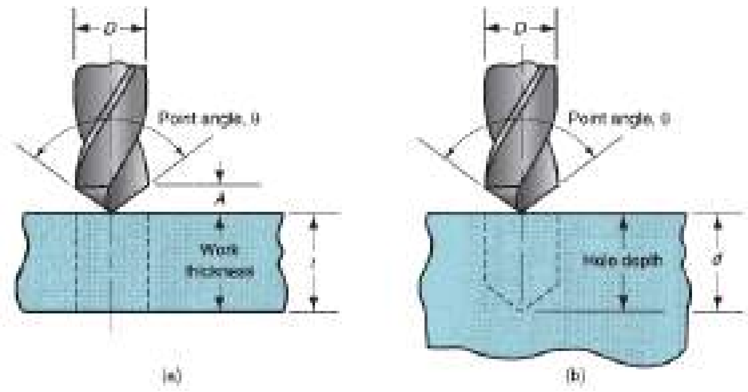


*Figure 3.5: Through hole and blind hole*

where A = approach allowance, mm (in); and u = drill point angle. In drilling a through hole, the feed motion usually proceeds slightly beyond the opposite side of the work, thus making the actual duration of the cut greater than Tm by a small amount.

In a blind-hole, hole depth d is defined as the distance from the work surface to the depth of the full diameter, Figure 3.5. Thus, for a blind hole, machining time is given by

$$T_m = \frac{d + A}{f_r}$$

where A = the approach allowance by The rate of metal removal in drilling is determined as the product of the drill cross sectional area and the feed rate:

$$R_{MR} = \frac{\pi D^2 f_r}{4}$$

This equation is valid only after the drill reaches full diameter and excludes the initial approach of the drill into the work.

### 3.6.3: Milling

Milling is a machining operation in which a work part is fed past a rotating cylindrical tool with multiple cutting edges The axis of rotation of the cutting tool is perpendicular to the direction of feed. This orientation between the tool axis and the feed direction is one of the features that distinguishes milling from drilling.

The cutting speed is determined at the outside diameter of a milling cutter. This can be converted to spindle rotation speed using a formula that should now be familiar:

$$N = \frac{v}{\pi D}$$

The feed fin milling is usually given as a feed per cutter tooth; called the chip load, it represents the size of the chip formed by each cutting edge. This can be converted to feed rate by taking into account the spindle speed and the number of teeth on the cutter as follows:

$$f_r = N n_t f$$

where fr=feed rate, mm/min(in/min); N=spindle speed, rev/min; nt=number of teeth on the cutter; and f =chip load in mm/tooth (in/tooth).

Material removal rate in milling is determined using the product of the cross sectional area of the cut and the feed rate. Accordingly, if a slab-milling operation is

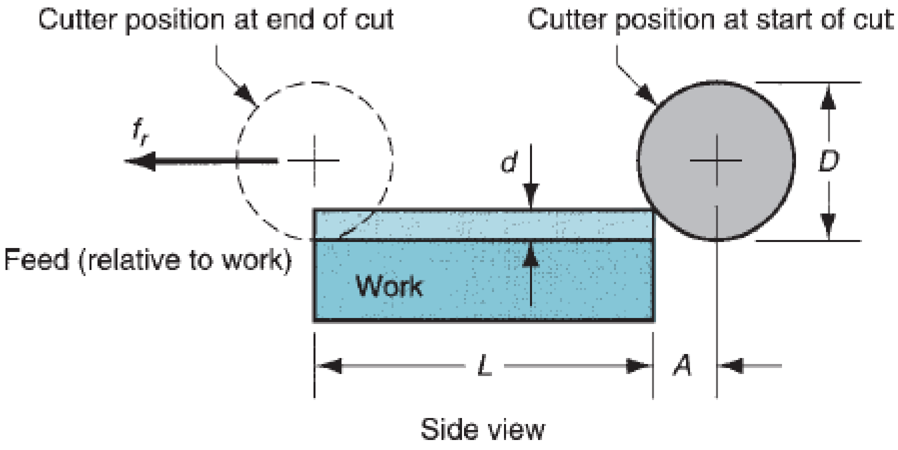


*Figure 3.6: Slab Milling operation*

cutting a workpiece with width w at a depth d, the material removal rate is

$$R_{MR} = wdf_r$$

This neglects the initial entry of the cutter before full engagement. The Eq can be applied to end milling, side milling, face milling, and other milling operations, making the proper adjustments in the computation of cross-sectional area of cut.

The time required to mill a workpiece of length L must account for the approach distance required to fully engage the cutter. First, consider the case of slab milling, Figure 3.6. To determine the time to perform a slab milling operation, the approach distance A to reach full cutter depth is given by

$$A = \sqrt{d(D-d)}$$

where d=depth of cut, mm(in); and D=diameter of the milling cutter, mm(in). The time Tm in which the cutter is engaged milling the workpiece is therefore

$$T_m = \frac{L+A}{f_r}$$

For face milling, let us consider the two possible cases pictured in Figure 3.7. The first case is when the cutter is centered over a rectangular workpiece as in Figure 3.7(a). The cutter feeds from right to left across the workpiece. In order for the cutter to reach the full width of the work, it must travel an approach distance given by the following:

$$A = 0.5\left(D - \sqrt{D^2 - w^2}\right)$$

where D=cutter diameter, mm(in) and w=width of the workpiece, mm(in). If D=w, then Eq. reduces to A=0.5D. And if D=w, then a slot is cut into the work and A=0.5D.

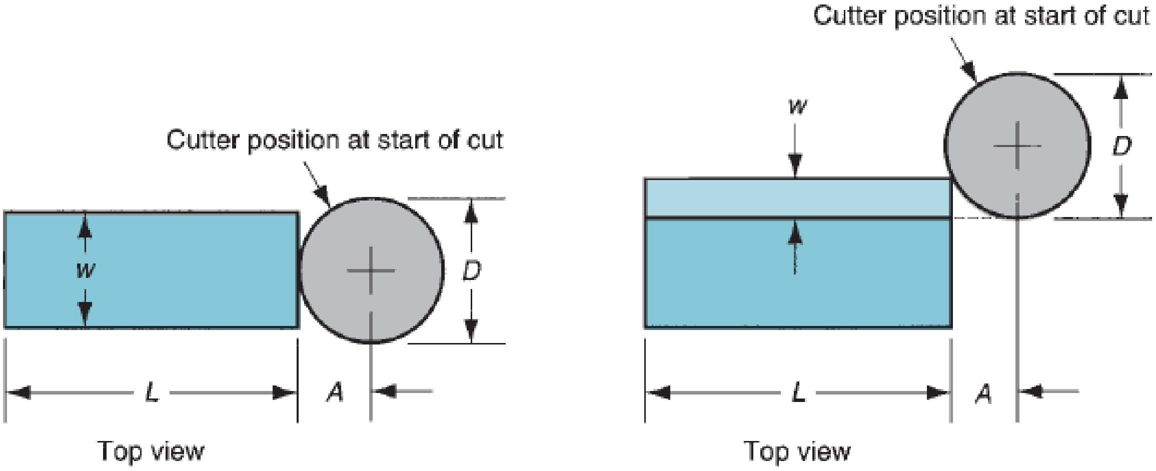


*Figure 3.7: Face milling showing approach and over travel distances for two cases: (a) when cutter is centered over the workpiece, and (b) when cutter is offset to one side over the work.*

The second case is when the cutter is offset to one side of the work, as in Figure 3.7(b). In this case, the approach distance is given by

$$A = \sqrt{w(D - w)}$$

where w ¼ width of the cut, mm (in). In either case, the machining time is given by

$$T_m = \frac{L + A}{f_r}$$

It should be emphasized in all of these milling scenarios that Tm represents the time the cutter teeth are engaged in the work, making chips. Approach and over travel distances are usually added at the beginning and end of each cut to allow access to the work for loading and unloading. Thus the actual duration of the cutter feed motion is likely to be greater than Tm.

## OUTCOMES:

- Student can able understand the various cutting tool materials and the tool signature of single point cutting tool, milling cutter and twist drill.
- Student gather the information about the importance of cutting fluids in machining operations.
- Students can solve the numerical on various aspects of machining of turning operation, milling operation and drilling operations.

## QUESTIONS:

1. List and explain the various cutting tool materials used in machining operations.
2. What is tool Signature. With the help of a neat sketch describe the single point cutting tool nomenclature.
3. Describe the nomenclature of milling cutter.
4. Obtain the tool signature of a twist drill with a neat sketch.
5. List the functions of Cutting Fluids.
6. What are the different cutting fluids and method of application of cutting fluid?

# MODULE-4

# MECHANICS OF MACHINING PROCESSES

**LESSON CONTENTS:**

Introduction, Chip formation, Orthogonal cutting, Merchants model for orthogonal cutting, Oblique cutting, Mechanics of turning process, Mechanics of drilling process, Mechanics of milling process, Numerical problems.

**OBJECTIVES:**

- To Study the mechanism of chip formation and differentiate between orthogonal and oblique cutting.
- To obtain equations to calculate various cutting forces using merchants tool diagram.
- To study the mechanics of turning, drilling and milling process.

## 4.0 Introduction:

Metal removal process is a machining process in which excess amount of material is removed in the form of chips in order to shape the material to the required dimension and size.

Machining is not just one process; it is a group of processes. The common feature is the use of a cutting tool to form a chip that is removed from the work-part. 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.

## 4.1 Theory of chip formation:

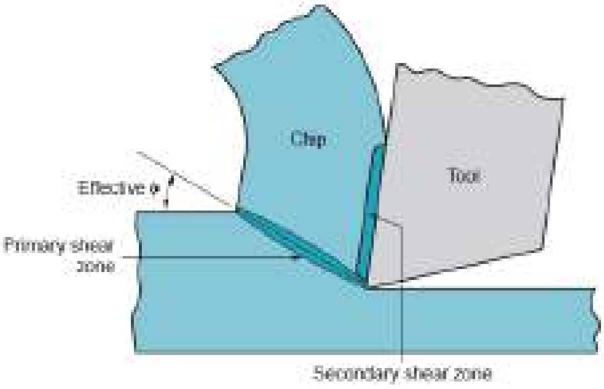


*Figure 4.1: Formation of chip in metal cutting*

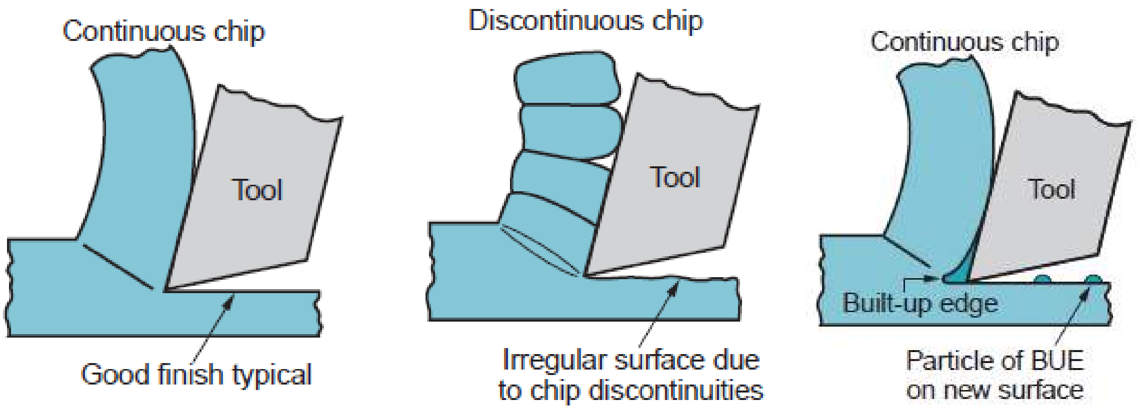
The above figure illustrates the basic geometry of a to dimensional chip formation. When the cutting tool is forced to move against the workpiece, the tool exerts a compressive force on the workpiece. The material of the workpiece is stressed beyond its yield point causing it to deform plastically and shear off. The plastic flow take place in the localised region called Shear plane. The sheared portion of the metal begins to flow along cutting tool face in the form a small pieces called chips.

### 4.2 Mechanics of chip formation:

As the cutting tool pressed against the workpiece, the tool removes the material in the form a chip at shear zone. Three different types of chips are formed during the process. They are:

1. Continuous chips
2. Discontinues Chips
3. Continuous with built up edges.

**Continuous chip:** When ductile work materials are cut at high speeds and relatively small feeds and depths, long continuous chips are formed. A good surface finish typically results when this chip type is formed. A sharp cutting edge on the tool and low tool–chip friction encourage the formation of continuous chips. Long, continuous chips (as in turning) can cause problems with regard to chip disposal and/or tangling about the tool. To solve these problems, turning tools are often equipped with chip breakers.



(a) (b) (c)

*Figure 4.2: (a) Continuous (b) Discontinuous (c) Continuous with built up edges*

**Discontinuous chip:** When relatively brittle materials (e.g., cast irons) are machined at low cutting speeds, the chips often form into separate segments (sometimes the segments are loosely attached). This tends to impart an irregular texture to the machined surface. High tool–chip friction and large feed and depth of cut promote the formation of this chip type.

**Continuous chip with built-up edge**: When machining ductile materials at low-to medium cutting speeds, friction between tool and chip tends to cause portions of the work material to adhere to the rake face of the tool near the cutting edge. This formation is called a built-up edge (BUE). The formation of a BUE is cyclical; it forms and grows, then becomes unstable and breaks off. Much of the detached BUE is carried away with the chip, sometimes taking portions of the tool rake face with it, which reduces the life of the cutting tool. Portions of the detached BUE that are not carried off with the chip become imbedded in the newly created work surface, causing the surface to become rough.

## 4.3 Orthogonal and Oblique cutting

### 4.3.1 Orthogonal Cutting Model:

In orthogonal cutting, the cutting edge inclination is zero and chip is expected to flow along the orthogonal plane. The cutting tool is passes such a way that the cutting edge is normal to the tool feed direction. In orthogonal cutting, the radial force is zero, and it involves only two component of force.

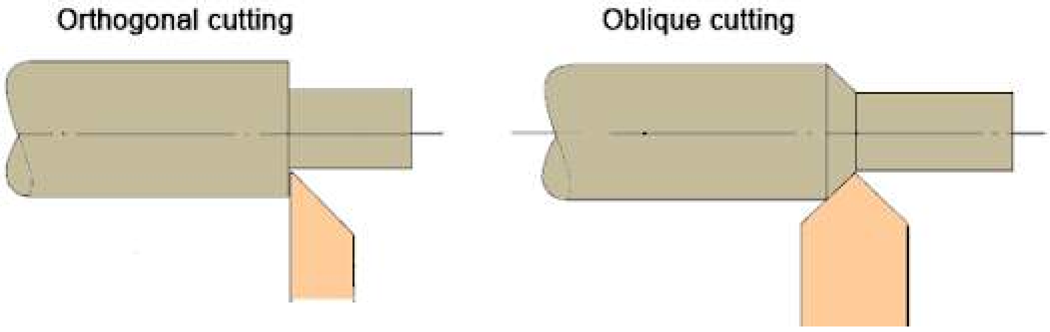


*Figure 4.3: Orthogonal and oblique cutting*

### 4.3.2 Oblique Cutting Model:

In oblique cutting, chip flow deviates from the orthogonal plane. Tool passes to workpiece at an acute angel to the tool feed motion. The analysis of cutting includes three mutually perpendicular component of force.

### 4.3.3 Comparison between Orthogonal and oblique cutting

| Sl. No | Orthogonal metal cutting | Oblique metal cutting |
|---|---|---|
| 1 | Cutting edge of the tool is perpendicular to the direction of tool travel. | The cutting edge is inclined at an angle less than 90° to the direction of tool travel. |
| 2 | The direction of chip flow is perpendicular to the cutting edge. | The chip flows on the tool face making an angle. |
| 3 | The chip coils in a tight flat spiral | The chip flows sideways in a long curl. |
| 4 | For same feed and depth of cut the force which shears the metal acts on a smaller area. So the life of the tool is less. | The cutting force acts on larger area and so tool life is more. |
| 5 | Produces sharp corners. | Produces a chamfer at the end of the cut |
| 6 | Smaller length of cutting edge is in contact with the work. | For the same depth of cut greater length of cutting edge is in contact with the work. |
| 7 | Generally parting off in lathe, broaching and slotting operations are done in this method. | This method of cutting is used in almost all machining operations. |

### 4.4 Earnest Merchant's Tool equation:

- Merchant circle diagram is used to analyse the forces acting in metal cutting.
- The analysis of three forces system, which balance each other for cutting to occur. Each system is a triangle of forces.

**Assumptions made in drawing Merchant's circle:**

- Shear surface is a plane extending upwards from the cutting edge.
- The tool is perfectly sharp and there is no contact along the clearance force.
- The cutting edge is a straight line extending perpendicular to the direction of motion and generates a plane surface as the work moves past it.

- The chip doesn’t flow to either side, that is chip width is constant.
- The depth of cut remains constant.
- Width of the too, is greater than that of the work.
- Work moves with uniform velocity relative tool tip.
- No built up edge is formed.

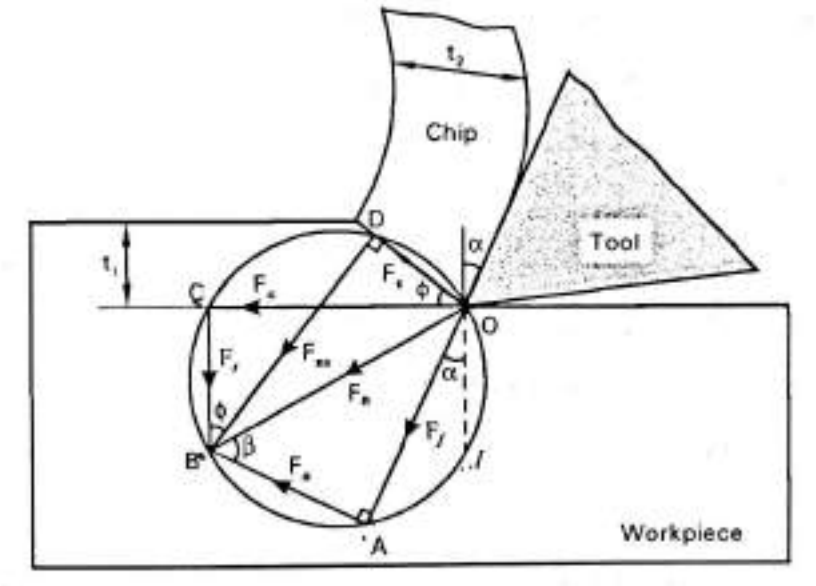


*Figure 4.4. Merchant Tool Diagram*

*Shear plane angel = φ*

*Tool rake angel = α*

*Friction angel = β*

*Fn = normal force*

*Fs= Shear force*

*Fns= Force normal to shear force*

*Fc= horizontal cutting force*

*Ft= Thrust force*

*$F_R$= Resultant Force*

From the figure,

$$\llcorner BOA = 90 - \beta$$

$$\llcorner COB = \beta - \alpha$$

$$\llcorner DOB = \varphi + \beta - \alpha$$

**To find Fc and Ft:**

From triangle BOC, $\cos(\beta-\alpha) = F_c/F_r$

$$\mathbf{F_c = F_r \cos(\beta-\alpha)}$$

From triangle BOC, $\sin(\beta-\alpha) = F_t/F_r$

$$\mathbf{F_t = F_r \sin(\beta-\alpha)}$$

**To find Fs and Fns:**

From triangle OBD, $\cos(\varphi+\beta-\alpha)=F_s/F_r$

$$\mathbf{F_s=F_r \cos(\varphi+\beta-\alpha)}$$

From triangle OBD, $\sin(\varphi+\beta-\alpha)= F_{ns}/F_r$

$$\mathbf{F_{ns}= F_r \sin(\varphi+\beta-\alpha)}$$

To find Ff and Fn, Ff and Fn are expressed in terms of Fc and Ft

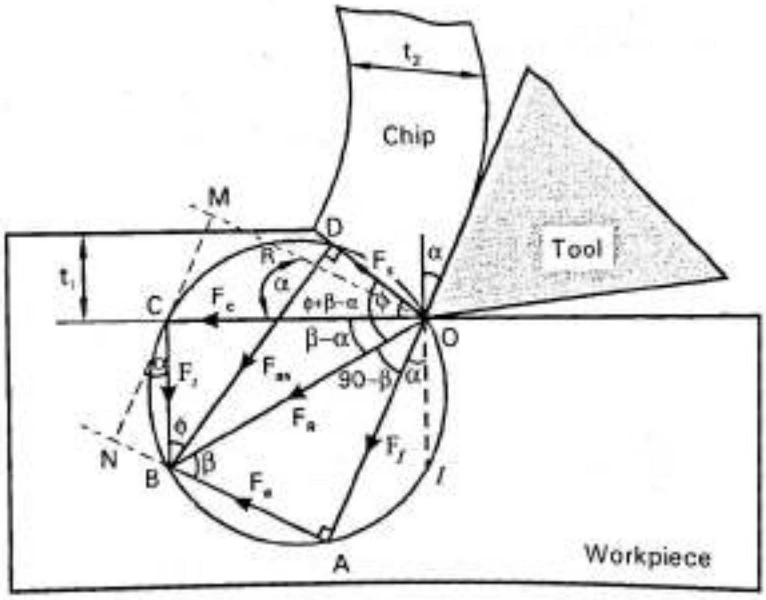


*Figure 4.5: Merchants Tool diagram*

From the diagram, $F_f = OA= MN$

$F_f= MN= MC+CN$

But from MC=? And CN=?

From triangle MCO, $\sin\alpha= MC/OC= MC/ F_c$

**To find CN=?**

From triangle CNB cosα= CN/CB= CN/Ft

**CN= Ft cosα**

**Therefore, Ff= Fc Sinα+Ft Cosα**

Fn = AB = OR

Fn =OR = OM – MR

OM =? And MR =?

From triangle OMC cosα= OM/ OC = OM/ Fc

**OM= Fc cosα**

MR= NB

From triangle CNB, Sinα= NB/BC= NB/Ft

**NB= Ft Sinα**

**Therefore, Fn= Fc Cosα+Ft Sinα**

**To calculate Co-efficient of friction,**

Ff= μFn

μ = Ff/ Fn

$$\mu = \frac{Fc\ Sin\alpha + Ft\ Cos\alpha}{Fc\ Cos\alpha - Ft\ Sin\alpha}$$

**OUTCOMES:**

- Students can understand the mechanism of chip formation and differentiate between orthogonal and oblique cutting.
- Students able to derive an expression for various cutting forces using merchants tool diagram.
- Students can solve numerical problems on various cutting forces through merchants diagram.

**QUESTIONS:**

1. Describe the mechanism of Chip formation.
2. Differentiate between orthogonal and oblique cutting
3. What are the different types of chips formed during machining orthogonal cutting?

4. Derive an expression to obtain cutting forces Fc, Ft, Fs, Fns, Ff, Fn and Coefficient of friction in orthogonal cutting using merchant tool diagram.

**FURTHER READING:**

**1. “Metal cutting principles”**, Milton C. Shaw, Oxford University Press, Second Edition,2005.
**2. “Manufacturing Technology”,** Vol 2, P N Rao, McGraw Hill Education, 3rd Edition

**3. “Workshop Technology, Vol-II”,** by Hazara Chowdary

# MODULE-5

## TOOL WEAR & TOOL LIFE

**LESSON CONTENTS:**

**Tool Wear, Tool Life:** Introduction, tool wear mechanism, tool life equations, effect of process parameters on tool life, machinability, Numerical problems.
**Economics of Machining Processes:** Introduction, choice of feed, choice of cutting speed, tool life for minimum cost and minimum production time, machining at maximum efficiency.

**OBJECTIVES:**

- To study the wear mechanism and types of wear
- Understand about the factors affecting tool life and Taylor's tool life equation.
- To study the machinability and machinability index.
- To know about the Economics of machining process and the factors affecting it.

### 5.1 Introduction:

We have seen that cutting tools are subjected to

(a) high localized stresses at the tip of the tool,
(b) high temperatures, especially along the rake face,
(c) sliding of the chip along the rake face, and
(d) sliding of the tool along the newly cut workpiece surface.

These conditions induce tool wear, which is a major consideration in all machining operations. Tool wear adversely affects tool life, the quality of the machined surface and its dimensional accuracy, and, consequently, the economics of cutting operations.

Wear is a gradual process. The rate of tool wear depends on tool and workpiece materials, tool geometry, process parameters such as speed, feed and depth of cut, cutting fluids, and the characteristics of the machine tool.

There are three possible modes by which a cutting tool can fail in machining:

1. **Fracture failure**. This mode of failure occurs when the cutting force at the tool point becomes excessive, causing it to fail suddenly by brittle fracture. (Mechanical Chipping)

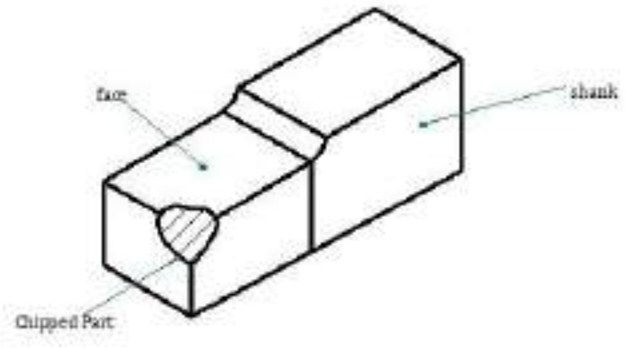


*Figure 5.1: Fracture failure*

2. **Temperature failure**. This failure occurs when the cutting temperature is too high for the tool material, causing the material at the tool point to soften, which leads to plastic deformation and loss of the sharp edge.

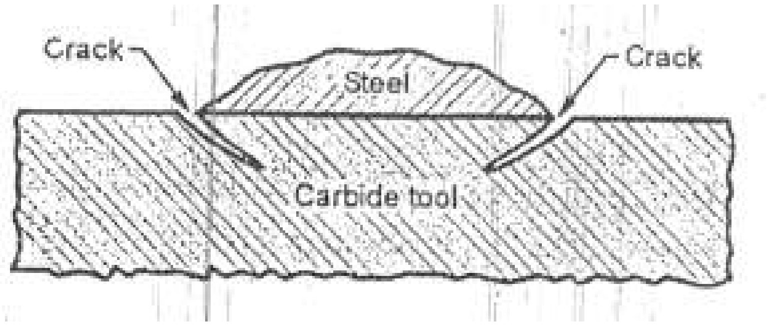


**Figure 5.2: Thermal Cracking**

3. **Gradual wear.** Gradual wearing of the cutting edge causes loss of tool shape, reduction in cutting efficiency, an acceleration of wearing as the tool becomes heavily worn, and finally tool failure in a manner similar to a temperature failure.
    a. **Wear on the flank of the tool:** Flank wear occurs on the relief (flank) face of the tool. It generally is attributed to rubbing of the tool along the machined surface, thereby causing adhesive or abrasive wear and high temperatures, which adversely affect tool-material properties.

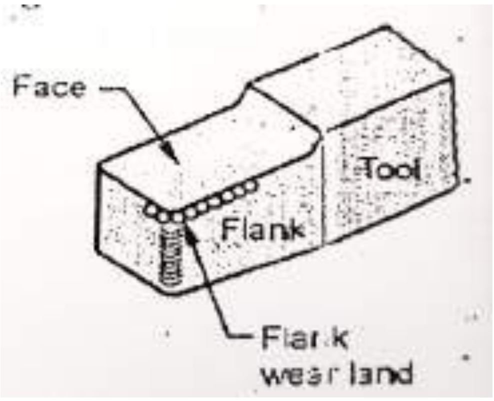


**Figure 5.3: Flank Wear**

b. **Crater Wear:** It consists of a cavity in the rake face of the tool that forms and grows from the action of the chip sliding against the surface. High stresses and temperatures characterize the tool–chip contact interface, contributing to the wearing action. The crater can be measured either by its depth or its area.

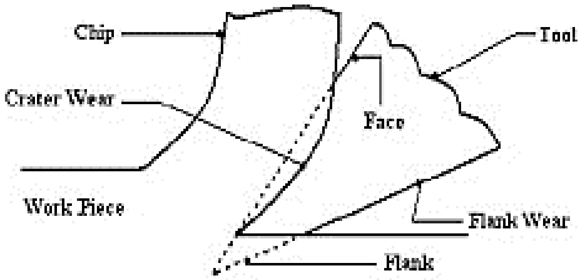


***Figure 5.4: Crater Wear***

## 5.2 Tool wear Mechanism:

The mechanisms that cause wear at the tool–chip and tool–work interfaces in machining can be summarized as follows:

- **Abrasion.** This is a mechanical wearing action caused by hard particles in the work material gouging and removing small portions of the tool. This abrasive action occurs in both flank wear and crater wear; it is a significant cause of flank wear.
- **Adhesion.** When two metals are forced into contact under high pressure and temperature, adhesion or welding occur between them. These conditions are present between the chip and the rake face of the tool. As the chip flows across the tool, small particles of the tool are broken away from the surface, resulting in attrition of the surface.
- **Diffusion:** This is a process in which an exchange of atoms takes place across a close contact boundary between two materials. In the case of tool wear, diffusion occurs at the tool–chip boundary, causing the tool surface to become depleted of the atoms responsible for its hardness. As this process continues, the tool surface becomes more susceptible to abrasion and adhesion. Diffusion is believed to be a principal mechanism of crater wear.
- **Chemical reactions:** The high temperatures and clean surfaces at the tool–chip interface in machining at high speeds can result in chemical reactions, in particular, oxidation, on the rake face of the tool. The oxidized layer, being softer than the parent tool material, is sheared away, exposing new material to sustain the reaction process.

- **Plastic deformation:** Another mechanism that contributes to tool wear is plastic deformation of the cutting edge. The cutting forces acting on the cutting edge at high temperature cause the edge to deform plastically, making it more vulnerable to abrasion of the tool surface. Plastic deformation contributes mainly to flank wear.

### 5.3 Tool Life:

As cutting proceeds, the various wear mechanisms result in increasing levels of wear on the cutting tool. The general relationship of tool wear versus cutting time is shown in Figure Although the relationship shown is for flank wear, a similar relationship occurs for crater wear. Three regions can usually be identified in the typical wear growth curve. The first is the break-in period, in which the sharp cutting edge wears rapidly at the beginning of its use. This first region occurs within the first few minutes of cutting. The break-in period is followed by wear that occurs at a fairly uniform rate. This is called the steady-state wear region. In our figure, this region is pictured as a linear function of time, although there are deviations from the straight line in actual machining. Finally, wear reaches a level at which the wear rate begins to accelerate. This marks the beginning of the failure region, in which cutting temperatures are higher, and the general efficiency of the machining process is reduced. If allowed to continue, the tool finally fails by temperature failure 5.5.

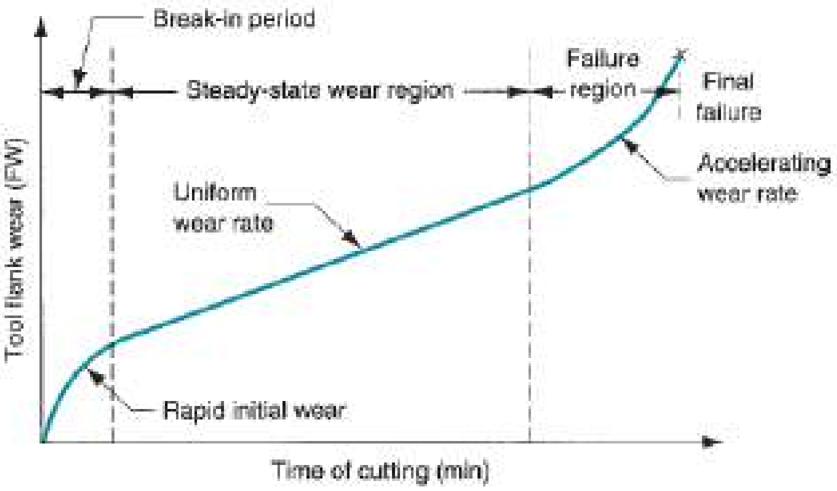


*Figure 5.5: relationship between Tool Life Vs cutting time*

**5.4 Tool Life and Taylors Equation:** Tool life is defined as the length of cutting time that the tool can be used between the two successive grinding.

In a classic study by F.W Taylor on the machining of steels conducted in the early 1890, the following approximate relationship for tool life, known as the Taylor tool life equation, was established:

$$V\,T^n = C$$

where V = cutting speed, m/min (ft/min); T = tool life in min; and n and C are parameters whose values depend on feed, depth of cut, work material, tooling (material in particular), and the tool life criterion used.

n= 0.1 for HSS

n= 0.20 to 0.25 for Carbide Tools

n= 0.4 to 0.55 for ceramic tools

### 5.5 Factors affecting Cutting tool life:

The life of tool is affected by many factors such as: cutting speed, depth of cut, chip thickness, tool geometry, material or the cutting fluid and rigidity of machine. Physical and chemical properties of work material influence tool life by affecting form stability and rate of wear of tools. The nose radius tends to affect tool life.

**1. Cutting speed:** Cutting speed has the greatest influence on tool life. As the cutting speed increases the temperature also rises. The heat is more concentrated on the tool than on the work and the hardness of the cutting tool changes so the relative increase in the hardness of the work accelerates the abrasive action. The criterion of the wear is dependent on the cutting speed because the predominant wear may be wear for flank or crater if cutting speed is increased.

**2. Feed and depth of cut:** The tool life is influenced by the feed rate also. With a fine feed the area of chip passing over the tool face is greater than that of coarse feed for a given volume of metal removal.

**3. Tool Geometry:** The tool life is also affected by tool geometry. A tool with large rake angle becomes weak as a large rake reduces the tool cross-section and the amount of metal to absorb the heat.

**4. Tool material:** Physical and chemical properties of work material influence tool life by affecting form stability and rate of wear of tool.

4. **Cutting fluid:** It reduces the coefficient of friction at the chip tool interface and increases tool life.

## 5.6 Machinability:

The machinability of a material is usually defined in terms of four factors:
l. Surface finish and surface integrity of the machined part.
2. Tool life.
3. Force and power required.
4. The level of difficulty in chip control.

***However, machinability is defined as the ease with which the work material can be machined.***

### 5.6.1 Machinability Index:

**5.6.1 Machinability Index:** Machinability index is used for comparing machinability of various materials. Machinability index of free cutting steel serves as a reference to which other machinability indexes are compared.

Machinability index of free cutting steel is taken as 100 and for calculating machinability of any other material, following relation is used:

$$M.I = \frac{Cutting\ Speed\ of\ metal\ for\ 20 min.tool\ life}{Cutting\ speed\ of\ standard\ free\ cutting\ steel\ for 20\ \min of\ tool\ life}\ X\ 100$$

Machinability index as compared to free Cutting steel for other materials are:
Stainless Steel=25%
Low carbon Steel = 55-65%
Copper = 70%
Brass = 180%
Aluminium = 300-1500%
Magnesium = 500-2000%

## 5.7 Economics of Machining process:

Machining or metal cutting is one important aspect of the production system. Ultimate objective of machining is to give intended shape, size and finish by gradually removing material from workpiece.

The primary goal of industries is to manufacture the product at a faster rate but at minimal cost and that too without sacrificing product quality. As long as conventional machining is utilized, in order to fulfil first requirement (faster production rate), the cutting speed and feed rate should have to be increased. However, this may lead to reduced cutting tool life due to faster wear rate and higher heat generation. Hence, cutting tool is required to change frequently, which will ultimately impose a loss for the industry as a result of idle time for changing tools. Cost of tool is also not negligible. Therefore, abrupt increase of cutting speed and feed rate is not a feasible solution; rather, an optimization is necessary.

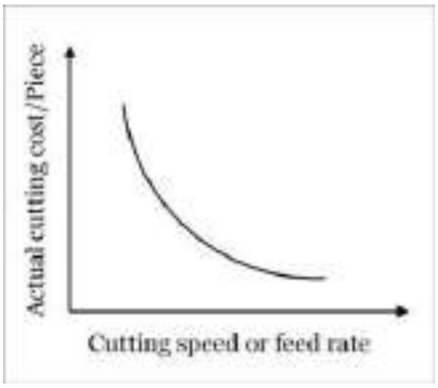
## 5.8 Overall machining time and cost

Overall or total machining time ($T_m$) is the summation of three different time elements closely associated with the machining or metal cutting process. These three elements include actual cutting time ($T_c$), total tool changing time ($T_{ct}$) and other handling or idle time ($T_i$). Beside these three time elements, cost of cutting tool is also required to incorporate for any optimization. All these time or cost elements, except handling time, are affected by the variation of cutting speed and feed rate as explained below. Mathematically, total time for machining ($T_m$) can be expressed as:

$$Tm = Tc + Tct + Ti$$

### 5.8.1 Actual cutting time (Tc):

Cutting time is the time taken during actual material removal action, i.e., from the beginning of chip production to the end for uninterrupted machining. In case of any planned or unplanned stoppage in cutting, the idle duration will not come under this time element. Therefore, increase in cutting speed and feed rate will result in reduction of actual cutting time as material removal rate (MRR) will increase. Hence, cost associated with cutting time will decrease if speed or feed is increased. The adjacent diagram depicts how cost associated with the actual cutting time varies with speed or feed employed during machining.



If, Lc is the total length of cut (mm), N is the spindle speed (rpm) and s is the feed rate (mm/rev), then estimated uninterrupted cutting or machining time can be expressed as:

$$\text{Actual cutting Time } \boldsymbol{Tc} = \frac{\boldsymbol{Lc}}{\boldsymbol{N.\ S}}$$

In most of the cases, where either workpiece or cutting tool is rotating, the spindle speed (N) and cutting velocity (Vc) are interchangeable. However, cutting velocity also depends on the diameter of the job/cutter (D). Cutting velocity can be expressed, in terms of speed and diameter of job or cutter (whichever is rotating), as follows. For better understanding of this conversion, you may read: Cutting speed and cutting velocity in machining.

$$\textbf{Cutting velocity } Vc = \frac{\pi DN}{1000}$$

In case of turning or milling, actual cutting time can be expressed as:

$$T_c = \frac{L}{N.s}$$

$$T_c = \frac{L}{\left(\frac{1000V_C}{\pi D}\right)s}$$

$$T_c = \frac{\pi DL}{1000V_C}$$

**Mathematical Expression for Tool Changing time:**

$$T_{ct} = \left(\frac{T_c}{TL}\right) \times TCT$$

**Tooling Cost:**

$$\text{Tooling cost} = \left(\frac{T_c}{TL}\right) \times K_2$$

### 5.8.2 Overall Machining Cost:

Since every time elements pertinent to machining contributes towards machining cost, so some factors are required to convert time to cost. On the basis of these factors, time elements can be converted to cost elements and estimation of machining economy becomes easier. Such factors include:

K1 = Cost-time conversion factor for machining

K2 = Cost-time conversion factor for tool sharpening or price of new tool.

Therefore, overall machining cost per piece (Cp)

= (Actual cutting cost/piece) + (Tool changing cost/piece) + (Handling cost/piece) + (Tooling cost/piece)

= K1 {(Actual cutting time/piece) + (Tool changing time/piece) + (Handling time/piece)} + K2 {(Tooling cost/piece)}

$$= K_1 \{T_c + T_{ct} + T_l\} + \left(\frac{T_c}{TL}\right) \times K_2$$

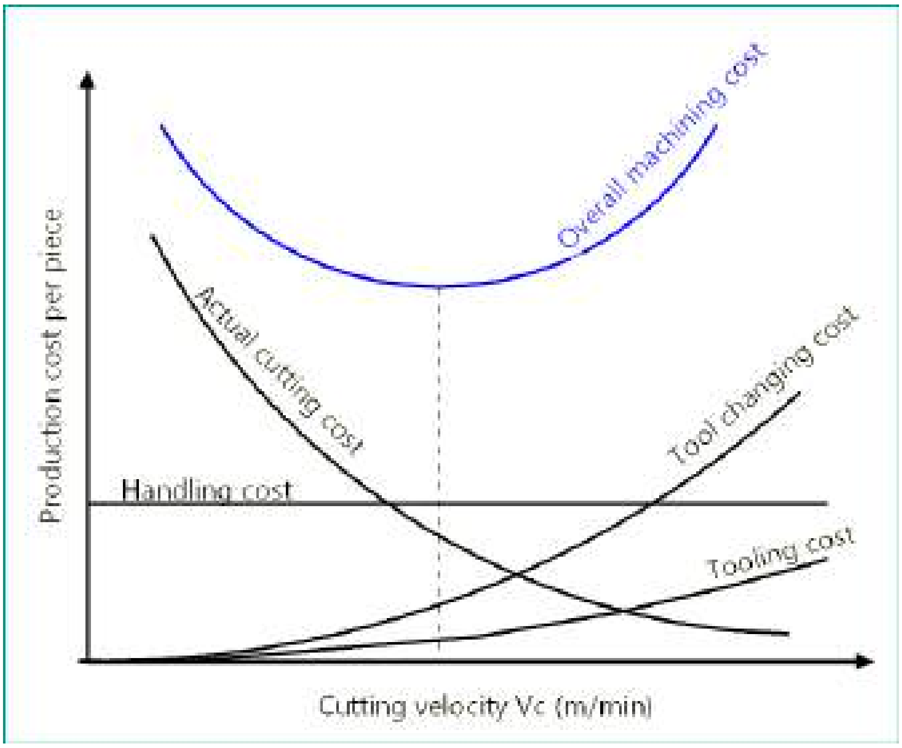


**Figure: Effect of variations in cutting speed on various cost factors**

### 5.8.3 Finding out economic condition and Gilbert's Model

Undoubtedly the final thing is to find out the optimum condition for either maximizing profit or minimizing time requirement. For paper based optimization, only cutting velocity or speed is considered in order to keep the analytical process less complicated. Moreover, cutting velocity is the main parameter that affects machining performance. A number of constraints can be handled effectively using computer programming based optimization techniques.

Now there exist a number of objectives for optimization, among which Gilbert's Model (1952) for Maximum Production Rate and Minimum Production Time are more prominent. These models are based on the Taylor's Tool Life equation, which consider only cutting velocity to determine tool life. Various economic models for optimizing machining process parameters for different objectives are provided below.

## OUTCOMES:

- Student will able to describe the wear mechanism and different types of wear.
- Student will understand about the factors affecting tool life and Taylor's tool life equation.
- Student will know the importance of machinability and factors affecting to it.
- Student can enumerate the importance of tool life and solve simple numerical problems.
- Student can know the importance of economic point of machining and parameters affecting the machining cost.

## QUESTIONS:

1. What are the factors affecting the tool life?
2. What is machinability? Define machinability index.
3. Write a short note on economics of metal machining
4. State a relationship of cutting speed and tool life for minimum cost and maximum production.
5. What do you mean by crater wear and flank wear?
6. Describe briefly tool wear mechanism.
7. A tool life of 80min is obtained at a speed of 30mpm and 8 minute at 60mpm. Determine the following:
    a. Tool life equation.
    b. Cutting speed for 4-minute tool life.
8. The following equation for tool life is given for turning operation $VT^{0.13}f^{0.77}d^{0.37}=C$. A 60 min tool life as obtained while cutting at V=30 m/min, f=0.30mm/rev and depth of cut d=2.5mm. calculate the change in tool life if the cutting speed, feed and depth of cut are increased by 20% of individually and also taken together. What will be their effect on the tool life?
9. A certain cutting tool during rough turning gave a tool life of 1 hour at a cutting speed of 30m/min. what will be the life of the tool when it is used at the same cutting speed for finish turning. Take n=0.125 for rough turning and n=0.1 for finish cut.
10. The tool life for a HSS tool is expressed by the relation $VT^{1/7}=C1$ and for tungsten carbide $VT^{1/5}=C2$. If the tool life for a cutting speed of 24m/min is 128min, compare the tool life of the two tools at a speed of 30 m/min

## FURTHER READING:

**1. "Metal cutting principles"**, Milton C. Shaw, Oxford University Press, Second Edition,2005.
**2. "Manufacturing Technology",** Vol 2, P N Rao, McGraw Hill Education, 3rd Edition

**3. "Workshop Technology, Vol-II",** by Hazara Chowdary