Theory of Metal Cutting

1.1 INTRODUCTION

Production or manufacturing of any object is a value addition process by which raw material of low utility and value due to its irregular size, shape and finish is converted into a high utility and valued product with definite size, shape and finish imparting some desired function ability.

Machining is an essential process of semi-finishing and often finishing by which jobs of desired shape and dimensions are produced by removing extra material from the preformed blanks in the form of chips with the help of cutting tools moved past the work surfaces in machine tools. The chips are separated from the workpiece by means of a cutting tool that possesses a very high hardness compared with that of the workpiece, as well as certain geometrical characteristics that depend upon the conditions of the cutting operation. Among all of the manufacturing methods, metal cutting, commonly called machining; is perhaps the most important. Forgings and castings are subjected to subsequent machining operations to acquire the precise dimensions and surface finish required. Also, products can sometimes be manufactured by machining stock materials like bars, plates, or structural sections.

1.2 BASIC FUNCTIONAL PRINCIPLES OF MACHINE TOOL OPERATIONS

Machine Tools produce desired geometrical surfaces on solid bodies (preformed blanks) and for that they are basically comprised of;

- Devices for firmly holding the tool and work
- Drives for providing power and motions to the tool and work
- Kinematic system to transmit motion and power from the sources to the tool-work
- Automation and control systems
- Structural body to support and accommodate those systems with sufficient strength and rigidity.

For material removal by machining, the work and the tool need relative movements and those motions and required power are derived from the power source(s) and transmitted through the kinematic system(s) comprised of a number and type of mechanisms.

1.3 INTRODUCTION TO METAL MACHINING

Three main areas of metal machining are Turning, Milling and Grinding.

In each topic, the basic essential concepts of the types of machines and their structures, the cutting tools and the operations will be discussed.

Turning

- Applications : **Usually to form round or taper cylindrical profiles**.
- Work-pieces : **Rotating, usually cylindrical shape**
- Cutters : **Single-point cutter, move in linear direction**.
- Machines : **Turning machine, or lathe**

Milling

- Applications : for Flat or formed profiles.
- Work pieces : Move linearly
- Cutters : **Rotating about its own axis, Multiple cutting edges**.
- Machines : Milling machines (Horizontal, Vertical and universal Milling Machines).

Grinding

- Applications : For better surface finishing, better accuracy or close tolerances.
- Work pieces : For hardened steels (materials), Very little metal removal
- Cutters : Grinding wheels
- Machines :
  - Surface grinding machines (for flat or formed profiles)
  - Cylindrical Grinding machines
  - Universal grinding machines

1.4 BROAD CLASSIFICATION OF MACHINE TOOLS

Number of types of machine tools gradually increased till mid 20th century and after that started decreasing based on Group Technology.

However, machine tools are broadly classified as follows:

- **According to direction of major axis :**
  - Horizontal center lathe, horizontal boring machine etc.
  - Vertical – vertical lathe, vertical axis milling machine etc.
  - Inclined – special (e.g. for transfer machines).
- **According to purpose of use :**

- General purpose – e.g. center lathes, milling machines, drilling machines etc.
- Single purpose – e.g. facing lathe, roll turning lathe etc.
- Special purpose – for mass production.

➢ **According to degree of automation**

- Non-automatic – e.g. center lathes, drilling machines etc.
- Semi-automatic – capstan lathe, turret lathe, hobbing machine etc.
- Automatic – e.g., single spindle automatic lathe, swiss type automatic lathe, CNC milling machine etc.

➢ **According to size :**

➢ Heavy duty – e.g., heavy duty lathes (e.g. ≥ 55 kW), boring mills, planning machine, horizontal boring machine etc.

- Medium duty – e.g., lathes – 3.7 ~ 11 kW, column drilling machines, milling machines etc.
- Small duty – e.g., table top lathes, drilling machines, milling machines.
- Micro duty – e.g., micro-drilling machine etc.

➢ **According to precision :**

- Ordinary – e.g., automatic lathes
- High precision – e.g., Swiss type automatic lathes

➢ **According to number of spindles :**

- Single spindle – center lathes, capstan lathes, milling machines etc.
- Multi-spindle – multispindle (2 to 8) lathes, gang drilling machines etc.

➢ **According to blank type :**

- Bar type (lathes)
- Chucking type (lathes)
- Housing type

➢ **According to type of automation :**

- Fixed automation – e.g., single spindle and multispindle lathes
- Flexible automation – e.g., CNC milling machine

➢ **According to configuration :**

- Stand alone type – most of the conventional machine tools.
- Machining system (more versatile) – e.g., transfer machine, machining center, FMS etc.

1.5 COMPARISONS OF VARIOUS METAL MACHINING PROCESSES

	Work pieces	Cutters	Applications	Remarks
Turning	Rotating	Move linearly, single point	Round or cylindrical shapes	Turning machine
Milling	Linear	Rotating Multi point	Flat or formed profiles	Milling machine
Shaping	Linear	Linear Single Point	Flat	Shaper
Planer	Move Linear	Single point	Flat	Planer
Surface Grinding	Linear	Rotating Grinding Wheel	Better finishing Better accuracy For hardened surfaces	Small metal removal
Cylindrical Grinding	Rotating	Rotating Grinding Wheel	Better finishing Better accuracy For hardened surfaces	Small metal removal
Drilling	Stationery	Rotating and Move linear along the axis	Cylindrical holes	Drilling machine
Reaming	Stationery	Rotating and Move linear along the axis	Better Finishing	Drilling machine
Boring	Stationery or rotating	Single point rotating or linear motion	Better straightness or correcting the ovality	Jig borer, Milling machine, Lathe

1.6 OBJECTIVES OF METAL MACHINING

➢ **Quality:**

- Better precision, smaller tolerances
- Better surface finishing

➢ **Productivity**:

- Faster machining time
- Shorter delivery

➢ **Cost :**

- Lower cost, cheaper price

1.7 CUTTING PARAMETERS

Cutting Speed: Cutting speed is the distance traveled by the work surface in unit time with reference to the cutting edge of the tool. The cutting speed, V is simply referred to as speed and usually expressed in m/min.

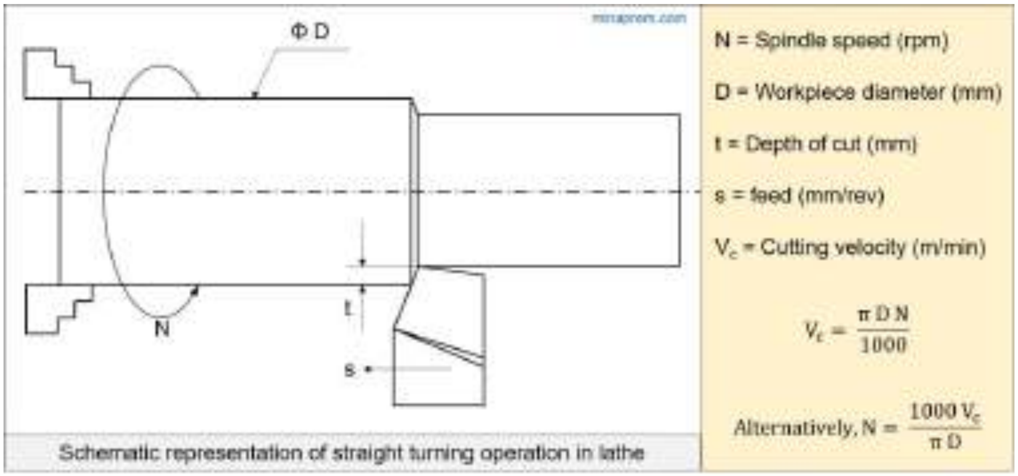


Fig.1 Cutting speed

Feed: The feed is the distance advanced by the tool into or along the workpiece each time the tool point passes a certain position in its travel over the surface. In case of turning, Feed rate is defined as tool's distance travelled during one spindle revolution.

Feed *f* is usually expressed in mm/rev. Sometimes it is also expressed in mm/min and is called feed rate.

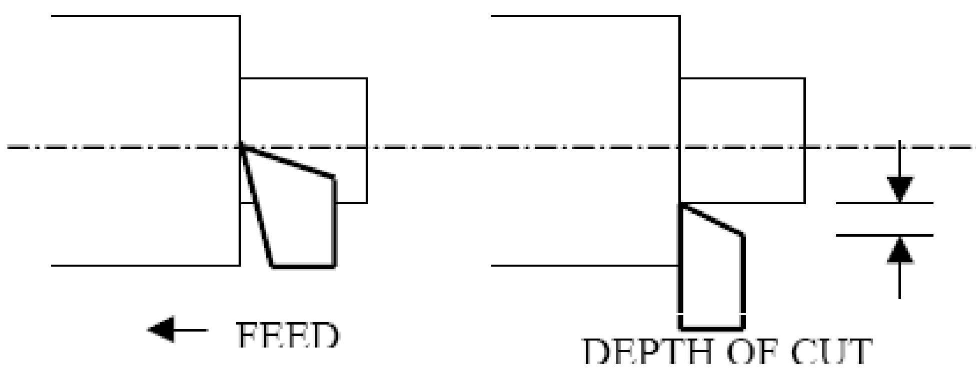


Fig. 2. Feed and depth of cut

Depth of cut: It is the distance through which the cutting tool is plunged into the workpiece surface. Thus it is the distance measured perpendicularly between the machined surface and the unmachined (uncut) surface or the previously machined surface of the workpiece. The depth of cut *d* is expressed in mm.

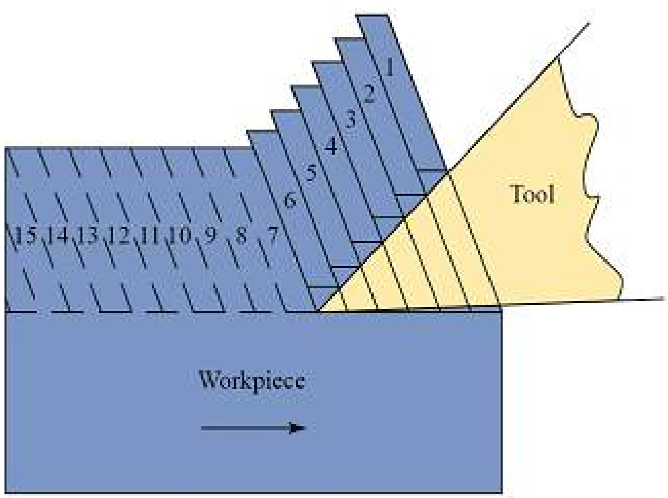
1.8 SELECTION OF CUTTING SPEED AND FEED

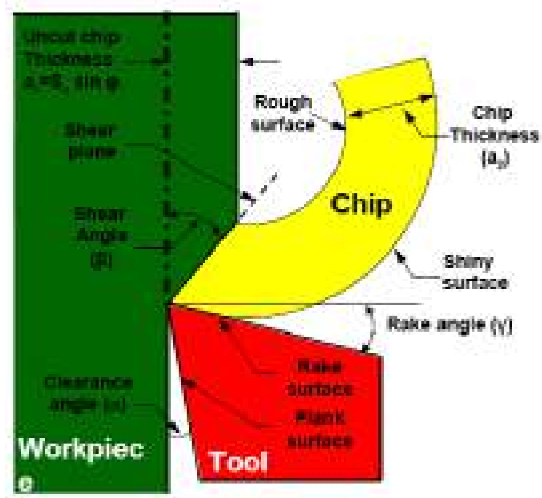
The selection of cutting speed and feed is based on the following parameters:

- Workpiece material
- Tool Material
- Tool geometry and dimensions
- Size of chip cross-section
- Types of finish desired
- Rigidity of the machine
- Types of coolant used

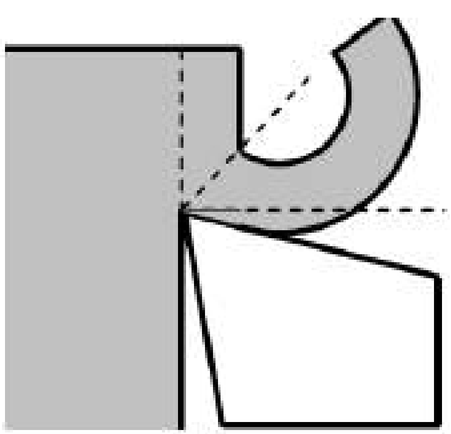
1.9 CHIP FORMATION

Regardless of the tool being used or the metal being cut, the chip forming process occurs by a mechanism called plastic deformation. This deformation can be visualized as shearing. That is when a metal is subjected to a load exceeding its elastic limit.

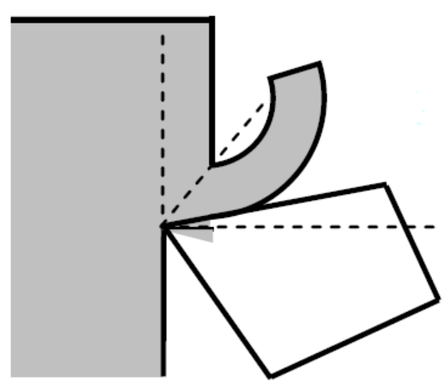




This action, shown in Figure is similar to the action that takes place when a deck of cards is given a push and sliding or shearing occurs between the individual cards.



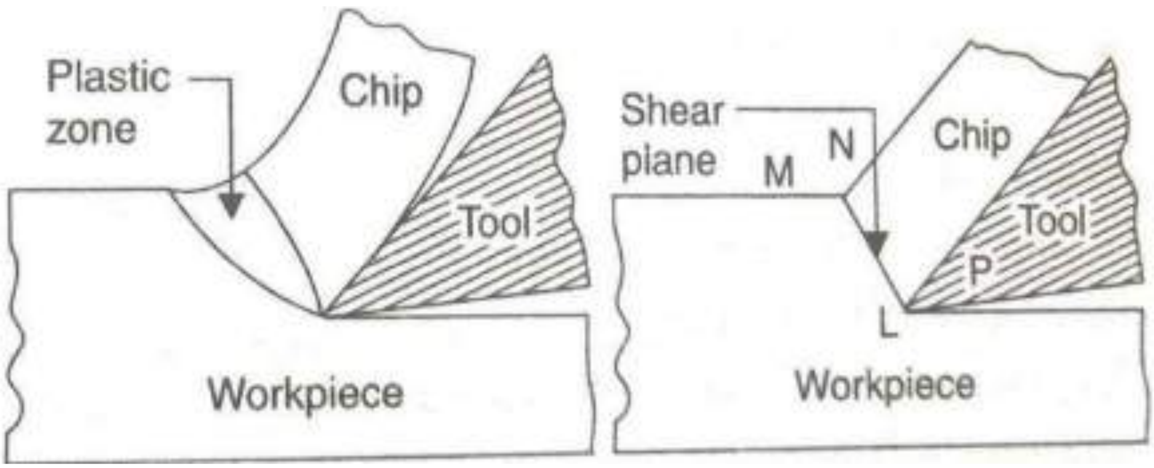
Positive rake



Negative rake

1.10 MECHANICS OF CHIP FORMATION

- During metal cutting, the metal is severely compressed in the area in front of the cutting tool.
- This causes high temperature shear and plastic flow if the metal is ductile.
- When the stress in the workpiece just ahead of the cutting tool reaches a value exceeding the ultimate strength of the metal, particles will shear to form a chip element, which moves up along the face of the work.
- The outward or shearing movement of each successive element is arrested by work hardening and the movement transferred to the next element.
- The process is repetitive and a continuous chip is formed.
- The plane along which the element shears, is called shear plane.



A chip has two surfaces:

1. One that is in contact with the tool face (rake face). This surface is shiny, or burnished.
2. The other from the original surface of the work piece.

This surface does not come into contact with any solid body. It has a jagged, rough appearance, which is caused by the shearing mechanism.

1.11 TYPES OF CHIP

Every Machining operation involves the formation of chips. The nature of which differs from operation to operation, properties of work piece material and the cutting condition. Chips are formed due to cutting tool, which is harder and more wearer-resistant than the work piece and the force and power to overcome the resistance of work material. The chip is formed by the deformation of the metal lying ahead of the cutting edge by a process of shear.

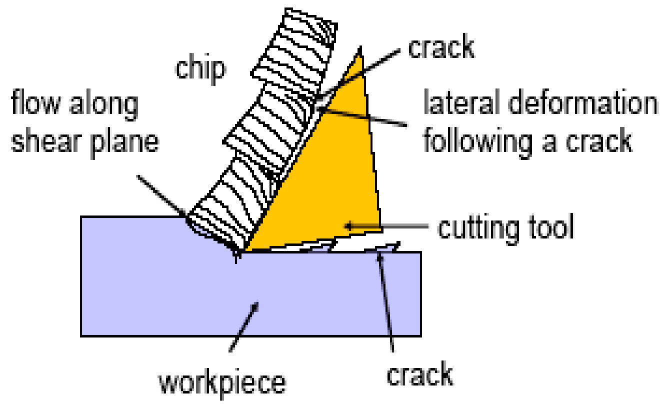
1. Discontinuous or segmented chips
2. Continuous chips

3. Continuous Chips with a built-up edge (BUE)
4. Serrated Chips

1. DISCONTINUOUS CHIP

Discontinuous or segmented chips are produced when brittle metal such as cast iron and hard bronze are cut or when some ductile metals are cut under poor cutting conditions.

They are formed when the amount of deformation to which chips undergo is limited by repeated fracturing. Discontinuous chips consist of segments that may be firmly or loosely attached to each other.



Brittle failure takes place along the shear plane before any tangible plastic flow occurs. Discontinuous chips will form in brittle materials at low rake angles (large depths of cut).

Discontinuous Chips usually form under the following conditions:

- Brittle work piece materials
- Work piece materials that contain hard inclusions and impurities, or have structures such as the graphite flakes in gray cast iron.
- Very low cutting speeds.
- Large depths of cut.
- High feed rate
- Low rake angles.
- Lack of an effective cutting fluid.
- Low stiffness of the machine tool.

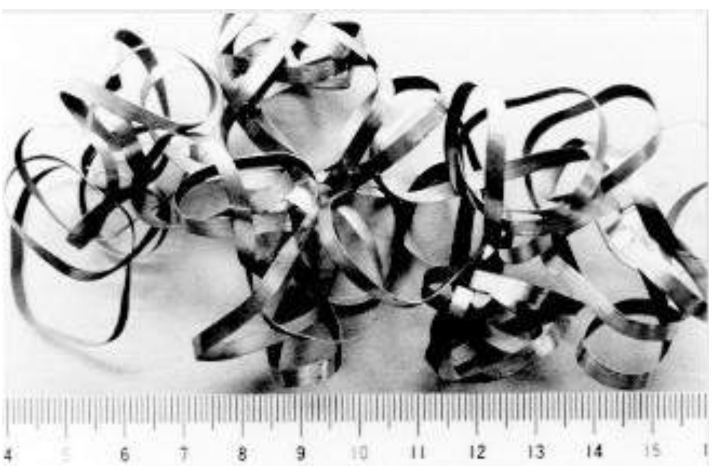
Because of the discontinuous nature of chip formation, forces continually vary during cutting. Hence, the stiffness or rigidity of the cutting-tool holder, the Work holding devices, and the machine tool are important in cutting with both DC and serrated-chip formation.

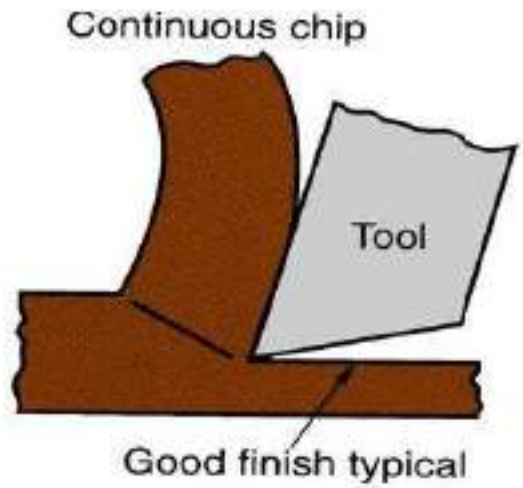
2. CONTINUOUS CHIPS

Continuous chips are usually formed with ductile materials such as mild steel, copper etc, at high rake angles and/or high cutting speeds. A good surface finish is generally produced. continuous chips are not always desirable, particularly in automated machine tools, it tends to get tangled around the tool and operation has to be stopped to clear away the chips.

Continuous chips usually form under the following conditions:

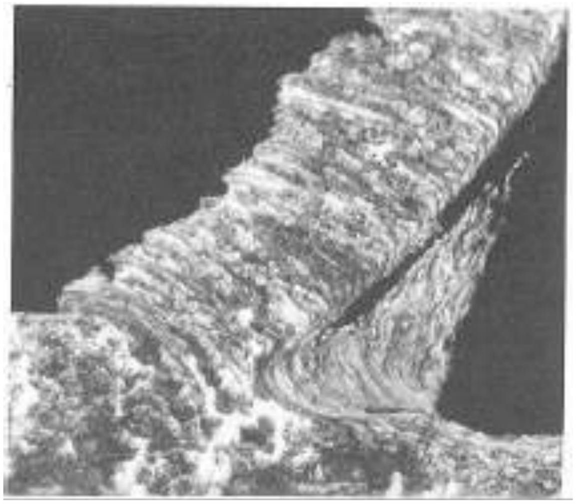
- Small chip thickness (fine feed)
- Small cutting edge
- Large rake angle
- High cutting speed
- Ductile work materials
- Less friction between chip tool interface through efficient lubrication

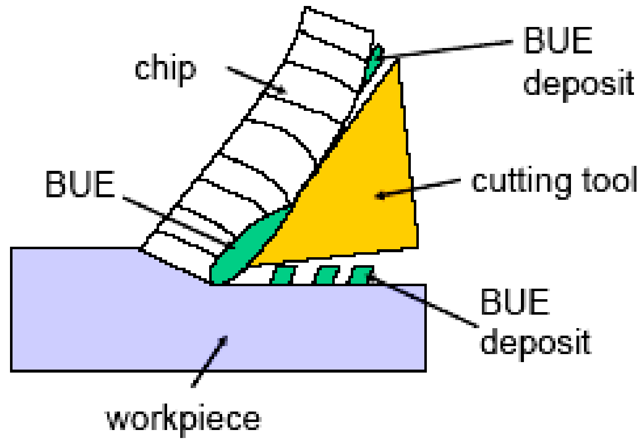




3. CONTINUOUS CHIPS WITH BUILT UP EDGE (BUE)

This type of chip is very similar to the continuous chip. With the difference that it has a built up edge adjacent to tool face and also it is not so smooth. It is obtained by machining on ductile material. Due to **high local temperature, extreme pressure in the cutting and high friction in the tool chip interference,** it may cause the work material to adhere or weld to the cutting edge of the tool on the rake surface. Successive layers of work material are then added to form the built up edge. When this edge becomes larger and unstable, it breaks up and it carries some part of the tool along with the chip while the remaining is left over the surface being machined, which contributes to the roughness of the surface. The built up edge changes its size during the cutting operation. It first increases, then decreases, and then again increases etc.





The tendency for a BUE to form is reduced by any of the following practices:

- Increase the cutting speeds
- Decreasing depth of cut
- Increasing the rake angle
- Using a sharp tool
- Using an effective cutting fluid
- Use a cutting tool that has lower chemical affinity for the work piece material.

➢ **Effects of BUE formation**

Formation of BUE causes several harmful effects, such as:

❖ It unfavorably changes the rake angle at the tool tip causing increase in cutting forces and power consumption

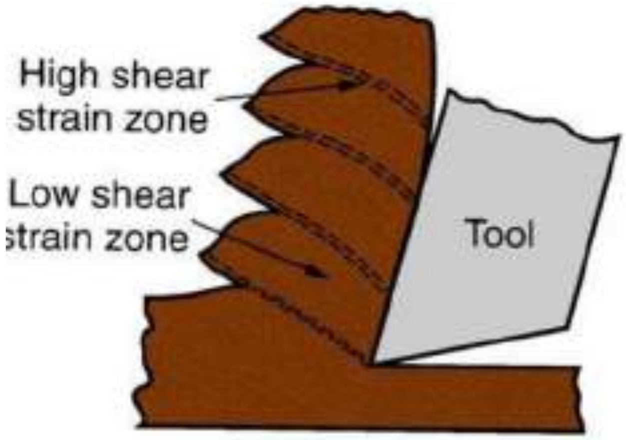
❖ Repeated formation and dislodgement of the BUE causes fluctuation in cutting forces and thus induces vibration which is harmful for the tool, job and the machine tool.

❖ Surface finish gets deteriorated

❖ May reduce tool life by accelerating tool-wear at its rake surface by adhesion and flaking

4. SERRATED CHIPS

Serrated chips: semi-continuous chips with alternating zones of high shear strain then low shear strain. Metals with low thermal conductivity and strength that decreases sharply with temperature, such as titanium, exhibit this behavior.



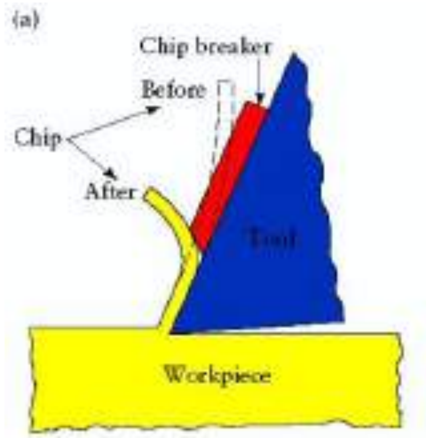
The Semi-continuous chips have a saw-tooth-like appearance and associated with difficult-to machine metals at high cutting speeds.

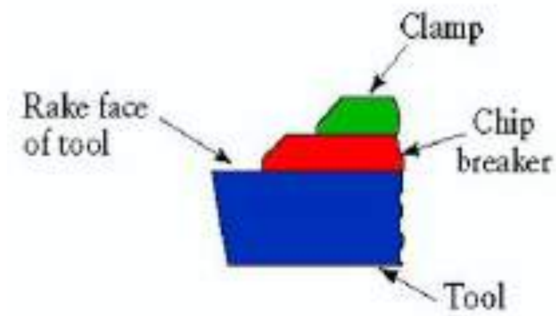
Summary of conditions favorable for the different types of chips

Parameters	Continuous chip	Continuous chip with BUE	Discontinuous chips
Material	Ductile material such as Mild steel, aluminium, copper	Ductile Material	Brittle material such as cast iron
Cutting Speed	High	Low	Low
Feed rate	Less	High	high
Depth of cut	Less	High	High
Rake angle	Positive / More rake angle	Positive / More rake angle	Less Rake angle / Negative rake angle
Friction at the chip tool interface	Less	More	More
Rigidity of Machine tool	More rigid	More rigid	Less rigid
Cutting fluid	Cutting fluid should act as lubricant to reduce the friction	Lack of cutting fluid	Lack of effective cutting fluid / sometimes cutting fluid at high velocity may act as chip breaker

1.11 CHIP BREAKERS

A continuous chip flows away from the work at high speed. If this chip is allowed to continue, it may wrap around the tool post, the workpiece, the chuck, and perhaps around the operator's arm. Not only is the operator in danger of receiving a nasty laceration, but if the chip winds around the workpiece and the machine, he must spend considerable time in removing it. A loss of production will be encountered. Therefore it is imperative that this chip be controlled and broken in some manner. Hence chip breakers are used to break up the long continuous chip in small pieces.



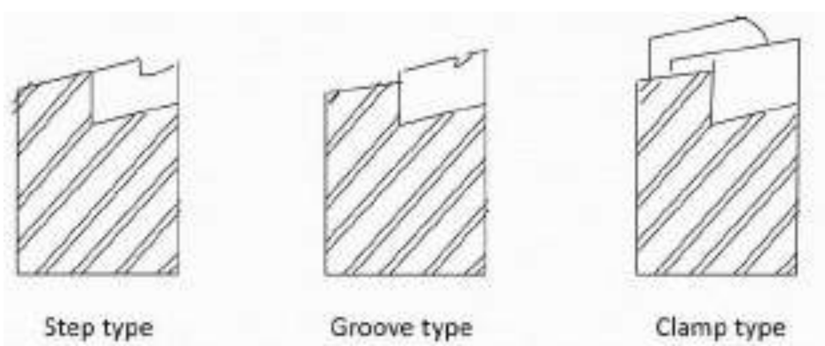


Chip breakers may be of the following types:

Step type: A step is ground on the face of the tool along the cutting edge.

Groove type: A small grove is ground behind the cutting edge.

Clamp type: A thin carbide plate or clamp is brazed or screwed on the face of the tool

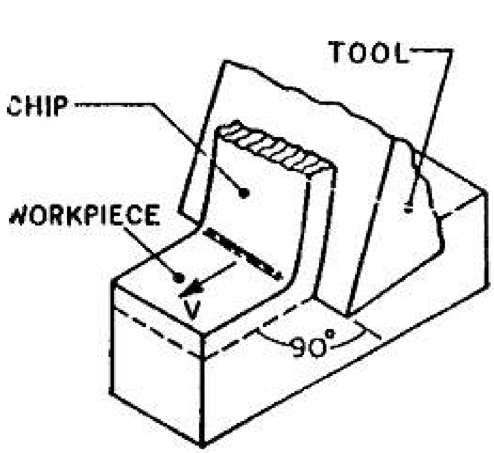
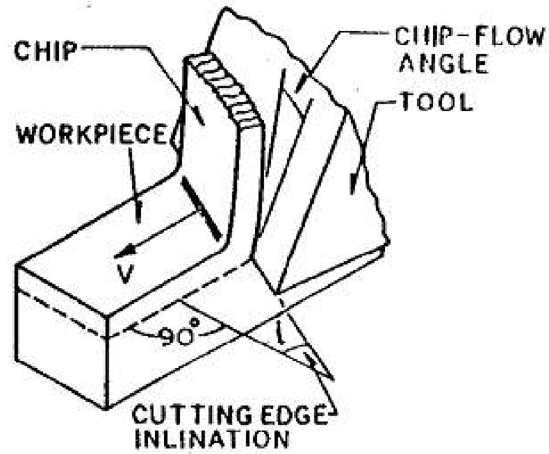
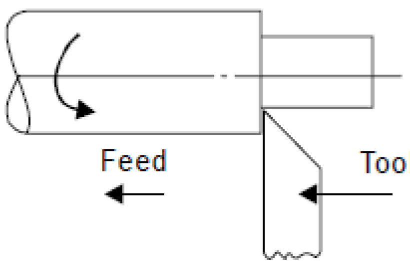
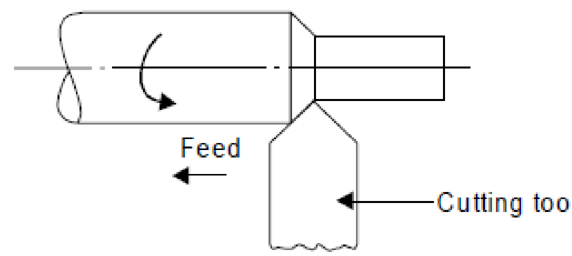


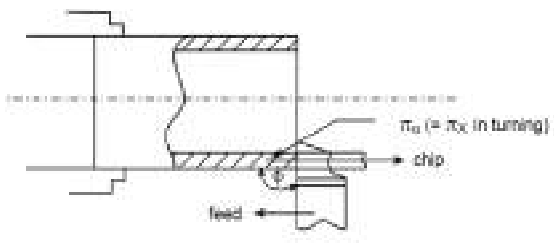
1.12 METHODS OF MACHINING

In the metal cutting operation, the tool is wedge-shaped and has a straight cutting edge. Basically, there are two methods of metal cutting, depending upon the arrangement of the cutting edge with respect to the direction of relative work-tool motion:

- **Orthogonal cutting or two dimensional cutting**
- **Oblique cutting or three dimensioning cutting.**

ORTHOGONAL CUTTING (2D)	OBLIQUE CUTTING (3D)

	
 Orthognal cutting	 Oblique cutting
Orthogonal cutting takes place when the cutting face of the tool is 90^0 to the line of action or path of the tool.	If the cutting face is inclined at an angle less than 90^0 to the path of the tool, the cutting action is known as oblique.
The direction of the chip flow velocity is normal to the cutting edge of the tool.	The direction of the chip flow velocity is at an angle with the normal to the cutting edge of the tool. The angle is known as chip flow angle.
The cutting edge clears the entire width of the workpiece on either ends	The cutting edge may or may not clear the entire width of the workpiece on either ends
Here only two components of forces are acting: Cutting Force and Thrust Force. So the metal cutting may be considered as a two dimensional cutting.	Here three components of forces are acting: Cutting Force, Radial force and Thrust Force or feed force. So the metal cutting may be considered as a three dimensional cutting.
For a given depth of cut, the contact line between the tool and the workpiece will be less and it will be equal to the depth of cut	For a given depth of cut, the contact line between the tool and the workpiece will be more and it will be more than the depth of cut
For the same feed and depth of cut, the force which shears the metal acts on a smaller area on the cutting tool. So the tool life is less	For the same feed and depth of cut, the force which shears the metal acts on a larger area on the cutting tool. So the tool life is more

Orthogonal cutting in the machine shop is confined mainly to such operations as knife turning, broaching and slotting	The bulk of machining being done by oblique cutting.
 *Fig. 6.3 Pure orthogonal cutting (pipe turning)*	

1.13 NOMENCLATURE / GEOMETRY OF SINGLE-POINT TURNING TOOL

A single point cutting tool consist Flank, face, cutting edge, nose, rack angle, clearance angle, cutting edge angle etc. All these parts control the cutting condition, tool life and cutting speed of tool. These parts are described as follows.

1. Shank:

The main body of the tool is known as shank. It is the backward part of tool which is hold by tool post.

2. Face:

The top surface tool on which chips passes after cutting is known as face. It is the horizontal surface adjacent to the cutting edges.

3. Flank:

Sometime flank is also known as cutting face. It is the vertical surface adjacent to cutting edge. Accordingly, there are two flank viz. side flank and end flank.

4. Nose or Cutting point:

The point where both cutting edge meets known as cutting point or nose. It is at the front of the tool.

5. Base:

The bottom surface of tool is known as base. It is just opposite surface of face.

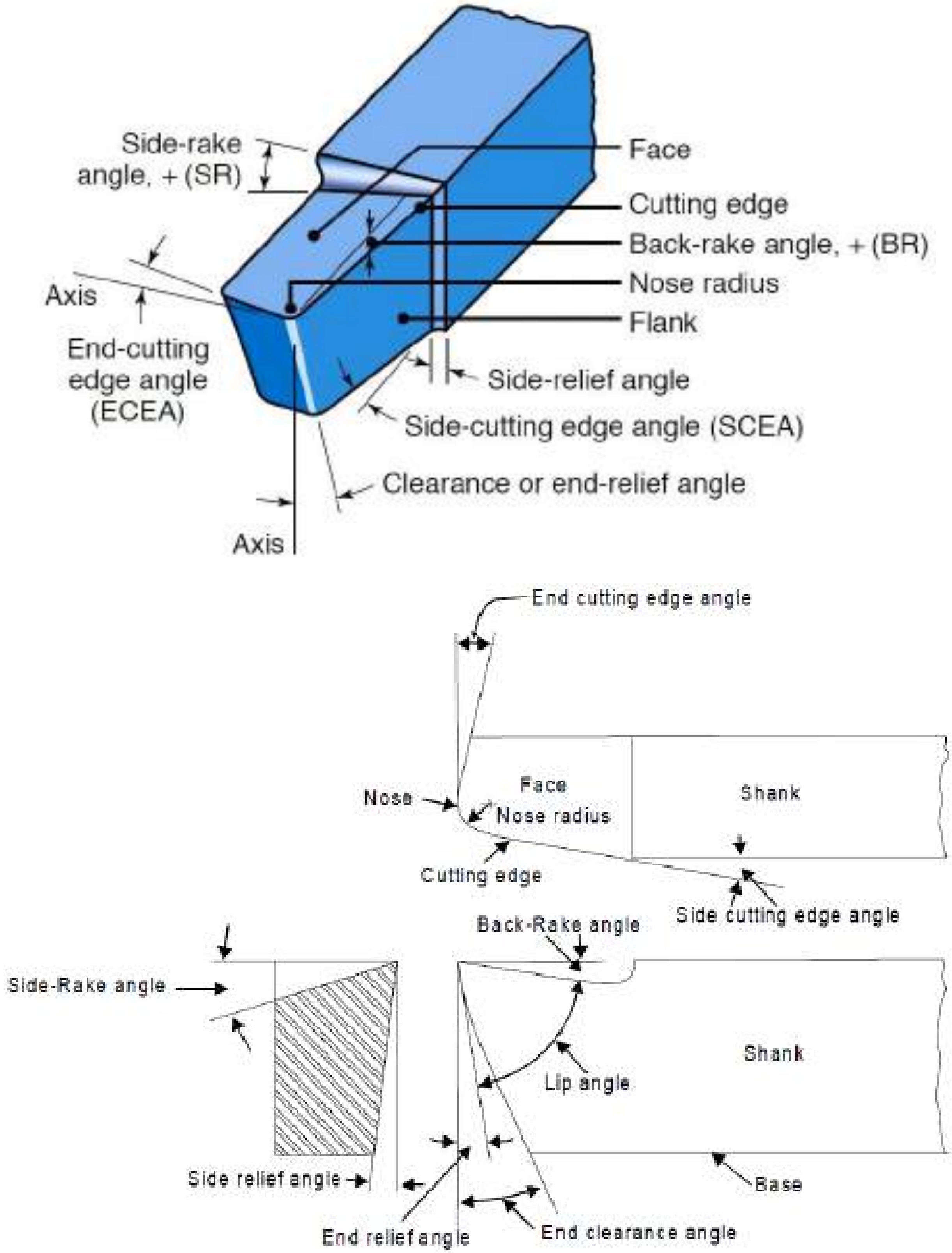


Fig. 1.6 Geometry of single-point turning tool.

6. Back rake angle

Back rake angle is the angle between the face of the single point cutting tool and a line drawn parallel to the base of the tool from the tool tip towards the backside of the tool and. Rake angles may be positive, zero or negative, respectively.

If the slope face is downward from the nose, it is positive rake angle and if it is upward away from the nose, it is negative back rake angle. Back rake angle helps in removing the chips away from the workpiece. When the sloping is zero, it is called as zero rake angle.

The strength of the tool is a function of rake angle. A tool with positive rake angle, Fig.1.7 (a) , has got less cross-sectional area for resisting the cutting forces when compared with tools in Fig. 1.7(b) and 1.7(c). Hence, the strength of the tool with positive rake is less as compared to other tools. Tool shown in Fig. 1.7(b) has zero rake angle. Positive rake angle is provided for easy removal of chip, but chip of brass is discontinuous so it will not affect tool, so no need of positive angle

Tool with negative rake angle is illustrated In Fig. 1.7(c) and it has more cross-sectional area for resisting the cutting forces: Hence, the strength of the tool is maximum when rake angle is negative. Further negative rake is used for machining at higher cutting speeds

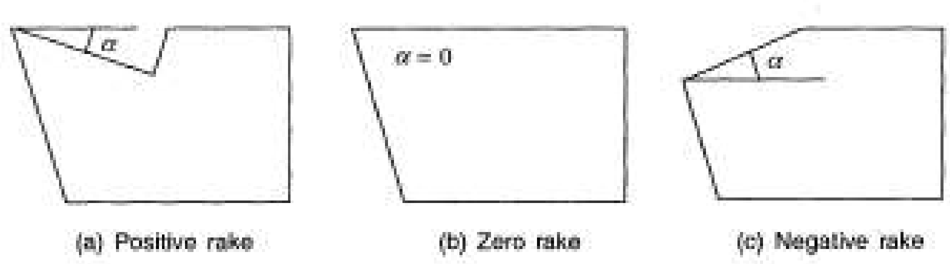


Fig. 1.7 Positive, zero and negative rake angles on a tool.

7. Side rake angle

The angle between the face and plane perpendicular to the side cutting edge is known as side rack angle. It allows chips to flow smoothly when material cut by side cutting edge.

Relief angle

The relief angle is the angle between the flank of the cutting tool and the tangent to the machined surface at the cutting edge. The side and the end face of flank form side and end relief angles, respectively.

The relief angles enable the flank of the cutting tool to clear the workpiece surface and prevent rubbing. These relief angles are also referred to as clearance angles.

8. 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.

9. 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.

10. Side cutting edge angle

Angle formed by the side cutting edge with the normal to machined surface is known as side cutting edge angle. It is essential for enabling the cutting tool at the start of cut to first contact the work back from the tool tip. A large side cutting edge angle increases the force component, which tends to force the cutting tool away from the workpiece.

11. End cutting edge angle

The angle formed by the end cutting edge with the machined surface is called end cutting edge angle. It provides a clearance for that portion of the cutting edge, which is behind the nose radius. This reduces the length of the cutting edge in contact with the work. Also, it is undesirable to have a cutting edge, just contact the work surface without actually cutting. These results in rubbing action causing more tool wear and may spoil the surface finish.

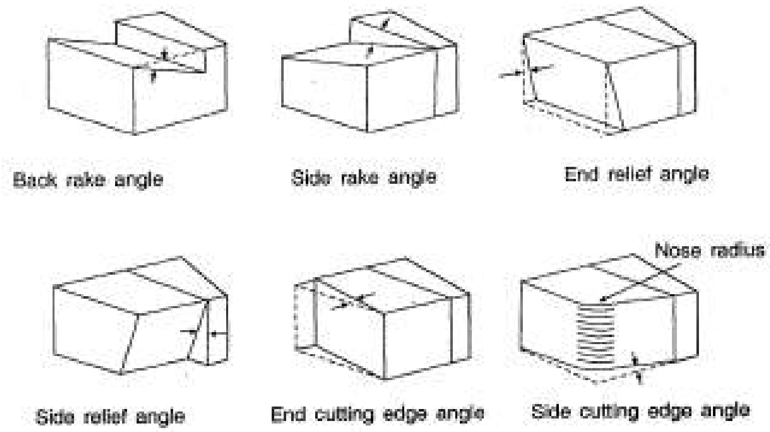
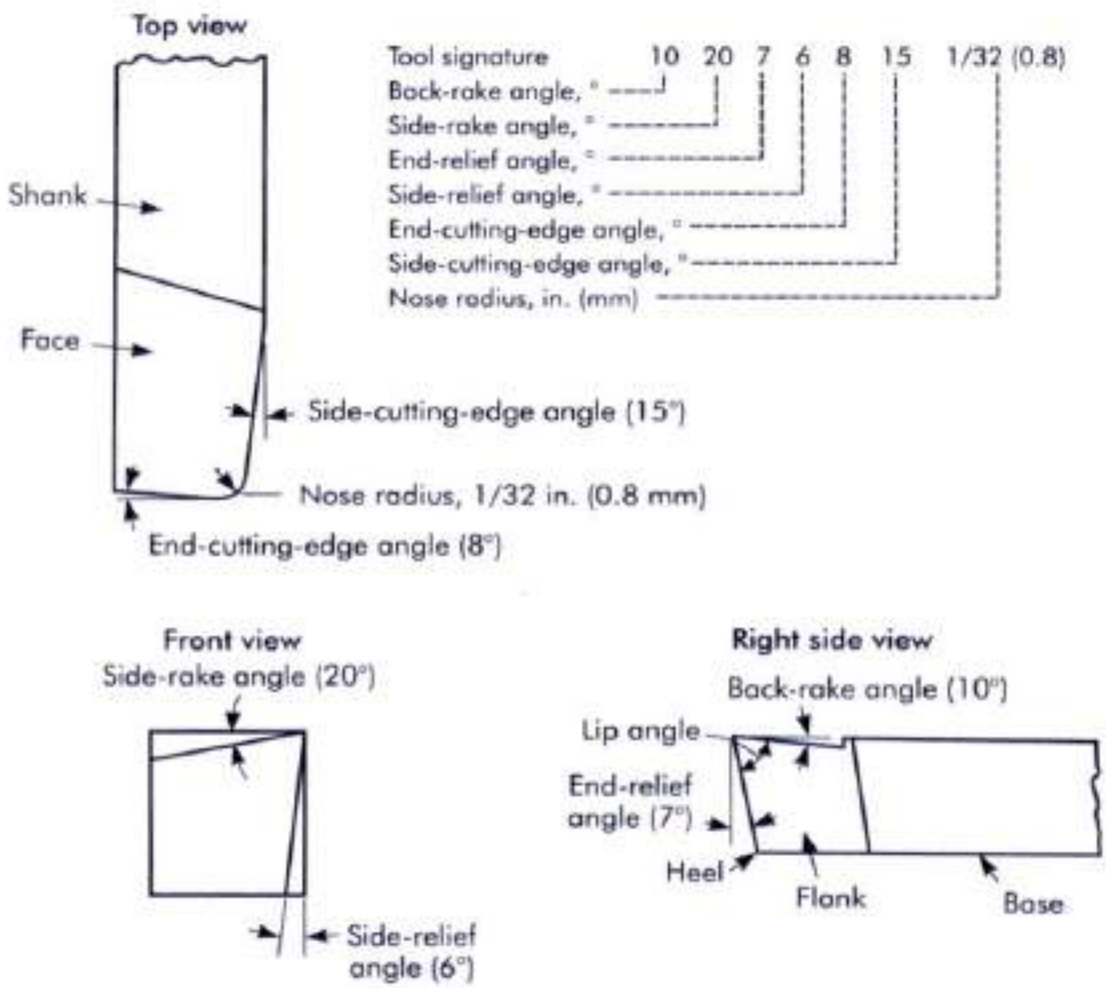


Fig. 1.8 Different cutting angles of single-point tool.

The different angles explained above are shown in **Fig. 1.8.** It is important to note that all the tool angles are defined relative to the machined surface of workpiece. Hence, their magnitude may be different if the tool is not properly set.

1.14 TOOL SIGNATURE



Tool signature is a numerical code that describes all the key angles of a given cutting tool or a convenient way to specify tool angles by use of standardized abbreviated system is known as tool signature or tool nomenclature.

Tool signature of a single point cutting tool consists of Seven elements:

1. Back rake angle (0°)
2. Side rake angle (7°)
3. End relief angle (6°)
4. Side relief angle (8°)
5. End cutting edge angle (15°)
6. Side cutting edge angle (16°) and
7. Nose radius (0.8 mm)

It is usual to omit the symbols for degrees and mm, simply listing the numerical value of each component in single point cutting tool:

A typical tool signature is 0-7-6-8-15-16-0.8

1.15 RIGHT HAND AND LEFT HAND TOOLS

With respect to the direction of feed, single point tools may be classified as either left hand or right hand tools. As shown in Fig. 1.9 they have their cutting edge on the specified side and will cut when moved from left to right or right to left.

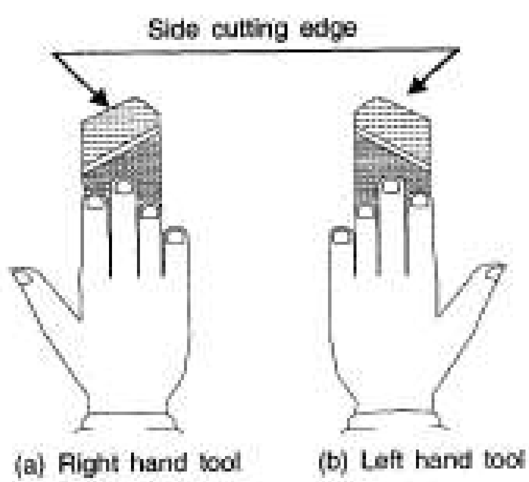


Fig. 1.9 Right hand and left hand tools.

A right hand tool (Fig. 1.9 (a)) is one in which the side cutting edge is on the side of the thumb when the right hand is placed on the tool with the fingers pointed towards the tool nose. In a lathe, a right hand tool cuts from right to left.

A left hand tool (Fig. 1.9(b)) is one in which the side cutting edge is on the thumb side when the left hand is applied.

1.16 CUTTING TOOL MATERIALS

Cutting tool is a device, used to remove the unwanted material from given work piece. For carrying out the machining process, cutting tool is fundamental and essential requirement. The cutting tool material of the day and future essentially require the following properties to resist or retard the phenomena leading to random or early tool failure:

- High mechanical strength; compressive and tensile
- Fracture toughness – high or at least adequate
- High hardness for abrasion resistance
- High hot hardness to resist plastic deformation and reduce wear rate at elevated temperature
- Chemical stability or inertness against work material, atmospheric gases and cutting fluids
- Resistance to adhesion and diffusion
- Thermal conductivity – low at the surface to resist incoming of heat and high at the core to quickly dissipate the heat entered
- High heat resistance and stiffness
- Manufacturability, availability and low cost.

1.16.1 PROPERTIES OF TOOL MATERIALS

Hardness	**:**	The tool material must be harder than the work piece material. Higher the hardness, easier it is for the tool to penetrate the work material.
Hot hardness	**:**	Hot Hardness is the ability of the cutting tool to maintain its Hardness and strength at elevated temperatures. This property is more important when the tool is used at higher cutting speeds, for increased productivity.
Toughness	**:**	In spite of the tool being tough, it should have enough toughness to withstand the impact loads that come in the start of the cut due to force fluctuations, due to imperfections in the work material. Toughness of cutting tools is needed so that tools don't chip or fracture, especially during interrupted cutting operations like milling.
Wear Resistance	**:**	The tool-chip and chip-work interface are exposed to severe conditions that adhesion and abrasion wear is very common. Wear resistance means the attainment of acceptable tool life before tools need to be replaced

Low friction	**:**	The coefficient of friction between the tool and chip should be low. This would lower wear rates and allow better chip flow.
Thermal characteristics	**:**	Since a lot of heat is generated at the cutting zone, the tool material should have higher thermal conductivity to dissipate the heat in shortest possible time; otherwise the tool temperature would become high, reducing its life.

1.16.2 NEEDS AND CHRONOLOGICAL DEVELOPMENT OF CUTTING TOOL MATERIALS

With the progress of the industrial world it has been needed to continuously develop and improve the cutting tool materials and geometry;

- To meet the growing demands for high productivity, quality and economy of machining
- To enable effective and efficient machining of the exotic materials that are coming up with the rapid and vast progress of science and technology
- For precision and ultra-precision machining
- For micro and even nano machining demanded by the day and future.

It is already stated that the capability and overall performance of the cutting tools depend upon,

- The cutting tool materials
- The cutting tool geometry
- Proper selection and use of those tools
- The machining conditions and the environments

Out of which the tool material plays the most vital role. The relative contribution of the cutting tool materials on productivity, for instance, can be roughly assessed from Fig. 3.3.1

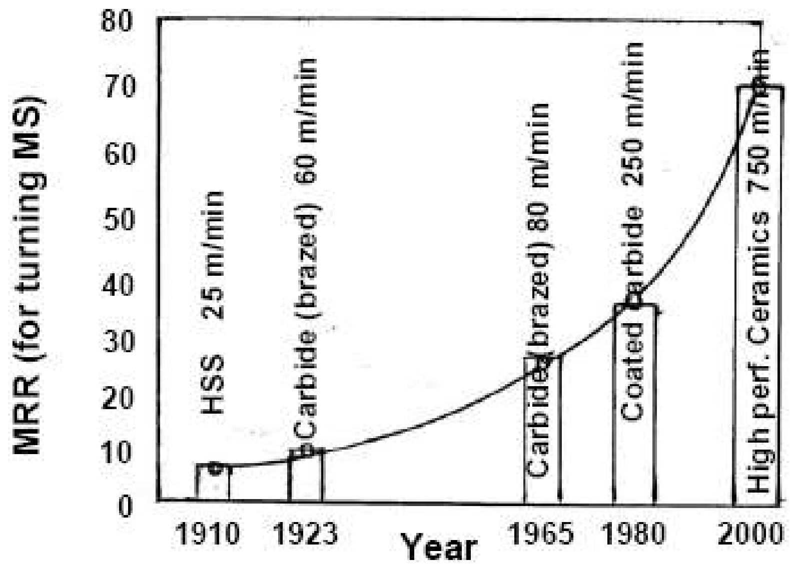


Fig. 3.3.1 *Productivity raised by cutting tool materials.*

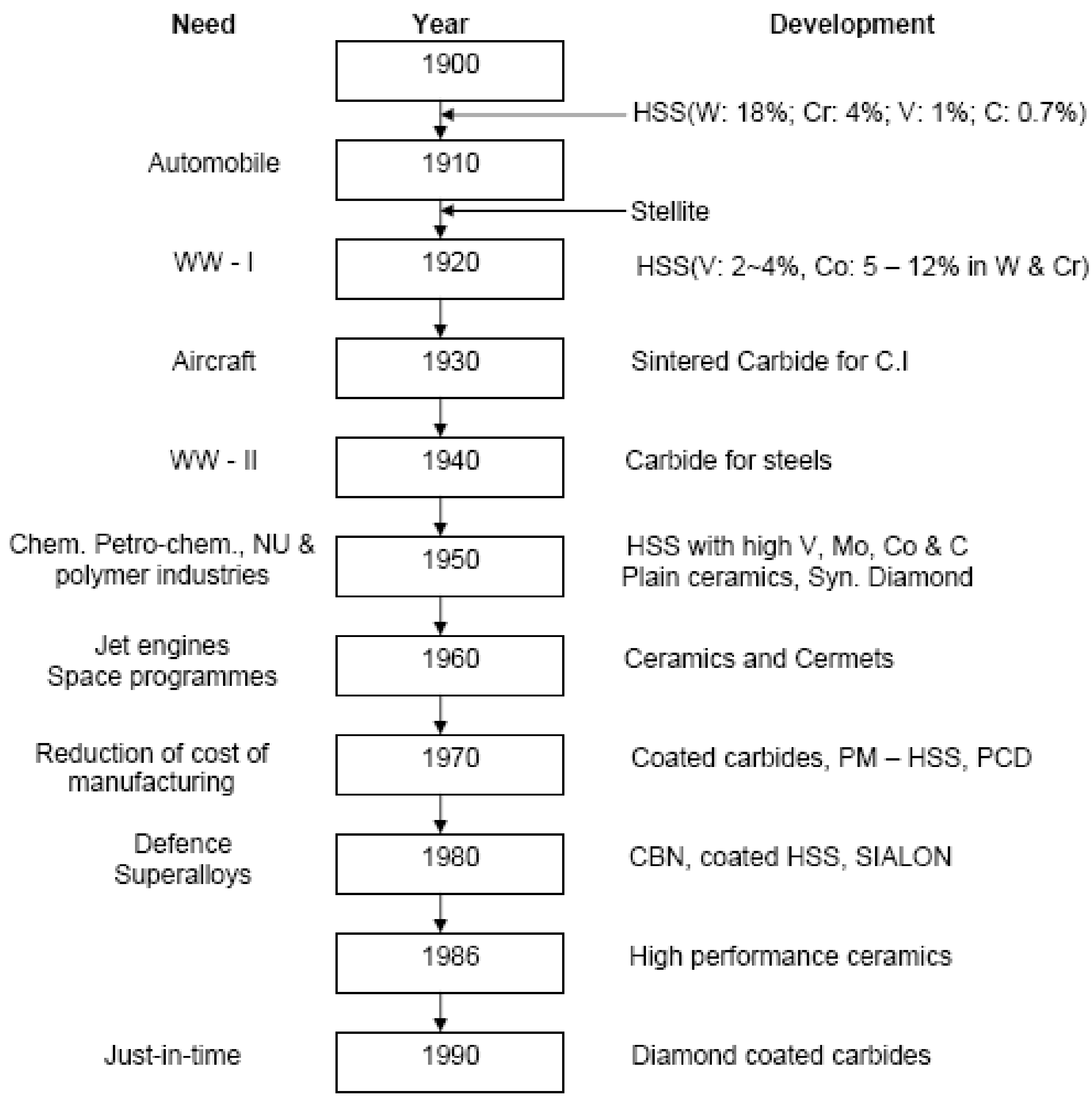


Fig. 3.3.2 Chronological development of cutting tool materials

1.16.3 DIFFERENT ELEMENTS USED IN CUTTING TOOL MATERIALS

Different elements used in cutting tool materials and their properties are

Element	Properties
Tungsten	Increases hot hardness Hard carbides formed Abrasion resistance
Molybdenum	Increases hot hardness Hard carbides formed Improving resistance
Chromium	Depth hardenability Improving abrasion resistance some corrosion resistance
Vanadium	combines with carbon for wear resistance retards grain growth for better toughness
Cobalt	Increases hot hardness, toughness
Carbon	Hardening element forms carbides

1.16.4 CUTTING TOOL MATERIALS

1. **Carbon and Medium alloy steels :** These are the oldest of the tool materials dating back hundreds of years. In simple terms it is a high carbon steel (steel which contains about 0.9 to 1.3% carbon). Inexpensive, easily shaped, sharpened. No sufficient hardness and wear resistance. Limited to low cutting speed operation

2. **High Speed Steel (1900):** The major difference between high speed tool steel and plain high carbon steel is the addition of alloying elements (manganese, chromium, tungsten, vanadium, molybdenum, cobalt, and niobium) to harden and strengthen the steel and make it more resistant to heat (hot hardness). They are of two types: Tungsten HSS (denoted by T), Molybdenum HSS (denoted by M).

General use of HSS is 18-4-1.

18- Tungsten is used to increase hot hardness and stability.

4 – Chromium is used to increase strength.

1- Vanadium is used to maintain keenness of cutting edge.

In addition to these 2.5% to 10% cobalt is used to increase red hot hardness.

Rest iron

- H.S.S is used for drills, milling cutters, single point cutting tools, dies, reamers etc.
- It loses hardness above 600°C.
- Sometimes tungsten is completely replaced by Molybdenum.
- Molybdenum based H.S.S is cheaper than Tungsten based H.S.S and also slightly greater toughness but less water resistance.

3. **Non – ferrous cast alloys**

It is an alloy of

Cobalt – 40 to 50%,
Chromium – 27 to 32%,
Tungsten – 14 to 29%,
Carbon – 2 to 4%

- Also known as cast cobalt alloys or stellites
- It cannot be heat treated and are used as cast form.
- It loses its hardness above 800°C
- It will give better tool life than H.S.S and can be used at slightly higher cutting speeds.
- They are weak in tension and like all cast materials tend to shatter when subjected to shock load or when not properly supported.

4. **Cemented Carbides or Sintered Carbides (1926-30):**

- Produced by powder metallurgy technique with sintering at 1000°C.
- Speed can be used 6 to 8 times that of H.S.S.
- Can withstand up to 1000°C.
- High compressive strength
- They are very stiff and their young's modulus is about 3 times that of the steel.
- High wear resistance.
- High modulus of elasticity.
- Low coefficient of thermal expansion.
- High thermal conductivity, low specific heat, low thermal expansion.

According to ISO the various grades of carbide tool materials grouped as

1. For cutting CI and non ferrous metals are designated as K10 to K50
2. For cutting steel are designated as P10 to P50
3. For general purpose application are designated as M10 to M50.

The advantages of carbide tools are

- They have high productivity capacity.
- They produce surface finish of high quality.
- They can machine hardened steel.
- Their use leads to reduction in machining costs.

Carbide tool are available as brazed tip tools (carbide tip is brazed to steel tool) and inserts (inserts are of various shapes- triangular, square diamond and round).

5. Ceramics and sintered oxides

- Ceramics and sintered oxides are basically made of Al_2O_3, These are made by powder metallurgy technique.
- Used for very high speed (500m/min).
- Used for continuous cutting only.
- Can withstand upto 1200°C.
- Have very abrasion resistance.
- Used for machining CI and plastics.
- Has less tendency to weld metals during machining.
- Generally used ceramic is **sintered carbides**.
- Another ceramic tool material is silicon nitride which is mainly used for CI.

6. Cermets

- Cermets are the combination of ceramics and metals and produced by Powder Metallurgy process.
- When they combine, ceramics will give high refractoriness and metals will give high toughness and thermal shock resistance.
- TiC, nickel, TiN, and other carbides are used as binders.
- Usual combination 90% ceramic, 10% metals.
- Increase in % of metals reduces brittleness some extent and also reduces wear resistance.

7. Diamond

- Diamond has

1. Extreme hardness
2. Low thermal expansion.
3. High thermal conductivity.
4. Very low coefficient of friction.

- Cutting tool material made of diamond can withstand speeds ranging from 1500 to 2000m/min.
- Because diamonds are pure carbon, they have an affinity for the carbon of ferrous metals. Therefore, they can only be used on non-ferrous metals.
- Can withstand above 1500°C.
- A synthetic (man made) diamond with polycrystalline structure is recently introduced and made by powder metallurgy process.
- Feeds should be very light and high speeds Rigidity in the machine tool and the setup is very critical because of the extreme hardness and brittleness of diamond.

8. Cubic Boron Nitride (CBN)

- The trade name is Borozone.
- Consists of atoms of Nitrogen and Boron and produced by power metallurgy process.
- Used as a substitute for diamond during machining of steel.
- Used as a grinding wheel on H.S.S tools.
- Excellent surface finish is obtained.

9. UCON

- UCON is developed by union carbide in USA.
- It consists of Columbium 50%, Titanium 30 % and Tungsten 20%.
- This is refractory metal alloy which is cast, rolled into sheets and slit into blanks. Though its hardness is only 200 BHN, it is hardened by diffusing nitrogen into surface producing very hard surface with soft core. It is not used because of its higher costs.

10. Sialon (Si-Al-O-N)

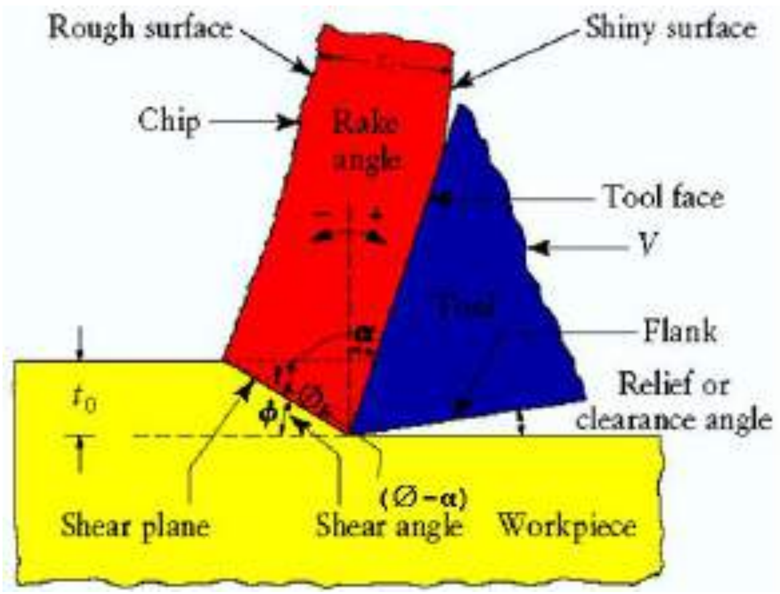
- Sialon is made by powder metallurgy with milled powders of Silicon, Nitrogen, Aluminium and oxygen by sintering at 1800°C.
- This is tougher than ceramics and so it can be successfully used in interrupted cuts. Cutting speeds are 2 to 3 times compared to ceramics.
- At present this is used for machining of aerospace alloys, nickel based gas turbine blades with a cutting speed of 3 to 5 m/sec.

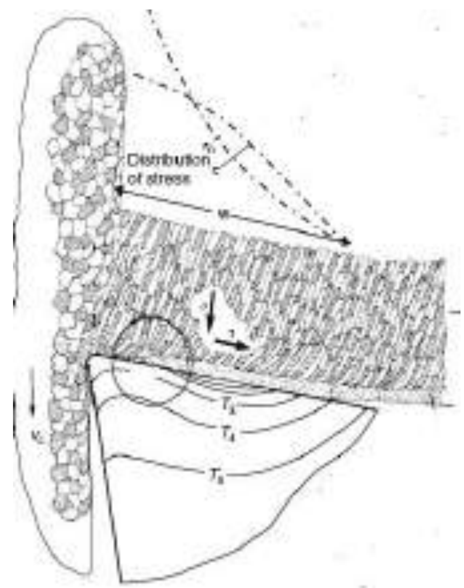
1.17 MECHANICS OF METAL CUTTING

1.17.1 Chip thickness ratio / Cutting ratio

Chip Thickness ratio is defined as the ratio of the undeformed thickness of the chip to the chip thickness after cutting.

- The outward flow of the metal causes the chip to be thicker after the separation from the parent metal. That is the chip produced is thicker than the depth of cut.





The chip separated from work by the action of cutting tool ,under goes severe plastic deformation. This means the chip cannot return to original dimension ,hence there shall always be strain associated with it. so separated chip will have larger dimension than uncut chip thickness**. Hence the chip thickness ratio will always be lesser than one**

Thus, Cutting ratio, r = $\frac{t}{t_c}$

Where

t = undeformed chip thickness (i.e. before cutting) and

t_c = mean thickness of chip (i.e., after cutting)

Coefficient of chip contraction or Chip reduction Coefficient

Coefficient of chip contraction or Chip reduction Coefficient is invese of Chip Thickness ratio. It is a quantitative measurement of plastic deformation occurred during the cutting process.

Chip reduction coefficient k = $\frac{1}{r} = \frac{t_c}{t}$

1.17.1.1 Methods to determine cutting ratio

1. The cutting ratio "r" can be obtained by direct measurement of "t" & "tc". However since underside of chip is rough the correct value of "t_c" is difficult to obtain and hence t_c can be calculated by measuring length of chip (L_c) and weight of the piece of chip "W_c".

   Weight of the chip = W_c = Volume of the chip x density

   = W_c = $L_c \times b_c \times t_c \times \rho$

$$t_c = \frac{W_c}{L_c \times b_c \times \rho}$$

   Where, b_c = width of the chip

   Lc = length of the chip

   ρ = Density of material assumed to be unchanged during chip formation.

2. Alternatively, the length of chip (1c) & length of work (l) can be determined. The length of work can be determined by using a work piece with slot, which will break the chip for each revolution of work piece. The length of chip can be measured by string.

   When metal is cut, there is no change in volume of metal cut. Hence volume of chip before cutting is equal to volume of chip after cutting i.e.

$$L \times d \times t = L_c \times b_c \times t_c$$

$$\frac{t}{t_c} = r = \frac{L_c}{L}$$ [for orthogonal cutting b_c = d]

   L = Length of cut

   L_c = Length of the chip

   d = depth of cut

   b_c = width of the chip

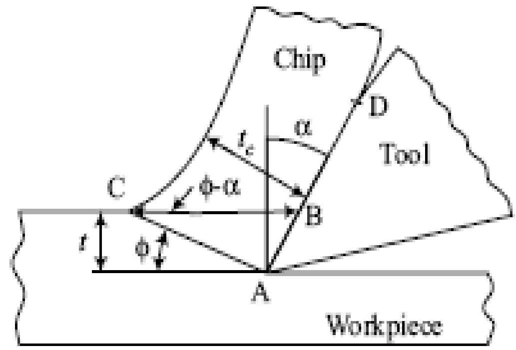
3. Cutting ratio can also be determined by finding chip velocity (Vc) and cutting speed (V). The chip velocity (V_c) can be accurately determined by determining length of chip with a string for a particular cutting time measured with the help of a stopwatch.

From the continuity equation, we know that volume of metal flowing per unit time before cutting is equal to volume of metal flowing per unit time after cutting.

$V.d.t. = V_c .b_c.t_c$

$$\frac{t}{t_c} = r = \frac{V_c}{V}$$ [for orthogonal cutting b_c = d]

1.17.2 Shear Plane Angle



The shear angle is the angle made by shear plane with the direction of tool travel. In fig 1.7a it is the angle made by the line AC with direction of tool travel. The value of this angle depends on cutting conditions, tool geometry, tool material & work material. If the shear angle is small, the plane of shear is larger, the chip is thicker and therefore higher fore is required to remove the chip.

From the figure,

$$t = AC\sin\varphi$$

$$t_c = AC\cos(\varphi - \alpha)$$

$$r = \frac{t}{t_c} = \frac{\sin\varphi}{\cos(\varphi - \alpha)}$$

$$r = \frac{\sin\varphi}{\cos\varphi\cos\alpha + \sin\varphi\sin\alpha}$$

From

$$r\cos\varphi\cos\alpha + r\sin\varphi\sin\alpha = \sin\varphi$$

$$r\cos\varphi\cos\alpha = \sin\varphi\,(1 - r\sin\alpha) \qquad .....(1)$$

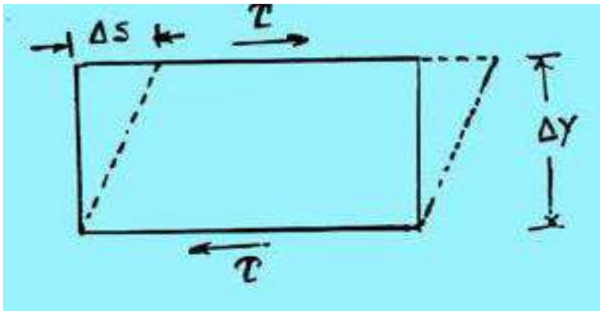
$$\div \text{ eqn } 1 \text{ by } \cos\varphi$$

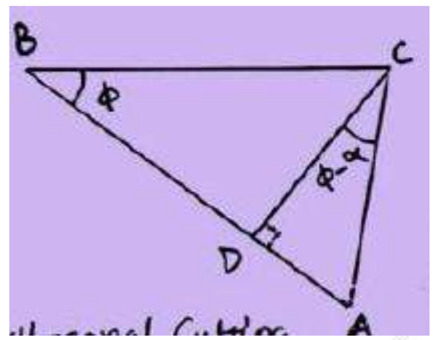
$$r\cos\alpha = \tan\varphi\,(1 - r\sin\alpha)$$

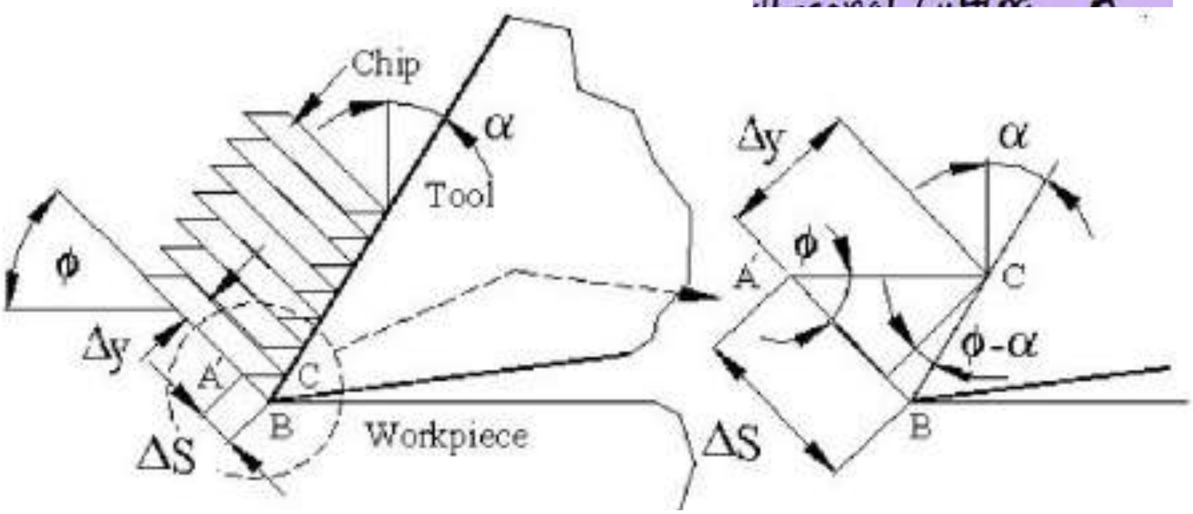
$$\tan\varphi = \frac{r\cos\alpha}{(1 - r\sin\alpha)}$$

1.17.3 SHEAR STRAIN IN CHIP

The total shear strain for chip formation can be calculated by assuming that the chip formation process is in pure shear, in the direction of V_s as shown in **Figure 8(b).**







The shear strain ε is defined as the ratio of the tangential displacement of an element of the material to its height and given as :

$$\text{Shear Strain} = \varepsilon = \frac{\text{Tangential Displacement}}{\text{Length}}$$

$$\varepsilon = \frac{\Delta S}{\Delta Y} = \frac{AB}{CD} = \frac{AD + DB}{CD} = \frac{AD}{CD} + \frac{DB}{CD}$$

$$\varepsilon = \tan\,(\phi - \alpha) + \cot\phi$$

$$\varepsilon = \frac{\sin(\phi-\alpha)}{\cos(\phi-\alpha)} + \frac{\cos\phi}{\sin\phi} = \frac{\sin\phi\sin(\phi-\alpha) + \cos\phi\cos(\phi-\alpha)}{\sin\phi\,\cos(\phi-\alpha)}$$

$$\varepsilon = \frac{\cos\,(\phi - \phi + \alpha)}{\sin\phi\,\cos(\phi-\alpha)}$$

$$\boxed{\varepsilon = \frac{\cos\ \alpha}{\sin\phi\ \cos(\phi-\alpha)}}$$

1.17.4 SHEAR STRAIN RATE IN CHIP

Shear Strain rate = $\dot{\varepsilon} = \frac{\varepsilon}{dt} = \frac{\Delta S}{\Delta Y\, dt} = \frac{V_s}{\Delta Y} \qquad \because \frac{\Delta S}{dt} = V_s$

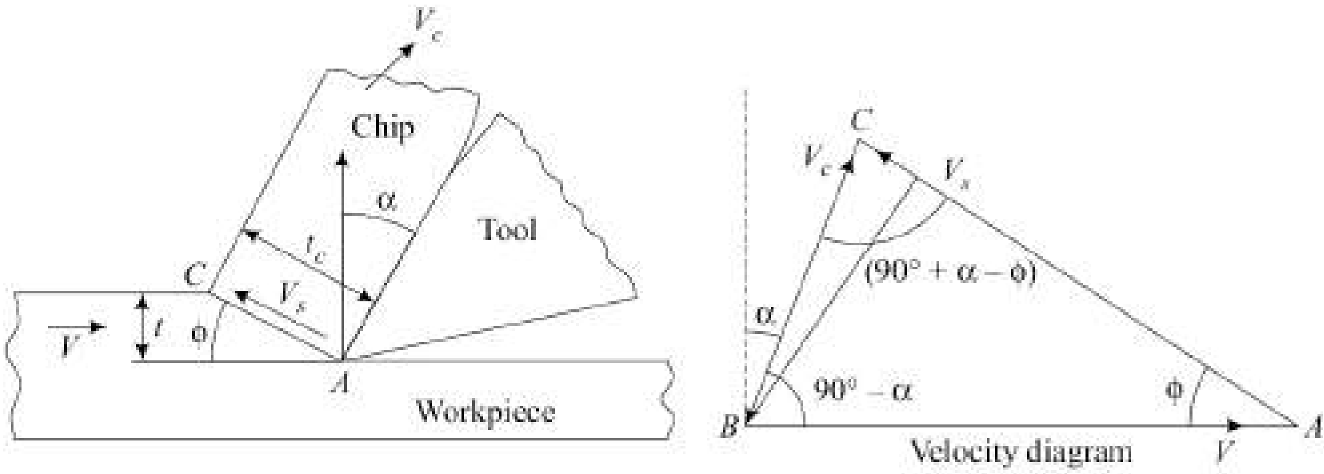
$$\boxed{\dot{\varepsilon} = \frac{V\, Cos\alpha}{Cos\,(\phi-\alpha)\ \Delta Y}}$$

Mean thickness of primary shear zone (ΔY) can be taken as = 25 microns

1.17.5 VELOCITY RELATION IN METAL CUTTING

In orthogonal machining, cutting velocity (V_C) , chip flow velocity (V_f) and shear velocity (V_S) are interrelated. These three velocity vectors together form a triangle, which is called **velocity triangle in machining**. A typical velocity triangle is depicted below. Here length of the sides of the triangle indicates the magnitude of corresponding velocity. The triangle is based on the assumptions of orthogonal machining (chip is flowing in orthogonal direction) and single shear plane (indicates shearing is occurring in a concentrated 2-D shear plane rather than through a region).

Since all three angles of the triangle is known, so Sine Rule can be applied. Law of sine states that, for an arbitrary triangle, the ratio between the length of a particular cord and the sine of its opposite angle is a constant. Hence, for this particular velocity triangle, the following equation can be written. This equation is very important in mechanics of machining as it provides an interrelation between cutting velocity, chip velocity and shear velocity.



$$\frac{V}{\sin(90-(\varphi-\alpha))} = \frac{V_c}{\sin\varphi} = \frac{V_s}{\sin(90-\alpha)}$$

$$\frac{V}{\cos(\varphi-\alpha)} = \frac{V_c}{\sin\varphi} = \frac{V_s}{\cos\alpha}$$

$$V_c = \frac{V\ \sin\phi}{\cos(\phi-\alpha)} \qquad \because \left[r = \frac{\sin\phi}{\cos(\phi - \alpha)} \right]$$

$$\boxed{V_c = V \times r}$$

$$\boxed{V_s = \frac{V\ \cos\alpha}{\cos(\phi-\alpha)} \quad ....(1)}$$

Multiply and divide the above expression by sinΦ

$$V_s = \frac{V\ Sin\ \phi \cos\alpha}{Sin\ \phi \cos(\phi - \alpha)}$$

$$\boxed{V_s = V \quad \varepsilon \ \ Sin\phi}$$

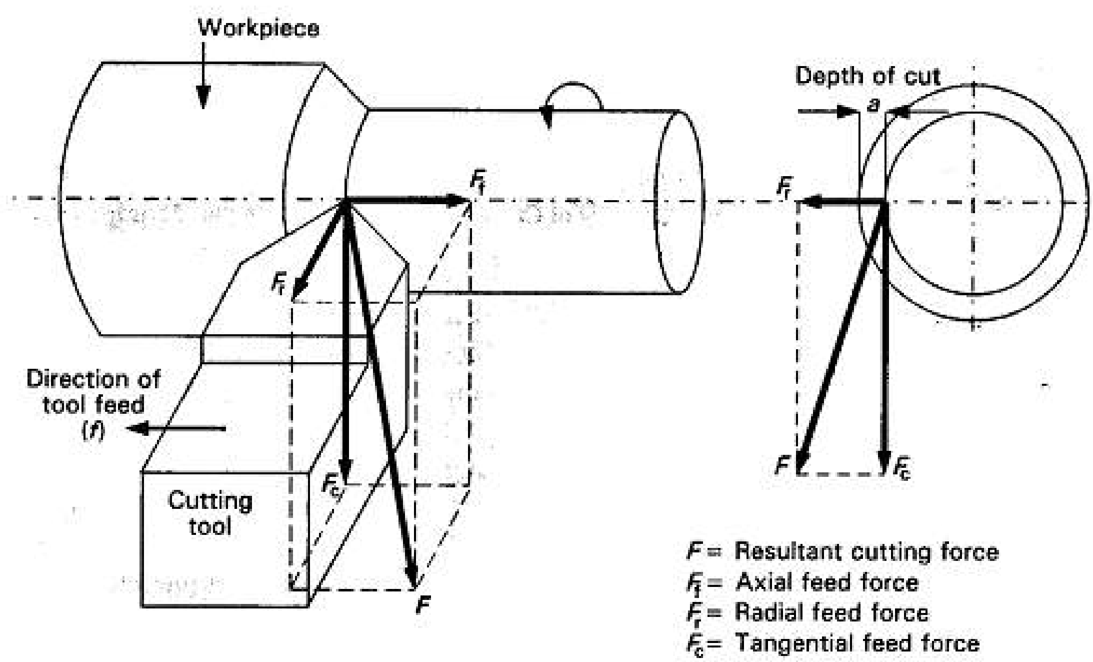
1.18 FORCE MEASUREMENT IN METAL CUTTING

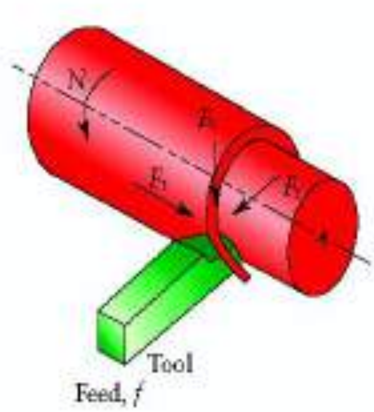
The force acting on a cutting tool during the process of metal cutting are the fundamental importance in the design of cutting tools. The determination of cutting forces necessary for deformation the work material at the shear zone is essential for several important requirements:

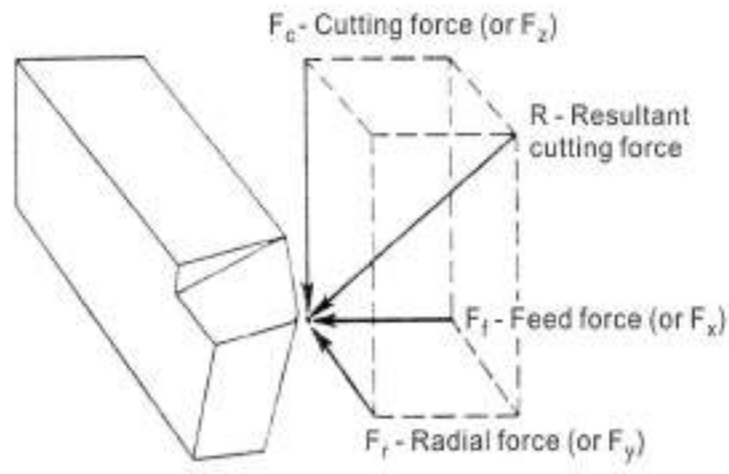
- to estimate the power requirements of a machine tool
- to estimate the straining actions that must be resisted by the machine tool components, bearings, jigs and fixtures
- to evaluate the role of various parameters in cutting forces
- to evaluate the performance of any new work material, tool material, environment, techniques etc. with respect to machinability (cutting forces)

Cutting forces

The force system in the general case of conventional turning process is shown in the following **Figure.**







The largest magnitude is the vertical force Fc which in turning is larger than feed force F_F, and F_F is larger than radial force Fr. For orthogonal cutting system Fr is made zero by placing the face of cutting tool at 90 degree to the line of action of the tool

$$\overline{R} = \overline{F}_x + \overline{F}_y + \overline{F}_z$$
$$= \overline{F}_c + \overline{F}_f + \overline{F}_r$$

Tangential Force (Cutting Force) ($\overline{F}_c$), Feed Force ($\overline{F}_f$), Radial Force ($\overline{F}_r$)

In orthogonal cutting

$$\overline{R} = \overline{F}_c + \overline{F}_t$$

Thrust Force ($\overline{F}_t$)

1.18.1 MERCHANTS FORCE RELATION

For establishing the relationship between measurable and actual forces Merchant's circle diagram will be used.

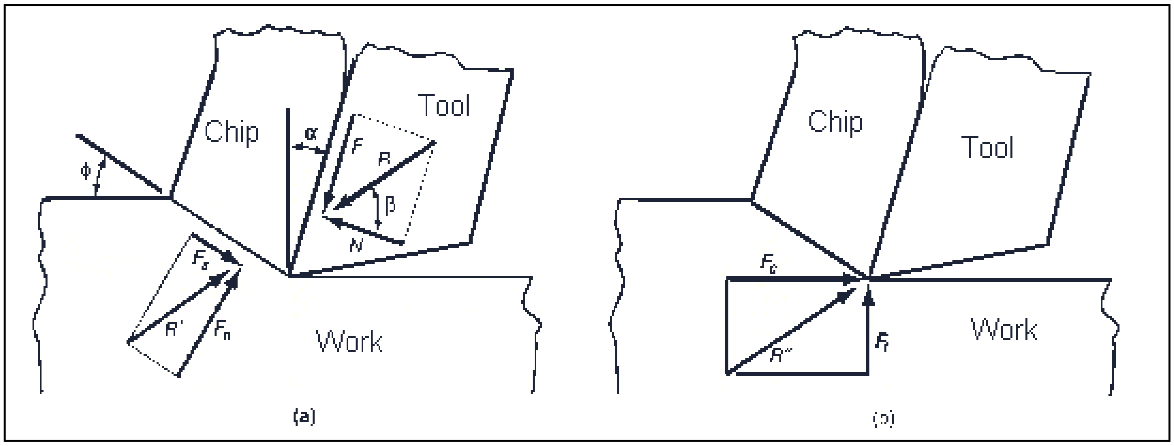
- Merchant circle diagram is used to analyze the forces acting in metal cutting.
- The analysis of three forces system, which balance each other for cutting to occur. Each system is a triangle of forces.

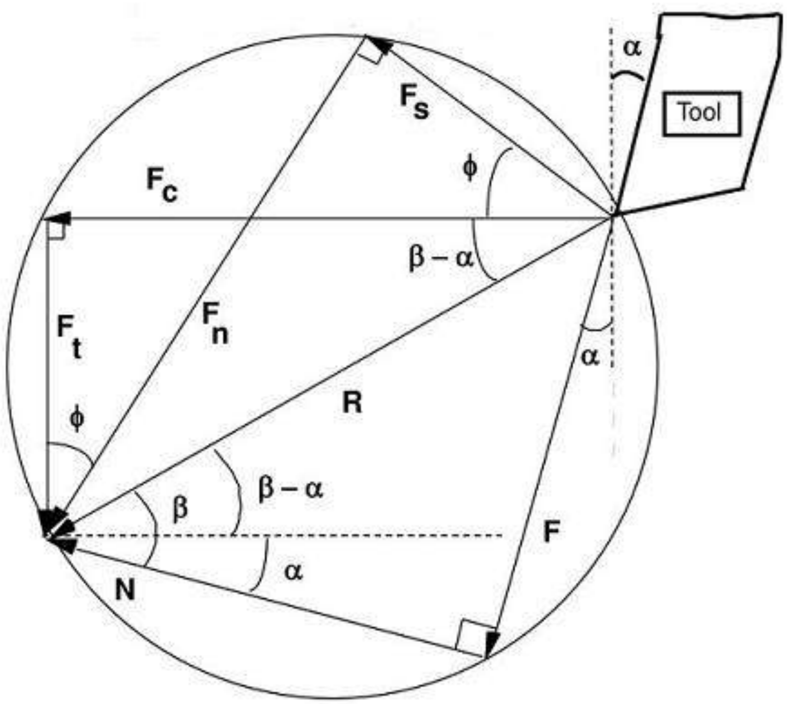
Assumptions made in drawing Merchant's circle:

1. Shear surface is a plane extending upwards from the cutting edge.
2. The tool is perfectly sharp and there is no contact along the clearance force.
3. 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.
4. The chip doesn't flow to either side, that is chip width is constant.
5. The depth of cut remains constant.
6. Width of the too, is greater than that of the work.
7. Work moves with uniform velocity relative tool tip.
8. No built up edge is formed.

The three triangles of forces in merchant's circle diagram are

1. A triangle of forces for the cutting forces,
2. A triangle of forces for the shear forces,
3. A triangle of forces for the frictional forces.





Fs = *Shear Force, which acts along the shear plane, is the resistance to shear of the metal in forming the chip.*

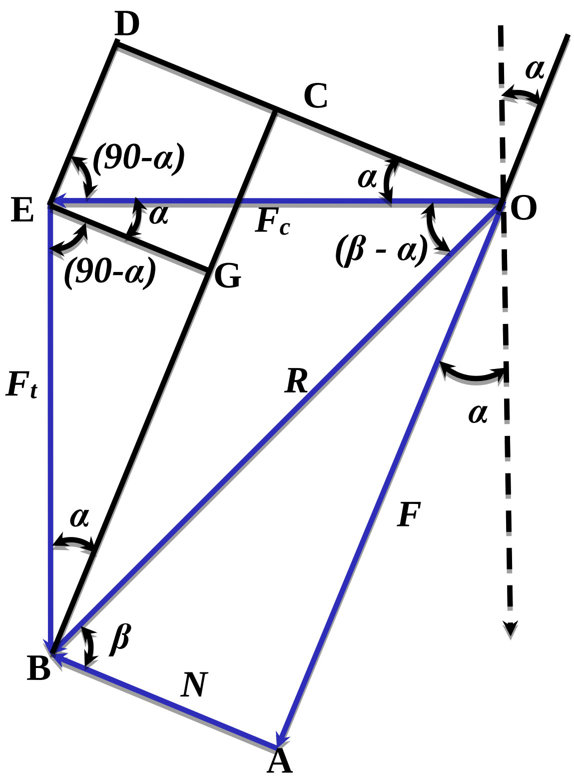
Fn = *Force acting normal to the shear plane, is the backing up force on the chip provided by the workpiece.*

F = *Frictional resistance of the tool acting against the motion of the chip as it moves upward along the tool.*

N = *Normal to the chip force, is provided by the tool.*

Relationship of various forces acting on the chip with the horizontal and vertical cutting force from Merchant circle diagram

Frictional Plane / Tool Plane Forces



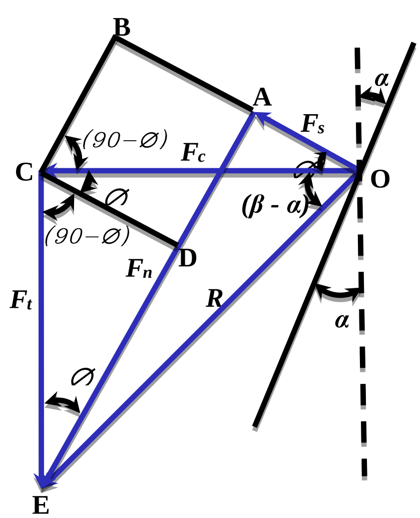
$$F = OA = CB = CG + GB = ED + GB$$
$$\Rightarrow F = F_C \sin\alpha + F_t \cos\alpha$$
$$N = AB = OD - CD = OD - GE$$
$$\Rightarrow N = F_C \cos\alpha - F_t \sin\alpha$$

The coefficient of friction

$$\mu = \tan\beta = \frac{F}{N}$$

Where β = Friction angle

Shear Force System



$$F_S = OA = OB - AB = OB - CD$$
$$\Rightarrow F_S = F_C \cos\phi - F_t \sin\phi$$
$$F_N = AE = AD + DE = BC + DE$$
$$\Rightarrow F_N = F_C \sin\phi + F_t \cos\phi$$

$$F_N = F_S \tan(\phi + \beta - \alpha)$$

$$F = F_C \sin\alpha + F_t \cos\alpha$$
$$N = F_C \cos\alpha - F_t \sin\alpha$$
$$F_S = F_C \cos\phi - F_t \sin\phi$$
$$F_N = F_C \sin\phi + F_t \cos\phi$$
$$F_N = F_S \tan(\phi + \beta - \alpha)$$

1.19 POWER REQUIRED IN METAL CUTTING

Power or energy consumed per unit time is the product of the cutting force and cutting velocity. Hence power at the cutting spindle / cutting power = **$P_c = F_c * V$** N-m/s or watts

The cutting power is dissipated in the shear zone and on the rake surface

The Power consumed/ work done per sec *in shear*: = **$P_s = F_s * V_s$** N-m/s or watts

The Power consumed/ work done per sec *in friction* = **$P_f = F * V_c$** N-m/s or watts

Total Power required = P_c = **$P_s + P_f$**

P_c = **$F_s * V_s + F * V_c$**

Motor power required = P_m =

$$\frac{P_c}{\eta_m}; \quad \text{where } \eta_m = \text{Efficiency of the motor}$$

1.19 SPECIFIC CUTTING ENERGY

Specific Energy, U_c is defined as the total energy required to remove a per unit volume of material removed.

$$U_C = \frac{\text{Energy}}{\text{Volume Removed}} = \frac{\text{Energy per unit time}}{\text{Volume Removed per unit time}}$$

$$U_C = \frac{\text{Cutting Power } (P_c)}{\text{Material Removal Rate (MRR)}}$$

$$U_c = \frac{F_C\, V}{b\ t\ V} = \frac{F_C}{b\, t} \quad \text{N} - \text{m/mm}^3 \text{ or Joule/mm}^3$$

If u_f and u_s be specific energy for friction and specific energy for shearing , then

$$U_c = U_f + U_s = \frac{F\, V_c}{b\ t\ V} + \frac{F_s\ V_s}{b\ t\ V} = \frac{F\ r}{b\ t} + \frac{F_s v_s}{b\ t\ V}$$

1.20 STRESS AND STRAIN ACTING ON THE CHIP

$$\text{Mean shear stress } \tau_s = \frac{F_s}{A_s}$$

As = Area of the shear Plane

= L * b

$$\text{Length of the shear plane } L = \frac{t}{\sin\phi}$$

$$A_s = \frac{b\ t}{\sin\varphi}$$

$$\text{Mean shear stress } \tau_s = \frac{F_s}{A_s} = \frac{(F_C\cos\varphi - F_t\sin\varphi)\ \sin\varphi}{b\ t}$$

$$\text{Mean shear stress } \sigma_s = \frac{F_N}{A_s} = \frac{(F_C\sin\varphi + F_t\cos\varphi)\sin\varphi}{b\ t}$$

We know, work done in shearing unit volume of the material = Shear stress * Shear strain

$$\frac{F_s \times V_s}{b\ t\ V} = \tau_s \times \varepsilon$$

$$\varepsilon = \frac{F_s \times V_s}{\tau_s\ b\ t\ V} \quad ; \quad \varepsilon = \frac{F_s \times V_s}{\frac{F_s}{A_s}\ b\ t\ V}$$

$$\varepsilon = \frac{F_s \times V_s}{\frac{F_s}{\frac{b\ t}{\sin\varphi}}\ b\ t\ V} \quad ; \quad \varepsilon = \frac{V_s}{V}\ x\ \frac{1}{\sin\phi}$$

$$\varepsilon = \frac{\cos\ \alpha}{\sin\phi\ \cos(\phi - \alpha)}$$

$$[\text{Note - } \frac{V_s}{V} = \frac{\cos\alpha}{\cos(\phi-\alpha)}]$$

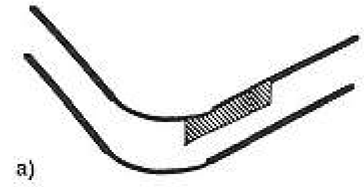
1.21 TOOL WEAR

Cutting tools are subjected to extremely severe cutting conditions such as:

- Metal to metal contact with chip and work
- Very high stress
- Very high temperature
- Very high temperature gradients

- Very high stress gradients

Because of all the above-mentioned factors, the tool-chip interface exhibits a loss in tool material which is known as tool wear. As tool wear progresses, cutting forces increase and vibrations increase. Tool tip softens and flows plastically and gets blunt edge which will result in further progressing of plastic deformation from the tool tip to the interior.



Cutting tools generally fail by :

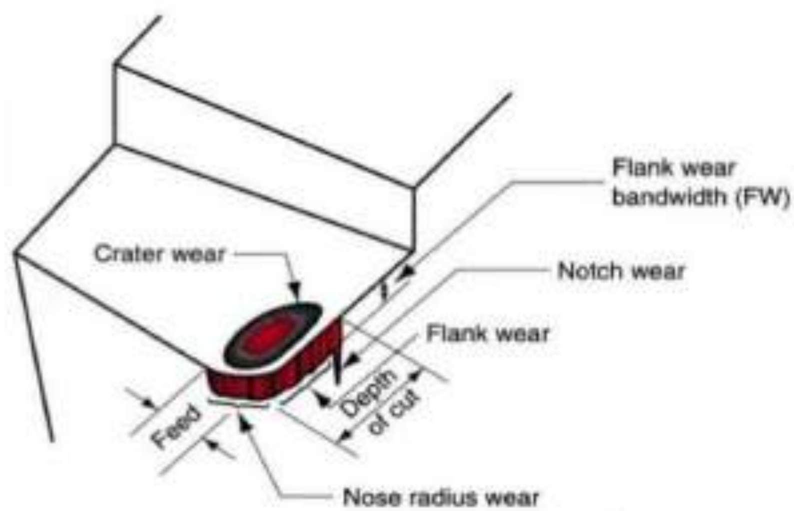
1. Mechanical breakage due to excessive forces and shocks. Such kind of tool failure is random and catastrophic in nature and hence is extremely detrimental.
2. Quick dulling by plastic deformation due to intensive stresses and temperature. This type of failure also occurs rapidly and is quite detrimental and unwanted.
3. Gradual wear of the cutting tool at its flanks and rake surface.

The first two modes of tool failure are very harmful not only for the tool but also for the job and the machine tool. Hence these kinds of tool failure need to be prevented by using suitable tool materials and geometry depending upon the work material and cutting condition.

But failure by gradual wear, which is inevitable, cannot be prevented but can be slowed down only to enhance the service life of the tool.

1.21.1 Types of Wear

- **Flank Wear**
- **Crater Wear**
- **Notch Wear**
- **Nose wear**



a. Flank Wear

Flank wear occurs on the clearance / relief face of the tool and is mainly caused by the rubbing of the newly machined workpiece surface on the contact area of the tool edge. This type of wear occurs on all tools while cutting any type of work material.

Flank wear begins along the lead cutting edge and generally moves downward, away from the cutting edge. The edge wear is also commonly known as the wear land. During the initial and steady wear phase (stage I & II), the root cause is due to abrasion, whereas during stage III, it is by diffusion. Flank Wear generally occurs when the speed of cutting is very high. It causes many losses but one of them is increased roughness of surface of the final product. Also when the cutting speed is increased, the wear curve shifts towards left side, thereby decreasing the tool life

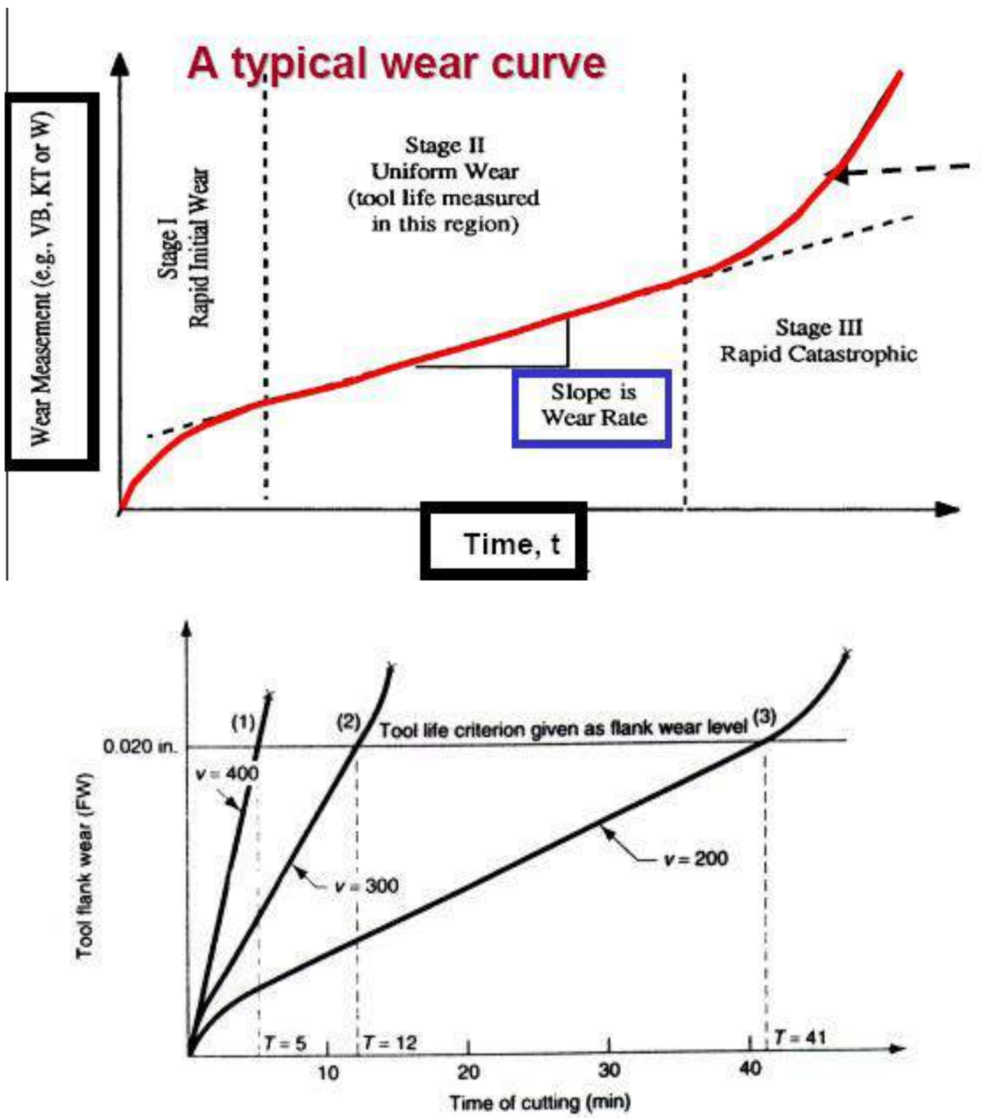


Figure : Flank Wear at different cutting speed

The main reasons for occurrence of flank wear and increase in rate of flank wear are listed below:

i) One of the main reason for flank wear is increase in cutting speed that causes flank wear to grow rapidly.

ii) Another reason for flank wear is high value of feed and depth of cut.

iii) Abrasion by hard panicles of the workpiece.

iv) Shearing of micro welds between tool and work material.

Effects and Losses due to Flank Wear:

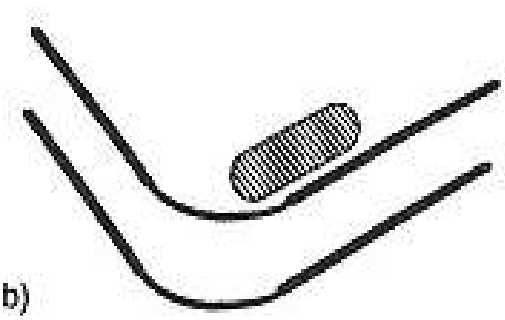
1. Flank wear increases the total cutting force required to cut.
2. It affects component's dimensional accuracy.
3. It also increases the final product surface roughness.
4. Sometimes flank wear also changes the shape of the components produced.

Flank Wear can be prevented by following ways:

1. Flank wear can be prevented by reducing the cutting speed.
2. Reducing the feed and depth of cut.
3. By using good quality of carbide in the cutting tool.

b. Crater Wear

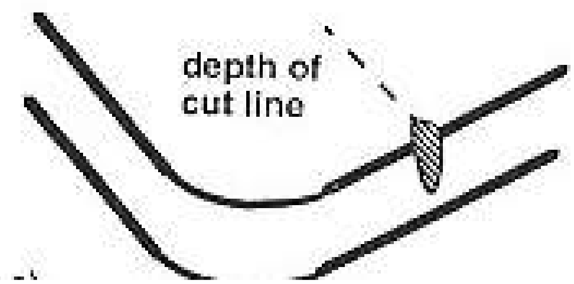
Typically, crater wear occurs on the rake face of the tool. It is essentially the erosion of an area parallel to the cutting edge. This erosion process takes place as the chip being cut, rubs the top face of the tool. Under very high-speed cutting conditions and when machining tough materials, crater wear can be the factor which determines the life of the tool. Crater wear is caused mainly by diffusion and adhesion.


b)

Crater formation increases the effective rake angle of the tool and thus may reduce cutting forces. However, excessive crater wear weakens the cutting edge and can lead to deformation or fracture of the tool, and should be avoided because it shortens tool life and makes resharpening the tool difficult. Severe crater wear usually results from temperature-activated diffusion or chemical wear mechanisms. Crater wear can be minimized by increasing the chemical stability of the tool material or by decreasing the tool's chemical solubility in the chip;

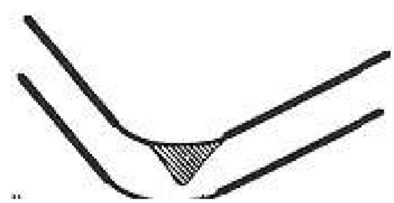
c. Notch Wear

Tools used in rough turning often develop notch wear on the tool face, especially at the point of contact between the tool and the unmachined part surface or free edge of the chip. Depth of cut notching usually results from abrasion and is especially common when cutting parts with a hard surface layer or scale, or work hardening materials which produce an abrasive chip (e. g. stainless steels and nickel-based superalloys). Severe notch wear makes resharpening the tool difficult and can lead to tool fracture, especially with ceramic tools. Notch wear can be reduced by increasing the lead angle, which increases the area of contact between the tool and part surface, by varying the depth of cut in multipass operations, and by increasing the hot hardness and deformation resistance of the tool material.



d. Nose Wear

Nose radius wear occurs on the nose radius of the tool, on the trailing edge near the end of the relief face. It resembles a combined form of flank and notch wear, and results primarily from abrasion and corrosion or oxidation Severe nose radius wear degrades the machined surface finish.



1.21.2 WEAR MECHANISMS

Five basic wear mechanisms are categorized as: Abrasion, adhesion, diffusion, oxidation and chemical corrosion

a. Abrasion:

Abrasive wear occurs when hard particles abrade and remove material from the tool. The abrasive particles may be contained in the chip, as with adhering sand in sandcast parts, carbide inclusions in steel, or free silicon particles in aluminum-silicon alloys. They may also result from the chip form or from a chemical reaction between the chips and cutting fluid, as with powder metal steels (which form a powdery chip) or cast irons alloyed with chromium. Abrasion occurs primarily on the flank surface of the tool. **Abrasive wear by hard particles entrained in the cutting fluid is sometimes called erosive wear.** Abrasive wear is usually the primary cause of flank wear, notch wear, and nose radius wear, and as such is the often, the form of wear which controls tool life, especially at low to medium cutting speeds.

b. Adhesion:

Adhesive or attritional wear occurs when small particles of the tool adhere or weld to the chip due to friction and are removed from the tool surface. It occurs primarily on the rake face of the tool and contributes to the formation of a wear crater. Adhesive wear rates are usually low, so that this form of wear is not usually practically significant. However, significant adhesive wear may accompany built-up edge (BUE) formation, since the BUE is also caused by adhesion, and can result in chipping of the tool.

c. Diffusion:

When a metal is in sliding contact with another metal and the temperature at their interface is high, conditions may become right for the alloying atoms from the harder metal to diffuse into the softer matrix; thereby increasing the latter's hardness and abrasiveness. On the other hand atoms from the softer metals may also diffuse into harder metal, thus weakening the surface layer of the latter to such an extent that particles on it are dislodged and are carried away by flowing chip material. Because of high temperatures and pressures in diffusion wear, micro transfer on an atomic scale takes place. The rate of diffusion increases exponentially with increases in temperature. .

d. Oxidation:

Oxidation occurs when constituents of the tool (especially the binder) react with atmospheric oxygen. It most often occurs near the free surface of the part, where the hot portion of the tool in and around the tool-chip contact region is exposed to the atmosphere. Oxidation often results in severe depth-of-cut notch formation and can be recognized by the fact that the tool material is typically discolored in the region near the notch. Oxidation of wear debris or particles of the work material may also result in the production of hard oxide particles which increase abrasive wear. **Oxidation does not occur with aluminum oxide-based ceramic tools**.

e. CHEMICAL WEAR OR CORROSION

Chemical wear or *corrosion*, caused by chemical reactions between constituents of the tool and the workpiece or cutting fluid, produces both flank and crater wear, with flank wear dominating as the cutting speed is increased. Chemical wear scars are smooth compared to wear scars produced by other mechanisms and may appear to be deliberately ground into the tool. This type of wear is commonly observed when machining highly reactive materials such as titanium alloys.

Chemical wear may also result from reactions with additives (e.g., free sulfur or chlorinated EP additives) in the cutting fluid. (EP additives, in fact, are used to reduce adhesive wear by producing controlled chemical wear. The surface layer of the tool is changed to the reaction product, which is typically soft and wears rapidly by abrasion. Changing the tool material (or coating) or the additives in the cutting fluid will often reduce this type of wear.

1.22 TOOL LIFE

Tool life is a most important factor in the evaluation of machinability, it is the period of time in which the tool cuts effectively and efficiently. Tool life is defined as the time period between two successive regrinding of tool and two successive replacement of tool. A cutting tool should have lone tool life. The cost of grinding and replacement is very high, so the short tool life will be uneconomical. Now a day's tool material improvement increases the tool life.

When a tool no longer performs the desired function then it is said that tool reaches end of useful life. The following sign indicates that the tool life is over.

- Poor surface finish, and dimensional error and presence of chatter marks on the workpiece
- Overheating of workpiece – tool interface due to friction'
- Spoiled cutting edges
- A sudden increase in power consumption

The methods of expressing the tool life are

- Time unit - it is the most commonly used tool life unit
- Number of Workpieces machined by a tool.
- Total length of cut.
- Volume of material removed y tool during its total life span

Volume of metal removed per minute = $\pi \cdot D \cdot t \cdot f \cdot N$ mm^3 / min

Total volume of metal removed for a given time = $\pi \cdot D \cdot t \cdot f \cdot N \cdot T$ mm^3

Total vol. of metal removed for tool failure = $V \times 1000 \times t \times f \times T$ mm^3

[note $V = \frac{\pi D N}{1000}$]

where,

D	=	workpiece dia in mm
t	=	Depth of cut in mm
f	=	Feed rate in mm/rev
N	=	no. of revolutions of workpiece per minute.
T	=	Time of operation in min

V = Cutting speed in m/min

The tool life is most commonly expressed in minute, expected life of some tool material is given below

Cast tool steel	=	120 min
High speed steel tool	=	60 to 120 min
Cemented carbide tool	=	420 to 480 min

1.22.1 FACTORS AFFECTING TOOL LIFE

The tool life will be affected by various factors, which are mentioned below

- Machining variables – Feed, cutting speed and depth of cut
- Tool material and its properties
- Properties of workpiece material
- Tool geometry – Profile of the cutting tool
- Machining conditions like temperature, rigidity of the machine tool, nature of cutting

a. 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 tool matrix 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 flank or crater if cutting speed is increased

A common method of forecasting tool wear is to use Taylor's equation; his study on tool life was done in 1907. Taylor thought that there is an optimum cutting speed for best productivity. This is reasoned from the fact that at low cutting speeds, tools have higher life but productivity is low, and at higher speeds the reverse is true. This inspired him to check up the relationship of tool life and cutting speed. Based on the experimental work he proposed the formula for tool life.

Taylor's Empirical Equation: $VT^n = C$

Where,

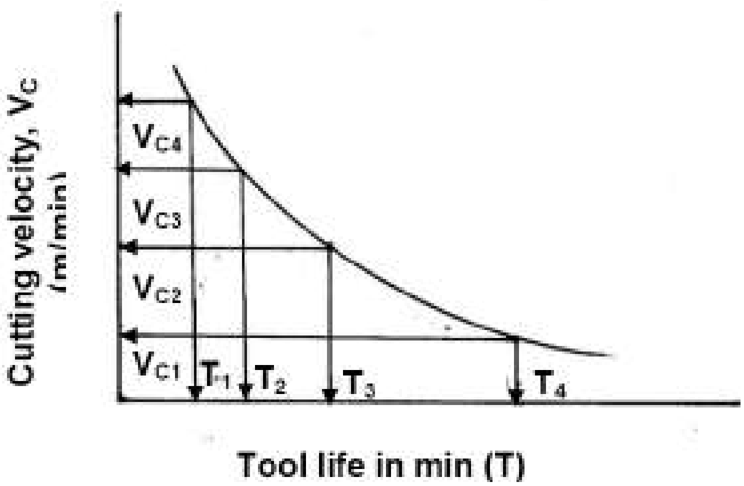
T	=	tool lifetime; usually in minutes
V	=	cutting velocity, m/min
C	=	constant; the cutting velocity for 1 minute of elapsed time before reaching the wear limit of the tool
n	=	constant which is considered a characteristic of the tool material, called tool life index.

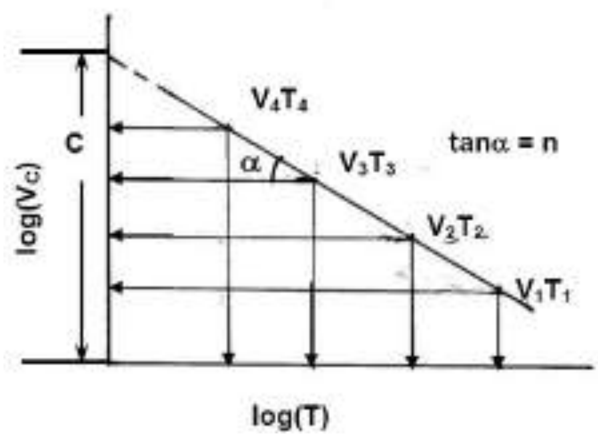
Note: at T = 1 minute, C becomes equal to the cutting speed

Each combination of workpiece, tool material and cutting condition has its own *n* and *C* values, both of which are determined experimentally. The Value of "C" and "n' for different tool materials are listed in the table.

Sl No.	Work materials	Values of "C" for different Tool Materials		
		HSS	Carbide	Ceramic
1	Carbon steel	40 – 100	200 – 160	2500
2	Cast Iron	30 – 60	100 – 150	9000
3	Stainless Steel	20 – 35	120 – 200	
4	Titanium	10 – 20	100 – 150	
5	Tungsten	120 - 160	400 - 600	

Sl No	Tool Materials	Values of "n"
1	HSS	0.08 - 0.2
2	Cemented Carbide	0.3 - 0.49
3	Ceramic	0.5 - 0.7





The tool life obviously decreases with the increase in cutting velocity keeping other conditions unaltered as indicated in Fig. If the tool lives, T_1, T_2, T_3, T_4 etc are plotted against the corresponding cutting velocities, V_1, V_2, V_3, V_4 etc as shown in Fig. a smooth curve like a rectangular hyperbola is found to appear. When F. W. Taylor plotted the same figure taking both V and T in log-scale, a more distinct linear relationship appeared as schematically shown in Fig.

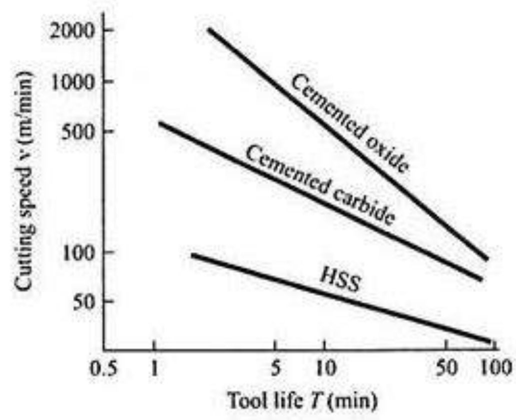


Figure shows the typical variation of tool life with speed for HSS, WC, and ceramic tools, keeping the other conditions the same. It is clear that the tool life for a given speed is normally much higher with WC than that with HSS. A ceramic tool performs better at a high cutting speed.

b. FEED AND DEPTH OF CUT

Tool life is a direct function of temperature. At higher feed, the cutting force per unit area of chip tool contact on rake face & work tool contact on flank face is increased there by increasing the temperature and hence wear rate.

Similarly, at higher depth of cut, the area of chip tool contact is increased roughly in proportion to change in depth of cut (such is not the case with feed change where the chip tool contact area changes by larger proportion than change in depth of cut), increasing the temperature & consequently the wear rate.

The cumulative effect of speed, feed & depth of cut can be seen from the modified Taylor's tool life equation. Increase in any one of the above reduces the tool life, but cutting speed has more impact on tool life followed by feed & depth of cut.

Modified Taylor's Equation:

$$VT^n f^p d^q = c$$

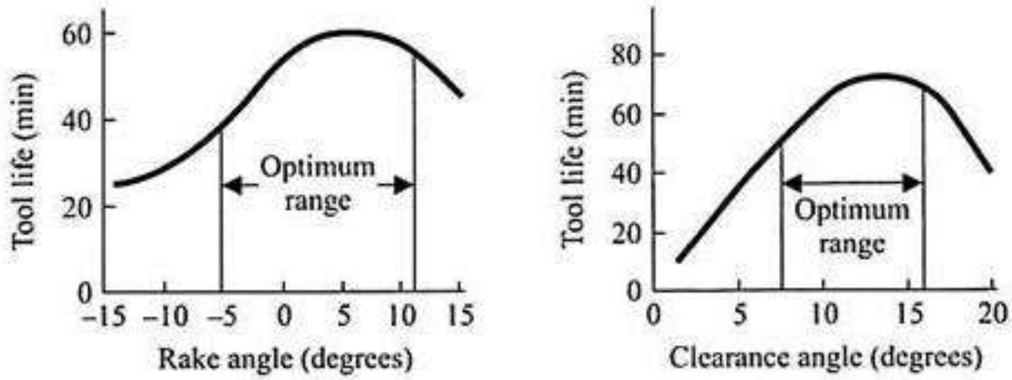
f = Feed mm/rev; d = depth of cut in mm

p,q are constants < 1

q < p indicates that tool life is more sensitive to the uncut slip chip thickness than to the width of cut.

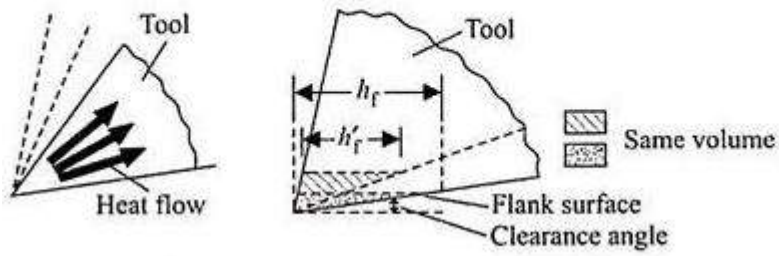
c. TOOL GEOMETRY

Rake angles and relief angles & nose radius affect the tool life by varying degree.



(a) Effect of rake angle

(b) Effect of clearance angle

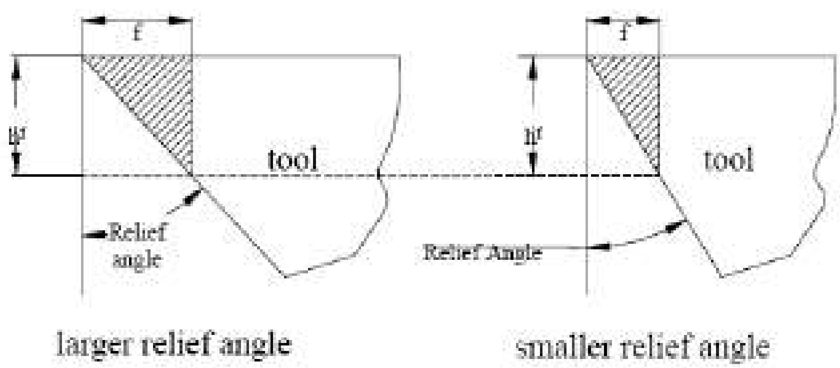


(a) Dependence of heat conduction on tool angle

(b) Dependence of flank wear on clearance angle

Increasing the rake angle decreases the cutting force and heat produced at the tool tip. However increasing the rake angle to a large value reduces the tool material available at the tool tip for conducting heat generated, thus increasing the tool tip temperature. This would decrease tool life, thus again an optimum value has to be selected.

Large relief angle increases volume of wear required to reach a particular width of flank wear land as seen from fig and also reduces the tendency of rubbing between flank & work piece surface, there by increasing the tool life. However, on the other hand, larger the relief angle, smaller is the mechanical strength of cutting edge & more liable the tool is to chipping fracture. Thus there is maximum tool life for optimum relief angle as seen for fig.



d. Work material:

The properties of the work material that tend to increase the tool life are as follows,

a) softness (or lack of hardness) to reduce cutting forces, cutting temperature & abrasive wear,
b) absence of abrasive component such as slag inclusions, surface scale & sand,
c) presence of desirable additives like lead to act as boundary lubricants and sulphur to reduce cutting forces & temperatures by acting as stress raiser, and
d) lack of work hardening tendency that tend to reduce cutting forces and temperatures and also abrasive wear and
e) Occurrence of favorable microstructure, e.g. presence of spheroidized pearlite instead of lamellar pearlite in high carbon steel improves tool life.

Similarly in cast irons, a structure that contains large amount of free graphite & ferrite leads to greater tool life than one, which contains free iron carbide.

e. TOOL MATERIAL

Tool material which can withstand maximum cutting temperature without losing its mechanical properties and geometry will ensure maximum tool life. Hence higher the mechanical properties (mostly hardness and toughness) in the tool materials, longer will be the tool life.

f. CUTTING FLUID

The cutting fluid cools the tool & work piece, acts as lubricant and reduces friction at chip tool interface. Therefore the cutting temperatures are decreased & the use of cutting fluid in the tool materials with low value of hot hardness (e.g.) shows appreciable increase in tool life. However in carbides & oxides, which have high value of hot hardness, the cutting fluid has negligible effect on tool forces or tool life.

g. NATURE OF CUTTING:

It has also great influence on tool life; e.g. in the case of continuous cutting the tool life is much better than in intermittent cutting. The intermittent cutting gives regular impacts on the tool leading to its failure much earlier.

h. VIBRATION / RIGIDITY BEHAVIOR OF MACHINE TOOL WORK SYSTEM

If the machine is not properly designed, if the work piece is long and thin or if the tool overhang is excessive, chatter may occur during cutting. It is known that chatter may cause fatigue failure or catastrophic failure of tool due to mechanical shock

1.23 MACHINABILITY: -

Machinability is the property of material to be machined, which governs the case or the difficulty with which it can be machined under a given set of conditions. In spite of the efforts made by the number of investigators, so far, there has been no exact quantitative definition of Machinability. It is due to large number factors involved & their complexity in metal cutting process viz. forces & power, tool life, surface finish etc. These are dependent upon number of variable such as work material, cutting conditions, M/C tool rigidity tool geometry. Due to this, it is impossible to combine these factors so as to give a suitable definition for Machinability. It is of a considerable economic importance for production engineer to know in advance the Machinability of work material so that he can its processing in an efficient manner.

1.25 TYPES OF TOOLS

All the cutting tools used in metal cutting can be broadly classified into two categories viz. single point tools and multipoint tools. Fig.1.4 shows examples of single point and multipoint tools.

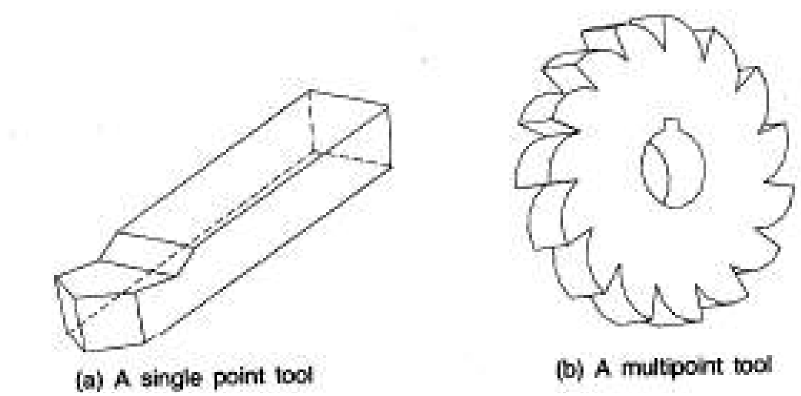


Fig. 1.4 Two types of cutting tools.

a. Single point tools

A single point tool has only one cutting edge. An example of a single point tool is shown in Fig. 1.4(a). These types of tools are used in lathes, shapers and planers.

b. Multipoint tools

As the name implies, multipoint tools have more than one cutting edge. Fig. 1.4(b) shows a multipoint tool. The tool has cutting teeth on its periphery. During the process of machining, each tooth does the cutting action independently on the workpiece. The tool shown In Fig. 1.4(b) is called a milling cutter and is used in milling operation. Examples of other multipoint tools include drills and reamers.

1.26 TEMPERATURE DISTRIBUTION IN METAL CUTTING

During machining heat is generated at the cutting point from three sources, as indicated in Fig. 2.7.1. Those sources and causes of development of cutting temperature are:

- **Primary shear zone (1)** where the major part of the energy is converted into heat
- **Secondary deformation zone (2)** at the chip – tool interface where further heat is generated due to rubbing and / or shear
- **At the work tool interface (3)** due to rubbing between the tool and the finished surfaces.

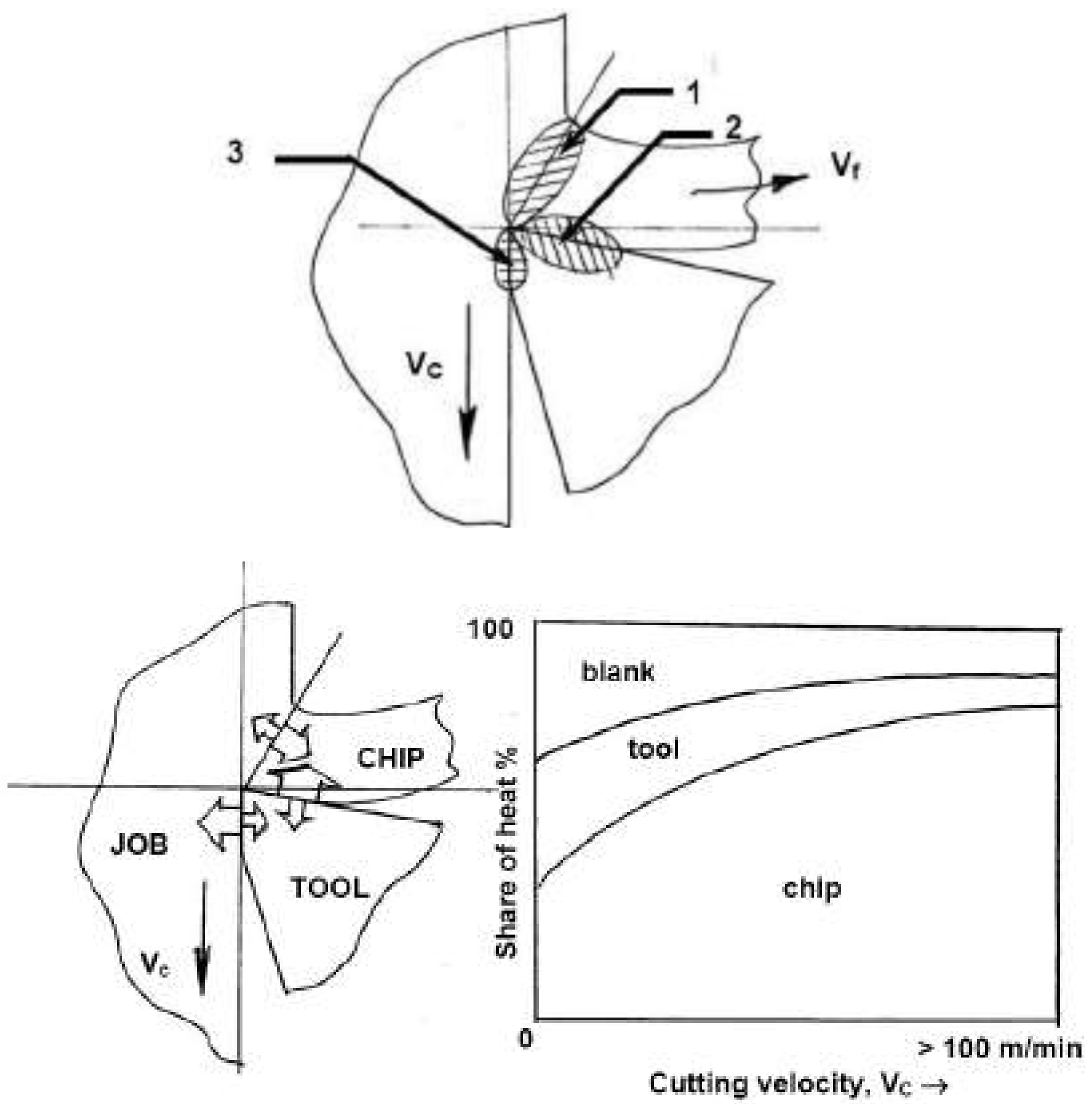


Figure : Apportionment of heat amongst chip, tool and work

The heat generated is shared by the chip, cutting tool and the blank. The apportionment of sharing that heat depends upon the configuration, size and thermal conductivity of the tool – work material and the cutting condition. Fig. 2.7.2 visualises that maximum amount of heat is carried away by the flowing chip. From 10 to 20% of the total heat goes into the tool and some heat is absorbed in the blank. With the increase in cutting velocity, the chip shares

For example, in a typical study of machining mild steel at 30 m/min at about 750 deg of

cutting temperature at tool-chip interface, the distribution of total energy developed at the shear zone is as follows

- Energy at chip – 60 percent
- Energy to workpiece – 30 percent
- Energy to tool - 10 percent

1.26.1 EFFECTS OF THE HIGH CUTTING TEMPERATURE ON TOOL AND WORK

High cutting temperatures are detrimental to both the tool and the job. The major portion of the heat is taken away by the chips. But it does not matter because chips are thrown out. So attempts should be made such that the chips take away more and more amount of heat leaving small amount of heat to harm the tool and the job. The possible detrimental effects of the high cutting temperature on cutting are:

On tool

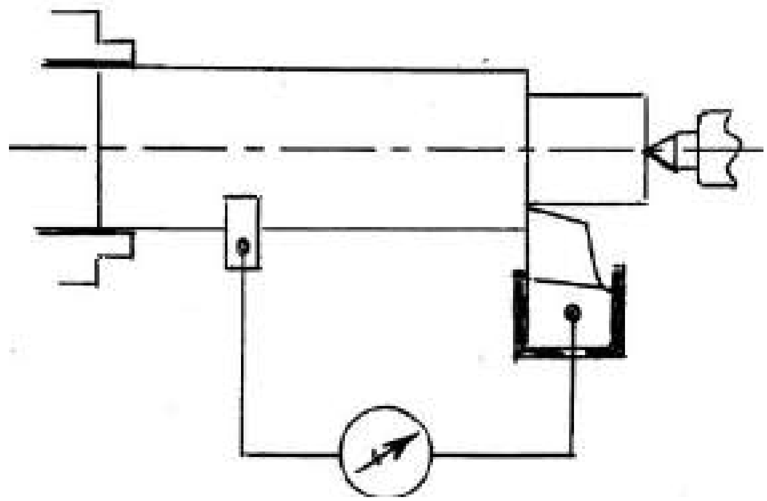
- Rapid tool wear , which reduces tool life
- Cutting edges plastically deform and tool may loose its hot hardness
- Thermal flaking and fracturing of cutting edges may take place due to thermal
- shock
- Built up edge formation

On work

- Dimension inaccuracy of work duet to thermal distortion and expansion and
- contraction during and after machining
- Surface damage by oxidation, rapid corrosion, burning etc.
- Tensile residual stresses and microcracks at the surface and sub surfaces.

1.26.2 Tool work thermocouple technique for measuring Temperature

Fig. 2.7.3 shows the principle of this method. In a thermocouple two dissimilar but electrically conductive metals are connected at two junctions. Whenever one of the junctions is heated, the difference in temperature at the hot and cold junctions produce a proportional current which is detected and measured by a milli-voltmeter. In machining like turning, the tool and the job constitute the two dissimilar metals and the cutting zone functions as the hot junction. Then the average cutting temperature is evaluated from the mV after thorough calibration for establishing the exact relation between mV and the cutting temperature.



Tool-work thermocouple technique of measuring cutting temperature.

1.26.3 Control of cutting temperature

It is already seen that high cutting temperature is mostly detrimental in several respects. Therefore, it is necessary to control or reduce the cutting temperature as far as possible. Cutting temperature can be controlled in varying extent by the following general methods:

- Proper selection of material and geometry of the cutting tool(s)
- Optimum selection of cutting speed and feed rate without sacrificing MRR
- Proper selection and application of cutting fluid
- Application of special technique, if required and feasible.

1.27 CUTTING FLUIDS

Cutting fluids, sometimes referred to as lubricants or coolants are liquids and gases applied to the tool and work piece to assist the cutting operation.

1.27.1 Functions of cutting fluids:

- To cool the tool.
- To cool the work piece.
- To lubricate and reduce friction
- To improve surface finish.
- To protect the finished surface from corrosion.

•To wash the chips away from the tool.

1.27.2 Properties of cutting fluids:

- High heat absorption capability.
- Good lubricating quality.
- High flash point so as to eliminate the hazard of fire.
- Stability so as not to get oxidized in presence of air.
- Neutral so as not to react chemically.
- Odourless so as not to produce bad smell even when heated.
- Harmless to the skin of the operators.
- Non corrosive to the work or the machine.
- Transparency so that the cutting action of tool may be observed by the operators.
- Low viscosity to permit free flow of the liquid.
- Low priced to minimize production cost.

1.27.3 TYPES OF CUTTING FLUIDS

4 general types:

- ❖ **Oils -** mineral, animal, vegetable, compounded, and synthetic oils,
- ❖ **Emulsions -** a mixture of oil and water and additives
- ❖ **Semi synthetics -** chemical emulsions containing little mineral oil
- ❖ **Synthetics -** chemicals with additives

1. **Straight oils**

These oils are non-emulsifiable and very useful in machining operations where they function in undiluted form. Their composition is a base mineral or even petroleum oil. Often they contain polar lubricants like vegetable oils, fats and esters.

They may also contain extreme pressure additives including sulphur, chlorine, and phosphorus. To achieve the best lubrication use straight oils however they may have poor cooling characteristics.

2. **Synthetic fluids**

They do not contain mineral oil base or petroleum. Instead, they're formulated from the alkaline organic and inorganic compounds alongside additives to prevent corrosion. They function well in their diluted form. Of all the varieties of cutting fluids, synthetic fluids offer the best cooling performance.

3. **Soluble oils**

Soluble Oils usually form an emulsion after mixing them with water. The resulting concentrate contains emulsions and a base mineral oil to produce a stable emulsion. They function well in their diluted form and offer a great lubrication in addition to heat transfer performance. They are the least expensive and are the most widely used fluids in the industry.

4. **Semi-synthetic fluids**

These fluids are basically a combination of the soluble oils and synthetic fluids. Besides, the heat transfer performance and cost of the semi-synthetic fluids falls between those of the soluble and synthetic fluids.

1.28 TOOL DYNAMOMETERS

Dynamometers are devices used to measure cutting forces in machining operation. The cutting force cannot be detected or quantified directly but their effect can be sensed using Transducer. For example, a force which can neither be seen nor be gripped but can be detected and also quantified respectively by its effect and the amount of those effects (on some material) like ***elastic deflection, deformation, pressure, strain*** etc. These effects, called signals, often need proper conditioning for easy, accurate and reliable *detection* and measurement. In other words, Measurement involves three stages

- ❖ Conversion into another suitable variable (deflection, expansion etc)
- ❖ Amplification, filtration and stabilization
- ❖ Reading or recording

Measurement of cutting force(s) is based on three basic principles:

1. Measurement of *elastic deflection* of a body subjected to the cutting force
2. Measurement of elastic deformation, i.e. *strain induced by the force*
3. Measurement of pressure developed in a medium by the force.

1.28.1 Measuring deflection caused by the cutting force(s)

Under the action of the cutting force, say **F_C** in turning, the tool or tool holder elastically deflects as indicated in Fig.1 Such tool deflection, δ is proportional to the magnitude of the cutting force, **FC**, simply as, Treating the tool as a cantilever beam, we can write the deflection of the tool as

$$\delta = \mathbf{F_C}\left(\frac{L^3}{3EI}\right)$$

Since for a given cutting tool and its holder, E and I are fixed, we can write, **$\delta \propto F_c$**

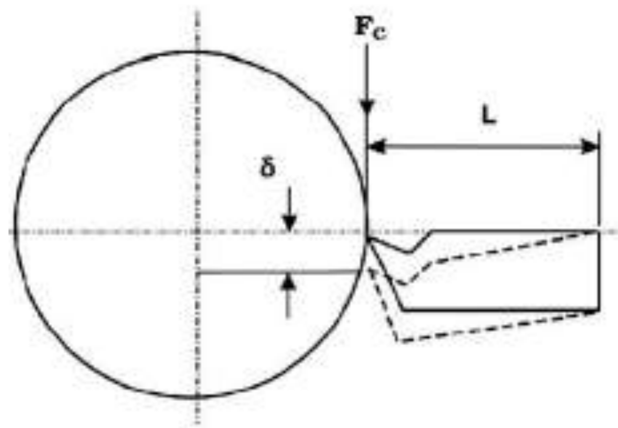


Figure :- Deflection of cutting tool

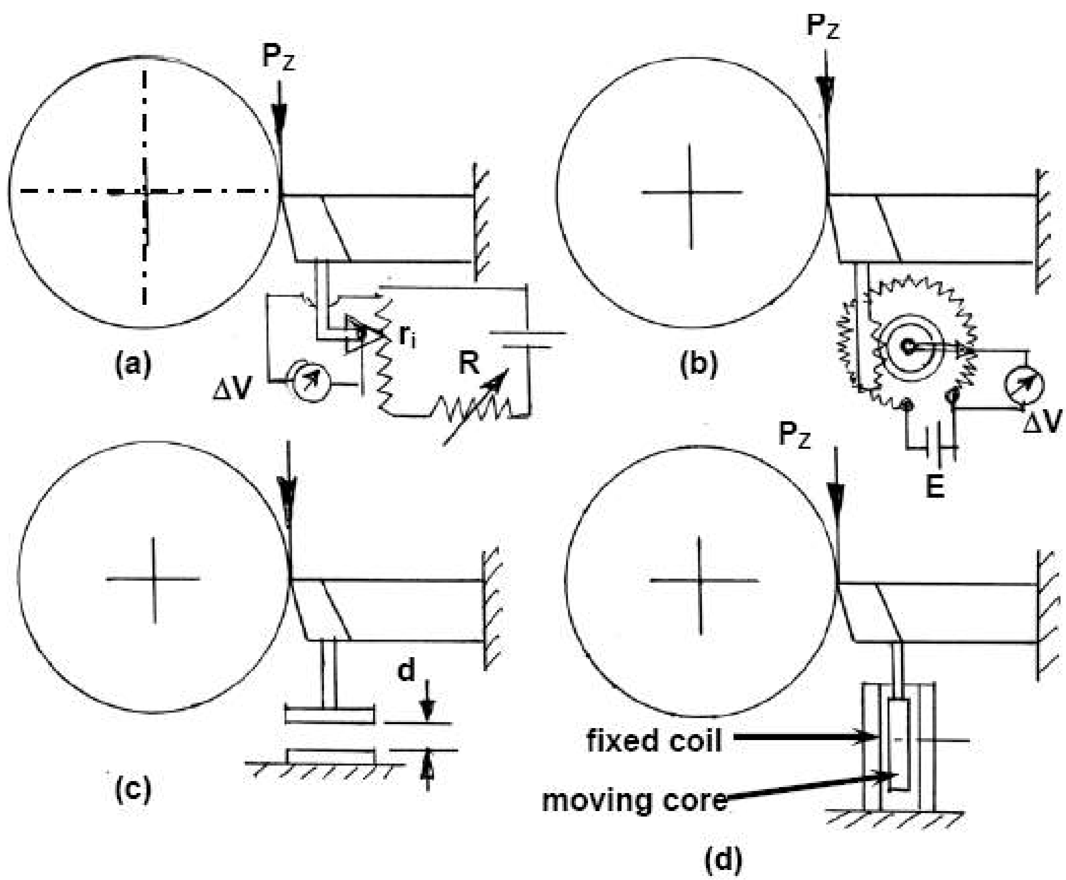


Fig. 10.3 *Electrical transducers working based on deflection measurment (a) linear pot (b) circular pot (c) capacitive pick up (d) LVDT type*

The deflection, δ, can be measured

1. mechanically by dial gauge (mechanical transducer)
2. electrically by using several transducers like; – potentiometer; linear or circular –
3. capacitive pickup – inductive pickup

4. LVDT (Linear Variable Differential Tranformer)

1.28.2 Measuring cutting force by monitoring elastic strain caused by the force.

Increasing deflection, δ enhances sensitivity of the dynamometer but may affect machining accuracy where large value of δ is restricted, the cutting forces are suitably measured by using the change in strain caused by the force. Fig. 10.5 shows the principle of force measurement by measuring strain, ε, which would be proportional with the magnitude of the force, F (say P_Z) as,

$$\varepsilon = \frac{\sigma}{E} = \frac{M/Z}{E} = \frac{P_Z . l}{Z.E} = k_1 P_Z$$

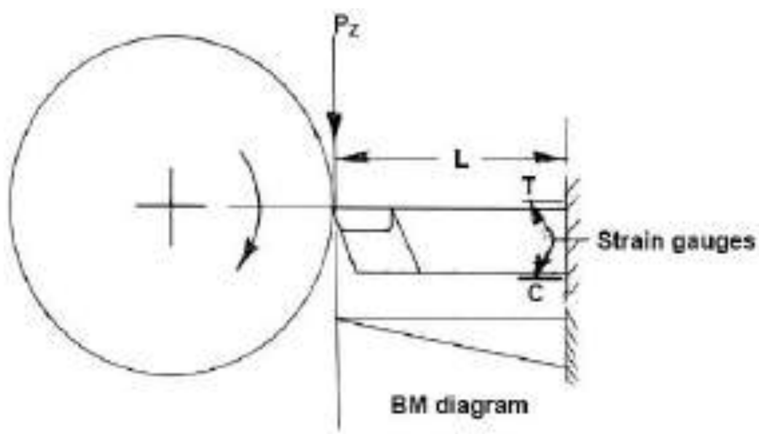
where,

M = bending moment

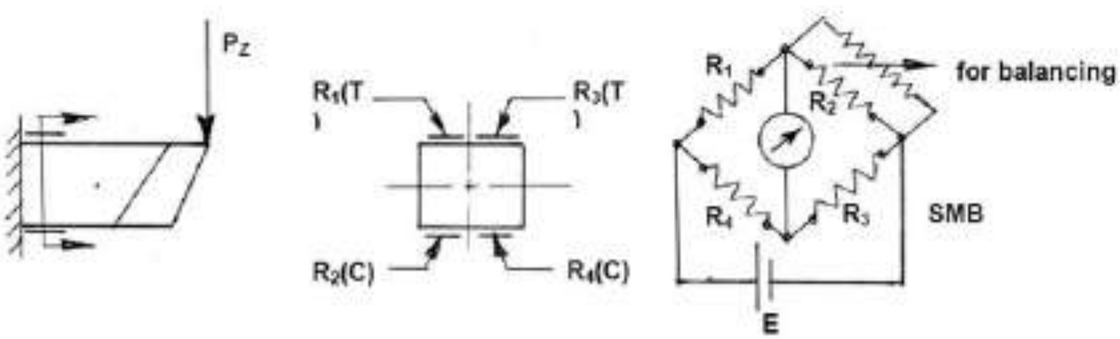
Z = sectional modulus (I/y) of the tool section

I = plane moment of inertia of the plane section

y = distance of the straining surface from the neutral plane of the beam (tool)



Measuring cutting forces by strain gauges



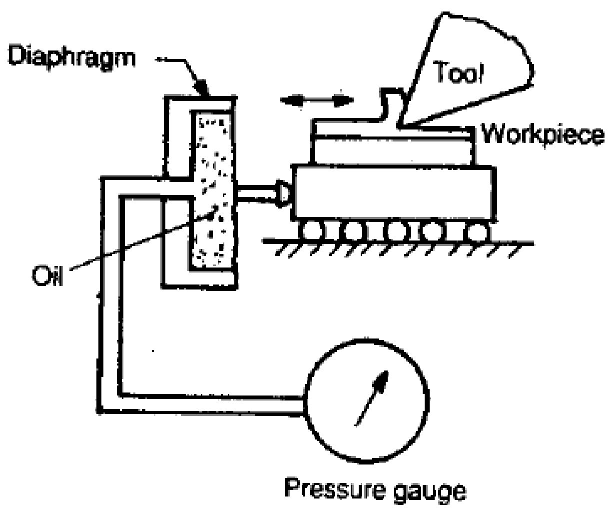
Force measurement by strain gauge based transducer.

The change in resistance of the gauges connected in a wheatstone bridge produces voltage output ΔV, through a strain measuring bridge (SMB) as indicated in Fig. 10.6. Out of the four gauges, R_1, R_2, R_3 and R_4, two are put in tension and two in compression as shown in Fig. 10.6. The output voltage, ΔV, depends displacement of the strain gauge.,

1.28.3 Measuring cutting forces by pressure caused by the force

This type of transducer functions in two ways :

1. the force creates hydraulic or pneumatic pressure (through a diaphragm or piston) which is monitored directly by pressure gauge as indicated in Figure 6.
2. the force causes pressure on a piezoelectric crystal and produces an emf proportional to the force or pressure as indicated in Figure 7



METAL CUTTING PRINCIPLES – PROBLEMS

1. In orthogonal turning of mild steel bar of 65 mm diameter on a lathe a feed of 0.8 mm/rev was used. A continuous chip of 1.4 mm thickness was removed at a rotational speed of 85 rpm of work. Calculate the chip thickness ratio, chip reduction ratio and total length of the chip removed in one minute.

2. In orthogonal cutting of a 50 mm diameter MS bar on a lathe, the following data was obtained:

Cutting force	=	180 N,	Feed force	=	60N,
Rake angle	=	15^0,	Cutting speed	=	100 m/min,
Feed	=	0.2 mm/rev,	Chip thickness	=	0.3 mm

Calculate:

1. The shear plane angle
2. Coefficient of friction
3. Cutting power
4. Chip flow velocity
5. Shear force

3. In an orthogonal turning operation on a lathe, the following observations were obtained

Cutting force	=	120 N	Feed rate	=	0.2 mm/rev
Feed force	=	30 N	Chip /Cutting thickness	=	0.3 mm
Back rake angle	=	15^0	Cutting speed	=	100m/min
Workpiece diameter	=	120 mm	Depth of cut	=	0.4 mm

Calculate:

1. Chip thickness ratio
2. Shear angle
3. Friction angle
4. Coefficient of friction
5. Shear stress
6. Shear strain
7. Strain energy
8. Chip flow velocity

4. A carbide tipped tool of designation 0-15-5-5-8-90-1 mm (ORS) is used to turn a steel workpiece of 50 mm diameter with the following parameter

cutting speed	=	240 m/min	Cutting Feed	=	0.25 mm/rev
Cutting force	=	180 N	Feed force	=	100 N
Chip thickness	=	0.32 mm			

Calculate

1. Shear angle 2. Shear force
3. Normal force acting on shear plane 4. Friction force
5. Coefficient of friction 6. Friction angle
7. Chip flow velocity

5. During orthogonal cutting of MS bar at 210 m/min with a tool of rake angle 12^0, the width of cut and uncut thickness are 1.8 mm and 0.2 mm respectively. If coefficient of friction between the tool and the chip is 0.55 and shear stress is 390 N/mm^2, calculate:

   1. Shear angle 2. Cutting and thrust components of the machining force

6. In an experiment on a MS tube of 200 mm diameter and 3 mm thick, an orthogonal cut was taken with a cutting speed of 80 m/min and 0.15 mm per revolution feed with a cutting tool having back rake angle of -10^0. It was determined that the cutting force = 1500 N, feed force = 400 n, net horse power for cutting was 3 Hp and chip thickness was 0.25 mm

   Calculate the shear strain and strain energy per unit volume

7. During orthogonal cutting process, a chip length of 75 mm was obtained with an uncut chip length of 195 mm and the rake angle used was 20^0 with depth of cut 0.45 mm. The horizontal and vertical components of cutting force were 1950 N and 190 N respectively. Calculate the shear plane angle, chip thickness, friction angle and resultant cutting force.

8. In an orthogonal cutting operation, following observations were made:

Cutting speed	=	25m/min	Width of cut	=	2.5 mm
Feed	=	0.24 mm/rev	Chip thickness	=	0.4 mm
Cutting force	=	1400 N	Thrust force	=	400 N
Tool rake angle	=	5^0			

   Calculate

   1. Shear angle 2. Friction angle
   3. Chip flow velocity 4. Shear strain
   5. Power consumed at tool in Kw

9. In an orthogonal cutting test with a tool of rake angle 8^0, the following observations were made:

   Chip thickness ratio = 0.2

   Horizontal component of the cutting force = 1050 N

Vertical component of the cutting force = 1450 N

From, Merchant's theory, calculate the various components of the cutting force and the coefficient of friction at the chip tool interface.

10. The following data was obtained from an orthogonal cutting test. Rake angle = 20^0, depth of cut = 6 mm, feed rate = 0.25 mm/rev, cutting speed = 0.6 m/s, chip length before cutting = 29.4 mm, vertical cutting force = 1050 N, horizontal cutting force = 630 N, chip length after cutting = 12.9 mm. Using merchant's analysis, calculate

    1. Magnitude of resultant force 2. Shear plane angle
    3. Friction force and friction angle
    4. Various energies consumed

11. The following data from an orthogonal cutting test is available

    a. Rake angle = 15^0
    b. Chip thickness ratio = 0.383
    c. Uncut chip thickness = 0.5 mm
    d. Width of cut = 3 mm
    e. Yield stress of material in shear = 280 N/mm^2
    f. Average coefficient of friction = 0.7

    Determine the normal and tangential forces on the tool face

12. Prove that in orthogonal cutting, the kinetic coefficient of friction (μ) is given by

$$\mu = \frac{F_c \, Sin\alpha \; + F_t \, Cos\alpha}{F_c \; Cos\alpha \; - F_t \, Sin\alpha}$$

13. An orthogonal cutting of steel is done with 10^0 rake tool, with a depth of cut 2 mm and feed rate of 0.20 mm/rev. The cutting speed is 200 m/min. The chip thickness ratio is 0.31. The vertical cutting force is 1200 N and the horizontal cutting force is 650 N. Calculate from Merchant theory the various work done in metal cutting and shear stress.

14. A tool with 18^0 rake angle is making an orthogonal cut, 3 mm wide at a speed of 31 mpm and a feed of 0.25mm/rev. The chip thickness ratio is 0.55, cutting force is 1392 N and feed force is 363 N. Find

    1. Chip thickness 2. Shear plane angle
    3. Coefficient of friction on tool face 4. Shear force on shear plane
    5. Energy consumed in KW min per cubic centimeter of metal removed

15. Following observation were made while machining steel with a tool of 0-1—6-6-75-1mm ORS shape tool.

Spindle speed	=	400 rpm	Work diameter	=	60 mm
Depth of cut	=	2.5 mm	Tool feed rate	=	80 m/min
Cut chip thickness	=	0.40 mm			

Determine

1. Chip thickness ratio
2. Shear plane angle
3. Dynamic shear strain
4. Cross sectional area of the chip
5. Theoretical continuous chip length per minute

16. A seamless tube of 50 mm outside diameter is turned on lathe with a cutting speed of 20 m/min. The tool rake angle is 15^0 and feed rate is 0.2 mm/rev. The length of continuous chip in one revolution measures 80 mm. Calculate :

1. Chip thickness ratio
2. Shear plane angle
3. Shear flow speed
4. Shear strain
5. Shear strain rate

17. While machining a mild steel rod on a lathe, following results were obtained

Width of cut = 2.5 mm   Uncut chip thickness = 0.27 mm

Chip thickness = 0.7 mm   Rake angle = 0 degree

Cutting force = 900 N   Thrust force = Feed force = 450 N

Determine

1. Chip thickness ratio
2. Chip reduction coefficient
3. Shear angle
4. Coefficient of friction
5. Ultimate shear stress
6. Friction angle

18. In an orthogonal cutting operation, the following observations are obtained

1. Cutting speed : 120 m/min
2. Uncut chip thickness: 0.127 mm
3. Rake angle : 10^0
4. Width of cut : 6.35 mm
5. Cutting force : 567 N
6. Thrust force : 227 N
7. Chip thickness : 0.228 mm

Calculate

a. Shear angle
b. Friction angle
c. Shear stress along the shear plane
d. Chip velocity

e. Shear strain   f. Cutting power

19. In an orthogonal cutting operation, the following observations are obtained

1. Cutting speed : 70 m/min   2. Uncut chip thickness: 0.20 mm
3. Rake angle : 15^0 4. depth of cut : 2 mm
5. Cutting force : 200 N   6. Thrust force : 80 N
7. Chip thickness : 0.4 mm

Calculate

a. Chip thickness ratio   b. shear angle
c. Shear force and normal force to shear plane
d. Shear stress and shear strain   e. Cutting power

20. In an orthogonal cutting test with a tool of rake angle 10^0, the following observations were made:

Chip thickness ratio = 0.4
Horizontal component of the cutting force = 1200 N
Vertical component of the cutting force = 1600 N

From, Merchant's theory, calculate

1. Shear plane angle   2. Shear force along the shear plane
3. Normal force to the shear plane   4. Coefficient of friction at the chip tool interface   5. Friction angle
6. Resultant cutting force.

21. Following data was collected from an orthogonal machining test on steel

Cutting speed = 18m/min   Rake angle = 20^0
Clearance angle = 10^0 Width of cut = 3.2 mm
Undeformed chip thickness = 0.1 mm
Deformed chip thickness = 0.25mm
Cutting force in the cutting velocity direction = 800 N
Normal force in a direction normal to the cutting velocity = 500 N

Draw the merchant circle diagram of forces and evaluate

1. shear angle,   2. Friction angle
3. Friction coefficient against chip flow and   4. Friction force on the rake surface

22. During machining of C-25 steel with 0-10-6-6-90-1 mm ORS shaped carbide cutting tool, the following observations have been made:

Depth of cut	=	2 mm	Feed	=	0.2 mm/rev
Speed	=	200 m/min	Tangential cutting force	=	1600 N
Feed thrust force	=	800 N	Chip thickness	=	0.39 mm

Calculate

1. Shear force
2. Friction force
3. Kinetic coefficient of friction
4. Shear strain
5. Shear strain rate
6. Specific power consumption

23. Show that in an orthogonal cutting process with rake angle $\alpha = 0^0$, shear strain is given by $e = \frac{1+k^2}{k}$, where k = Chip reduction factor

24. A pipe 38 mm diameter is being turned on a lathe with a tool having a rake of 33^0 and a feed of 0.15 mm/rev. The length of chip over one revolution of workpiece is 72 mm. The cutting speed is 12.5 m/min. The tangential force is 410 N and feed force is 170 N. Calculate

1. Coefficient of friction
2. Thickness of chip
3. Angle of shear
4. Velocity of shear
5. Velocity of chip along the tool face

25. In orthogonal cutting operation, the feed is 0.10 mm and the chip thickness is 0.25 mm. The cutting force is 1360 N and the feed thrust force is 770N. The rake angle of the tool is $+10^0$. Find

1. The shear angle
2. The size of the force exerted by the tool on the chip
3. The coefficient of friction on the face of the tool
4. The size of the friction force and the normal force on the tool face
5. The size of the shearing force and the normal force on the shearing plane.

26. During machining of an alloy steel with a tool having a tool signature of 0-5-6-6-8-75-1mm ORS, the tangential force and axial force measured by the dynamometer are to be 280 N and 130 N respectively. Calculate the

1. Radial force
2. Frictional force

3. Coefficient of friction at the chip tool interface.

27. An orthogonal cut 2.5 mm wide is made at a speed of 0.5 m/sec and feed of 0.26 mm with a HSS tool having a 20^0 rake angle. The chip thickness ratio is found to be o.58, the cutting force is 1400 N and the feed thrust force is 360 N. Find the

1. Chip Thickness
2. Shear plane angle
3. Resultant force
4. Coefficient of friction
5. Friction force and normal force
6. Specific energy

28. A job of 40 mm in diameter is being turned on a lathe with a tool having a rake angle of 31^0 and feed 0.15 mm/rev. The length of chip over one revolution of workpiece is 76 mm. The cutting speed is 12 m/min. The tangential force is 415 N and feed force is 175 N.

Calculate

1. Coefficient of friction on the rake face
2. Thickness of chip
3. Angle of shear
4. Velocity of shear
5. Velocity of chip along tool face

29. Calculate power required for cutting a steel rod of 50 mm in diameter at 200 rpm. Assume cutting force of 160Kg

30. In an orthogonal cutting operation on a workpiece of width 2.5 mm, the uncut chip thickness was 0.25 mm and the tool rake angle was zero degree. It was observed that the chip thickness was 1.25 mm. The cutting force was measured to be 900 N and the thrust force was found to be 810 N

Find

1. Shear plane angle
2. If the coefficient of friction between the chip and the tool was 0.5, what is the machining constant C_m?

31. The tool life of turning tool obtained was 40 minutes and 25 minutes at cutting speed of 80m/min and 100 m/min respectively. Determine the tool life at 40 m/min and 120 m/min

32. During orthogonal cutting, the following data were recorded .

Back rake angle	=	25^0	Chip thickness	= 0.15 mm/rev
Length of chip	=	100 mm	Width of chip	= 4mm
Width of chip before cut	=	3.5 mm		

Coefficient of friction	=	0.75
Cutting speed	=	250 m/min
Average shear stress	=	$250 N/mm^2$

Determine the power consumption

33. In an experiment on orthogonal cutting a chip length of 85 mm was obtained from an uncut chip length of 202 mm while cutting with a tool of 20^0 rake angle using a depth of 0.5 mm. Determine shear plane angle.

34. During an orthogonal turning operation of C20 steel, the following data were recorded:

Rake angle	=	10^0	Chip Thickness	=	0.48 mm
Width of cut	=	2.0 mm	Feed	=	0.25 mm/rev

Tangential cutting force	=	1200 N
Feed thrust force	=	300 N
Cutting speed	=	2.5 m/s

Find the value of

1. Shear force at the shear plane
2. Find also the kinetic coefficient of friction at the chip tool interface

35. A specimen of 100 mm length along the stroke of shaper is machined with a tool with 15^0 rake angle. The uncut chip thickness is 1.5 mm. if a chip length of 40 mm is obtained during one stroke of machining, find the shear plane and the thickness of cut chip.

36. The end of a pipe was orthogonally cut with a tool of 20^0 rake angle. The cut chip length was 85 mm corresponding to uncut chip length of 202 mm. if the depth of cut was 0.5 mm, find the chip thickness and shear plane angle.

37. In an orthogonal cutting test with a tool rake angle of 10^0, the following observations were made

Chip thickness ratio	=	0.3
Horizontal component of cutting force	=	1290 N
Vertical component of the cutting force	=	1650 N

From Merchant's theory, calculate the various components of the cutting force and the coefficient of friction at the chip tool interface.

38. A metal is being cut orthogonally with a tool with zero rake angles. Show that the rate of heat generation in the shear plane can be expressed as $F_c V_c (1-\mu r)$, where F_c is the principle cutting force, V_c is the cutting speed, μ is the coefficient of friction at the chip tool interface and r is the cutting ratio.

39. When the rake angle is zero during orthogonal cutting, show that

$$\frac{\tau_s}{P_c} = \frac{(1-\mu r)r}{1+r^2}$$

40. The following equation for tool life has been obtained for HSS tool when machining AISI

$$V\,T^{0.13} f^{0.6} d^{0.3} = C$$

A 60 minute tool life was obtained while cutting at V = 40 m/min, f = 0.25 mm/rev and d= 2.0 mm

Determine the effect on the tool life

(i) for a 25% increase in the cutting speed
(ii) for a 25% increase in feed
(iii) for a 25% increase in depth of cut
(iv) if the 25% increase in cutting speed, feed and depth of cut are considered together.

41. The following cutting speed and cutting time observations have been noted in a machining process. Calculate

(v) 1. n and C
(vi) 2. Recommend the cutting speed for a desired tool life of 60 minutes

Cutting speed, V	25 m/min	35 m/min
Cutting Time	90 min	20 min

42. The following equation for tool life has been obtained for HSS tool when machining AISI

$$V\,T^{0.13} f^{0.77} d^{0.37} = C$$

A 60 minute tool life was obtained while cutting at V = 30 m/min, f = 0.30 mm/rev and depth of cut d = 2.5 mm

Determine the effect on the tool life

(i) for a 20% increase in the cutting speed

(ii) for a 20% increase in feed

(iii) for a 20% increase in depth of cut

(iv) if the 20% increase in cutting speed, feed and depth of cut are considered together.

43. If the Taylors tool life constants for a given operation are specified as n=0.5 and C=400, what is the percentage increase in tool life when cutting speed is reduced by half?

44. Tool life tests in turning yield the following data : (1) V= 110m/min, T=20 min, (2) V = 85m/min, T = 40 min. Determine

(a) The n and C values in the Taylor tool life equation

Based on the equation compute

(b) The tool life for a speed of 95m/min and

(c) The speed corresponding to a tool life of 30 min

45. The useful tool life of an HSS tool while machining mild steel at 25 m/min is 5 hours. Calculate the tool life when tool operates at 40 m/min

46. The useful tool life of an HSS tool while machining mild steel at 18 m/min is 3 hours. Calculate the tool life when tool operates at 24 m/min

47. A hollow workpiece of 50 mm diameter and 200 mm long is to be turned over in 4 passes. If the approach length is 20 mm, over travel 10 mm, feed 0.8 mm/rev and cutting speed 30 m/min. Find machining time.

ME18402 – MACHINE TOOLS AND MACHINING PROCESSES

UNIT-1 - IMPORTANT FORMULAE

1.	Chip thickness ratio	$r = \frac{t}{t_c} = \frac{l_c}{l} = \frac{V_c}{V}$
2.	Chip reduction factor or chip reduction ratio	$k = \frac{1}{r} = \frac{t_c}{t} = \frac{l}{l_c} = \frac{V}{V_c}$
3.	Length of uncut chip	$l = \pi D N \ mm/\min$
4.	Shear plane angle	$\phi = \tan^{-1}\left(\frac{r \, Cos\, \alpha}{1 - r \; Sin\alpha}\right)$
5.	Chip flow velocity	$V_c = V r$ m/min
6.	Shear Velocity	$V_s = \frac{V \; Cos\alpha}{Cos(\phi - \alpha)}$ $V_s = V \varepsilon \; Sin\phi$
7.	Shear strain	$\varepsilon = Cot\phi + \tan(\phi - \alpha)$ $\varepsilon = \frac{Cos\alpha}{Sin\phi \; Cos(\phi - \alpha)}$
8.	Material Removal Rate (MRR)	MRR = b t V mm³/min
9.	Shear Force	$F_s = F_C \; Cos\, \phi - F_F \; Sin\phi$
10.	Normal force at shear plane / Backing up force	$N_s = F_F \; Cos\phi + F_C \; Sin\phi$
11.	Friction Force	$F = F_F \; Cos\alpha + F_C \; Sin\alpha$
12.	Normal force at tool plane / Force exerted by the tool on the chip	$N = F_C \; Cos\, \alpha - F_F \; Sin\, \alpha$
13.	Coefficient of Friction	$\mu = \frac{F}{N} = \frac{F_F \; Cos\alpha + F_C \; Sin\alpha}{F_C \; Cos\, \alpha - F_F \; Sin\, \alpha}$ $\mu = \tan \beta$
14.	Friction angle	$\beta = \tan^{-1} \mu = \tan^{-1}\left(\frac{F_F \; Cos\alpha + F_C \; Sin\alpha}{F_C \; Cos\, \alpha - F_F \; Sin\, \alpha}\right)$

15.	Shear Stress	$\tau_s = \frac{F_s}{A_s} = \left(\frac{F_C \ Cos\,\phi - F_F \ Sin\phi}{b \ t}\right)\sin\phi$
16.	Shear strain rate	$\overset{*.}{\varepsilon} = \frac{Shear \ \ strain}{feed \ in \ mm/rev}$
17.	Cutting Power	$P_C = Cutting \ Force \ x \ Cutting Velocity$ $P_C = \frac{F_C \ V}{1000 \ x \ 60}, kW$
18.	Specific Cutting Energy	$E = \frac{Cutting \ Power}{MRR} \ kW \ \min / \ mm^3$ $E = \frac{F_C}{b \ t} \ N/mm^2$
19	Specific Shear Energy	$E_S = \frac{F_S \ x \, V_S}{b \, t \, V} \ kW \ \min / \ mm^3$ $E_S = \frac{F_S \ x \, V_S}{b \, t \, V} \ N/mm^2$
20	Specific Friction Energy	$E_F = \frac{F \ x \, V_C}{b \, t \, V} \ kW \ \min / \ mm^3$ $E_F = \frac{F \ x \, V_C}{b \, t \, V} \ N/mm^2$
21.	Strain energy	U = Shear Stress x Shear strain
22	Taylors Tool life equation	$V T^n = C$
23	Taylors Modified Tool life equation	$V \ T^n \ f^x \ d^y = C$

Conventional Lathes

2.1 Lathe:

The lathe is one of the oldest machine tools. The main functions of lathe is to remove metal from a piece of work to give it the required shape and size.

2.2 Classification of Lathes:

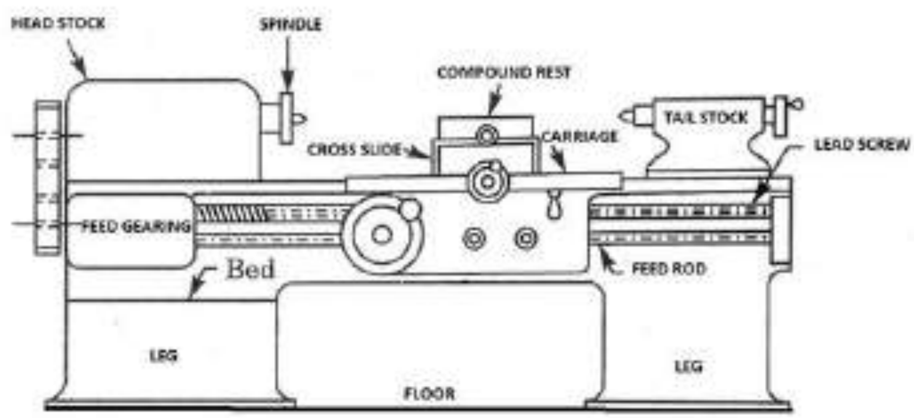
Types generally used are,

1. Speed lathe
2. Engine Lathe
3. Bench Lathe
4. Tool room Lathe
5. Capstan & Turret Lathe
6. Special purpose Lathe
7. Automatic Lathe

2.2 Lathe Construction:

Basic parts:

1. Bed
2. Headstock
3. Tailstock
4. Carriage
5. Feed Mechanism
6. Screw cutting mechanism



Bed:

Forms the base of the Machine. Headstock and tailstock are located at either end of the bed and the carriage rests over the bed. Bed should be rigid to prevent deflection while machining and should absorb vibration. To resist twisting stress developed due to the cutting forces diagonal ribbing will be provided. Cast iron alloyed with nickel is mostly used as bed material

Headstock:

Provides the mechanical means of rotating the work at multiple speeds. Comprises ahollow spindle and mechanism for driving and altering the spindle speed headstock casting houses all the parts. Spindle is made of carbon or nickel chrome steel.

Tailstock:

Located on the inner ways at the right-hand end of the bed. This unit supports other end of the workpiece when machined between centres. Also can be used to hold tools such as drill, reamer and tap for necessary operation.

Carriage:

Carriage support, move and control the cutting tool. It consists of the following parts such as saddle, cross-slide, compound rest, tool post and apron.

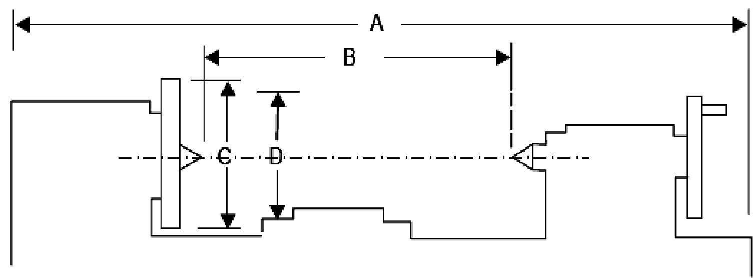
Feed mechanism: Explained in sections 2.4.3-2.4.6

Screw cutting mechanism: Explained in 2.4.7

2.3 Lathe Specifications

Lathe is specified as follows,

1. The height of the centres measured from the lathe bed
2. The swing diameter over bed
3. The length between centres
4. The swing diameter over carriage
5. The maximum bar diameter
6. The length of bed



A - Length of bed.
B - Distance between centres.
C - Diameter of the work that can be turned over the ways.
D - Diameter of the work that can be turned over the cross slide.

Fig. 21.7 Specifications of a lathe

2.4 Lathe Mechanisms

The following are different mechanisms available in atypical lathe machine for the head stock spindle drive.

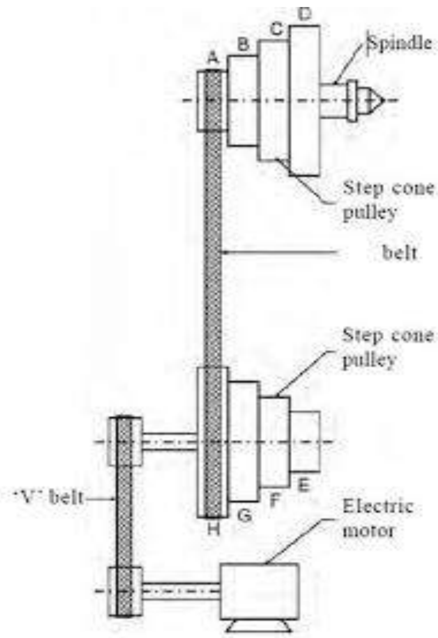
1. Belt drive on cone pulley
2. By gear drive
3. By variable speed motor

The relative movement of the tool with respect to the work is called as feed. Lathe also has mechanisms for providing various types of feed such as longitudinal, cross and Angular feed. To provide these feeds the feed mechanism has the following units,

1. End of bed gearing
2. Feed gear box
3. Feed rod & Lead screw
4. Apron Mechanism

Spindle drive mechanisms

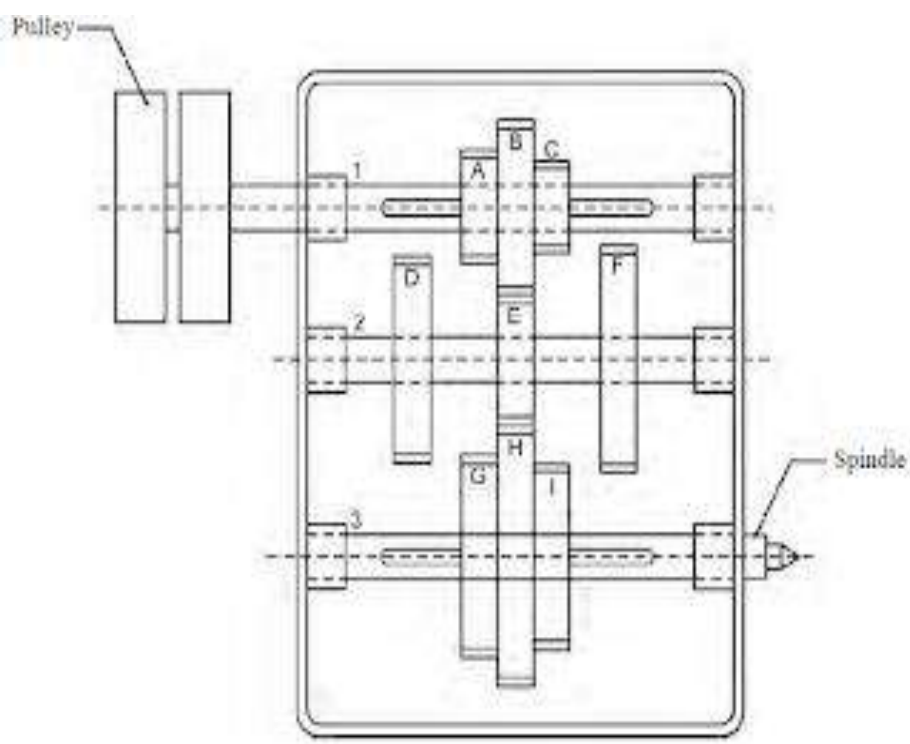
2.4.1 Belt drive on cone pulley



In this mechanism, there will be a step cone pulley coupled with the motor shaft. Normally, number of steps in the cone pulley will be four. One more cone pulley of the same size will be loosely placed over the spindle shaft to the back side of the head stock. There is a big gear called bull gear keyed to the spindle shaft. The motion from the motor will be transmitted to the stepcone pulley through belts, connecting both cone pulleys. A lock pin is used to connect the bull gear to the step cone pulley at the back side of the spindle shaft to transmit the rotation to the head stock spindle.

By varying the spindle step connections four different speeds are possible. Four different slow speeds can also be achieved using a back-gear mechanism, which connects a pair of gears to the gear attached with the step cone pulley and to the bull gear.

2.4.2 Gear Drive

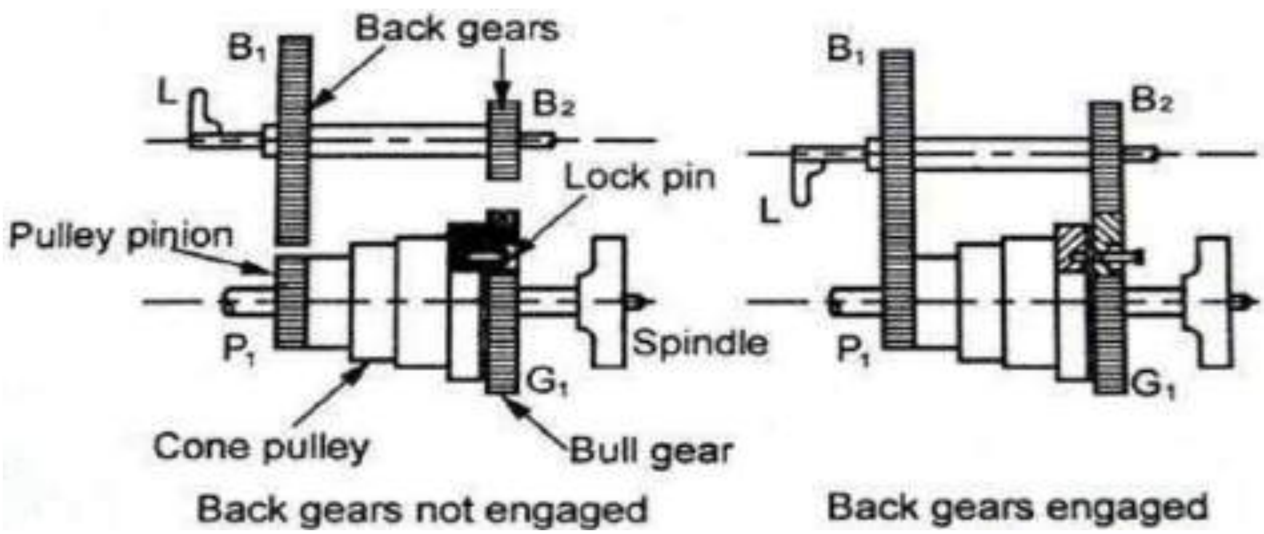


Lathe which is equipped with All Geared Headstock is easy to operate as there is no manual slip of Belt from one step to any other. All Geared Lathe have three or more than three shafts mounted within & it is shown in the picture below. The gears are equipped to obtain different speeds, which can be more than 18 spindle speeds. The middle shaft has got three gears D,E & F as a single unit, thus will rotate at similar speed. The spline shaft also have three gears mounted as A,B & C on it. The three gears can made to slide with the help of the lever mounted in front of the Headstock. This movement enables the Gear A to contact with the Gear D or with other gears. This sliding movement on the either of the shafts with the help of the levers, different gears get in touch with each other & these gears are connected to the main Spindle. So, with the rotation of the Electric Motor Pulley, these gears are also in drive and rotates the spindle, producing different RPM. All Speed Changes are done with the series of levers on the front of the Headstock & are like shifting gears in a Car.

Back gear Mechanism in lathe

Back gear arrangement is used for reducing the spindle speed, which is necessary for thread cutting and knurling. *The back gear arrangement is shown in Fig.2.3.* There is one stepped cone pulley in the lathe spindle. This pulley can freely rotate on the spindle. A pinion gear P1 is connected to small end of the cone pulley. P1 will rotate when cone pulley rotates. Bull gear G1 is keyed to lathe spindle such that the spindle will rotate when Gear G1 rotates. Speed changes can be obtained by changing the flat belt on the steps. A bull gear G1 may be locked or unlocked with this cone pulley by

a lock pin. There are two back gears B1 and B2 on a back shaft. It is operated by means of hand lever L; back gears B1 and B2 can be engaged or disengaged with G1 and P1. For getting direct speed, back gear is not engaged. The step cone pulley is locked with the main spindle by using the lock pin. The flat belt is changed for different steps. Thus three or four ranges of speed can be obtained directly.

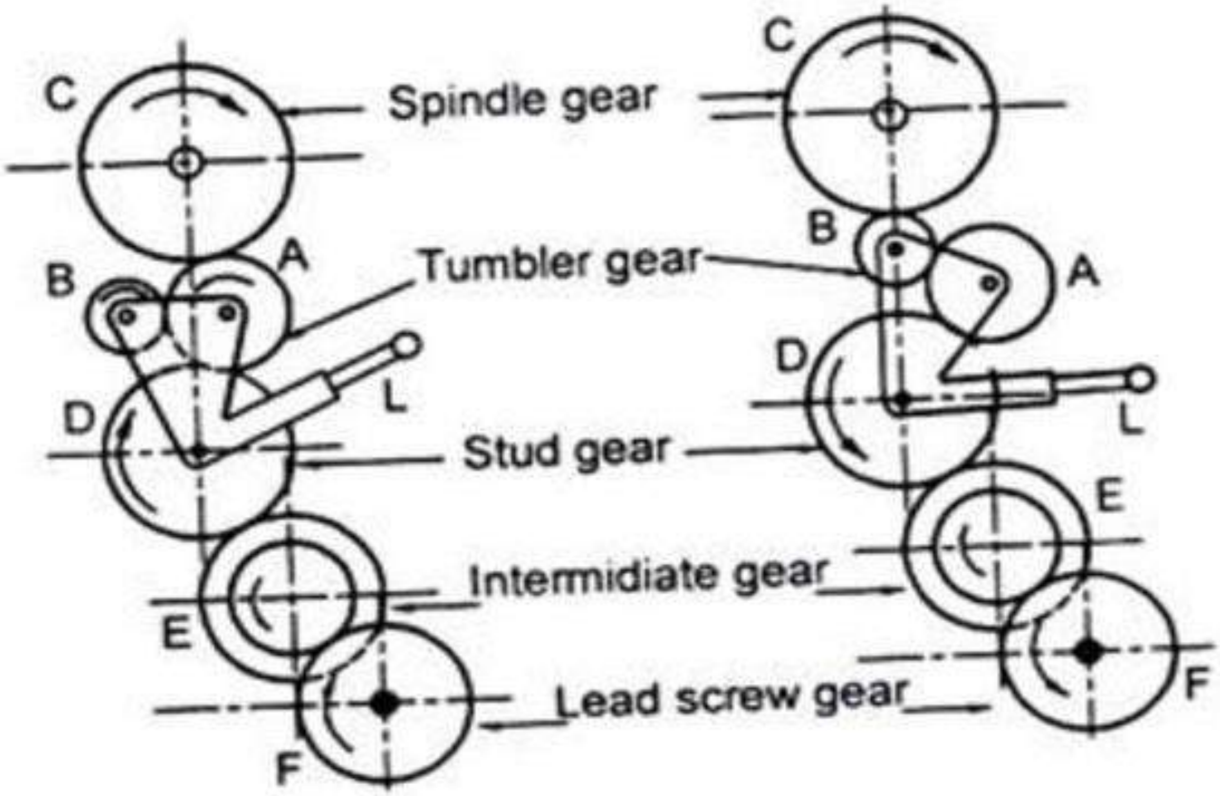


Feed Mechanisms

2.4.3 End of bed gearing

This gearing is used to transmit the drive to the feed rod & lead screw. This is done by following two types of Mechanisms.

1. Tumbler gear mechanisms
2. Bevel gear feed reversing mechanism



Tumbler gear mechanism is used to change the direction of carriage movement. This mechanism consists of a set of gears used to connect the work spindle to the lead screw by using a pair of gears called tumbler gears. The rotation of the lead screw can be made clockwise or anticlockwise which in turn makes the carriage to move either toward the head stock or away from the head stock. The limitation of tumbler gear mechanism is that the setup is non-rigid. Hence method called bevel gear feed reversing mechanism is often used which is clutch operated.

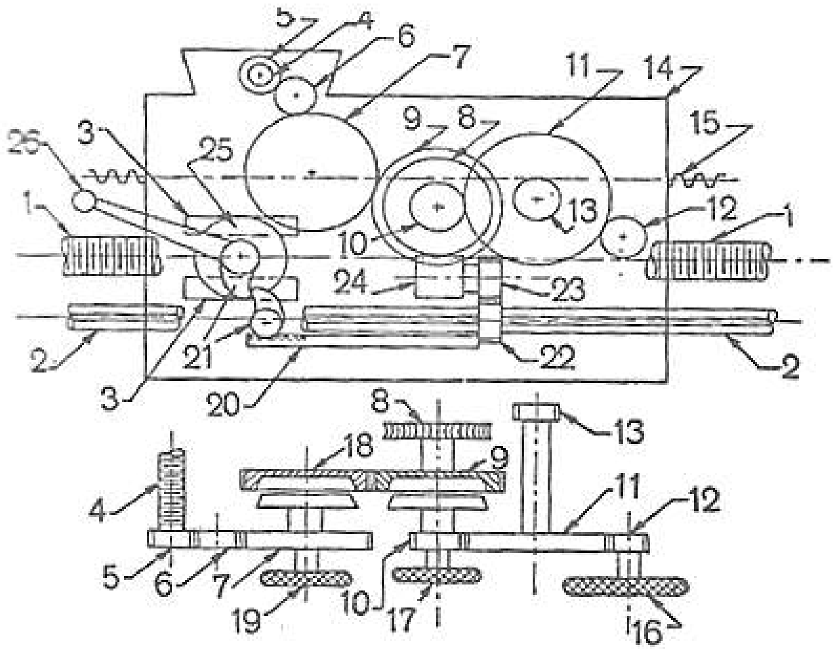
2.4.4 Feed gear box

Feed gear box is provided to change the feed rate as desired. The drive of the feed mechanism may be powered by a separate electric motor or from the head stock spindle through a gear, chain or belt transmission.

2.4.5 Feed rod and lead screw

Feed rod is a long shaft that has a keyway extending from the feed box across and in front of the bed, while lead screw is a long-threaded shaft and is used only when thread cutting is required. Feed rod is used for providing automatic feed movement for carriage. Lead screw is used for thread cutting operation so as to make the tool to move according to the pitch of the thread.

2.4.6 Apron Mechanism



Apron mechanism in the carriage provides manual and automatic longitudinal and cross feed. Hand wheel 16 will give necessary rotation to the pinion 13 through gear 12 & 11 to achieve manual longitudinal feed. Manual cross feed is achieved by rotating the hand wheel 19. Automatic longitudinal feed is achieved by engaging the cone clutch 9 using knob 17, thereby transmitting rotation from feed rod to the pinion 13 through worm gear 8. Similarly knob 19 will be used to get automatic cross feed by engaging cone clutch 18 to rotate the gear 5 through gears 7 & 6.

2.4.7 Thread cutting mechanism

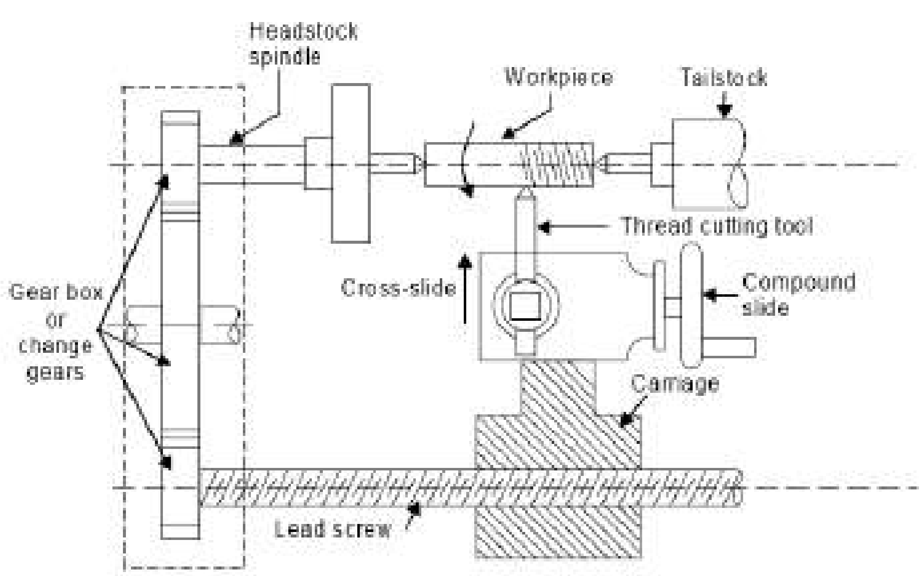


Fig. 21.14 Thread cutting

The principle is to produce helical groove on a cylindrical or conical surface by feeding the tool longitudinally by setting a relative motion between the rotating workpiece and the longitudinally travelling tool. The lead screw used has a definite pitch to be made in the workpiece. The gears can be selected as per the following relation.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the lead screw}}{\text{Pitch of the workpiece}}$$

A relative motion is set between the Rotation of the lead screw and the rotation of the head stock (workpiece) by engaging the half nut of the carriage. Both simple and the compound gear train arrangements can be made depending on the pitch required in the workpiece. Change gears set comprised with gears having teeth from 20 up to 120 in steps of 5, and a gear with 127 number of teeth are used.

2.4.7.1 Problems in thread cutting

1. Pitch of the lead screw is 5mm and the pitch of the thread to be cut is 1.25 mm. Find the change gears.

   $$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the lead screw}}{\text{Pitch of the workpiece}}$$

   =1.25/5=1/4=20/80 or 30/120 or 25/100

   Hence the change gears can be any of the above options

2. Pitch of the lead screw = 7mm, pitch of the thread to be cut = 1.5 mm. Find the change gears.

   $$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the lead screw}}{\text{Pitch of the workpiece}}$$

   =1.5/7=1.5x2/7x2=30*20/70*40. Hence a compound gear arrangement, of gears with 30 teeth and 20 teeth as driver gears and 70 teeth and 40 teeth as driven gears are required.

2.5 Lathe operations

For performing the various machining operations in a lathe, the job is being supported and driven by anyone of the following methods.

1. Job is held and driven by chuck with the other end supported on the tail stock centre.
2. Job is held between centers and driven by carriers and catch plates.
3. Job is held on a mandrel, which is supported between centers and driven by carriers and catch plates.
4. Job is held and driven by a chuck or a faceplate or an angle plate.

The above methods for holding the job can be classified under two headings namely job held between centers and job held by a chuck or any other fixture. The various important lathe operations are depicted through Fig. The operations performed in a lathe can be understood by three major categories

(a) Operations, which can be performed in a lathe either by holding the work piece between centers or by a chuck are:

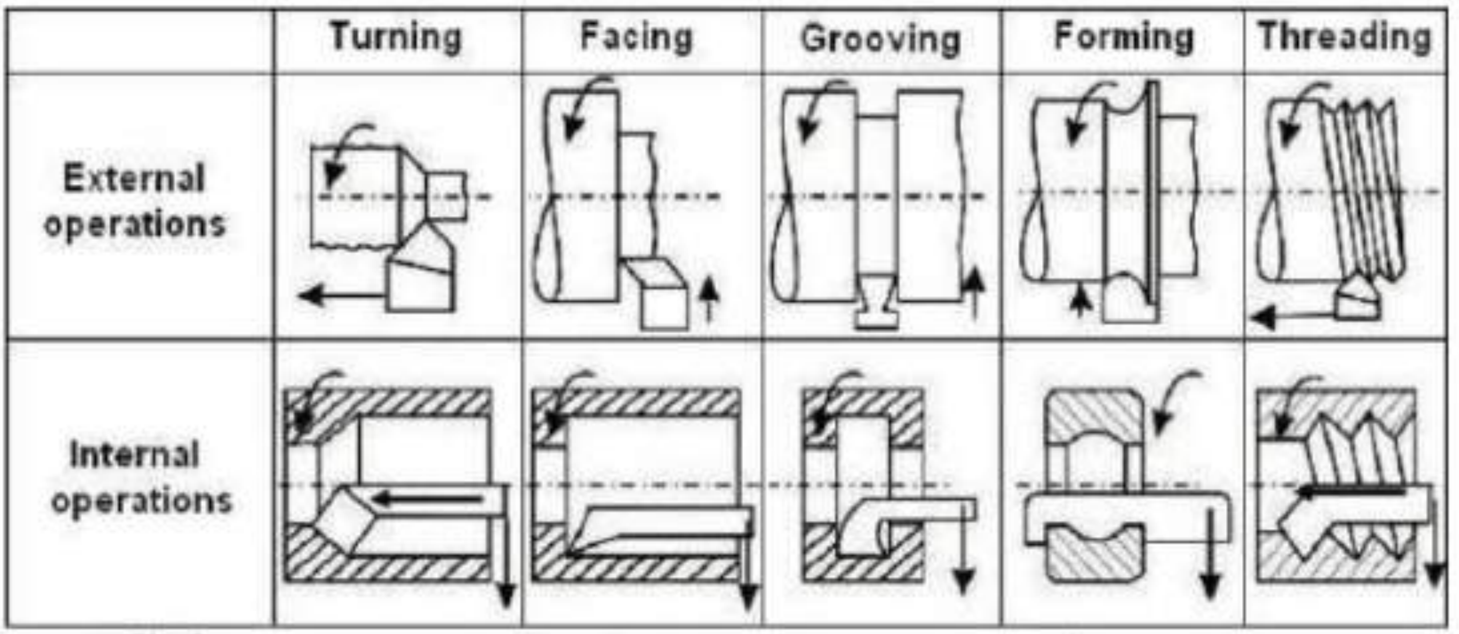
1. Straight turning
2. Shoulder turning
3. Taper turning
4. Chamfering
5. Eccentric turning
6. Thread cutting
7. Facing
8. Forming
9. Filing
10. Polishing
11. Grooving
12. Knurling
13. Spinning
14. Spring winding

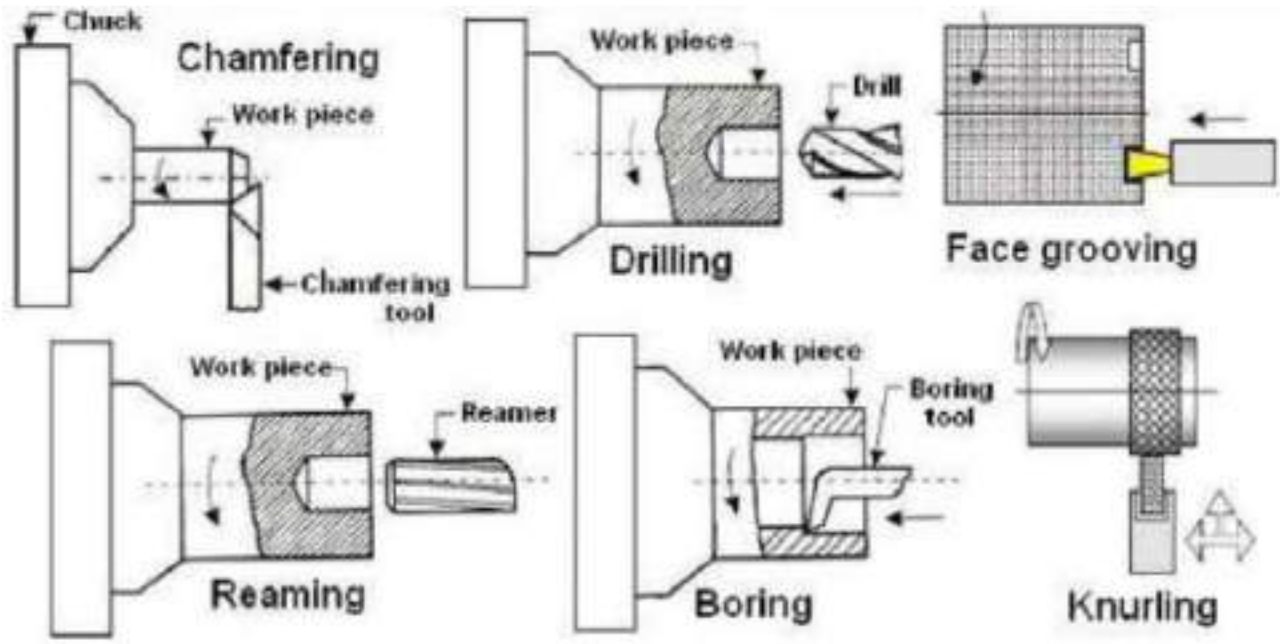
(b) Operations which are performed by holding the work by a chuck or a face plate or an angle plate are:

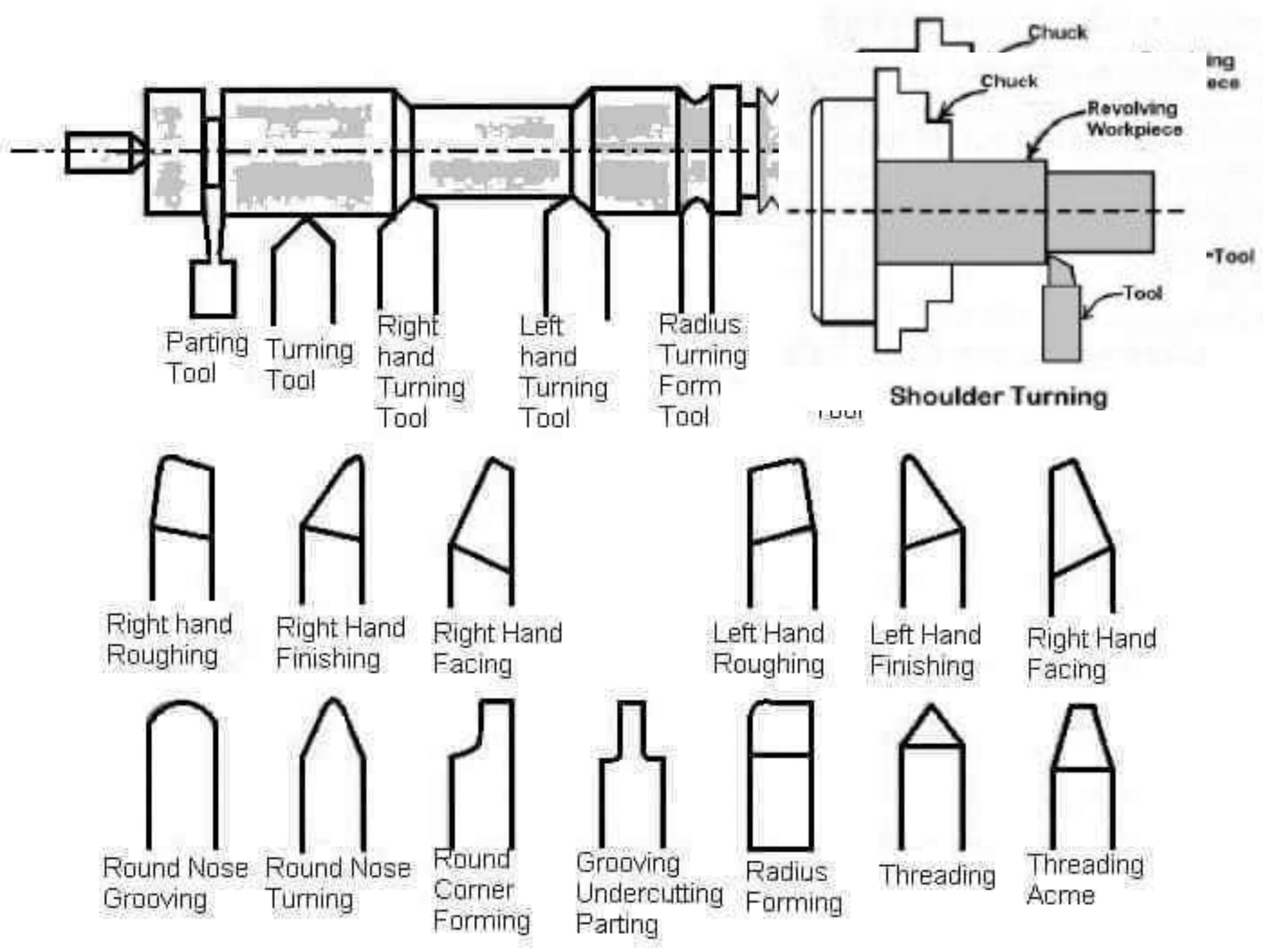
1. Undercutting
2. Parting-off
3. Internal thread cutting
4. Drilling
5. Reaming
6. Boring
7. Counter boring
8. Taper boring
9. Tapping

(c) Operations which are performed by using special lathe attachments are:

1. Milling   2. Grinding







Lathe Machine Cutting Tools

1. Operations Done by Holding Workpiece Between Centres

Turning:

It is the most common type of operation in all lathe machine operations. Turning is the operation of removing the excess material from the workpiece to produce a cylindrical surface to the desired length.

The job held between the centre or a chuck and rotating at a required speed. The tool moves in a longitudinal direction to give the feed towards the headstock with proper depth of cut. The surface finish is very good.

1. Straight Turning:

The workpiece is held on the chuck and it is made to rotate about the axis, and the tool is fed parallel to the lathe axis. The straight turning produces a cylindrical surface by removing excess metal from the workpiece.

2. Rough Turning:

It is the process of removal of excess material from the workpiece in minimum time by applying high rate feed and heavy depth of cut. in rough turning the average depth of cut 2mm to 4mm can be given and feed is from 0.3 to 1.5mm per revolution of the work.

3. Shoulder Turning:

When a workpiece has different diameters and is to be turned, the surface forming steps from one diameter to the other is called the shoulder, and machining this part of the workpiece is called shoulder turning.

Eccentric turning:

When a cylindrical surface has two separate axis of rotation, with the first axis, is offset to the other axis then such a workpiece is machined by the operation called eccentric turning. Here three sets of centre holes are drilled.

By holding the workpiece at these three centres the machining operation for each of the surface can be completed.

Taper Turning:

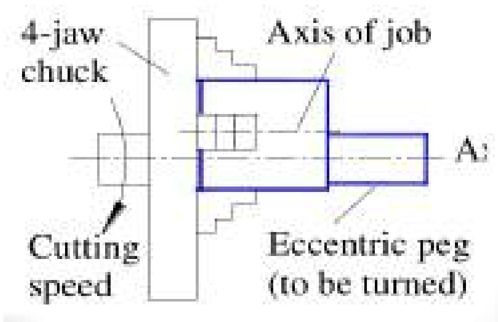
- A "taper" is the uniform increase or decrease in the diameter of the workpiece and measured along with its length.
- Taper turning means to produce a conical shape by a gradual reduction in diameter from a cylindrical workpiece.

The amount of taper in the workpiece is usually specified on the basis of the difference in diameter of the taper to its length. It is known as **a conicity** and it is indicated by the letter K.

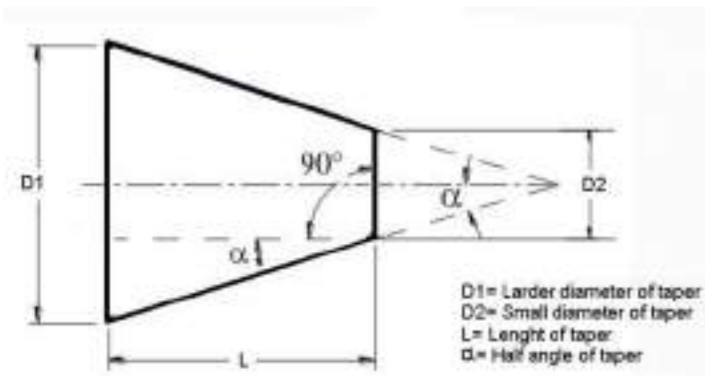
It has the formula K = D-d / 1 to produce the taper on the workpiece.

- D = Larger diameter of taper.
- d = Small diameter of taper.

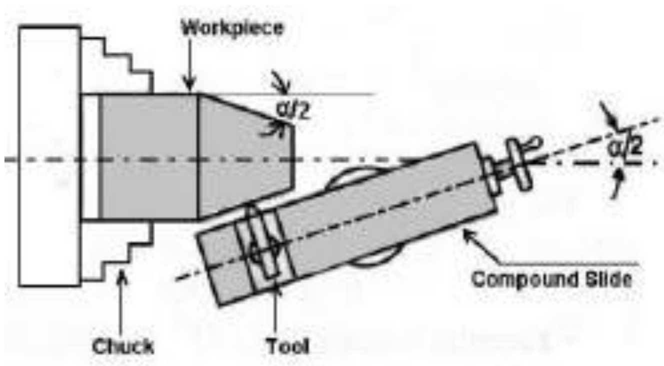
In the case of a lathe, the taper on a given workpiece is obtained by tuning the job and feeding the tool at an angle to produce a gradual increase or decrease in the diameter of the workpiece.



- The two important types of tapers are,
  - "Morse taper" here, the angle is very small and varies from 1.4 to 1.5°.
  - "Metric taper" is available in seven standard sizes with standard taper angles.
- **Methods of taper turning,**
  - Form tool method
  - Combined feeds method
  - Compound rest method or swivelling compound rest method
  - Tailstock set over method
  - Taper turning attachment method



1. Form tool method



Here the taper length obtain is equal to the width of the form tool. To obtain the required size of the taper the form tool is fed slowly straight into the workpiece by operating the cross slide perpendicular to the lathe axis.

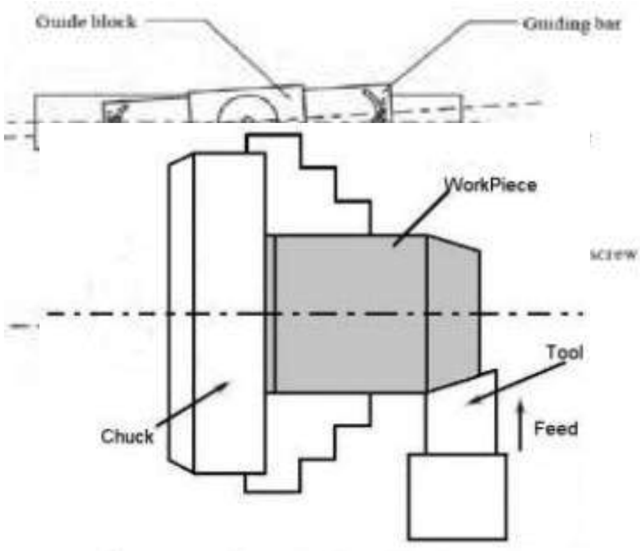
This is the simplest method of taper turning. It is limited to obtain small taper length such as chamfering the side of the workpiece. The method is done at a faster rate.

2. Combined feeds method

The combined feed is made with the movement of a tool in longitudinal and lateral direction simultaneously while moving the workpiece.

- Taper: $$\tan \alpha = \frac{D_1 - D_2}{2L}$$

The taper, which we are going to obtain, is equal to the resultant to the magnitude of the longitudinal and lateral feeds. Changing the feeds rates in both directions can change the direction and the taper angle.



Taper Turning Using Form Tools.

3. Compound rest swivel method

Here the workpiece rotates and the cutting tool is fed at an angle by swivelled compound rest. The base of the compound rest is graduated in degrees.

The taper angle is the angle at which the compound rest to be rotated is calculated by using the formula tanα = (D-d) / 21, where, D= bigger diameter, d = smaller diameter, l = length of the workpiece.

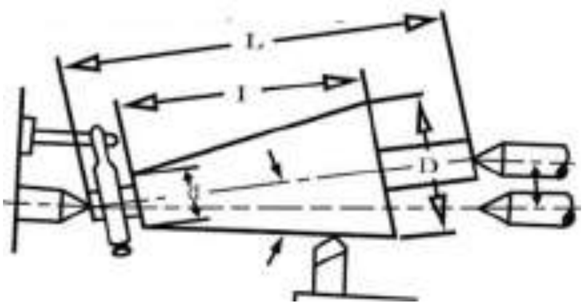
Compound rest can be swivelled to the required angle α. Once the compound rest is set to a particular angle then the tool is moved by compound rest and wheel.

4. Taper turning attachment method

- This method is similar to the compound rest method.
- Here the job or workpiece rotates and the tool is fed at the taper angle α.
- In this, arrangement, which has guide block graduated in degrees, with the help of this the block can be required taper angle to the lathe axis.

- The taper angle is calculated similarly to the compound rest method using the formula: tanα = D-d / 21.

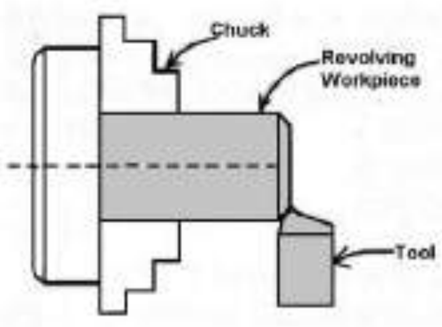
Advantages of taper turning attachment:



- Internal tapers can be obtained accurately.
- large size tapers can be easily obtained.
- Once the attachment is set the taper turning operation can do at a faster rate.
- By setting the taper angle to ‘zero’ we can carry out plain turning.

Disadvantages of taper turning attachment:

- It requires additional mounting facilities.
- Fitting and removing attachment consume more time.
- The attachment has to take large forces.



Chamfering

5. ***Tailstock set over method:***

Here the workpiece on the job is tilted at the required taper angle. The tool is fed parallel to the axis.
The tilting of the workpiece or the job to the required taper angle is achieved by the movement of the tailstock with the help of tailstock set over the screw. This method is useful for small tapers.

Set over = $\frac{D-d}{2l} \; x \; L$

l = length of taper

L = Length of the workpiece

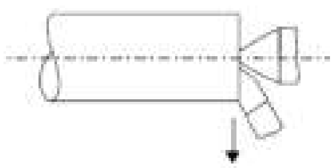
Facing:

It is an operation of reducing the length of the workpiece by feeding the perpendicular to the lathe axis. This operation of reducing a flat surface on the end of the workpiece. For this operation, regular turning tool or facing tool may use. The cutting edge of the tool should set to the same height as the centre of the workpiece.

- Facing consist of 2 operations
  - Roughing: Here the depth of cut is 1.3mm
  - Finishing: Here the depth of cut is 0.2-0.1mm.

Chamfering operation:

It is the operation of getting a bevelled surface at the edge of a cylindrical workpiece. This operation is done in case of bolt ends and shaft ends.
Chamfering helps to avoid damage to the sharp edges and protect the operation getting hurt during other operations.



Facing workpiece on centres

Chamfering on bolt helps to screw the nut easily.

Knurling operation:

It is an operation of obtaining a diamond shape on the workpiece for the gripping purpose. This is done to provide a better gripping surface when operated by hands. It is done using a knurling tool. The tool consists of a set of hardened steel roller, and it is held rigidly on the toolpost.

Knurling is done at the lowest speed available on a lathe. It is done on the handles and also in case of ends of gauges. The feed varies from 1 to 2 mm per revolution. Two or three cuts may be necessary to give the full impression.

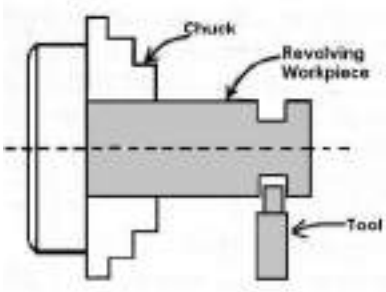
Filling:

It is the finishing operation performed after turning. This is done on a lathe to remove burrs, sharp corners, and feed marks on a workpiece and also to bring it to the size by removing the very small amount of metal.

The operation consists of passing a flat single-cut file over the workpiece which revolves at a high speed. The speed is usually twice that of turning.

Polishing:

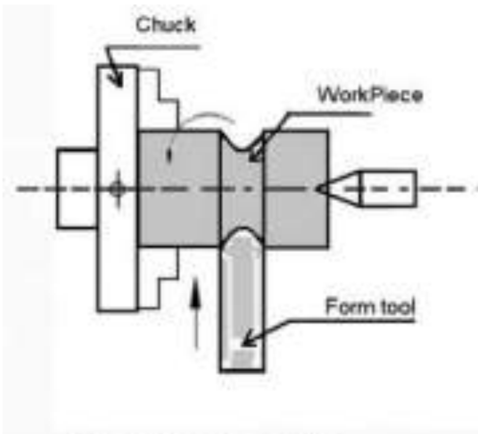
This operation is performed after filing to improve the surface quality of the workpiece. Polishing with successively finer grades of emery cloth after filing results in a very smooth, bright surface. The lathe is run at high speeds from 1500 to 1800m per min, and oil is used on the emery cloth.



Grooving

Grooving:

It is the process of reducing the diameter of a workpiece over a very narrow surface. It is done by a groove tool. A grooving tool is similar to the parting-off tool. It is often done at the end of a thread or adjacent to a shoulder to leave a small margin.



Forming Operation

Spinning:

It is the process of forming a thin sheet of metal by revolving the job at high speed and pressing it against a headstock spindle. Support is also given from the tailstock end.

Forming:

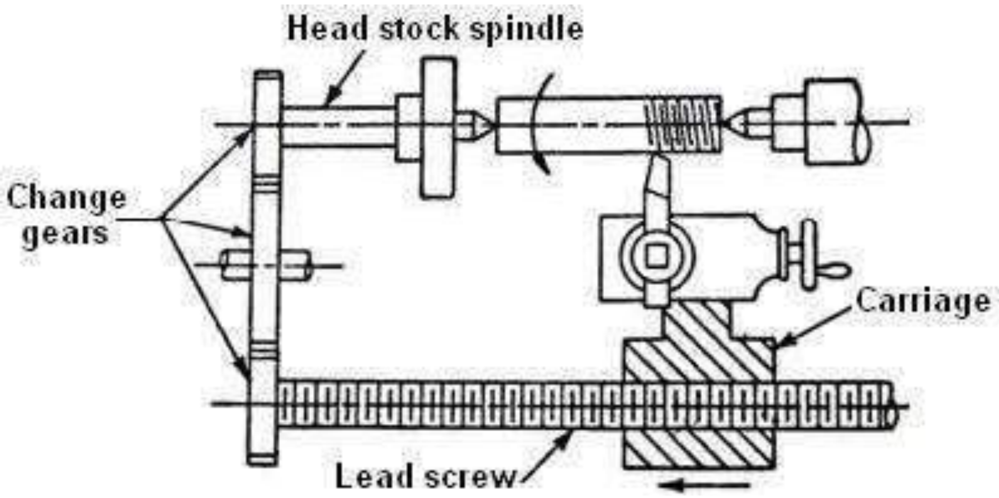
It is the process of turning a convex, concave or of any irregular shape. Form-turning may be accomplished by the following method:

1. Using a forming tool.
2. Combining cross and longitudinal feed.
3. Tracing or copying a template.

Forming tools are not supposed to remove much of the material and is used mainly for finishing formed surfaces. Generally, two types of forming tools are used straight and circular. Straight type is used for wider surface and the circular type for narrow surfaces.

T HREAD CUTTING OPERATION

Thread cutting is one of the most important operations performed in a centre lathe. It is possible to cut both external and internal threads with the help of threading tools. There are a large number of thread forms that can be machined in a centre lathe such as Whitworth, ACME, ISO metric, etc. The principle of thread cutting is to produce a helical groove on a cylindrical or conical surface by feeding the tool longitudinally when the job is revolved between centres or by a chuck (for external threads) and by a chuck (for internal threads). The longitudinal feed should be equal to the pitch of the thread to be cut per revolution of the workpiece.



The lead screw of the lathe has a definite pitch. The saddle receives its traversing motion through the lead screw. Therefore a definite ratio between the longitudinal feed and rotation of the headstock spindle should be found out so that the relative speeds of rotation of the work and the lead screw will result in the cutting of a thread of the desired pitch. This is effect by change gears arranged between the spindle and the lead screw or by the change gear mechanism or feed gear box used in a modern lathe. Thread cutting on a centre lathe is a slow process, but it is the only process of producing square threads, as other methods develop interference on the helix. *Fig.2.37 illustrates the principle of thread cutting.*

The gears can be selected as per the following relation.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the lead screw}}{\text{Pitch of the workpiece}}$$

A relative motion is set between the Rotation of the lead screw and the rotation of the head stock (workpiece) by engaging the half nut of the carriage. Both simple and the compound gear train arrangements can be made depending on the pitch required in the workpiece. Change gears set comprised with gears having teeth from 20 up to 120 in steps of 5, and a gear with 127 number of teeth are used.

Thread cutting procedure

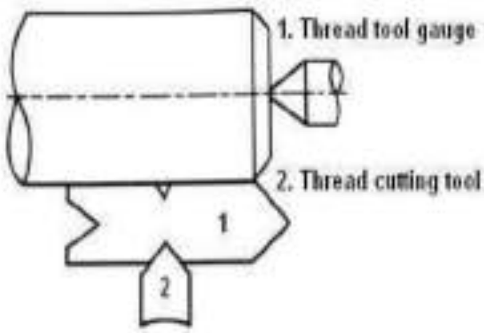
1. The work piece should be rotated in anticlockwise direction when viewed from the tail stock end.

2. The excess material is removed from the workpiece to make its diameter equal to the major diameter of the screw thread to be generated.

3. Change gears of correct size are fitted to the end of the bed between the spindle and the lead screw.

4. The thread cutting tool is selected such that the shape or form of the cutting edge is of the same

form as the thread to be generated. In a metric thread, the included angle of the cutting edge should be ground exactly 60^0.



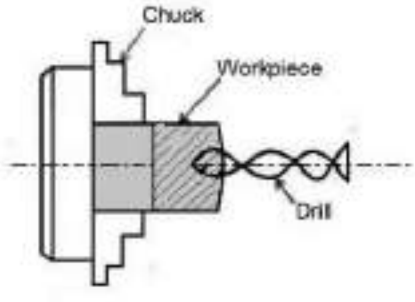
5. A thread tool gauge or a centre gauge is used against the turned surface of the workpiece to check the form of the cutting edge so that each face may be equally inclined to the centre line of the workpiece. *This is illustrated in Fig. 2.38.*

2. Operations Done By Holding The Work By A Chuck

Lathe machine operations performed by holding the work by a chuck or a faceplate or an angle plate are:

Drilling:

Drilling is the operation of producing a cylindrical hole in a workpiece. It is done by a rotating tool, the rotating side of the cutter, known as drilling drill. In this operation, The workpiece is revolving in a chuck or a faceplate and the drill is held in the tailstock drill holder or drill chuck.

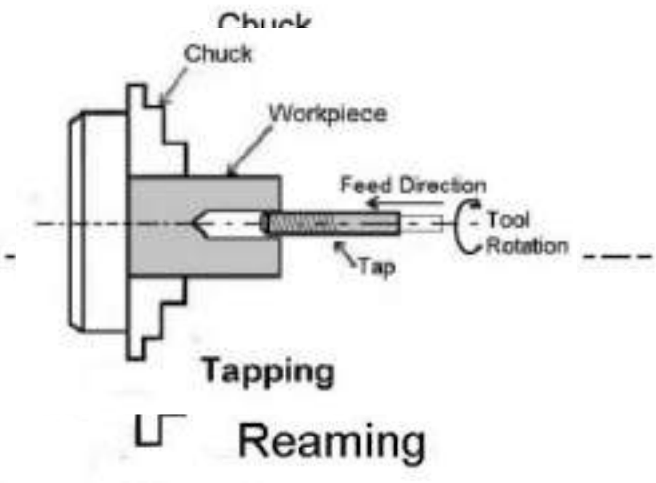


Drilling

The feeding is adopted is affected by the movement of the tailstock spindle. This method is adopted for the drilling regular-shaped workpiece.

Reaming:

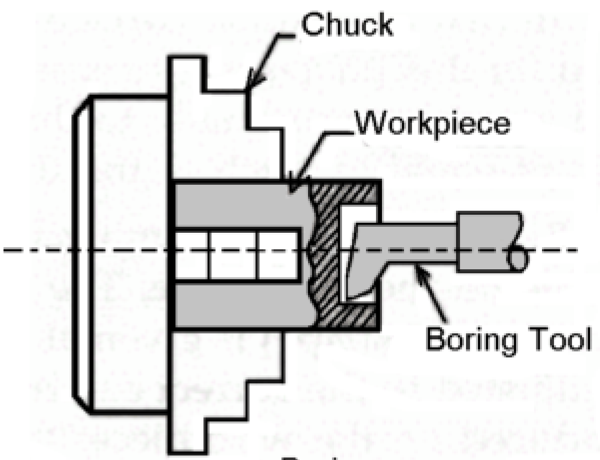
Reaming is the operation of finishing and sizing a hole which has been already drilled or bored. The tool is used is called the reamer, which has multi-plate cutting edges.



The reamer is held on the tailstock spindle, either directly or through a drill chuck and is held stationary while the work is revolved at a very slow speed.

Boring:

Boring is the operation of enlarging the hole which is already drilled, punched or forged. It cannot produce a hole. Boring is similar to the external turning operation and can be performed in a lathe. In this operation, the workpiece is revolved in a chuck or a faceplate and the tools which are fitted to the tool post is fed into the work.



Boring

It consists of a boring bar having a single-point cutting tool which enlarges the hole. It also corrects out of roundness of a hole. This method adopted for boring small-sized works only. The speed of this process is slow.

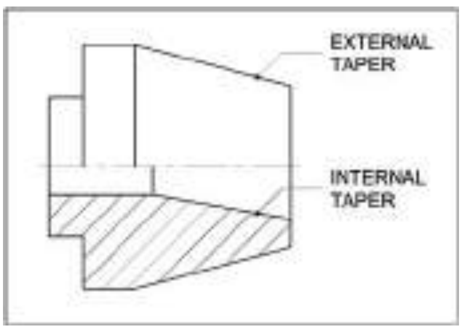
Counterboring:

Counterboring is the operation of enlarging the end of the hole through a certain distance. It is similar to a shoulder work in external turning.

The operation is similar to boring and plain boring tools or a counterbore may be used. The tool is used called a counterbore. The speed is slightly less than drilling.

Taper Boring:

The principle of turning a tapered hole is similar to the external taper turning operation and is completed by rotating the work on a chuck or a faceplate. The feeding tool is at an angle to the axis of rotation of the workpiece.



A boring tool is mounted on the tool post and by swivelling the compound slide to the desired angle, a short taper hole is machined by hand feeding.

Tapping:

Tapping is the operation of cutting internal threads of small diameter using a multipoint cutting tool called the tap. In a lathe, the work is mounted on a chuck or on a faceplate and revolved at a very slow speed. A tap of required size held on a special fixture is mounted on the tailstock spindle.

Undercutting:

Undercutting is similar to grooving operation when performed inside a hole. It is the process of boring a groove or a large hole at a fixed distance from the end of a hole.

This is similar to the boring operation, except that a square nose parting is used. Undercutting is done at the end of an internal thread or a counterbore to provide clearance for the tool or any part.

2.6 Machining time and power estimation – Problems

1. Calculate power required for cutting a steel rod of 50 mm in diameter at 200 rpm.

   Cutting force generated is 160kg.

   Fc= Cutting force in N = 160 *9.81 = 1569.6 N

   Vc=cutting velocity in m/s

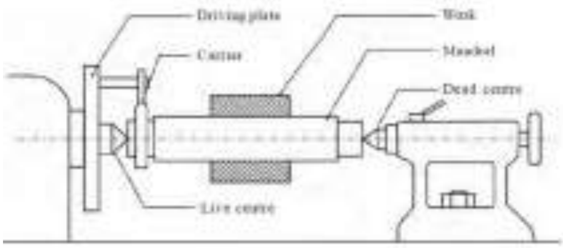
   D=Diameter in m = 0.05

   N = Number of revolution = 200 rpm

   Cutting power = Fc*Vc

   $=Fc*\frac{\pi DN}{60} = 1569.6 * \frac{\pi*0.05*200}{60} = 821.4$

   Nm/s = 821.4 W



2. Determine the cutting speed in machining a workpiece of 200 mm diameter rotating at a speed of 100 rpm. Also calculate machining time if workpiece length is 0.5 m and feed 0.45 mm/rev.

   V = cutting speed = $\frac{\pi DN}{1000} = \frac{\pi*200*100}{1000}$ = 62.83 mpm

   T= Machining time = $\frac{l}{f x N}$, where l= length of workpiece in mm and f=feed in mm/rev

   $= \frac{500}{0.45x100}$ = 11.11 minutes

2.8 WORK HOLDING DEVICES ON A LATHE

As we know the lathe is one of the oldest and highly important machine tool. There is vast number of applications of this machine tool. So for facilitating the easy machining of the work piece it should be held tightly and securely. For this purpose many types of accessories are being used for facilitating easy holding of the work piece.

Some of the work holding devices are

1) Carriers and catch plates

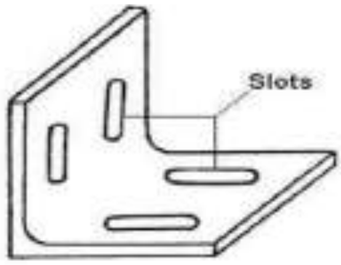
2) Face plates

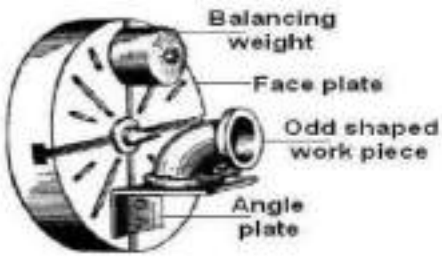
3) Angle plates

4) Mandrels

5) Rests

2.8.1. Carriers and catch plates





These are in general used for driving the work piece when it is held in between two centers namely head stock and tail stock. Carriers are also called as the driving dogs. These are attached to the work piece by the help of setscrews. Where as the catch plates are pinned to the headstock.

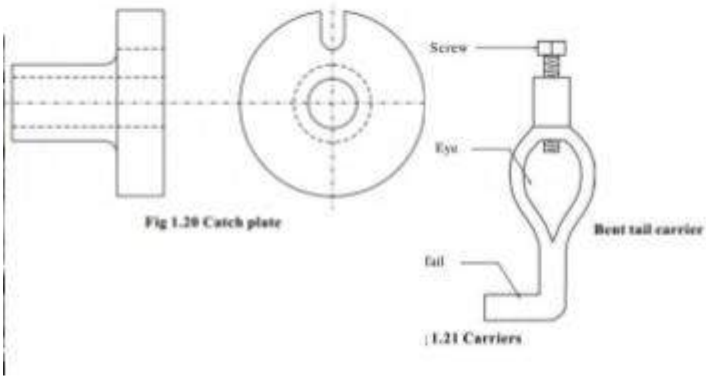
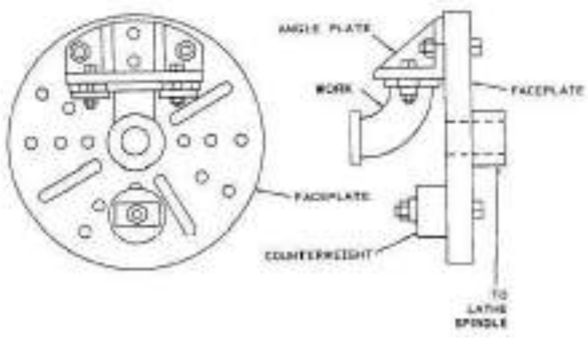


Fig 1.20 Catch plate

1.21 Carriers

2.8.2 Face plates

Faceplates are used for holding those work pieces, which cannot be held both by centers and by chucks. The construction of the faceplates is very simple. It consists of a center bore and plain and radial slots through the plate for facilitating the holding of the work piece. The central bore has a radius equal to that of the radius of the spindle of the lathe. And the plain and radial slots provide a healthy platform for holding the jobs by using T-bolts and clamps.



2.8.3 Angle plates

These are used along with faceplates for maintaining the given work piece horizontal i.e. perpendicular to the tool used. Angle plates consist of two faces, which are highly machined, and these also have the provision of holes for the easy clamping of the work piece to it.

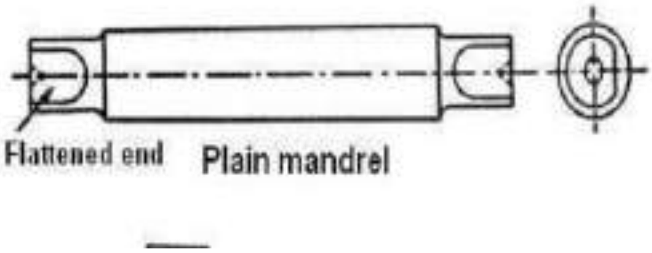
2.8.4 Mandrels

This type of work holding devices are employed for holding previously drilled or bored hole so as to facilitate effective outer surface machining. The work is loaded over the mandrels between the centers. The ends of the mandrels are made slightly smaller than the original diameter. This is done for effective gripping of the mandrel in the chuck or any other holding device. In general the material used for the manufacturing of the mandrels is plain carbon steel. Various types of mandrels are in usage. Various types of mandrels are

1) Plain mandrels
2) Step mandrels
3) Collar mandrels
4) Screwed mandrels
5) Cone mandrels
6) Gang mandrels
7) Expansion mandrels

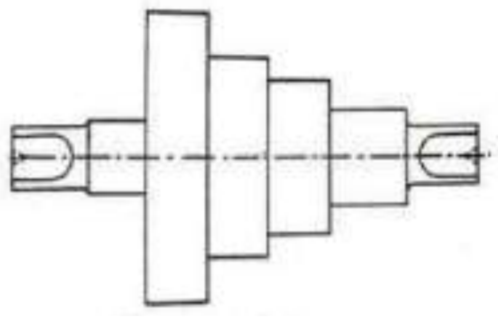
Plain mandrels

This type of mandrels finds a numerous number of applications in shops where identical pieces are to be generated. The body of these mandrels has generally a tapered shape. The difference in the tapped diameter is of 1 to 2 mm and the length varies between 55mm to 430mm. the tapper is provided for facilitating high end gripping for holding the work piece.



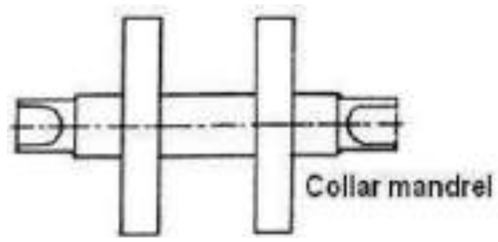
Step mandrel

A special type of mandrel, which facilitates faster processing's by holding various sized jobs with out replacing the mandrel. This type of mandrels finds applications in repair shops and generally used for turning collars, washers and odd sized jobs.



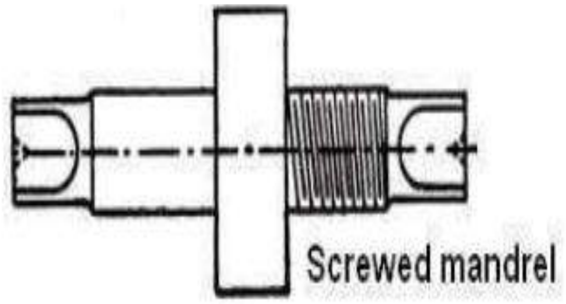
Collar mandrel

A collar mandrel has a collar arrangement, which is fixed and may have size larger than that of the 100mm. this type of construction is a type of optimizing the material being used.



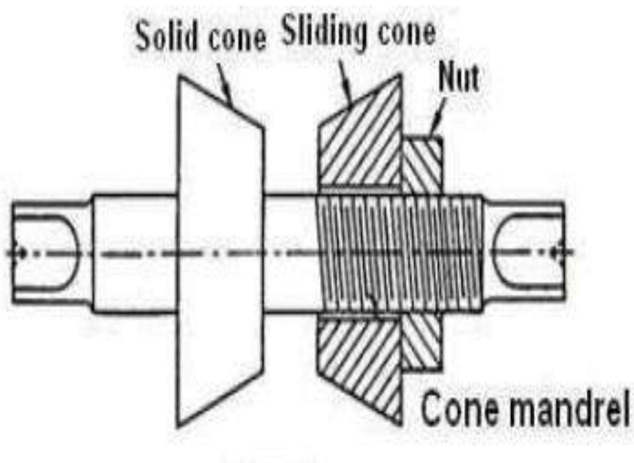
Screwed mandrels

These mandrels have a thread cut engraved on one side along with a collar. Such type of mandrels is called as screwed mandrels. Screwed mandrels are used when work pieces having internal threads are to be machined. The size of the threads to be engraved on the screwed mandrels depends on the type of work piece, which is going to fit over it.



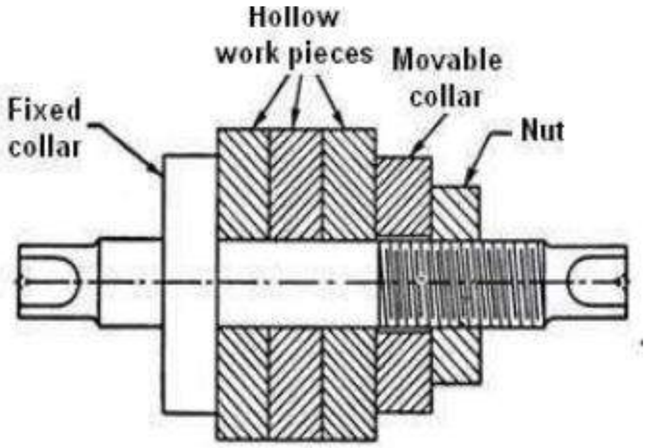
Cone mandrel

Cone mandrels have a cone shaped piece attached at one of the mandrel. This type of arrangement allows the mandrel to handle a variety of work pieces having a varying internal cross sectional diameter. The work piece is held tightly by fixing a nut at the other end of fixing the work piece to the mandrel. Too tight fitting of the work piece over the cone may damage the internal surface finish of the work piece along with the damage to the cone shape of the mandrel.



Gang mandrel

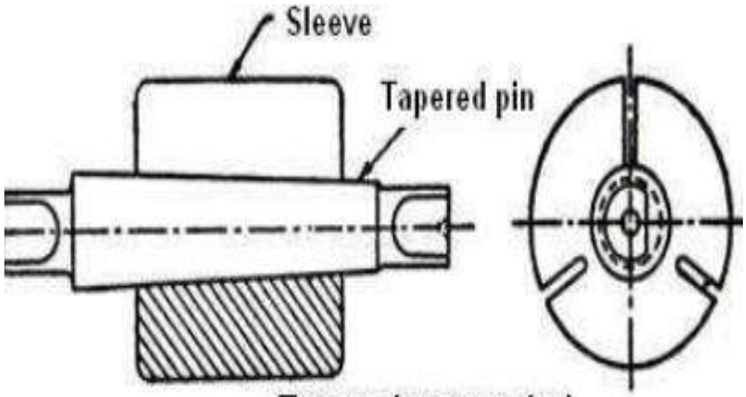
Gang mandrel is generated by some of the optimizers so as to reduce the material. This type of mandrel can facilitate machining for work pieces of various diameters. The gang mandrels consist of a fixed collar at one end and removable mandrels at the other end which is fixed by the help of the threads engraved both on the mandrel and also on the internal surface of a hallow mandrel. This mandrel can be used for machining various diameter pieces by just removing and fixing various collars over the thread. The friction between the walls of collar and sides of work piece is enough to hold the work piece tightly and hence facilitating is a high end machining.



Gang mandrel

Expansion mandrel

This is a special type of mandrel, which has a central tapered pin. Over this tapered pin a sleeve is arranged when this sleeve is moved over it form one corner to other the size increases or decreases. This type of mandrels is best used when a varying diameter pieces are to be hold without much difficulty.



Expansion mandrel

Chucks

A chuck is a work holding device. It is used for holding work over a lathe machine, which is having large length and small diameter, and also for jobs, which are unable to mount on between the centers i.e., head stock and tail stock centers. A chuck is also employed when a non-axis symmetrical object is to be mounted over the lathe. The chucks are most commonly used work holding devices. These are fixed directly to the spindle of the lathe by means of screws and a back plate. In general there are various types of chuck, which have their own importance and unique applications

Types of chucks

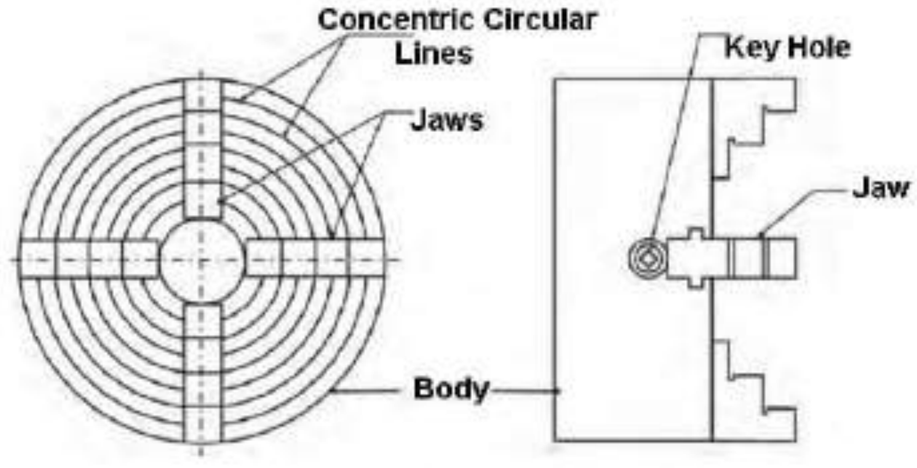
- Four jaw chuck or independent chuck
- Three jaw chuck or self-centering chuck
- Air or hydraulic operated chuck
- Magnetic chuck

• Collet chuck

• Combination chuck

• Drill chuck

Four jaws or independent chuck



Four Jaw Lathe Chuck

As the name indicates this chuck has four jaws for holding the work each jaw is independent to move.

Advantages

a. Any type of work piece can be held easily

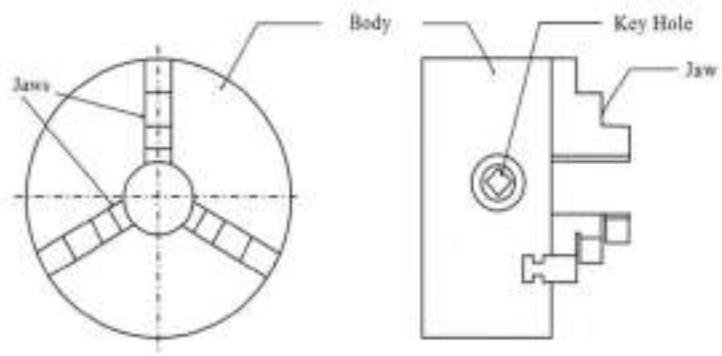
b. High grip is possible as four jaws are employed

Disadvantages

a. Centering is a little difficult process

b. Semi skilled labor are required

Three jaw chuck (or) self centering chuck



Three Jaw Chuck

Three jaw chuck or self-centering chuck is the chuck, which has three jaws for holding the work piece, and if one jaw is moved then all the other jaws also move by same distance. This mechanism is obtained by engaging pinions which meshes with the teeth cut on the under surface of the chuck. This chuck is also called *universal chuck*

Advantages

a. Centering process is not necessary as all the jaws moves at a time

b. Consumes less time for mounting and un mounting the work piece.

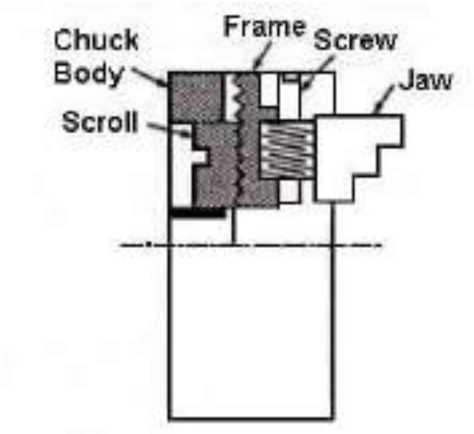
Disadvantages

a. Grip is not so high as only three jobs are employed

b. All type of jobs cannot be mounted on these chucks, as they are useful in machining only axis
symmetrical objects

Combinational chuck

As the name indicates it is a combination of both three-jaw chuck and four-jaw chuck. It acts as both self-centering and independent chuck. This is obtained by engaging a scroll disc at the backside of jaws.

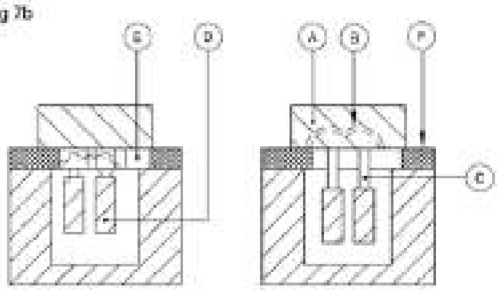


Combination Chuck

Magnetic chuck

MAGNETIC CHUCK

Fig 7b



INSIDE DETAILS OF MAGNETIC CHUCK

A WORKPIECE
B MAGNETIC FIELD
C CONNECTION
D MAGNETS
E KEEPERS
F FACE PLATE

In this chuck magnetic force is used for holding the work piece.

Advantages

• Work pieces in which damage due to the jaws of chucks is not tolerated can be effectively machined by using this chuck

• If the work to be held is of very small size and cannot be hold by the above three chucks then this chuck can be used

Disadvantages

a. All type of jobs cannot be held

b. Size of the job affects the efficiency of the holding

c. All type of materials cannot be held as it works on magnetic property only magnetic materials can

be held.

Air or hydraulic operated chuck

This is also called as pneumatic chuck as it is being operated by hydraulic or air energy. This chuck holds the work piece due to the pressure generated in the cylinder.

Drill chuck

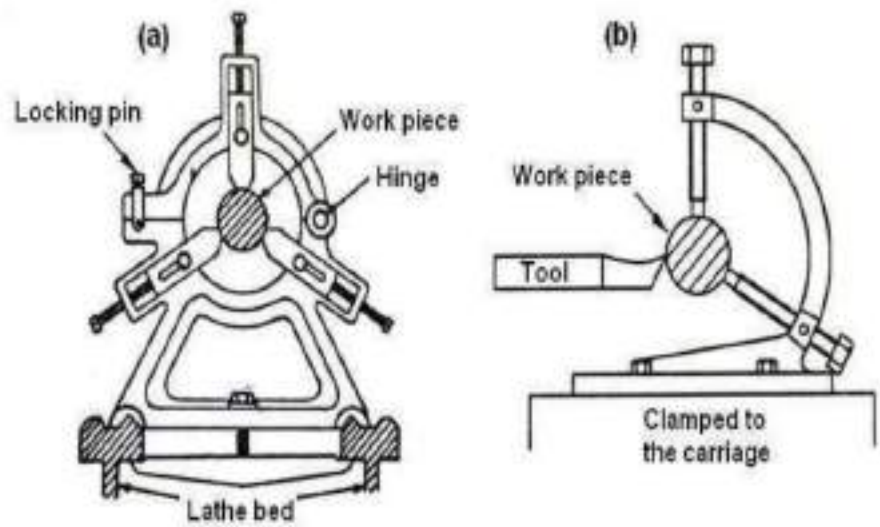
This chuck is used for holding the tool in the drilling machines. This also plays a role in holding taper shanks on a lathe for easy machining of the job on which holes are to be drilled.

Rests

Rest is a work holding device, which is used to hold the work piece when the work piece of very long length are to be held. In general when a long piece is to be held it may have directly held then there arises deflection in the work piece due its own weight. So to prevent the deflection in the work piece rests of various types are used. Some of the rests being used are

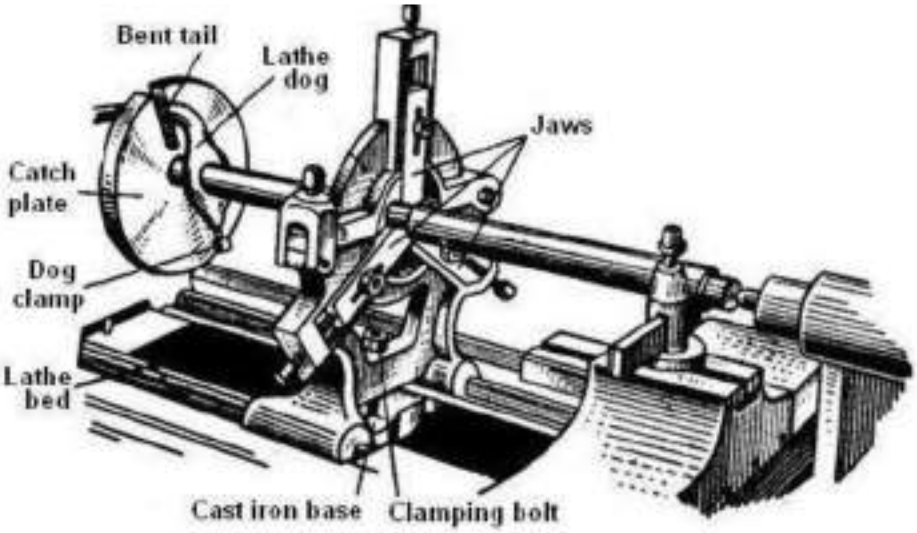
1) Steady rest

2) Follower rest



Steady rests and follower rests were developed to remedy the lathe turning deflection problem. Steady rests and follower rests hold a long workpiece steady during turning. Steady rests are mounted to the lathe bed and do not move with the lathe. They ensure concentricity, but limit the length of the supporting cut. Also, to achieve the best results in turning, vibration must be kept to a minimum. Vibration is problematic for any turning, but it is particularly problematic when turning long spindles, or hollowing out deep vessels. To reduce vibration, a steady rest provides needed support. Typically, steady rests are selected to provide support for longer cuts. Both steady rests and follower rests are used with cylindrical parts and round stock. They are also used to keep the workpiece from wobbling and to ensure that a drilled hole will be concentric with the outside diameter (OD) of the part.

Follower rests are so named because they attach to the saddle (the lathe component that holds the tool) and move along with or "follow" the lathe. Some long slender shafts that tend to whip and spring while they are being machined require the use of a follower. The follower rest is fastened to the carriage and moves with the cutting tool. The follower rest is often used when long, flexible shafts are threaded. At the completion of each threading cut, care must be taken to remove any burrs that may have formed to prevent them from causing the work to move out of alignment.



CAPSTAN & TURRET LATHE: INTRODUCTION

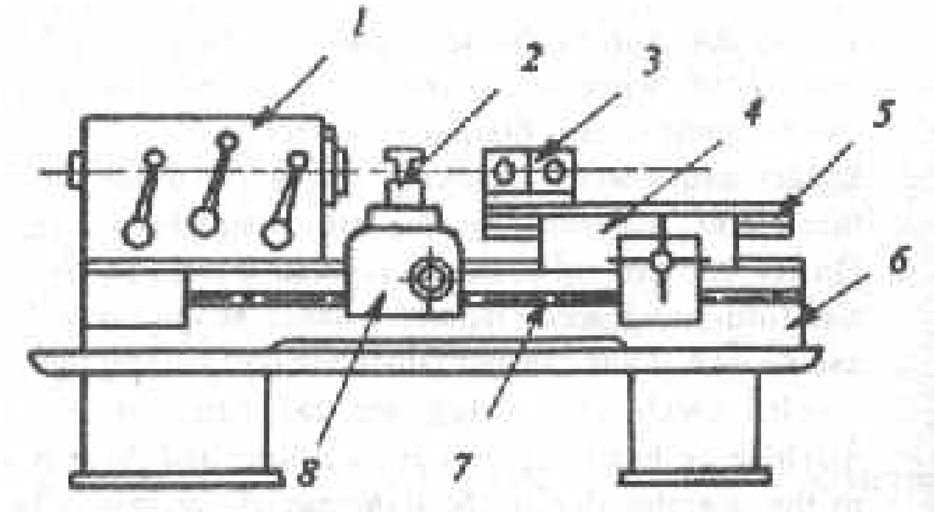
A capstan lathe or a turret lathe is a production lathe used to manufacture any number of identical pieces in the minimum time. These lathes are development of engine lathes. The capstan lathe was first developed in the United States of America by Pratt and Whitney sometimes in 1860.

Special characteristics of a capstan or turret lathe enable it to perform a series of operations such as drilling, turning, boring, thread cutting, reaming, chamfering, cutting-off and many other operations in a regular sequence to produce a large number of identical pieces in a minimum time.

Differences between a Capstan Lathe and Turret and an Engine Lathe:

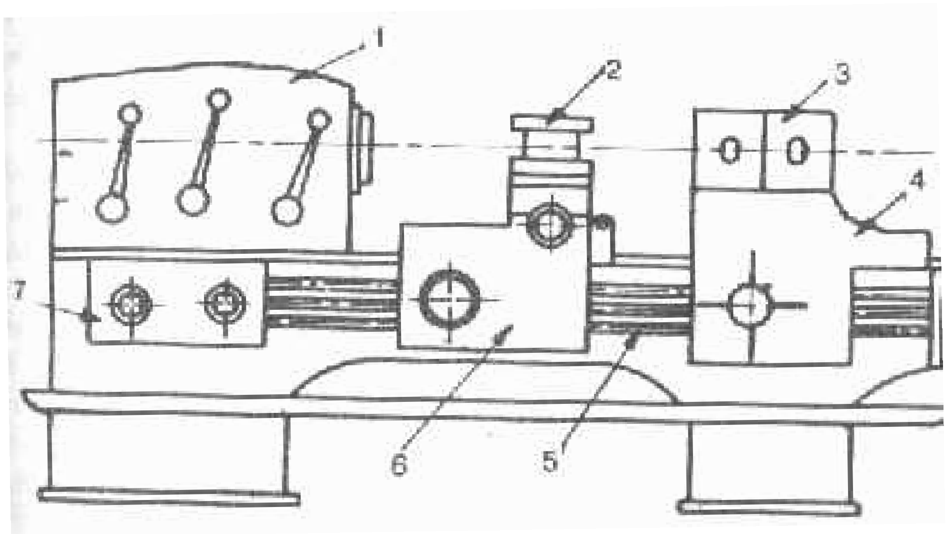
- The headstock of a turret lathe is similar to that of an engine lathe in construction but possesses wider range of speeds, and is of heavier in construction.
- Similar sizes of capstan and turret lathe and engine lathe, when an engine lathe will require a motor of 3h.p. to drive its spindle and other parts, a capstan and turret lathe will demand power as high as 15h.p. for high rate of production.
- In a turret lathe, the tailstock of an engine lathe is replaced by a turret. This is a six-sided block each of which may carry one or more tools. These tools may be indexed one after the other to perform different operations in a regular order. This is a decisive advantage in mass production.
- In a turret lathe, combination of cuts can be taken. Two or more tools may be mounted on the same face of the turret, making it possible to machine more than one surface at a time. This feature reduces total operational time.

- A semiskilled operator can operate a capstan or turret lathe after the machine has been set up by a skilled machinist. A skilled machinist may be requisitioned for setting up only for a large number of machines, where as actual production may be given by a semiskilled operator.



1. Headstock, 2. Cross – slide tool post, 3. Hexagonal turret, 4. Saddle for auxil slide, 5. Auxilary slide, 6. Lathe bed, 7. Feed rod, 8. Saddle for cross-slide.

Capstan lathe parts



1. Headstock, 2. Cross – slide tool post, 3. Hexagonal turret, 4. Turret saddle, 5. Feed rod, 6. Saddle for cross-slide.

Turret lathe parts

- Capstan and turret lathe is fundamentally a production machine, capable of producing large number of identical pieces in a minimum time. The centre lathe is suitable for odd jobs having different shapes and sizes.
- Capstan and turret lathes are not usually fitted with lead screws for cutting threads. A short length of lead screw called “Chasing screw” are sometimes provided for cutting threads by a chaser in a turret lathe.

Differences between a capstan and turret lathe

Capstan lathe	Turret lathe
Turret of a capstan lathe is mounted on a short slide or ram slide which slides on the saddle.	Turret of a turret lathe is mounted on a saddle which slides directly on the bed
Thus in a capstan lathe, travel of the turret is	Turret saddle moves on total length of bed,

dependent upon the length of the travel of the ram. This limits the maximum length of work to be machined in one setting.	this enables the turret to be moved in entire length of the bed and can machine longer work.
Maximum size of bar that a capstan lathe can accommodate is 60 mm in diameter.	Turret lathes are capable of turning bars 125 to 200 mm in diameter and absorbing up to 50h.p. in the main drive.
Capstan lathes are suitable for bar work.	Larger and heavier chucking works are usually handled on a turret lathe.
In capstan lathe hand feeding is easy as the hexagonal turret can be moved forth and back more rapidly without moving saddle unit.	In case of turret lathe hand feeding is a laborious process due to the movement of the entire saddle unit.

Principal parts of capstan and turret lathes

The turret has essentially the same parts as the engine lathe except the turret and the complex mechanism incorporated in it for making it suitable for mass production work.

Different parts of capstan lathe are:

- Head stock
- Cross-slide tool post
- Hexagonal turret
- Saddle for auxiliary slide
- Auxiliary slide
- Lathe head
- Feed rod
- Saddle for cross-slide.

Different parts of turret lathe are:

- Head stock
- Cross-slide tool post
- Hexagonal turret
- Turret saddle
- Feed rod
- Saddle for cross-slide

Bed:

The bed is a long box like casting provided with accurate guide ways upon which are mounted the carriage and turret saddle. The bed is designed to ensure strength, rigidity and permanency of alignment under heavy duty services. Like engine lathe precision surface finishing methods must be applied to keep it resistant to wear during service period.

Headstock:

The headstock is a large casting located at the left hand end of the bed. The headstock of capstan and turret lathe may be of the following types:

1. Step cone pulley driven headstock
2. Direct electric motor driven headstock
3. All geared headstock
4. Pre optive or pre selective headstock.

1. Step cone pulley driven headstock:

This is the simplest type of headstock and is fitted with small capstan lathes where the lathe is engaged in machining small and constant diameter work pieces. Three or four steps of pulleys can cater to the needs of the machine.

2. Direct electric motor driven headstock:

In this type of headstock the spindle of the machine and the armature shaft are one and the same. Any speed variation or reversal is effected by simply controlling the motor. The machine is suitable for smaller diameter of work pieces rotated at high speeds.

3. All geared headstock:

On the larger lathes, the headstocks are geared and different mechanisms are employed for sped changing by actuating levers. The speed changing may be affected without stopping the machine.

4. Pre optive or pre selective headstock:

- It is an all geared headstock with provisions for rapid stopping, starting and speed changing for different operations by simply pushing a button or pulling a lever.
- Different speeds are required for different operations. These speed variations are obtained by placing the speed changing lever in the required position.
- After the first operation is complete, a button or lever is simply actuated and the spindle starts rotating at the selected speed required for the second operation without stopping the machine.

CROSS-SLIDE & SADDLE:

In small size capstan lathes, hand operated cross-slide and saddle are used which are clamped on the lathe bed at the required position. The larger capstan lathes and heavy duty turret lathes equipped with usually two designs of carriage.

1. Conventional type carriage
2. Side hung type carriage

Conventional type carriage bridges the gap between the front and rear bed ways and is equipped with four station type tool post at the front, and one rear tool post at the back of the cross-slide.

The side hung type carriage is generally fitted with heavy duty turret lathes where the saddle rides on the top and bottom guide ways on the front of the lathe bed.

The design facilitates swinging of larger diameter of work pieces without being interfered by the cross slide. The saddle and the cross slide may be fed longitudinally or crosswise by hand or power.

The tools are mounted on the tool post and correct heights are adjusted by using rocking or packing pieces.

THE TURRET SADDLE AND AUXILLARY SLIDE:

In a capstan lathe, the turret saddle bridges the gap between two bed ways, and the top face is accurately machined to provide bearing surface for the auxiliary slide.

The saddle may be adjusted on the lathe bed ways and clamped at required position. In a turret lathe the turret is directly mounted on the top of the saddle and any movement of the turret is effected by the movement of the saddle.

The turret is the hexagonal tool holder intended for holding six or more tools. The centre line of the each hole is perfectly aligned with the axis of the lathe when aligned with the head stock spindle.

In addition to these holes, there are four tapped holes on each face of the turret for securing different tool holding attachments. After one operation is completed, as the turret is brought back away from the spindle nose, the turret index automatically by a mechanism incorporated on the bed and in turret saddle, so that the tool mounted on the next face is aligned with the work.

CAPSTAN AND TURRET LATHE MECHANISM

The carriage, cross-slide, turret-slide, and the saddle holding the turret may be fed in to the work by hand or power.

They are two main mechanisms

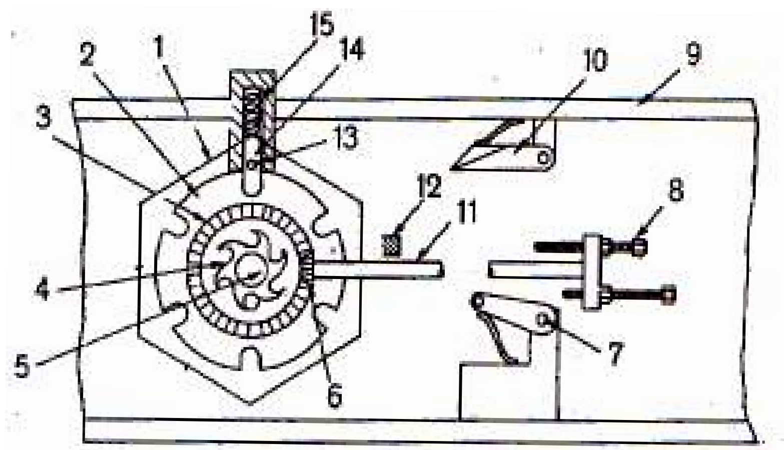
1. Turret indexing mechanism
2. Bar feeding mechanism

Turret Indexing Mechanism

A simple line sketch of the mechanism is show in figure.

- The turret 1 is mounted on the spindle 5, which rests on a bearing on the turret saddle.
- The index plate 2, the bevel gear 3 and an indexing ratchet 4 are keyed to the spindle 5.
- The plunger 14 fitted with in the housing and mounted on the saddle locks the index plate by spring 15 pressure and prevents any rotary movement of the turret as the tool feeds in to the work.

- A pin 13 is fitted on the plunger 14 projects out of the housing. An actuating cam 10 and the indexing pawl 7 are attached to the lathe bed 9 at the desired position.
- Both the cam and the pawl are spring loaded. As the turret reaches the backward position, the actuating cam10 lifts the plunger 14 out of the groove in the index plate due to the riding of the pin 13on the beveled surface of the cam 10 and thus unlocks the index plate2.



1. Hexagonal turret, 2. Index plate, 3. Bevel gear, 4. Indexing ratchet, 5. Turret spindle, 6. Bevel pinion, 7. Indexing pawl, 8. Screw stop rods, 9. Lathe bed, 10. Plunger actuating cam 11. Pinion shaft, 12. Stop, 13. Plunger pin, 14. Plunger, 15. Plunger spring

Turret Indexing Mechanism

- The spring loaded pawl 7 which by this time engages with a groove of the ratchet plate 4, causes the ratchet to rotate as the turret head moves backward.
- When the index plate or the turret rotates through one sixth of revolution, the pin 13 and the plunger 14 drops out of the cam 10 and the plunger locks the index plate at the next groove. The turret is thus indexed by one sixth of revolution and again locked in to the new position automatically.

Bar Feeding Mechanism

- Capstan and turret lathes while working on bar work require some feeding mechanism for bar feeding. A simple bar feeding mechanism is illustrated as show in figure.
- The bar 6 is passed through the bar chuck 3, spindle of the machine and then through the collet chuck.
- The bar chuck 3 rotates in the sliding bracket body 2 which is mounted on a long slide bar. The bar chuck 3 grips the bar centrally by two set screws 5 and rotates with the bar in the sliding bracket body 2.
- One end of the chain 8 is connected to the pin 9 fitted on the sliding bracket 10 and the other end supports a weight 4, the chain running over two fixed pulleys 7 and 11 mounted on the slide bar.
- The weight 4 constantly exerts end thrust on the bar chuck while it revolves on the sliding bracket and forces the bar through the spindle, the moment the Collet chuck is released.
- Thus the feeding may be accomplished without stopping the machine.

CAPSTAIN AND TURRET LATHE SIZE:

The size of a capstan or turret lathe is designated by the maximum diameter of rod that can be passed through the head stock spindle and the swing diameter of the work that can be rotated over the lathe bed ways. In order to specify the lathe fully, other important particulars such as number of spindle speeds, number of feeds available to the carriage and turret saddle, net weight of the machine , floor space and power required, etc. should also be stated.

Tool Layout for Capstan and Turret Lathe

Tool and Tool-Layout:

Tools for capstan and turret lathe are similar in construction to those of centre lathe tools, except material. The tools used are made of H.S.S. or tungsten carbide because the machines are more rigid and also operated at higher cutting speeds. The tools mounted on cross-slide are used for tuning, facing, necking, parting etc. and those mounted on the turret head are used for drilling, boring reaming, etc.

The tool layout for a job constitutes the pre-determined plan for machining operation of a particular component. The layout is dependent upon the number of pieces to be manufactured, i.e. lot size. As a general rule, standard tools should be used as much as possible and also for small batches of work, the layout should be simple.

For large quantities and long run special tools should be used as they minimise machining time and retain their cutting qualities for the maximum period. The accuracy and cost of component largely depends upon the tool layout.

For preparation of the tool layout, it is necessary to have the finished drawing of the part to be machined and if it is a forging or casting, the forged, or cast blank will determine how much machining has to be done on various faces.

After a preliminary list-giving the order of operations has been decided upon with details as the tools required, a tool layout to the scale is prepared on the tracing paper by super-imposing the layout of the machine capacity chart, drawn to the same scale with the component in position.

For any layout in which there may be a doubt regarding sufficient travel, clearances, etc., a simple preliminary trial is conducted on the drawing board before it is put on the machine. For many jobs, their comfortable adaptation to a machine will be obvious by simple estimates and more elaborate precautions will prove unnecessary.

Examples of Tool Layout:

A few typical examples of tool layout are given below:

TOOL LAYOUT FOR FRONT WHEEL AXLE:

Tool layout shows the types of tools required and the sequence in which they are to be used. While preparing tool layouts, the capacity chart of machine showing the capabilities of machine should not be lost sight of. It also ensures that tool movements and turret indexing, etc. clear the various machine parts. Machining time can be established by listing the operations required systematically in the form of tool layout.

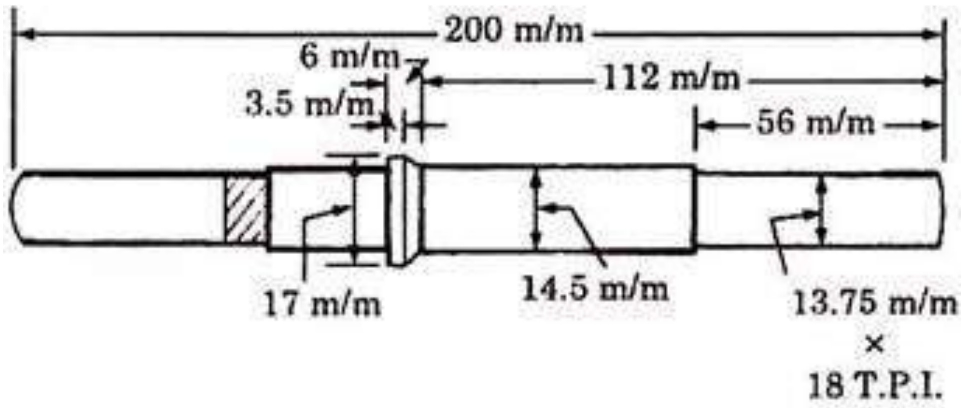


Fig. 32.20

To machine the front wheel axle from 18 mm steel bar, shown in Fig. 32.20, the following operations are required:

(1) Feed out the bar.

(2) Turn 14.5 mm diameter with the tool box.

(3) Turn 13.75 mm diameter with the tool box.

(4) Round end with roller steady end tool.

(5) Centre with centre drill.

(6) Cut thread with Coventry die.

(7) Form 17 mm diameter and chamfer with tool in the square turret.

(8) Part off with stepped cut-off tool in the rear-tool post.

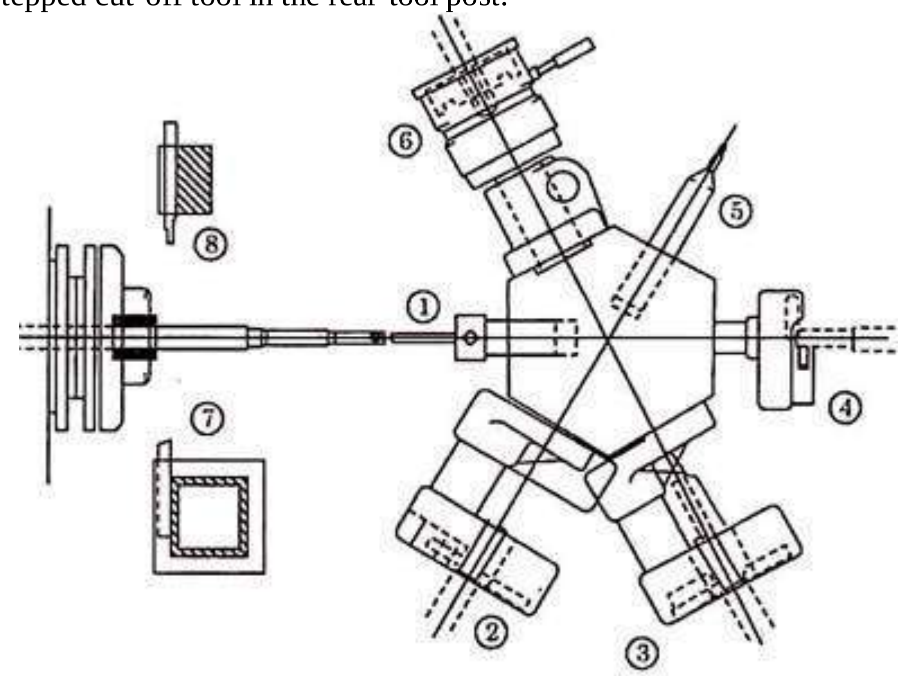


Fig. 32.21

A typical layout to be used for various operations is shown in Fig. 32.21.

TOOL LAY-OUT FOR BALL-BEARING PART:

To machine a ball-bearing part shown in Fig. 32.22 the following operations are performed:

(1) Rough face end with tool in the back tool post.

(2) Finish face end with tool in the front tool post.

(3) Bore diameters O and P and chamfer C with tools in boring bar in hexagonal turret.

(4) Recess diameters X and Y using double recess cutter held in recessing tool slide.

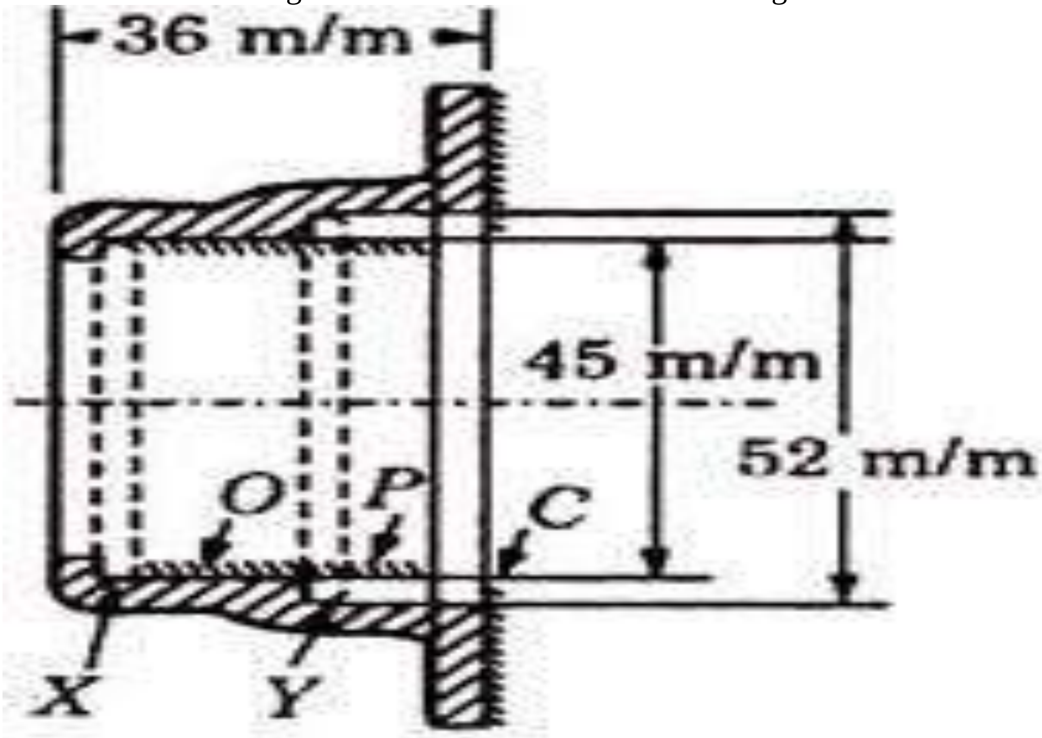


Fig. 32.22

(5) Size bore diameter O using fine adjustment boring bar.

(6) Tap bore P using tap and die holder. A typical layout of tools to be used for various operations is shown in Fig. 32.23.

Table 32.1 shows the description of operation at various tool stations, spindle speed, feed rate and machining time, etc.

Problem:

Draw a tool layout for the component shown in Fig. 32.24. Also determine the machining time for all the operations, the manipulation times and the overall machining time for producing the component on a turret lathe.

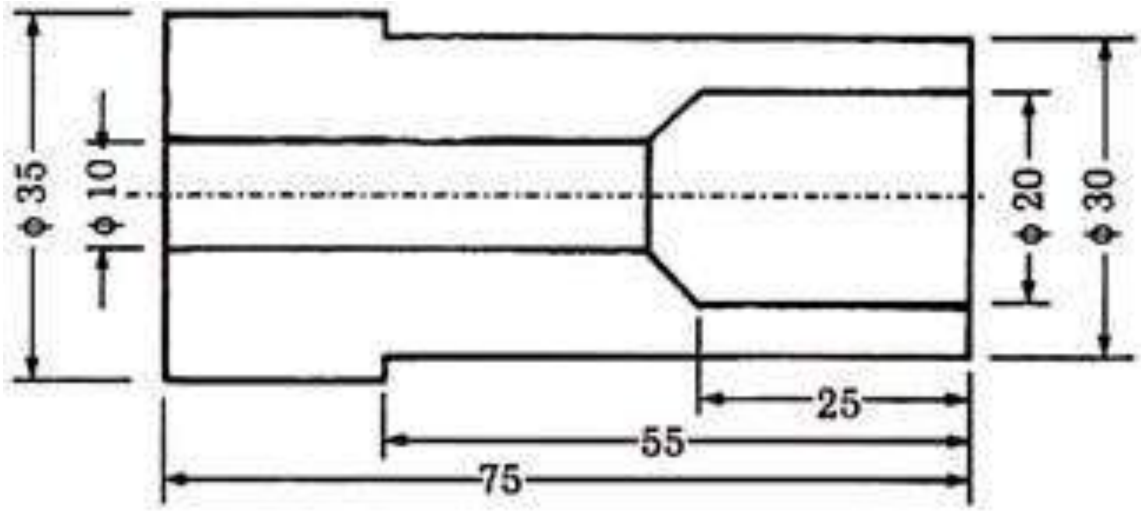


Fig. 32.24. Bar stock : 40 mm diameter.

Table 32.1

Operation	*Tooling station*	*Description of Operation*	*Spindle Speed rev/min*	*Feed mm/rev*	*Time (min)*
1.	—	Index turret to position 1	—	—	0.08
2.	Turret 1	Feed to bar stop	—	—	0.08
3.	—	Select spindle speed to 710 rpm	—	—	0.05
4.	—	Index turret to position 2	—	—	0.08
5.	Turret 2	Start drill and face	710	—	0.20 (Estimated)
6.	—	Select feed as 0.15 mm/rev	—	—	0.05
7.	—	Index turret to position 3	—	—	0.08
8.	Turret 3	Drill φ 10	710	0.15	$\frac{\text{Depth}}{\text{Feed} \times \text{RPM}} = \frac{75}{0.15 \times 710} = 0.70$
9.	—	Index turret to position 4	—	—	0.08
10.	—	Change speed to 350 rpm	—	—	0.05
11.	—	Change feed to 0.25 mm/rev.	—	—	0.05
12.	Turret 4	Drill φ 20 × 25 deep	350	0.25	$\frac{25}{0.25 \times 360} = 0.29$
13.	—	Index turret to position 5	—	—	0.08
14.	—	Change speed to 254 rpm	—	—	0.05
15.	—	Change feed to 0.30 mm/rev.	—	—	0.05
16.	Turret 5	Turn φ 35 × 75 long	254	0.30	$\frac{\text{Length}}{\text{Feed} \times \text{RPM}} = \frac{75}{254 \times 0.30} = 0.98$
17.	—	Index turret to position [illegible]	—	—	0.08
18.	Turret 6	Turn φ 30 × 55 long	254	0.30	$\frac{55}{254 \times 0.30} = 0.72$
19.	—	Change feed to 0.15 mm/rev.	—	—	0.05
20.	Rear Tool post 7	Part off	254	0.15	$\frac{15}{254 \times 0.15} = 0.4$

Out of above operations 2, 5, 8, 12, 16, 18, and 20 are machining times and rest are manipulation times. Total time 4.2 min
Allowing 1.2 for fatigue, total machining time = 4.2 × 1.2 = 5.04 min.

Solution:

Fig. 32.25 shows the various tools and the tool layout required to produce the component shown in Fig. 32.24.

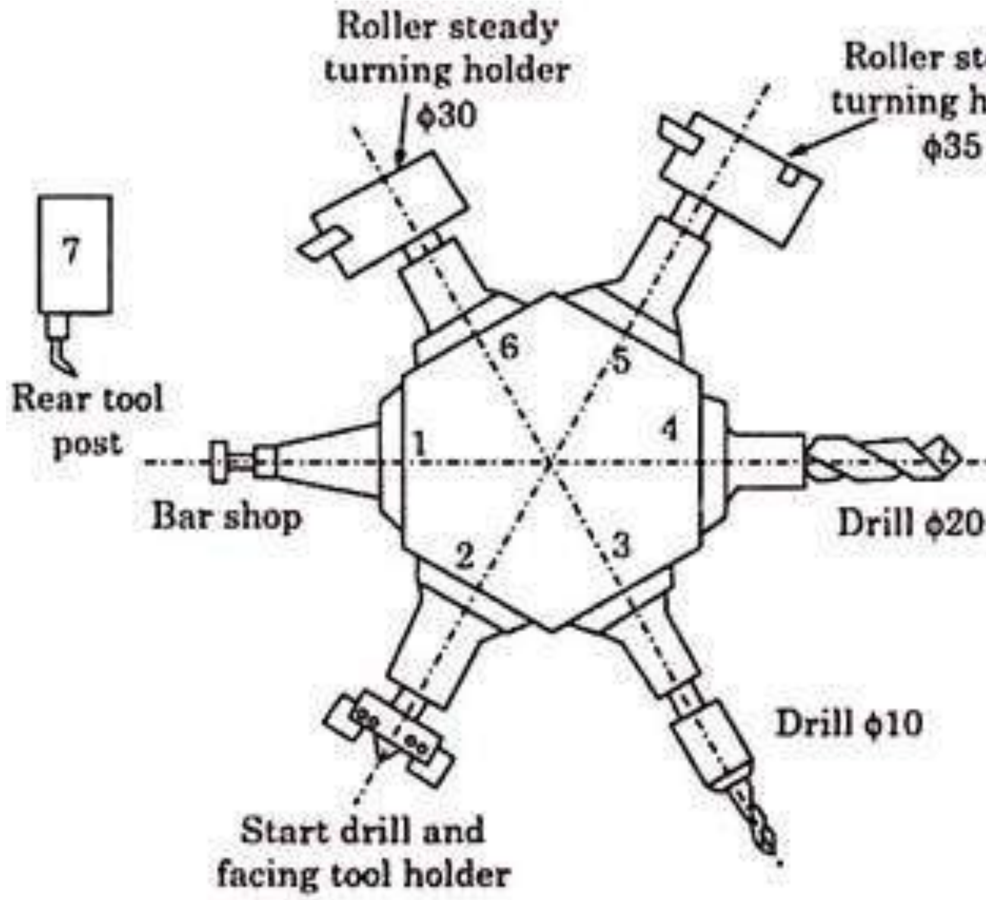


Fig. 32.25. Tool layout for the component.

SHAPER

1.1 SHAPER

Shaper is a reciprocating type of machine tool in which the ram moves the cutting tool Backwards and forwards in a straight line. The basic components of shaper are shown in Fig. 1. It is 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. The principal of shaping operation is shown in Fig. .2 (a, b). Modern shapers can also generate contoured surface as shown in Fig. 3. A shaper is used to generate flat (plane) surfaces by means of a single point cutting tool similar to a lathe tool.

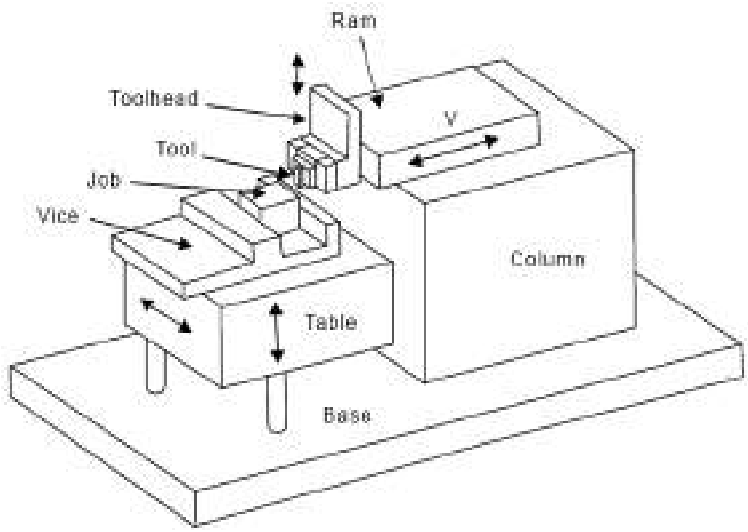


Fig.1 Principal components of a shaper

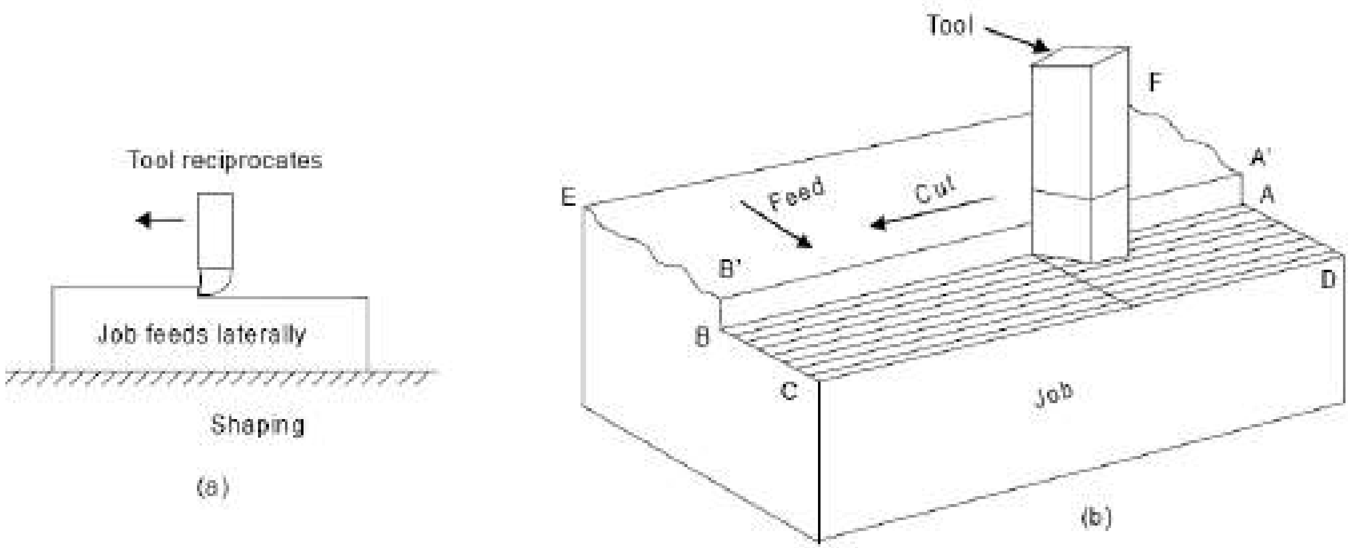


Fig. 2 (a, b) Working principal of shaping machine

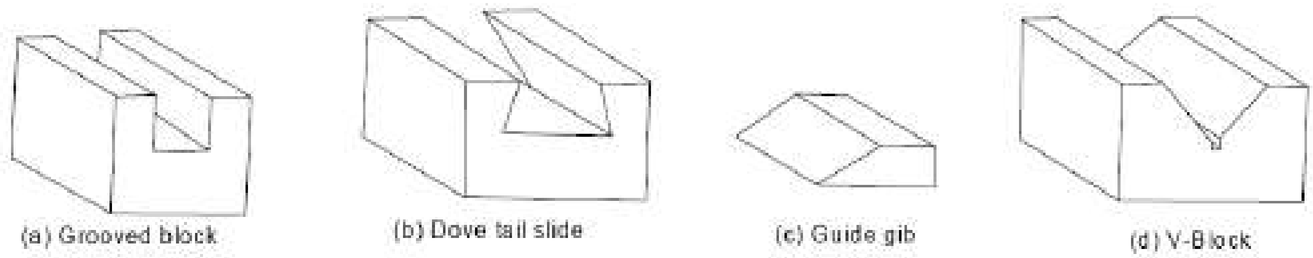


Fig. 3 Job surfaces generated by shaper

1.2 WORKING PRINCIPLE OF SHAPER

A single point cutting tool is held in the tool holder, which is mounted on the ram. The workpiece is rigidly held in a vice or clamped directly on the table. The table may be supported at the outer end. The ram reciprocates and thus cutting tool held in tool holder moves forward and backward over the workpiece. In a standard shaper, cutting of material takes place during the forward stroke of the ram. The backward stroke remains idle and no cutting takes place during this stroke. The feed is given to the workpiece and depth of cut is adjusted by moving the tool downward towards the workpiece. The time taken during the idle stroke is less as compared to forward cutting stroke and this is obtained by quick return mechanism. The cutting action and functioning of clapper box is shown in Fig. 4 during forward and return stroke.

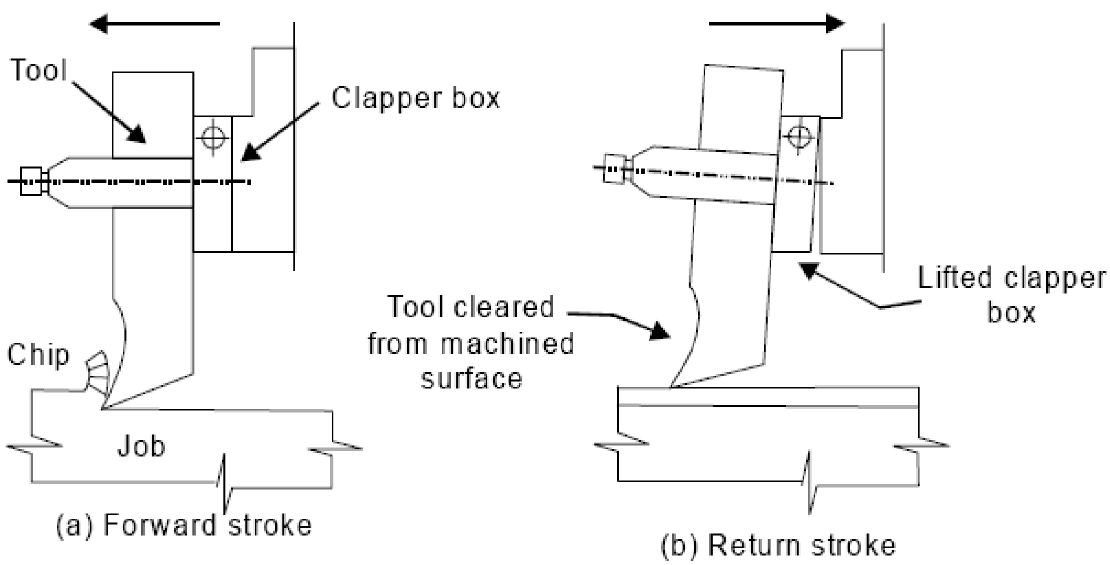


Fig. 4 Cutting action and functioning of clapper box

1.3 TYPES OF SHAPERS

Shapers are classified under the following headings:

1. According to the type of mechanism used for giving reciprocating motion to the ram

(*a*) Crank type

(*b*) Geared type

(*c*) Hydraulic type

2. According to the type of design of the table:

(*a*) Standard shaper

(*b*) Universal shaper

3. According to the position and travel of ram:

(*a*) Horizontal type

(*b*) Vertical type

(*c*) Traveling head type

4. According to the type of cutting stroke:

(*a*) Push type

(*b*) Draw type.

A brief description these shapers is given below-

1.3.1 Crank Shaper

This is the most common type of shaper. It employs a crank mechanism to change circular motion of a large gear called "bull gear" incorporated in the machine to reciprocating motion of the ram. The bull gear receives power either from an individual motor or from an overhead line shaft if it is a belt-driven shaper.

1.3.2 Geared Shaper

Geared shaper uses rack and pinion arrangement to obtain reciprocating motion of the ram. Presently this type of shaper is not very widely used.

1.3.3 Hydraulic Shaper

In hydraulic shaper, reciprocating motion of the ram is obtained by hydraulic power. For generation of hydraulic power, oil under high pressure is pumped into the operating cylinder fitted with piston. The piston end is connected to the ram through piston rod. The high pressure oil causes the piston to reciprocate and this reciprocating motion is transferred to the ram of shaper. The important advantage of this type of shaper is that the cutting speed and force of the ram drive are constant from the very beginning to the end of the cut.

1.3.4 Standard Shaper

In standard shaper, the table has only two movements, horizontal and vertical, to give the feed.

1.3.5 Universal Shaper

A universal shaper is mostly used in tool room work. In this type of shaper, in addition to the horizontal and vertical movements, the table can be swiveled about an axis parallel to the ram ways, and the upper portion of the table can be tilted about a second horizontal axis perpendicular to the first axis.

1.3.6 Horizontal Shaper

In this type of shaper, the ram holding the tool reciprocates in a horizontal axis.

1.3.7 Vertical Shaper

In vertical shaper, the ram reciprocates in a vertical axis. These shapers are mainly used for machining keyways, slots or grooves, and internal surfaces.

1.3.8 Travelling Head Shaper

In this type of shaper, the ram while it reciprocates, also moves **crosswise** to give the required feed.

1.3.9 Push Type Shaper

This is the most general type of shaper used in common practice, in which the metal is removed when the ram moves away from the column, i.e. pushes the work.

1.3.10 Draw Type Shaper

In this type of shaper, the cutting of metal takes place when the ram moves towards the column of the machine, i.e. draws the work towards the machine. The tool is set in a reversed direction to that of a standard shaper.

1.4 PRINCIPAL PARTS OF SHAPER

Fig. 5 shows the parts of a standard shaper. The main parts are given as under.

1. Base
2. Column
3. Cross-rail
4. Saddle
5. Table
6. Ram
7. Tool head
8. Clapper box
9. Apron clamping bolt
10. Down feed hand wheel

11. Swivel base degree graduations
12. Position of stroke adjustment hand wheel
13. Ram block locking handle
14. Driving pulley
15. Feed disc
16. Pawl mechanism
17. Elevating screw

Some of important parts are discussed as under.

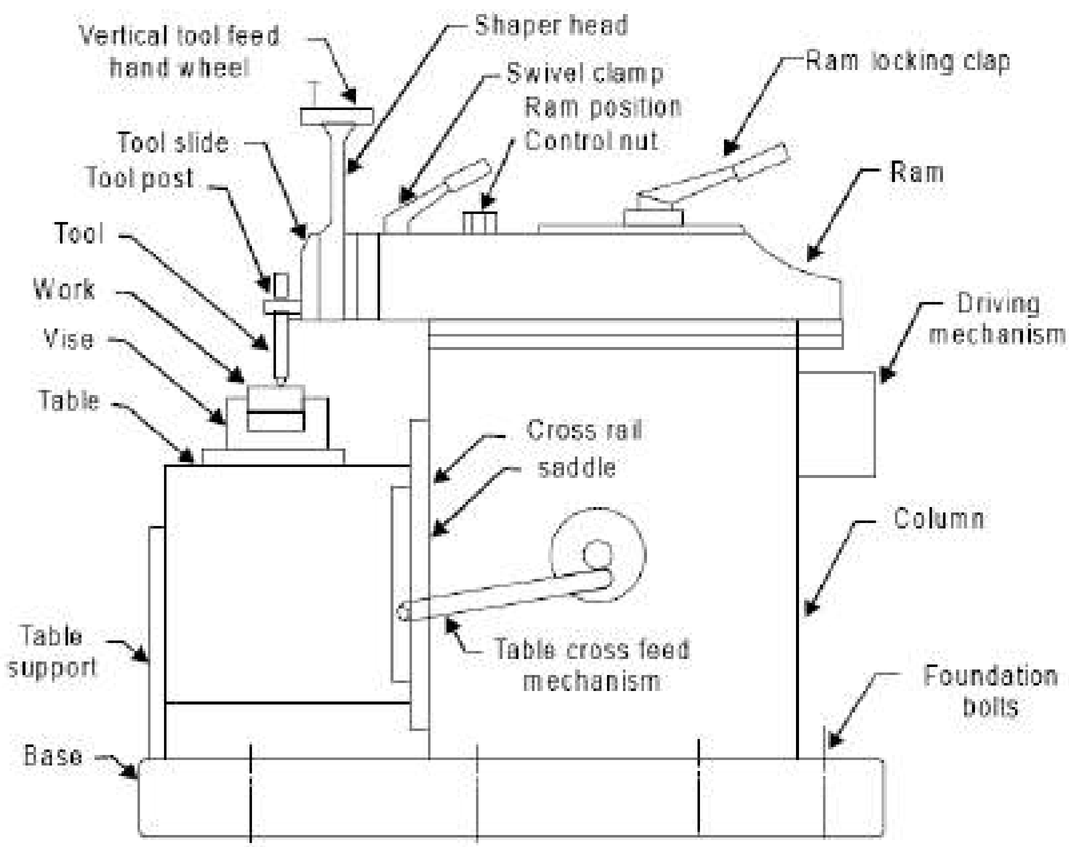


Fig. 5 Parts of a standard shaper

Base

It is rigid and heavy cast iron body to resist vibration and takes up high compressive load. It supports all other parts of the machine, which are mounted over it. The base may be rigidly bolted to the floor of the shop or on the bench according to the size of the machine.

Column

The column is a box shaped casting mounted upon the base. It houses the ram-driving mechanism. Two accurately machined guide ways are provided on the top of the column on which the ram reciprocates.

Cross rail

Cross rail of shaper has two parallel guide ways on its top in the vertical plane that is perpendicular to the rai1 axis. It is mounted on the front vertical guide ways of the column. It consists mechanism for raising and lowering the table to accommodate different sizes of jobs by rotating an elevating screw which causes the cross rail to slide up and down on the vertical face of the column. A horizontal cross feed screw is fitted within the cross rail and parallel to the top guide ways of the cross rail. This screw actuates the table to move in a crosswise direction.

Saddle

The saddle is located on the cross rail and holds the table on its top. Crosswise movement of the saddle by rotation the cross feed screw by hand or power causes the table to move sideways.

Table

The table is a box like casting having T -slots both on the top and sides for clamping the work. It is bolted to the saddle and receives crosswise and vertical movements from the saddle and cross rail.

Ram

It is the reciprocating part of the shaper, which reciprocates on the guideways provided above the column. Ram is connected to the reciprocating mechanism contained within the column.

Tool head

The tool head of a shaper performs the following functions-

1. It holds the tool rigidly,
2. It provides vertical and angular feed movement of the tool, and
3. It allows the tool to have an automatic relief during its return stroke.

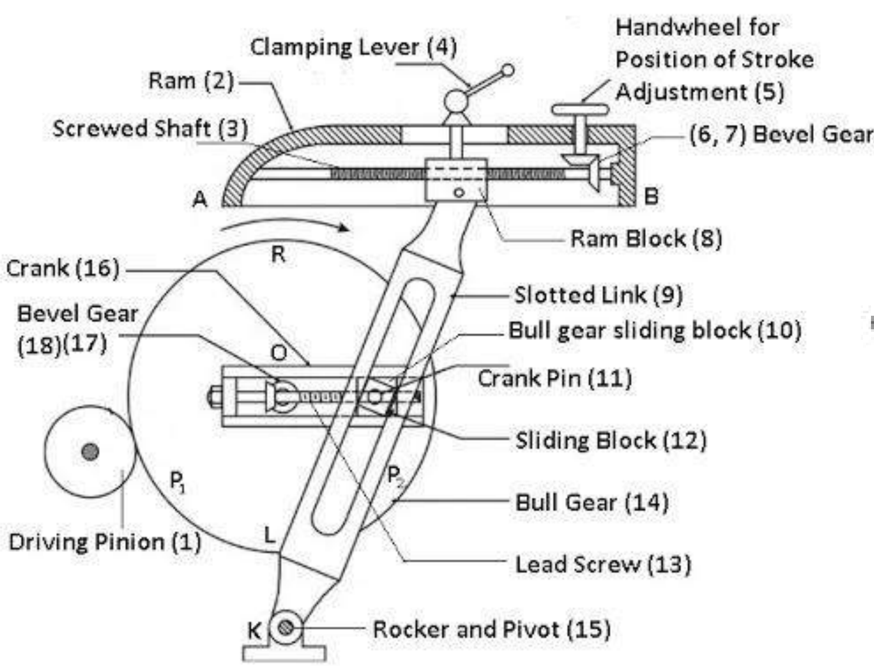
The various parts of tool head of shaper are apron clamping bolt, clapper box, tool post, down feed, screw micrometer dial, down feed screw, vertical slide, apron washer, apron swivel pin, and swivel base. By rotating the down feed screw handle, the vertical slide carrying the tool gives down feed or angular feed movement while machining vertical or angular surface. The amount of feed or depth of cut may be adjusted by a micrometer dial on the top of the down feed screw. Apron consisting of clapper box, clapper block and tool post is clamped upon the vertical slide by a screw. The two vertical walls on the apron called clapper box houses the clapper block, which is connected to it by means of a hinge pin. The tool post is mounted upon the clapper block. On the forward cutting stroke the clapper block fits securely to the clapper box to make a rigid tool support. On the return stroke a slight frictional drag of the tool on the work lifts the block out of the clapper box a sufficient amount preventing the tool cutting edge from dragging and consequent wear. The work surface is also prevented from any damage due to dragging.

1.5 SHAPER MECHANISM

In a shaper, **rotary motion of the drive is converted into reciprocating motion of the ram by the mechanism housed within the column or the machine**. In a standard shaper metal is removed in the forward cutting stroke, while the return stroke goes idle and no metal is removed during this period as shown in Fig. 4. The shaper mechanism is so designed that it moves the ram holding the tool at a comparatively slower speed during forward cutting stroke, whereas during the return stroke it allow the ram to move at a faster speed to reduce the idle return time. This mechanism is known as quick return mechanism. The reciprocating movement of the ram and the quick return mechanism of the machine are generally obtained by anyone of the following methods:

1. Crank and slotted link mechanism
2. Whitworth quick return mechanism, and
3. Hydraulic shaper mechanism

1. CRANK AND SLOTTED LINK MECHANISM



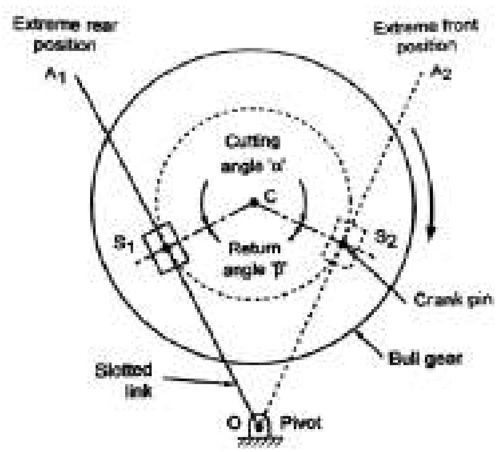


Fig. 6 Crank and slotted link mechanism

In crank and slotted link mechanism. The power is transmitted to the bull gear by a pinion which receives its power from an individual motor. In a two gear system, the smaller gear is called pinion and the larger gear is called bull gear.

The radial slide is bolted to the centre of the bull gear. This radial slide carries a sliding block into which the crank pin is fitted. As the bull gear will rotate, the crank will revolve at uniform speed.

The sliding block which is mounted upon the crank pin is fitted within the slotted link. This slotted link is pivoted at its bottom end attached to the frame of column. The upper end of the sliding link is attached to the ram block by a pin.

As the bull gear rotates causing the crank pin to rotate, the sliding block fastened to the crank pin will rotate on the crank pin circle. And at the same time will run up and down the slot in the slotted link providing it with a rocking movement which is communicated to the ram. Thus the rotary motion of the bull gear is converted to reciprocating movement of the ram.

Since the angle swept by the crank pin during the cutting stroke is more when compared to the return stroke, the ram will travel at a faster rate during idle stroke.

Stroke length adjustment:- Stroke length adjustment is required to handle different length of workpiece. For this the crank pin should be made to be moved towards the centre or away from the centre of the bull gear. When the crank pin is near the centre of the bull gear, the stroke length is less, whereas when the crank is moved away from the centre, the stroke length increases.

Positioning the Ram: To position the ram with respect to the workpiece, the clamping lever (4) is loosened, and the handwheel (5) is rotated which will make the entire ram body to move front and back thereby positioning the ram with respect to the workpiece.

2. **WHITWORTH QUICK RETURN MECHANISM**

This mechanism is also used in slotting machines other than a shaper. In this type of shaper machine mechanism, the link CD (link 2) forming the turning pair is fixed..

Arrangement of parts

The link 2 matches to a crank in a reciprocating steam engine. The driving crank “CA” (link 3) rotates at a similar angular speed. The slider (link 4) connected to the crankpin at “A” moves along the slotted bar “PA” (link 1) which oscillates at a pivoted point D.

The connecting rod PR carries the ram at R to which the cutting tool is fixed. The movement of the tool is constrained along the line RD produced. i.e. along a line passing through D and perpendicular to CD.

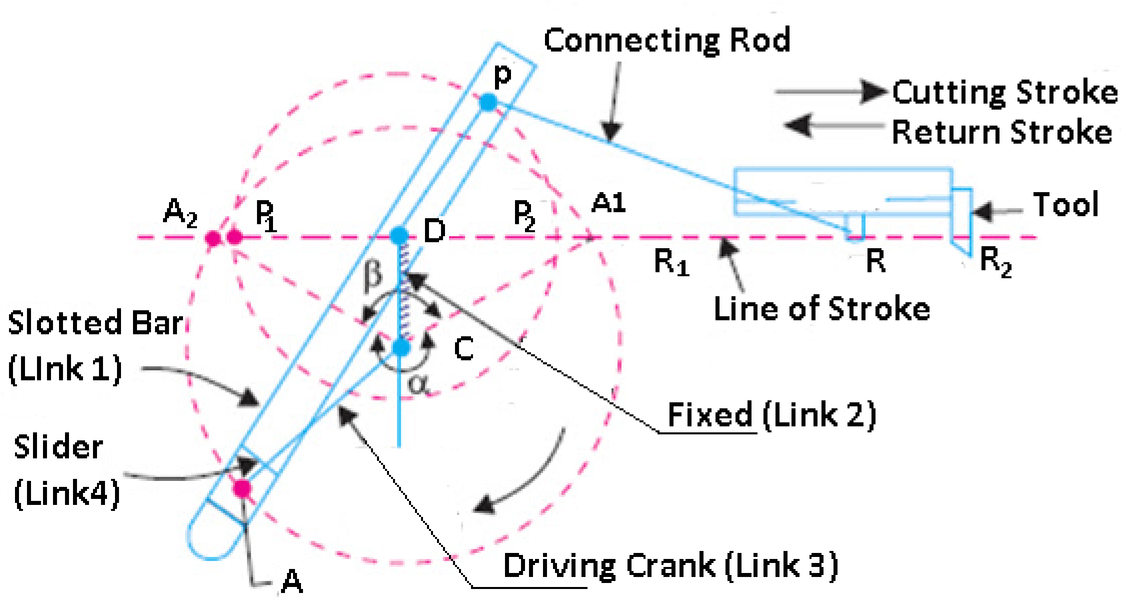


Fig. 7 Whitworth Quick Return Mechanism

Working

When the driving crank CA moves from the point CA1 to CA2. (or the link DP from the point DP1 to DP2) through an angle α in the clockwise direction, the tool moves from the left-hand end of its stroke to the right-hand end by a distance 2 PD. Now when the driving crank moves from the point CA2 to CA1. (or the link DP from DP2 to DP1) through an angle β in the clockwise direction, the tool moves back from the right-hand end of its stroke to the left-hand end.

A little consideration will show that the time taken during the right movement of the ram (i.e. during the forward stroke) will be equal to the time taken by the driving crank to move from CA1 to CA2.

Similarly, the time needed during the right to left movement of the ram (or during the idle or return stroke) will be equal to the time taken by the driving crank to move from CA2 to CA1. Since the crank link CA rotates at a uniform angular velocity, thus, the time taken during the cutting stroke (or forward stroke) is more than the time taken during the return stroke.

$$\frac{Time\ of\ Cutting\ Stroke}{Time\ of\ Return\ Stroke} = \frac{\alpha}{\beta} = \frac{\alpha}{360° - \alpha}\ or\ \frac{360° - \beta}{\beta}$$

In other words, the mean speed of the ram during a cutting stroke is less than the mean speed during the return stroke

3. **HYDRAULIC SHAPER MECHANISM**

In this type of shaper machine mechanism, the ram is moved forwards and backwards by a piston moving in a cylinder placed under the ram.

The machine is consists of a constant discharge oil pump, a cylinder, a valve chamber and a piston. The Piston-rod is bolted to the ram body.

Arrangement and working

The oil under high pressure is drawn from the reservoir. The oil is passed through the valve chamber to the right side of the oil cylinder exerting pressure on the piston. This cause the ram connected to the piston to perform forward stroke. Any oil present on the left side of the cylinder is discharged to the reservoir through the throttle valve.

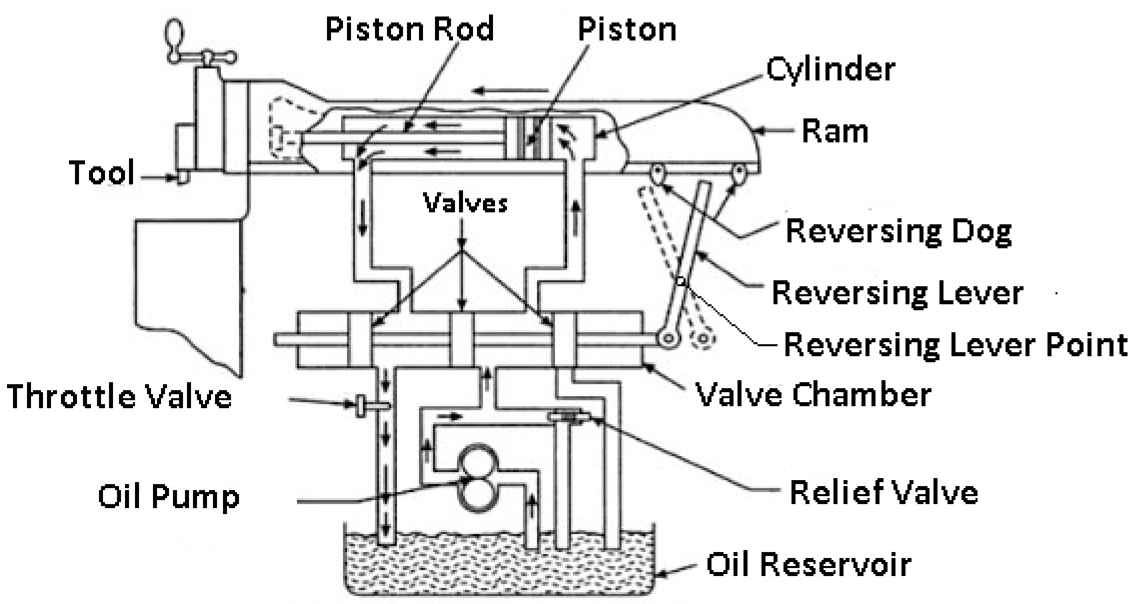


Fig. 8 Hydraulic Shaper Mechanism

At the end of a forward stroke, the shaper dog hit against the reversing lever causing the valves to alter their positions within the valve Chamber. Oil under high pressure is now pumped to the left side of the piston causing the ram to perform return stroke. Oil present on the right side of the piston is now discharged to the reservoir.

At the end of the return stroke, another shaper dog hits against the reversing lever altering the direction of stroke of the piston and the cycle is thus repeated.

The quick return is affected due to the difference in the stroke volume of the cylinder at both ends.

The left-hand end being smaller due to the presence of the piston rod. As the pump is a constant discharge one, within a fixed period, the same amount of oil will be a pump into the right or to the left-hand side of the cylinder. This will mean that the same amount of oil will be packed within a smaller stroke volume causing the oil pressure to rise automatically and increase the speed during the return stroke.

The length and position of stroke are adjusted by shifting the position of reversing dogs.

The cutting speed is changed by controlling the throttle valve which controls the flow of oil. When the throttle valve is lost the excess oil flow cut through the relief valve to the reservoir maintaining uniform pressure during the cutting stroke.

4. **AUTOMATIC TABLE FEEDING MECHANISM OF SHAPER**

It consists of a slotted disc, which carries a T-slot, as shown in the figure. In this slot is fitted an adjustable pin and to this is attached a connecting rod. The other end of the connecting rod is attached to the lower end of the rocker arm of the pawl mechanism. The rocker arm swings about the screw C, and at its upper end carries a spring-loaded pawl, as shown.

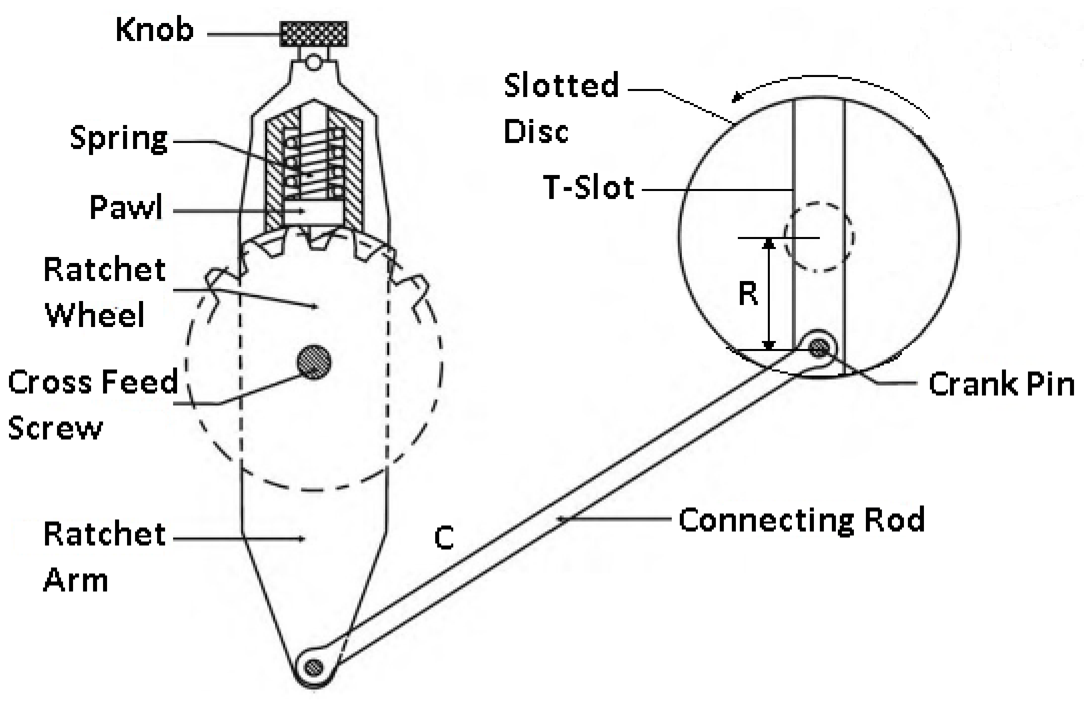


Fig. 9 Ratchet and Pawl Mechanism

Note, that the lower end of the pawl is bevelled on one side.

This arrangement helps the power feed to operate in either direction, but the same should be set to operate during the return stroke only. If otherwise, the mechanism will be subjected to severe stress. Variation in the feed can be provided by varying the distance R between the disc centre and the centre of the adjustable pin. Larger the said distance greater will be the feed and vice versa. The amount of feed to be given depends upon the type of finish required on the job. For rough machining, heavier cuts are employed, and thus, a coarse feed is needed. Against this, a finer feed is employed in finishing operations.

Working

As the driving disc rotates, the rocker arm rocks on the fulcrum. When the disc makes half revolution in clockwise direction the top part of the rocker arm moves in the clockwise direction. As the bevel side of pawl, fits on right side, the pawl slips over the teeth of ratchet wheel giving no motion to the table. During the other half rotation, the straight side of the pawl engages with the teeth, hence motion will be given to the table. Amount of cross feed is varied by changing the position of crankpin in the radial direction of disc.

1.6 SPECIFICATION OF A SHAPER

The size of a shaper is specified by the **maximum length of stroke or cut it can make**. Usually the size of shaper ranges from 175 to 900 mm. Besides the length of stroke, other particulars, such as the type of drive (belt drive or individual motor drive), floor space required, weight of the machine, cutting to return stroke ratio, number and amount of feed, power input etc. are also sometimes required for complete specification of a shaper.

1.7 OPERATIONS PERFORMED IN A SHAPER

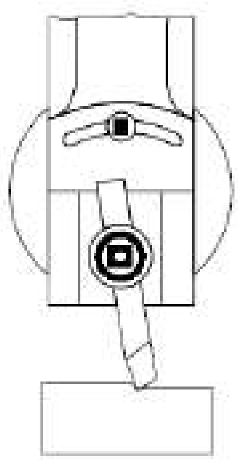


Fig. 10 Machining horizontal vertical surface on shaper

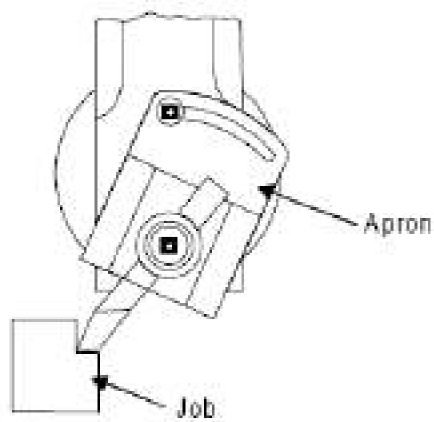


Fig. 11 Machining vertical surface on shaper

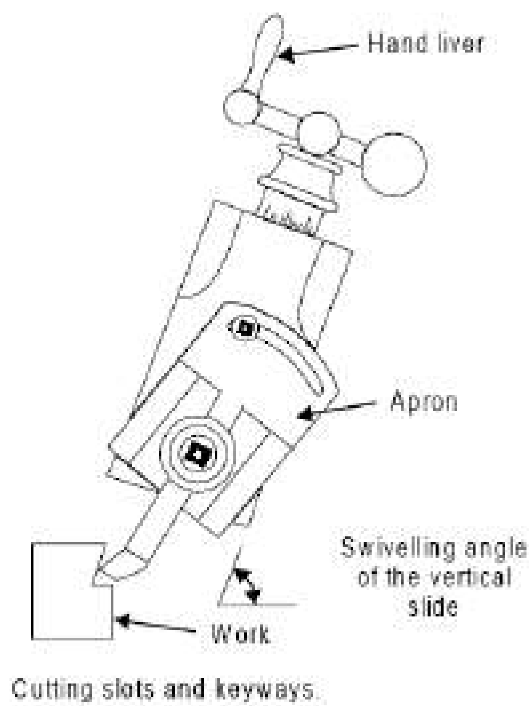


Fig. 12 Machining angular surface on shaper

A shaper is a machine tool primarily designed to generate a flat surface by a single point cutting tool. Besides this, it may also be used to perform many other operations. The different operations, which a shaper can perform, are as follows:

1. Machining horizontal surface (Fig. 10)
2. Machining vertical surface (Fig. 11)
3. Machining angular surface (Fig. 12)
4. Slot cutting (Fig. 13)
5. Key ways cutting (Fig. 14)
6. Machining irregular surface (Fig. 15)
7. Machining splines and cutting gears (Fig. 16)

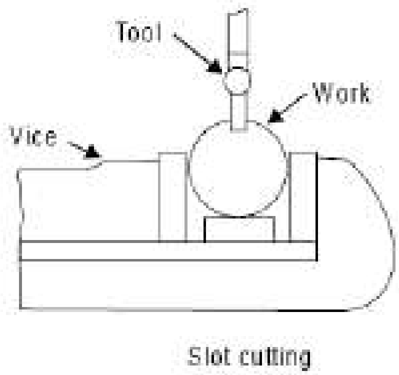


Fig. 13 Slot cutting on shaper

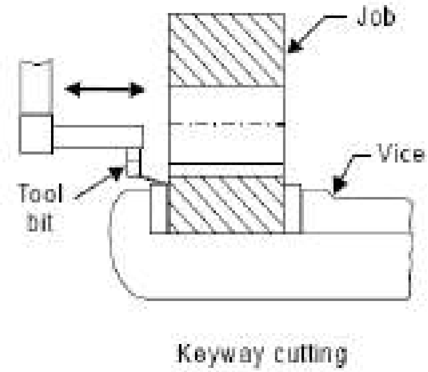


Fig. 14 Keyway cutting on shaper

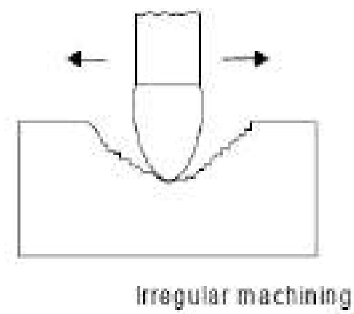


Fig. 15 Machining irregular surface on shaper

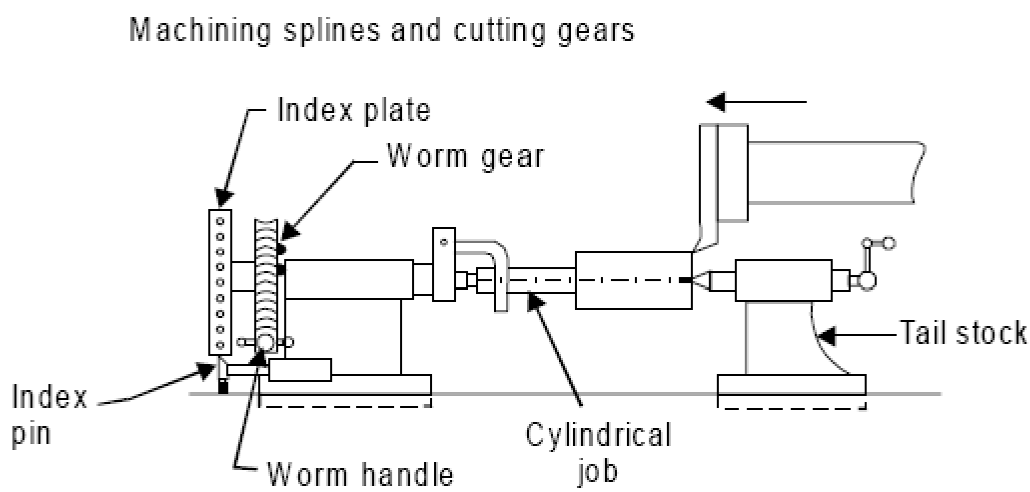


Fig. 17 Machining splines and cutting gears on shaper

MILLING MACHINES

1.1 INTRODUCTION

Milling is a machining operation in which a workpart is fed past a rotating cylindrical rotating 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 feature that distinguishes milling from drilling, however, in other operations like drilling, turning, etc. the tool is fed in the direction parallel to axis of rotation.

The cutting tool used in milling operation is called milling cutter, which consists of multiple edges called teeth. The machine tool that performs the milling operations by producing required relative motion between workpiece and tool is called milling machine. Normally, the milling operation creates plane surfaces. Other geometries can also be created by milling machine. Milling operation is considered an interrupted cutting operation in which the teeth of milling cutter enter and exit the work during each revolution. This interrupted cutting action subjects the teeth to a cycle of impact force and thermal shock on every rotation. The tool material and cutter geometry must be designed to bear the above stated conditions. Depending upon the positioning of the tool and workpiece the milling operation can be classified into different types.

1.2 TYPES OF MILLING MACHINES

Milling machines can be classified into different categories depending upon their construction, specification and operations. The choice of any particular machine is primarily determined by nature of the work to be done, its size, geometry and operations to be performed. The typical classification of milling machines on the basis of its construction is given below.

1. **Column and Knee Type Milling Machine**

   (a) Hand milling machine
   (b) Plain milling machine
   (c) Universal milling machine
   (d) Omniversal milling machine
   (e) Vertical milling machine

2. **Fixed Bed Type Milling Machine**

   (a) Simplex milling
   (b) Duplex milling
   (c) Triplex milling

3. **Special Type Milling Machine**

   (a) Rotary table milling

   (b) Drum milling

   (c) Planetary milling

   (d) Tracer controlled milling

In addition to above three types there is one more type of milling machine named as planner type milling machine which is rarely used.

1. **Column and Knee Type Milling Machine**

Main shape of column knee type of milling machine is shown in Figure 1.3. This milling machine consists of a base having different control mechanisms housed there in. The base consists of a vertical column at one of its end. There is one more base above the main base and attached to the column that serves as worktable equipped with different attachments to hold the workpiece. This base having worktable is identified as “knee” of the milling machine. At the top of the column and knee type milling machines are classified according to the various methods of supplying power to the table, different movements of the table and different axis of rotation of the main spindle. These are described in brief as below.

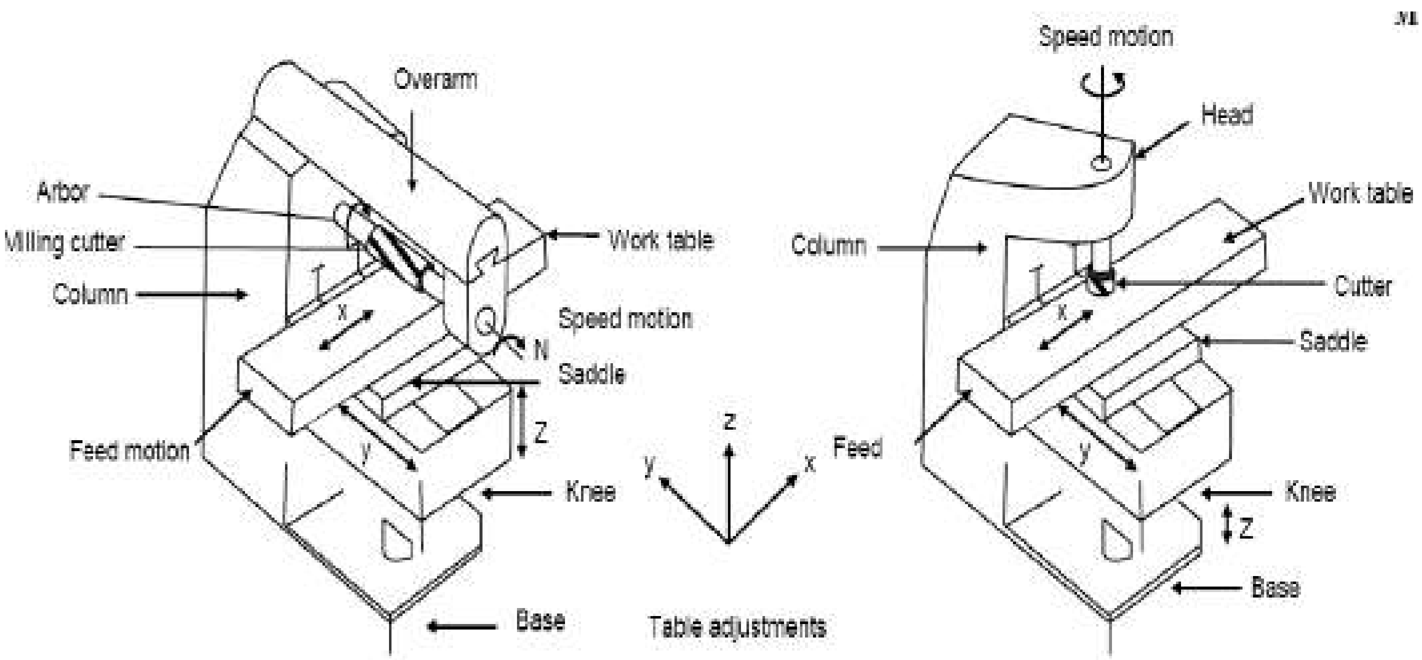


Fig. 1 . Column and Knee Type Milling Machine

A. **HAND MILLING MACHINE**

In case of head milling machine feed motion is given by hand and movements of the machine are provided by motor. This is simple and light duty milling machine meant for basic operations.

B. PLAIN MILLING MACHINE

Plain milling machine is similar to hand milling machine but feed movement can be powered controlled in addition to manual control. The table may be fed in a longitudinal, cross or vertical directions.

The feed is:

- Longitudinal - when the table is moved at right angles to the spindle.
- Cross - when the table is moved parallel to the spindle.
- Vertical - when the table is adjusted in the vertical plane.

C. UNIVERSAL MILLING MACHINE

It can be adapted to a wide range of milling operations. Here the table can be swiveled to any angle up to 45-degrees on either side of the normal position. In addition to 3 movements as mentioned earlier in a plain milling machine, the table may have the fourth movement when it is fed at an angle to the milling cutter. Helical milling operation can also be performed. The capacity of this type of machine is increased by using special attachments such as

- Dividing head or index head.
- Vertical milling attachment.
- Rotary attachment.
- Slotting attachment.

This machine can produce spur, bevel, spiral, twist drill, reamer, milling cutter. All operations that are performed on a shaper can be done using a universal milling machine.

D. OMINVERSAL MILLING MACHINE

Here the table also has four movements of the universal milling machine. It can also be tilted in a vertical plane by providing a swivel arrangement at the knee. The additional swiveling arrangement of the table helps in machining spiral grooves in reamers and bevel gears.

E. VERTICAL MILLING MACHINE

Here the position of the spindle is vertical or perpendicular to the table. This type of machine is adapted for machining grooves, slots, and flat surfaces.

The machine may be of the plain or universal type and has all the movements of the table for a proper setting and feeding the work.

The spindle head is clamped to the vertical column which is swiveled at an angle. It allows the milling cutter fixed on the spindle to work on angular surfaces.

2. FIXED BED TYPE MILLING MACHINE

It is also known as manufacturing type milling machine. Its table is mounted directly on the ways of fixed bed. Table movement is restricted to reciprocation only. Cutter is mounted on the spindle head which can move vertically on the column. Duplex milling machine has double spindle heads, one on each side of the table. Triplex milling machine has three spindle heads one each side of the table and third one is mounted on the cross rail. Bed type milling machine is shown in Figure 2..

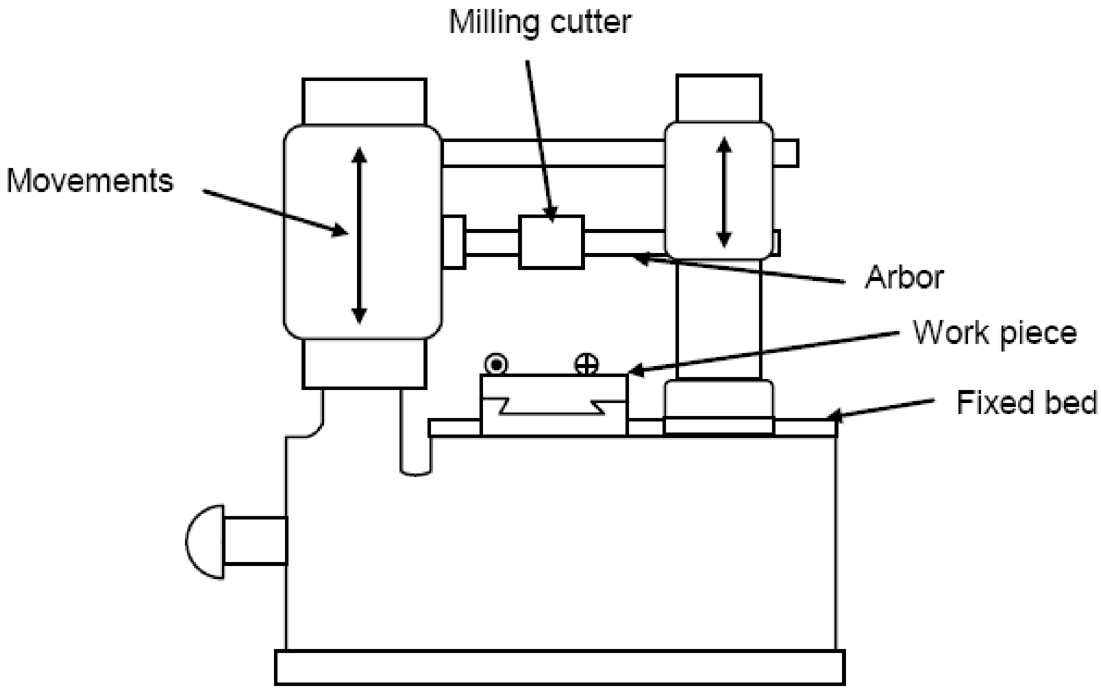


Fig. 2 Fixed bed type milling machine

3. PLANER TYPE MILLING MACHINE

This types of milling machine are also called as "Plano-Miller". It is a massive machine used for heavy-duty work having spindle heads adjustable in the vertical and transverse direction.

It relates a planer and like a planing machine. This machine has a cross rail capable of being raised or lowered carrying the cutters. It has their heads, and the saddles, all supported by rigid uprights.

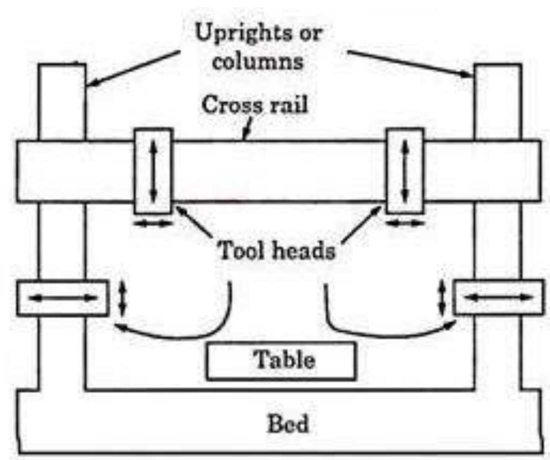


Fig. 3 Plano miller

This arrangement of driving multiple cutter spindles enables a number of work surfaces to be machined. Thereby it obtains the great reduction in production time.

The essential difference between a planer and a Plano-miller lies in the table movement. ***In a planer, the table moves to give the cutting speed. But in a Plano-milling machine, the table movement gives the feed.***

4. SPECIAL TYPES OF MILLING MACHINE

These are the special purpose milling machines, entirely different in design and construction from the conventional milling machines.

In case of **rotary table milling machine** face milling cutters are mounted on two or more vertical spindles and a number of workpieces are clamped on the horizontal surface of a circular table which rotates about a vertical axis. Different milling cutters are mounted at different heights. Loading and unloading are possible while milling is in progress.

In case of **drum milling machine** the worktable rotates about a horizontal axis and is called drum.

In a **planetary milling machine**, the work is held stationary while the revolving cutters in a planetary path. It is used to finish cylindrical surface of a workpiece internally or externally or both.

Pantograph milling machine reproduced the workpiece at any desired scale of pre-decided model.

Profiling machine duplicates full size of the template attached to the machine.

Tracer milling machine can produce any pre-decided irregular or complex shapes of dies, moulds by synchronizing movements of the cutter and tracing elements.

1.3 PRINCIPAL PARTS OF A MILLING MACHINE

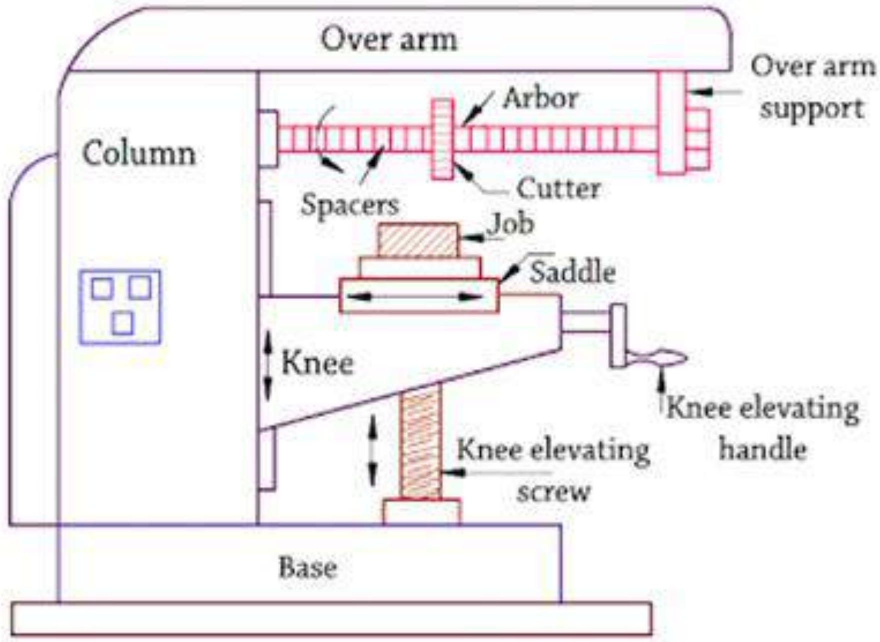


Fig. 4. Horizontal Type Column and Knee Type Milling Machine

In this machine, both the base and column are integral casting The plain milling machine has following parts:

Base:

The base is made up of cast iron. It acts as the foundation of the whole machine. It supports the entire structure of the machine. It gives strength and rigidity to the machine. It should absorb vibrations, if generated during machine. In some machines, the base is hollow and serves as a reservoir for cutting fluid.

Column:

Column is the main supporting frame mounted vertically on the base. The column houses all the driving mechanisms for the spindle and table feed. The front vertical face of the column is provided with guideways, on which the knee can slide up and down. The top of the column is accurately finished to hold an overarm that extends outward at the front of the machine.

***Knee**:*

The knee supports the saddle, table, workpiece and other clamping devices. It can slide up and down on the vertical guideways of the column. The height of knee can be adjusted by operating an elevating screw. The knee houses the feed mechanism of the table. The top face of the knee forms a slideways for saddle to provide cross travel of the table.

Saddle:

The saddle supports and carries the table. It is mounted on the guideways provided on the top of the knee and can be moved by hand wheel or by power.

Table:

The table is mounted on the saddle and it travels longitudinally. The top surface of the table is accurately machined and T-slots are provided, for clamping the workpiece and other fixtures on it. Table moves perpendicular to the direction of saddle movement.

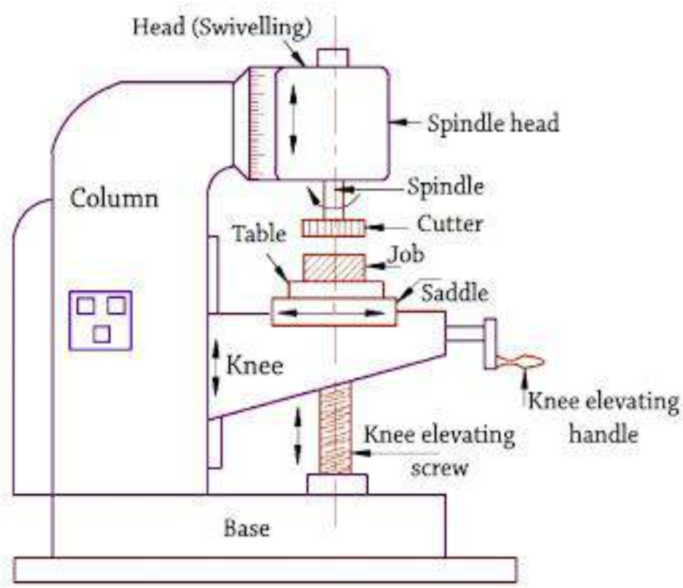
***Arbor**:*

An arbor is an extension of the machine spindle on which milling cutters are securely mounted and rotated. The arbor is provided with taper shanks for proper alignment with the machine spindle. The position of the cutter can be adjusted with spacer.

Spindle:

The spindle of the machine is located in the upper part of the column and receives power from the motor through belts and gears and transmits it to the arbor. The front end of the spindle projects from the column face and is provided with a tapered hole into which arbors may be inserted.

Working principle of Plain milling machine:



In this machine, the workpiece is placed on the table, and the table may be fed in longitudinal [back and forth], cross [in and out] and vertical [up and down] directions. The feed is longitudinal when the table is moved parallel to the spindle, and the feed is vertical when table is adjusted in a vertical plane. The table may be fed by hand or power. The plain milling machines are more rigid and heavier in construction than hand milling machine and accommodates heavy workpiece. The multi-point milling cutter is mounted on a rotating arbor. The cutter is rotating at the particular cutting speed. When the work is fed, the cutting edges of the cutter remove the metal from the workpiece. Thus the milling is done by a milling machine.

1.4 Vertical milling machine:

In Vertical milling machine, the position of the spindle head is vertical and the axis of the spindle is perpendicular to the work table. In this both the base and column are integral casting.

The vertical milling machine may have fixed head or swiveling head or slidable head spindle. The spindle head can be moved up and down over the guideways. The saddle is mounted over the knee guideways and it can be moved on the guideway horizontally either by hand or power. The knee can be moved up and down over the guideways provided on the column face. The work table is mounted on the saddle and can be moved longitudinally over the guideways provided on the top of the saddle. The workpiece can be moved both in vertical and horizontal plane. This machine is used to machine grooves, slots and flat surface using end mill and face mill cutter.

Fig. 5.

Vertical Milling machine

1.5 SPECIFICATIONS OF A MILLING MACHINE

Along with the type of a milling machine, it has to be specified by its size. Generally size of a typical milling machine is designated as given below :

- Size (dimensions) of the worktable and its movement range table length x table width as 900 x 275 mm.
- Table movements : Longitudinal travel x Cross x Vertical as 600x 200x 400 mm.
  - Above travels indicate maximum movement in a direction.
- Number of feeds available (specify their values).
- Number of spindle speeds (specify their values).
- Total power available.
- Spindle nose taper.
- Floor space required.
- Net weight.

1.6 CLASSIFICATION OF MILLING OPERATION

The milling process is broadly classified into **peripheral milling and face milling**.

In peripheral milling, the cutting edges are primarily on the circumference or periphery of the milling cutter and the milled surface is generally parallel to cutter axis.

In face milling, although the cutting edges are provided on the face as well as the periphery of the cutter, the surface generated is parallel to the face of the cutter and is perpendicular to the cutter axis. Refer to Fig. 6; in which both these process have been illustrated.

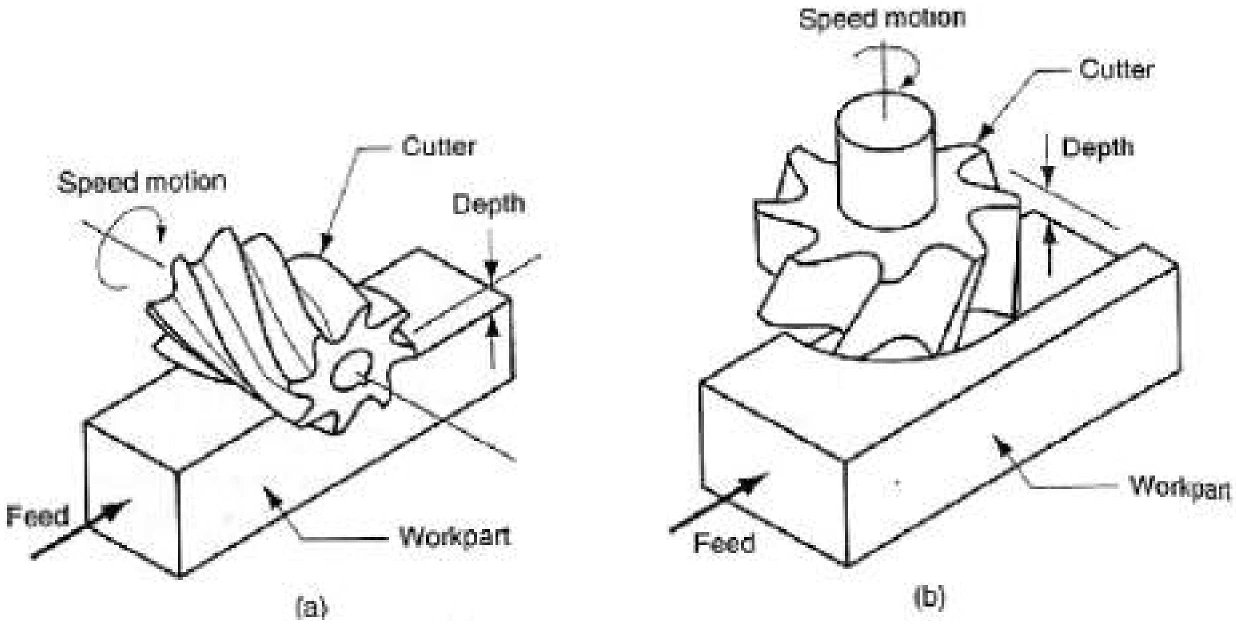


Fig. 6 Peripheral and Face Milling

1.7 PERIPHERAL MILLING

Peripheral milling is adopted for the following machining operations:

1. Slab milling to produce flat surfaces.
2. Slot milling to produce precision slots.
3. Side and face milling to machine adjacent horizontal and vertical surfaces simultaneously.
4. Form milling to produce prismatic shape of any form, *e.g.*, involute form in gear cutting.
5. Straddle milling to machine two parallel vertical faces.
6. Gang milling to machine a number of surfaces simultaneously with a set of cutters.

The various peripheral milling operations are illustrated in Fig. 7.

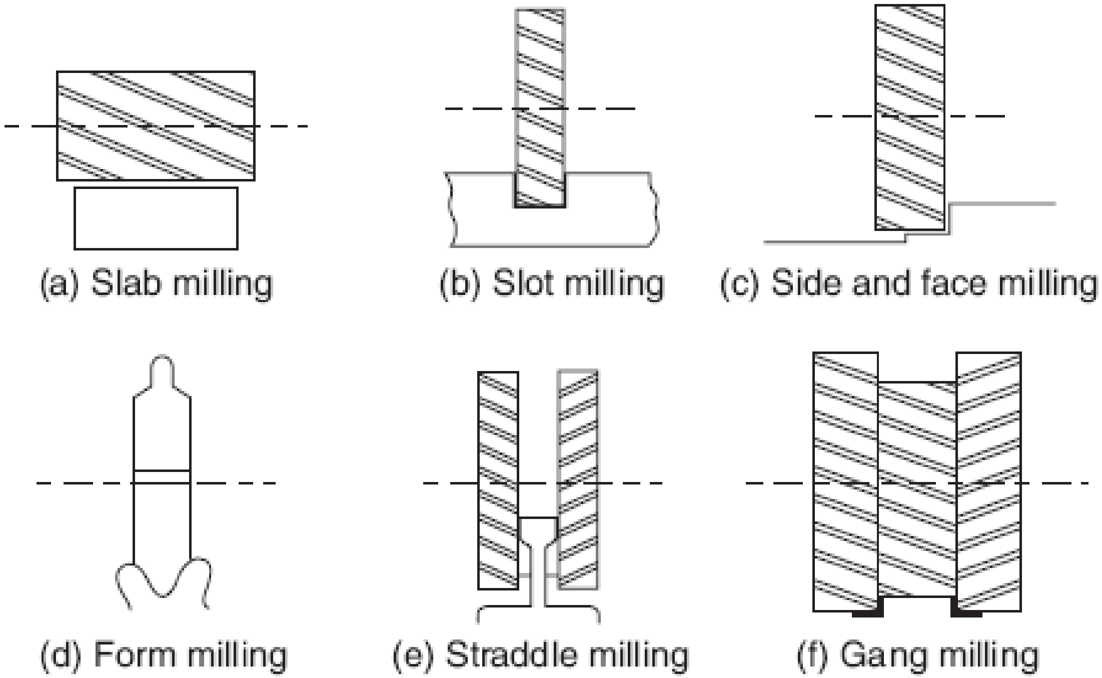


Fig. 7 Various Peripheral Milling Operations

A number of milling cutters of peripheral milling type are shown in Fig. 8. The hole and the keyway provided in the centre of all peripheral cutters is for mounting them on the arbor of a horizontal milling machine.

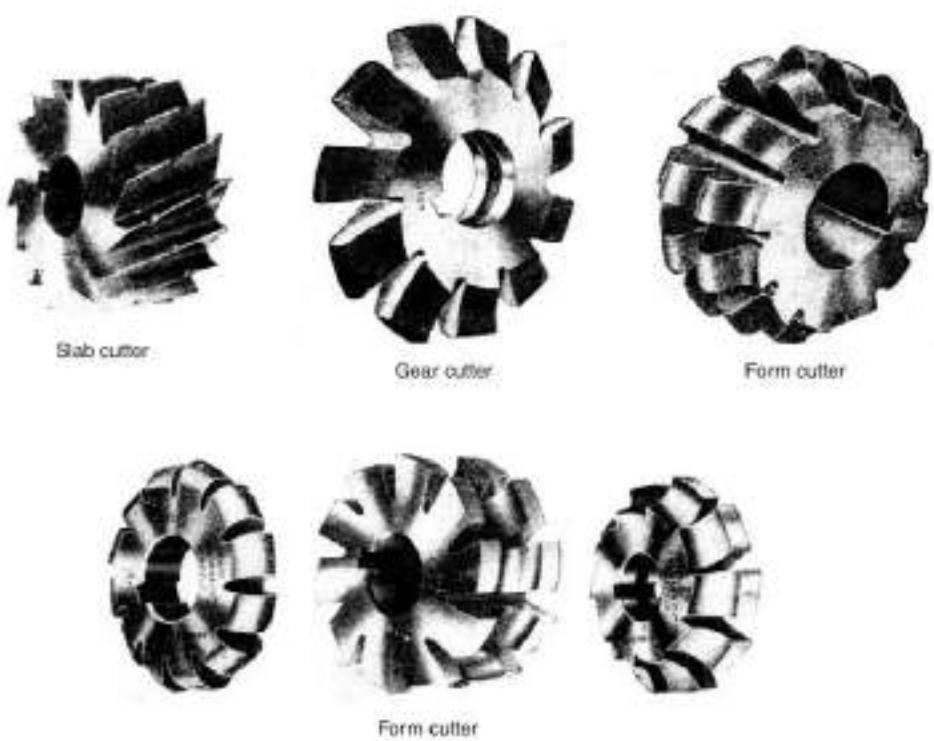


Fig. 8 Peripheral Milling Cutters

1.7.1 Classification of Peripheral Milling

Peripheral milling is also classified on the basis of the rotational direction of cutter, as up milling and down milling.

A. UP-Milling or Conventional Milling Procedure

In the up-milling or conventional milling, as shown in Fig. 9, the metal is removed in form of small chips by a cutter rotating against the direction of travel of the workpiece. In this type of milling, the chip thickness is minimum at the start of the cut and maximum at the end of cut. As a result the cutting force also varies from zero to the maximum value per tooth movement of the milling cutter. The major disadvantages of up-milling process are the tendency of cutting force to lift the work from the fixtures and poor surface finish obtained. But being a safer process, it is commonly used method of milling.

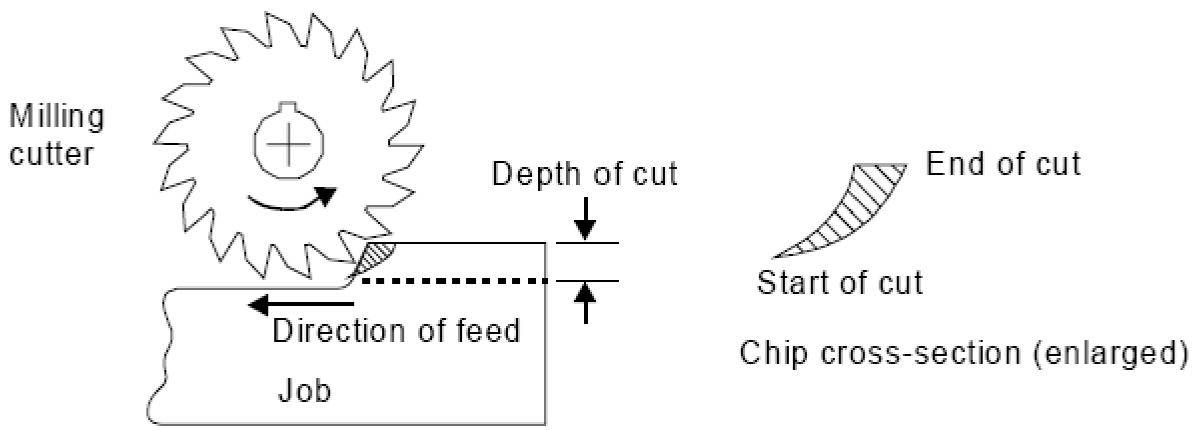


Fig. 9. Principle of Upmilling

B Down-Milling or Climb Milling

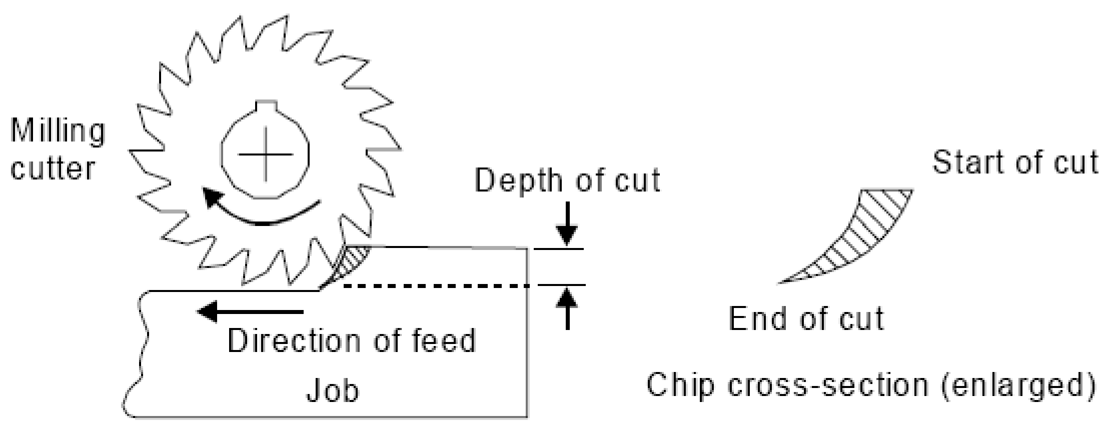


Fig.10 Principle of Down Milling

Down milling is shown in Fig. 10. It is also known as climb milling. In this method, the metal is removed by a cutter rotating in the same direction of feed of the workpiece. The effect of this is that the teeth cut downward instead of upwards. Chip thickness is maximum at the start of the cut and minimum in the end. Advantage is that slightly lower power consumption is obtainable by climb milling, since there is no need to drive the table against the cutter.

1.7.2. Difference Between Up Milling and Down Milling

no	Up milling	Down milling
1	Up milling is also termed as conventional milling. It is also called as climb up milling.	Down milling is also called as climb down milling.
2	The rotary cutter moves against the motion of feed.	The rotary cutter moves in the same direction as that of the feed.
3	Here, while cutting the workpiece from right to left, cutter rotates in anti-clockwise direction.	In down milling, while cutting the workpiece from right to left cutter rotates in clockwise direction.
4	The material from the surface or cutting chips are thrown in the upward direction that's why it is called as up milling.	Here the situation is opposite to that of up milling as cutting chips are removed in downward direction and therefore, we called it as down milling.
5	It requires a large amount of force compared to down milling.	For down milling, there is a requirement of less amount of force.
6	In up milling, you will get poorer finish.	Here, you will get the best surface finish.
7	The chip width increases from zero and then increases as the process goes on.	Chip width changes from maximum to minimum value.
8	Up milling is mostly used for rough cutting operations.	Down milling is used for finishing operations.
9	Tool wear rate is more as the direction of rotary cutter is against the feed.	Tool wear rate is less as the direction of cutter is in the same direction as that of the feed.
10	Due to increasing tool wear rate, there is reduction in tool life.	Due to reducing tool wear rate, there is enhancement in tool life.
11	It is also famous as traditional way of cutting the surface.	Nowadays, down milling is a better choice than up milling.
12	The cutting forces acts upward.	The cutting forces acts downward.
13	It is mostly used in the cutting of brass, bronze and ferrous materials.	It is mostly applicable to aluminium and aluminium alloys.

1.8 FACE MILLING

In the operation of face milling, axis of the milling cutter remains perpendicular to the surface being milled. In this case cutting action is done by cutting edges of both sides (end and out side) periphery of the milling cutter. (Fig.11)

Face milling is a combination of up cut and down cut milling operation. The points discussed earlier about up and down milling operations in peripheral milling, apply equally well to the face milling operation (refer to Fig. 12)

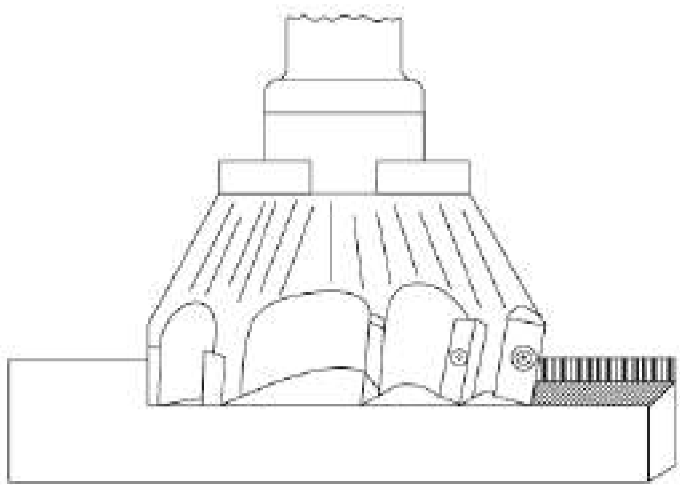


Fig. 11 Face milling

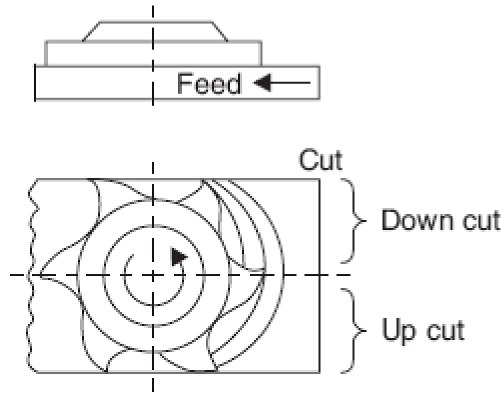


Fig. 12 Down and up cut in face milling

Depending upon the relative geometry of workpiece and milling cutter face milling is different types as described below.

1. Conventional Face Milling

In this case diameter of milling cutter is greater than the width of workpiece. The milling cutter remains over hanging on both sides of workpiece.

2. Partial Face Milling

In this case the milling cutter overhangs on the workpiece on one side only.

3. End Milling

In case of end milling thin (low diameter) cutter are used as compared to workpiece width. It is used to make slot in the workpiece.

4. Profile Milling

This is just like end milling in which the outer side periphery of a flat part is machined (milled).

5. Pocket Milling

This is a selective portion milling on the flat surface of workpiece used to make shallow packets there.

6. Surface Contouring

In this operation a ball nose cutter if feedback and forth across the workpiece along a curvilinear path at short intervals. This creates the required contours on the surface of workpiece. This operation is used to make contours of molds and dies and this time the operation is named as die sinking.

All the above described operations are indicated in Figure 13 at their respective number.

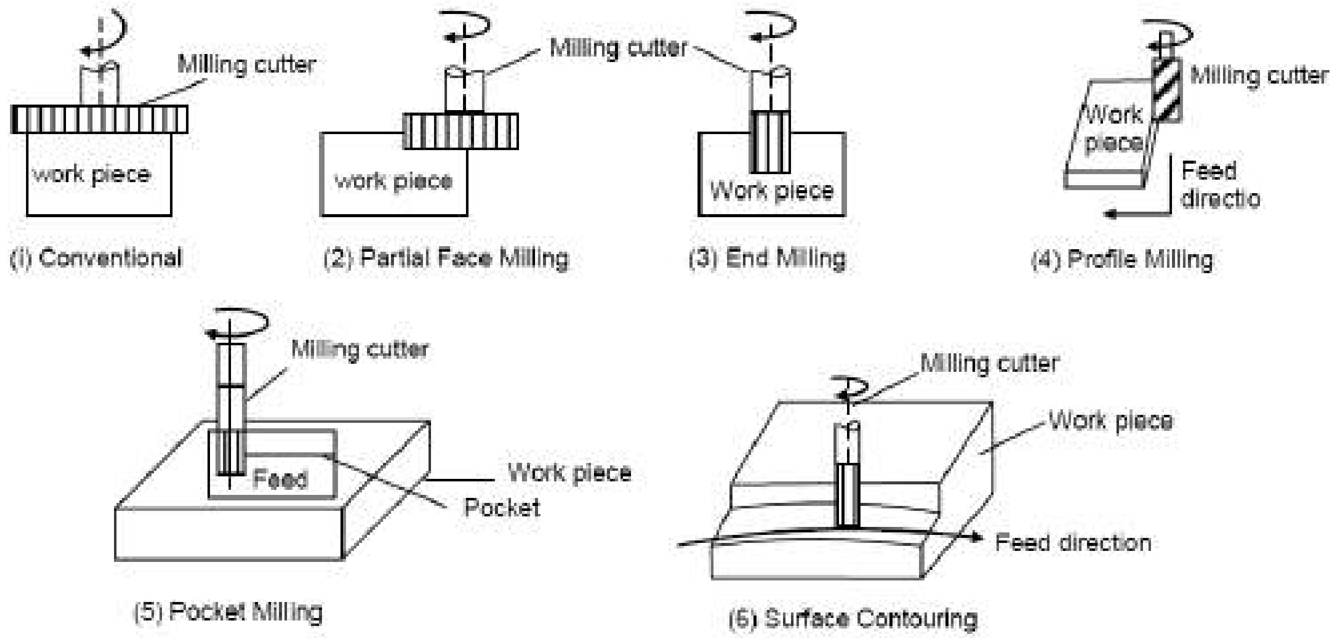


Fig. 13. Different types of Face milling

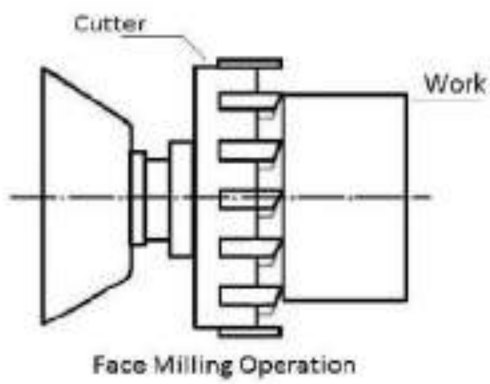
1.9. MILLING OPERATIONS

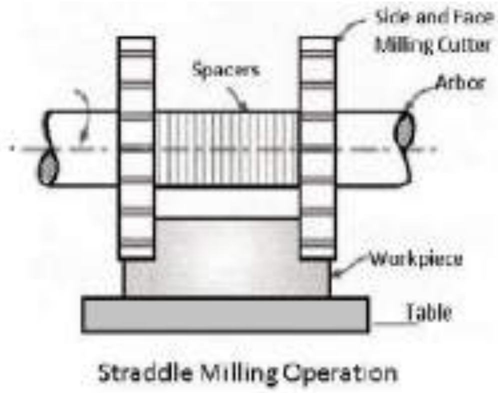
The 15 different **types of milling machine operations** are as follow:

1. Plain Milling Operation
2. Face Milling Operation
3. Side Milling Operation
4. Straddle Milling Operation
5. Angular Milling Operation
6. Gang Milling Operation
7. Form Milling Operation
8. Profile Milling Operation
9. End Milling Operation
10. Saw Milling Operation
11. Milling Keyways, Grooves and Slot
12. Gear Milling
13. Helical Milling
14. Cam Milling
15. Thread Milling

Plain Milling

- The plain milling is the most common types of milling machine operations.
- Plain milling is performed to produce a plain, flat, horizontal surface parallel to the axis of rotation of a plain milling cutter.
- The operation is also known as **slab milling.**
- To perform the operation, the work and the cutter are secured properly on the machine.
- The depth of cut is set by rotating the vertical feed screw of the table. And the machine is started after selecting the right speed and feed.





Face Milling

- The face milling is the simplest milling machine operations.
- This operation is performed by a face milling cutter rotated about an axis perpendicular to the work surface.
- The operation is carried in plain milling, and the cutter is mounted on a stub arbor to design a flat surface.
- The depth of cut is adjusted by rotating the crossfeed screw of the table.

Side Milling

- The side milling is the operation of producing a flat vertical surface on the side of a workpiece by using a side milling cutter.
- The depth of cut is set by rotating the vertical feed screw of the table.

Straddle Milling

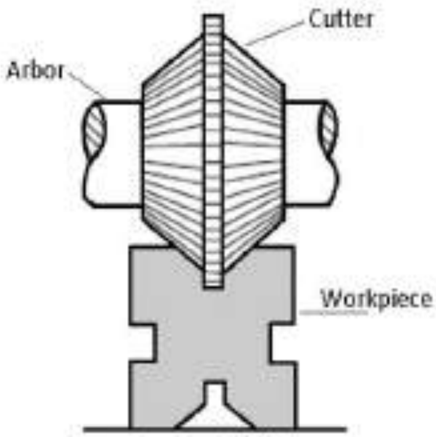
- The straddle milling is the operation of producing a flat vertical surface on both sides of a workpiece by using two side milling cutters mounted on the same arbor.
- Distance between the two cutters is adjusted by using suitable spacing collars.
- The straddle milling is commonly used to design a square or hexagonal surfaces.

Angular Milling

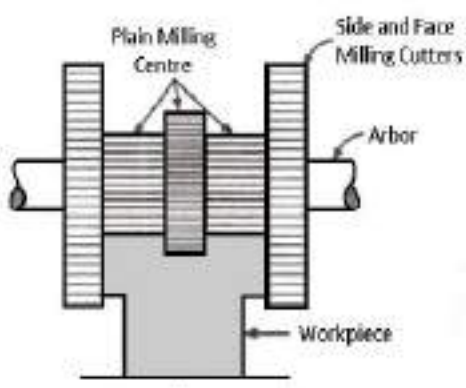
- The angular milling is the operation of producing an angular surface on a workpiece other than at right angles of the axis of the milling machine spindle.
- The angular groove may be single or double angle and may be of varying included angle according to the type and contour of the angular cutter used.
- One simple example of angular milling is the production of V-blocks.

Gang Milling

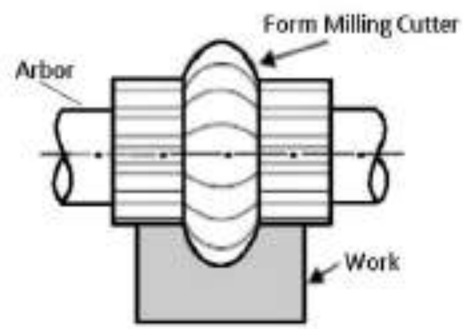
- The gang milling is the operation of machining several surfaces of a workpiece simultaneously by feeding the table against a number of cutters having the same or different diameters mounted on the arbor of the machine.
- The method saves much of machining time and is widely used in repetitive work.
- Cutting speed of a gang of cutters is calculated from the cutter of the largest diameter.



Angular Milling



Gang Milling Operation



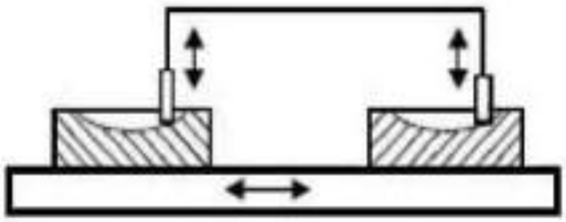
Form Milling Operation

Form Milling

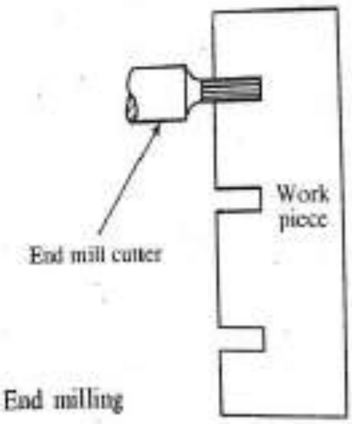
- The form milling is the operation of producing the irregular contour by using form cutters.
- The irregular shape may be convex, concave, or of any other shape. After machining, the formed surface is inspected by a template gauge.
- Cutting speed for form milling is 20% to 30% less than that of the plain milling.

Profile Milling

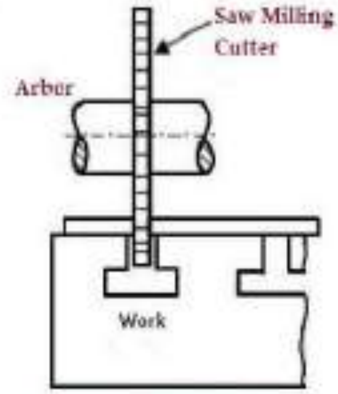
- The profile milling is the operation of reproduction an outline of a template or complex shape of a master dies on a workpiece.
- Different cutters are used for profile milling. An end mill is one of the widely used milling cutters in profile milling work.



Profile Milling Operation



End milling



Saw Milling Operation

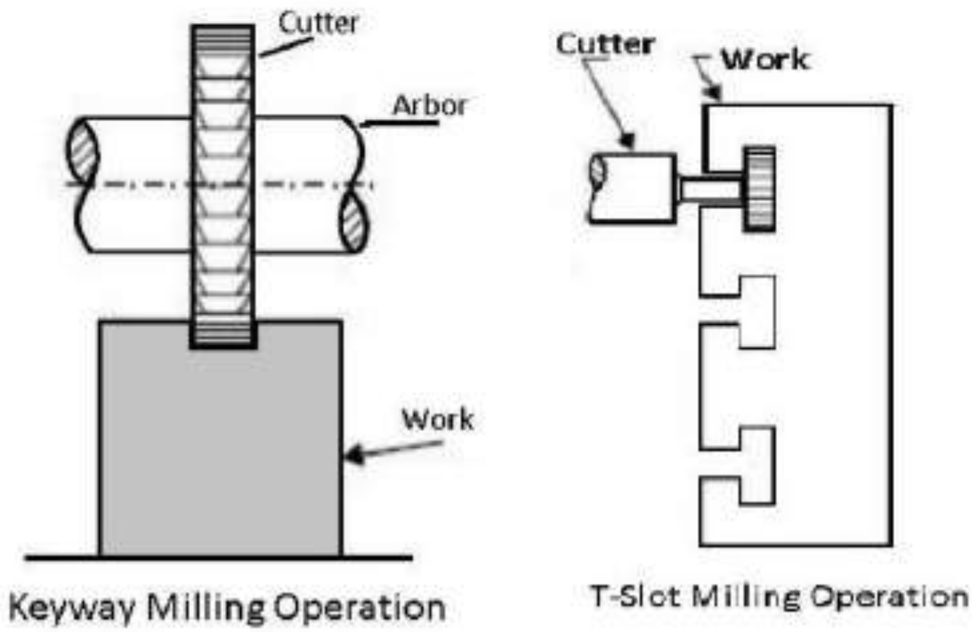
End Milling

- The end milling is the operation of producing a flat surface which may be vertical, horizontal or at an angle in reference to the table surface.
- The cutter used is an end mill. The end milling cutters are also used for the production of slots, grooves or keyways.
- A vertical milling machine is more suitable for end milling operation.

Saw Milling

- Saw-milling is the operation of producing narrow slots or grooves on a workpiece by using a saw-milling cutter.
- The saw-milling also performed for complete parting-off operation.
- The cutter and the workpiece are set in a manner so that the cutter is directly placed over one of the T-slots of the table.

Milling Keyways, Grooves and Slots

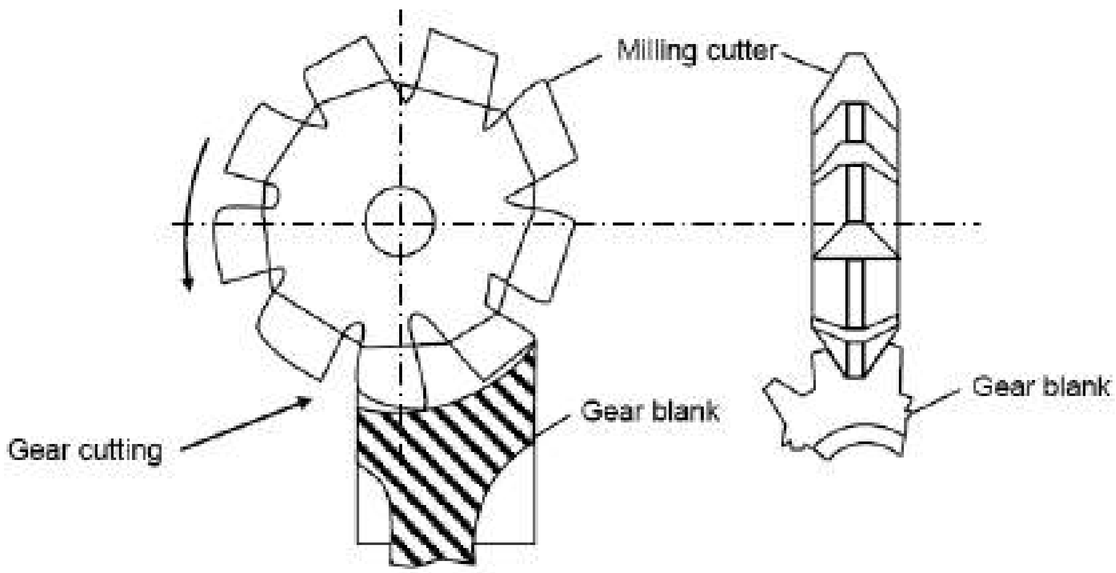


- The operation of producing of keyways, grooves and slots of varying shapes and sizes can be performed in a milling machine.
- It is done by using a plain milling cutter, a metal slitting saw, an end mill or by a side milling cutter.
- The open slots can be cut by a plain milling cutter, a metal slitting saw, or by a side milling cutter. The closed slots are produced by using endmills.

- A dovetail slot or T-slot is manufactured by using special types of cutters designed to give the required shape on the workpiece.
- The second slot is cut at right angles to the first slot by feeding the work past the cutter.
- A woodruff key is designed by using a woodruff key slot cutter.
- Standard keyways are cut on the shaft by using side milling cutters or end mills.
- The cutter is set exactly at the centre line of the workpiece and then the cut is taken.

Gear Cutting

- The gear cutting operation is performed in a milling machine by using a form-relieved cutter. The cutter may be a cylindrical type or end mill type.
- The cutter profile fits exactly with the tooth space of the gear.
- Equally spaced gear teeth are cut on a gear blank by holding the work on a universal diving head and then indexing it.



Gear Cutting Operation

Helical Milling

- The helical milling is the operation of producing helical flutes or grooves around the periphery of a cylindrical or conical workpiece.
- The operation is performed by rotating the table to the required helix angle. And then by rotating and feeding the workpiece against rotary cutting edges of a milling cutter.
- Production of the helical milling cutter, helical gears, cutting helical grooves or flutes on a drill blank or a reamer.

Cam Milling

- The cam milling is the operation of producing cams in a milling machine by the use of universal dividing head and a vertical milling attachment. The cam blank is mounted at the end of the dividing head spindle and an end mill is held in the vertical milling attachment.
- The axis of the cam blank and the end mill spindle should always remain parallel to each other when setting for cam milling. The dividing head is geared to the table feed screw so that the cam is rotated

Thread Milling Operation

The operation thread milling produces threads using thread milling centres. This operation needs three simultaneous movements revolving movement of cutter, simultaneous longitudinal movement of cutter, feed movement to the workpiece through table. For each thread, the revolving cutter is fed longitudinal by a distance equal to pitch of the thread. Depth of cut is normally adjusted equal to the full depth of threads.

1.10 MILLING CUTTERS

Milling cutters are classified into different categories depending on different criteria as described below :

a. According to the Construction of Milling Cutter

(a) Solid milling cutter

(b) Inserted teeth cutter

(c) Tipped solid cutter

Solid cutter consists of teeth integral with the cutter body, in tipped cutter, teeth are made of cemented carbide or satellite, teeth are brazed to steel cutter body called shank. Inserted teeth cutter are larger in diameter, teeth of hard material are inserted and secured in the shank.

b. According to Relief Characteristics of the Cutter Teeth

(a) Profile relieved cutter

(b) Form relieved cutter

In case of profile relieved cutter, a relief to cutting edges is provided by grinding a narrow land at their back. In case of form relieved cutters a curved relief is provided at the back of the cutting edges.

c. According to Method of Mounting the Cutters

(a) Arbor type

(b) Facing cutter

(c) Shank cutter

Arbor type cutters have a central hole and keyways for their mounting on arbor. Shank type cutters are provided with straight or tapered shanks inserted into the spindle nose and clamped there. Facing type milling cutter are used to produce flat surfaces. These are balled or attached to the spindle nose or the face of a short arbor (stub arbor).

d. According to Direction of Rotation of the Cutter

(a) Right hand rotational cutter

(b) Left hand rotational cutter

A right hand rotational cutter rotates in an anticlockwise direction when viewed from end of the spindle while left hand rotational cutter rotates in a clockwise direction.

e. According to the Direction of Helix of the Cutter Teeth

(a) Parallel straight teeth

(b) Right hand helical

(c) Left hand helical

(d) Alternate helical teeth

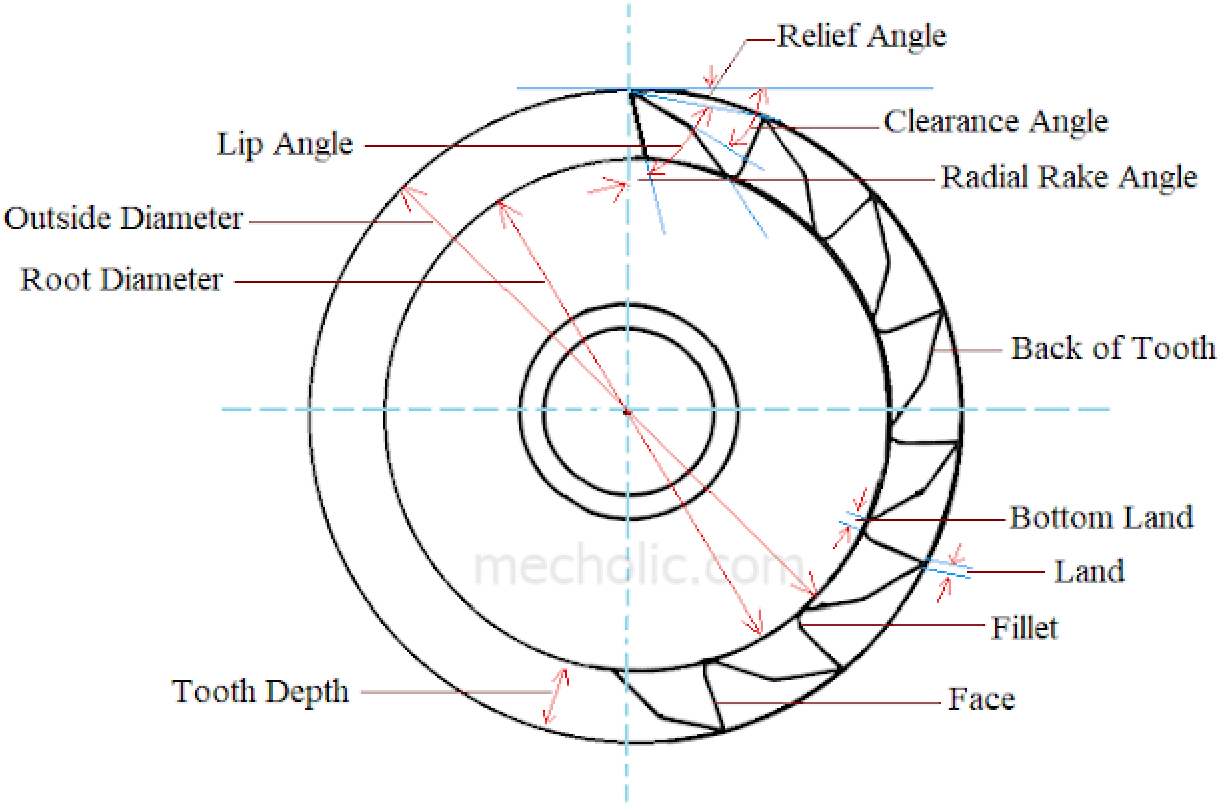
Parallel or straight teeth cutter consists of teeth parallel to axis of rotation of the cutter with zero helix angle. In case of right hand and left hand helical teeth cutters, teeth cut at an angle to the axis of rotation of the cutter. Teeth have opposite inclination in both the cutters. Alternate helical teeth cutter has alternate teeth of right hand and left hand helical teeth cutters.

f. According to Purpose of Use of the Cutter

(a) Standard milling cutter

(b) Special milling cutter

1.11 NOMENCLATURE OF A MILLING CUTTER



Periphery:

It is locus of the cutting edge of the cutter and is an imaginary cylindrical surface enveloping the tips of the cutting teeth. It determines the diameter of the cutter.

Cutting Edge:

Cutting edge of a milling cutter is the only portion that touches the work. It is the intersection of the tooth face and the tooth flank of back surface. The cutting edge is generally a line which may be straight, helical or some complex profile.

Gash:

It is the chip space or flute between the back of one tooth and the face of the next tooth.

Face:

It is that portion of the gash adjacent to the cutting edge on which the chip impinges as it is cut from the work.

Fillet:

It is the curved surface at the bottom of gash which joins the face of one tooth to the back of the tooth immediately ahead.

Land:

This is the narrow surface back of the cutting edge resulting from providing a clearance angle. It never touches the work and is less than 1.5 mm in width.

Tooth Face:

This is the surface upon which the chip is formed when the cutter is cutting. It may be either flat or curved.

Back of Tooth:

The back or flank of the tooth is created by the gullet and relief angle (secondary clearance). It may be flat or curved surface.

Lip Angle:

It is the inclined angle between the land and the face of the tooth. It is also equal to the angle between the tangent to the back of the cutting edge and the face of the tooth.

Clearance Angle (Primary Clearance):

This is the angle between a line through the surface of the land and a tangent to the periphery at the cutting edge. It is necessary to prevent the back of the tooth from rubbing against the work. It is always positive and should not be small so as to weaken the cutting edge of the tooth.

Relief Angle (Secondary Clearance):

A secondary clearance is generally ground at back of the land to keep the width of the land within the proper limits. It is necessary because after several sharpening of the cutter, the width of the land increases to a point where it begins to interfere with the work. It is usually 3° greater than the clearance angle (Primary clearance). Not all cutters have secondary clearance.

Rake:

If the face of a milling cutter lies along a radius of the cutter, it is said to have zero rake. If the face of cutter lies along a line on either side of the radius, it has a positive or negative rake. If this line lies on the same side of the radius as tooth, it is a positive rake. If it lies on the opposite side of the radius of tooth, it is a negative rake.

DRILLING MACHINES

1.1 INTRODUCTION

Drilling is a metal cutting process carried out by a rotating cutting tool to make circular holes in solid materials. Tool which makes hole is called as drill bitν or twist drill. The machine used for this purpose is called drilling machine. Although it was primarily designed to originate a hole, it can perform a number of similar operations. In a drilling machine holes may be drilled quickly and at a low cost. As the machine tool exerts vertical pressure to originate a hole it is also called drill press.

1.2 CONSTRUCTION OF DRILLING MACHINE

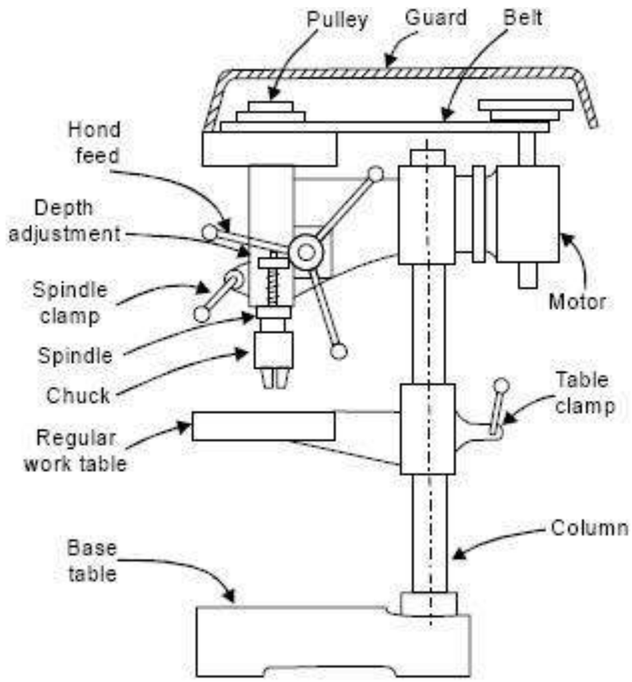


Fig. 1 Construction of Drilling Machine

In drilling machine the drill is rotated and fed along its axis of rotation in the stationary workpiece. Different parts of a drilling machine are shown in Fig. 1 and are discussed below:

(i) The head containing electric motor, V-pulleys and V-belt which transmit rotary motion to the drill spindle at a number of speeds.

(ii) Spindle is made up of alloy steel. It rotates as well as moves up and down in a sleeve. A pinion engages a rack fixed onto the sleeve to provide vertical up and down motion of the spindle and hence the drill so that the same can be fed into the workpiece or withdrawn from it while drilling. Spindle speed or the drill speed is changed with the help of V-belt and V-step-pulleys. Larger drilling machines are having gear boxes for the said purpose.

(iii) Drill chuck is held at the end of the drill spindle and in turn it holds the drill bit.

(iv) Adjustable work piece table is supported on the column of the drilling machine. It can be moved both vertically and horizontally. Tables are generally having slots so that the vise or the workpiece can be securely held on it.

(v) Base table is a heavy casting and it supports the drill press structure. The base supports the column, which in turn, supports the table, head etc.

(vi) Column is a vertical round or box section which rests on the base and supports the head and the table. The round column may have rack teeth cut on it so that the table can be raised or lowered depending upon the workpiece requirements. This machine consists of following parts

1. Base
2. Pillar
3. Main drive
4. Drill spindle
5. Feed handle
6. Work table

1.3 TYPES OF DRILLING MACHINE

Drilling machines are classified on the basis of their constructional features, or the type of work they can handle. The various types of drilling machines are:

(1) Portable drilling machine

(2) Sensitive drilling machine

- (a) Bench mounting
- (b) Floor mounting

(3) Upright drilling machine

- (a) Round column section
- (b) Box column section machine

(4) Radial drilling machine

- (a) Plain
- (b) Semiuniversal
- (c) Universal

(5) Gang drilling machine

(6) Multiple spindle drilling machine

(7) Automatic drilling machine

(8) Deep hole drilling machine

- (a) Vertical
- (b) Horizontal

Few commonly used drilling machines are described as under.

1. Portable Drilling Machine

A portable drilling machine is a small compact unit and used for drilling holes in workpieces in any position, which cannot be drilled in a standard drilling machine. It may be used for drilling small diameter holes in large castings or weldments at that place itself where they are lying. Portable drilling machines are fitted with small electric motors, which may be driven by both A.C. and D.C. power supply. These drilling machines operate at fairly high speeds and accommodate drills up to 12 mm in diameter.

Fig. 2 Portable Drilling Machine

2. **Sensitive Drilling Machine**

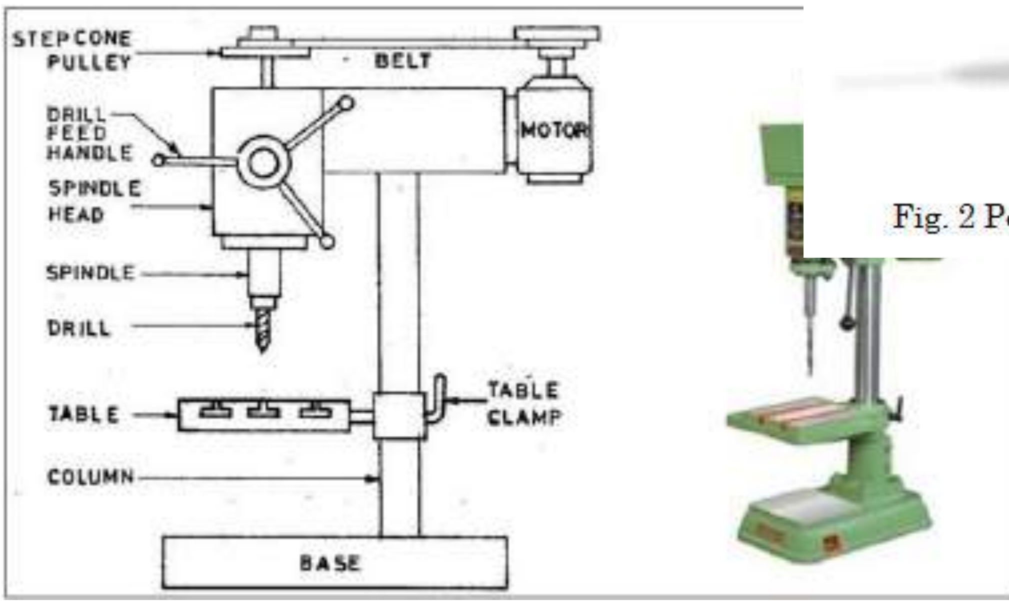


Fig. 3 Sensitive drilling Machine

It is a small machine used for drilling small holes in light jobs. In this drilling machine, the workpiece is mounted on the table and drill is fed into the work by purely hand control. High rotating speed of the drill and hand feed are the major features of sensitive drilling machine. As the operator senses the drilling action in the workpiece, at any instant, it is called sensitive drilling machine. A sensitive drilling machine consists of a horizontal table, a vertical column, a head supporting the motor and driving mechanism, and a vertical spindle.

Drills of diameter from 1.5 to 15.5 mm can be rotated in the spindle of sensitive drilling machine. Depending on the mounting of base of the machine, it may be classified into following types:

1. Bench mounted drilling machine, and
2. Floor mounted drilling machine

3. **Upright Drilling Machine**

The upright drilling machine is larger and heavier than a sensitive drilling machine. It is designed for handling medium sized workpieces and is supplied with **power feed arrangement**. In this machine a large number of spindle speeds and feeds may be available for drilling different types of work. Upright drilling machines are available in various sizes and with various drilling capacities (ranging up to 75 mm diameter drills). The table of the machine also has different types of adjustments. Based on the construction, there are two general types of upright drilling machine:

(1) Round column section or pillar drilling machine.

(2) Box column section.

The round column section upright drilling machine consists of a round column whereas the upright drilling machine has box column section. The other constructional features of both are same. Box column machines possess more machine strength and rigidity as compared to those having round section column.

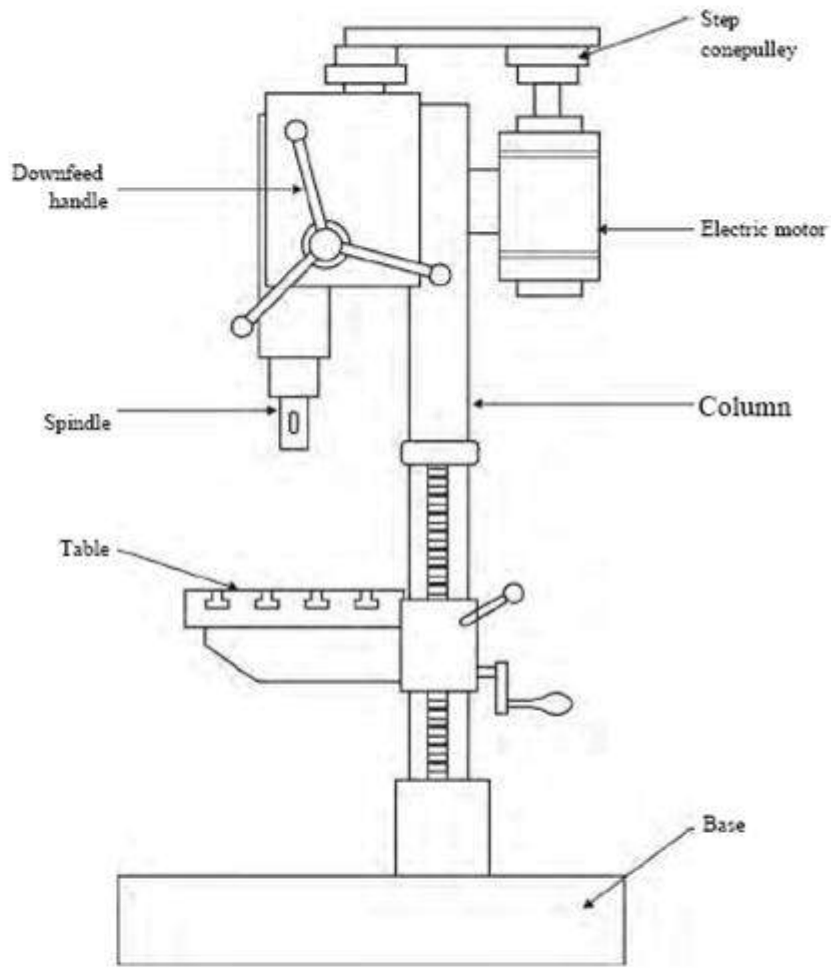


Fig.4 Upright Drilling Machine

The main parts of a upright drilling machine are : base, column, table and drill head.

Base : Base is made of cast iron as it can withstand vibrations set by the cutting action. It is erected on the floor of the shop by means of bolts and nuts. It is the supporting member as it supports column and other parts on it. The top of the base is accurately machined and has 'T'- slots. When large workpieces are to be held, they are directly mounted on the base.

Column: Column stands vertically on the base and supports the work table and all driving mechanisms. It is designed to withstand the vibrations set up due to the cutting action at high speeds.

Table: Table is mounted on the column and can be adjusted up and down on it. It is provided with 'T'-slots for Workpieces to be mounted directly on it. Table may have the following adjustments Vertical adjustment obtained by the rack on the column and a pinion in the table Circular adjustment about its own axis After the required adjustments are made, the table is clamped in position.

Drill head: The drill head is mounted on the top of the column. It houses the driving and feeding mechanism of the spindle. The spindle can be provided with hand or power feed. There are separate hand wheels for quick hand feed and sensitive hand feed. The handle is spring loaded so that the drill spindle is released from the work when the operation is over.

4. **Radial Drilling Machine**

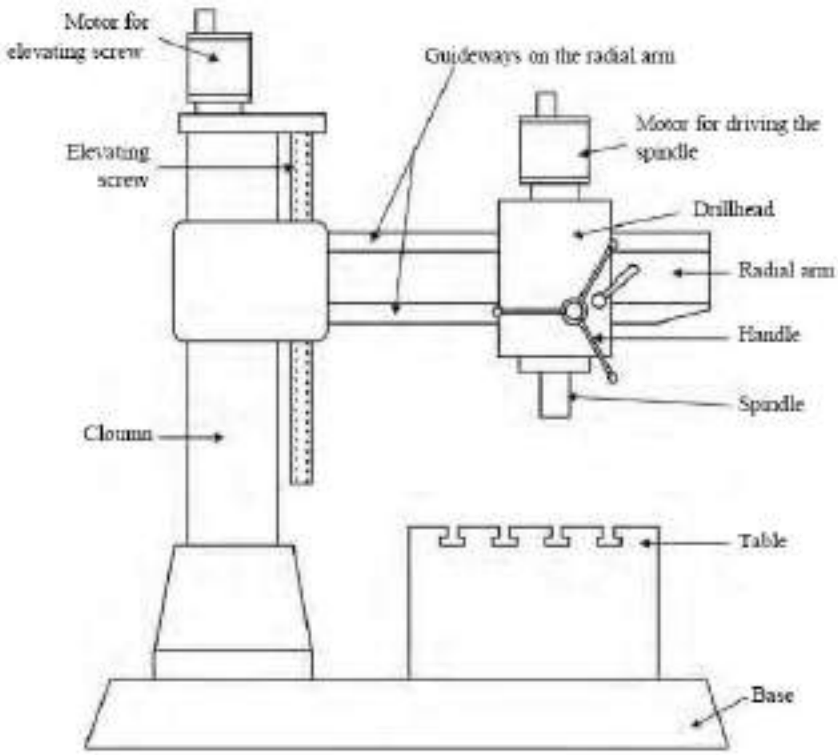


Fig. 5 Radial Drilling Machine

Fig. 5 illustrates a radial drilling machine. The radial drilling machine consists of a heavy, round vertical column supporting a horizontal arm that carries the drill head. Arm can be raised or lowered on the column and can also be swung around to any position over the work and can be locked in any position. The drill head containing mechanism for rotating and feeding the drill is mounted on a radial arm and can be moved horizontally on the guide-ways and clamped at any desired position. These adjustments of arm and drilling head permit the operator to locate the drill quickly over any point on the work. The table of radial drilling machine may also be rotated through 360 deg. The maximum size of hole that the machine can drill is not more than 50 mm. Powerful drive motors are geared directly into the head of the machine and a wide range of power feeds are available as well as sensitive and geared manual feeds. The radial drilling machine is used primarily for drilling medium to large and heavy workpieces. Depending on the different movements of horizontal arm, table and drill head, the upright drilling machine may be classified into following types-

1. Plain radial drilling machine
2. Semi universal drilling machine, and
3. Universal drilling machine.

In a plain radial drilling machine, provisions are made for following three movements -

1. Vertical movement of the arm on the column,
2. Horizontal movement of the drill head along the arm, and
3. Circular movement of the arm in horizontal plane about the vertical column.

In a semi universal drilling machine, in addition to the above three movements, the drill head can be swung about a horizontal axis perpendicular to the arm. In universal machine, an additional rotary movement of the arm holding the drill head on a horizontal axis is also provided for enabling it to drill on a job at any angle.

The main parts of a upright drilling machine are: base, column, radial arm, table and drill head

Base

The base is a large rectangular casting and is mounted on the floor of the shop. Its top is accurately finished to support a column at one end and the table at the other end. 'T'-slots are provided on it for clamping workpieces.

Column

The column is a cylindrical casting, which is mounted vertically at one end of the base. It supports the radial arm and allows it to slide up and down on its face. The vertical adjustment of the radial arm is effected by rotating a screw passing through a nut attached to the arm. An electric motor is mounted on the top of the column for rotating the elevating screw.

Radial arm

The radial arm is mounted on the column parallel to the base and can be adjusted vertically. The vertical front surface is accurately machined to provide guideways for the drillhead. The drillhead can be adjusted along these guideways according to the location of the work. In some machines, a separate motor is provided for this movement. The arm may be swung around the column. It can also be moved up and down to suit Workpieces of different heights.

Drill head

The drill head is mounted on the radial arm and houses all mechanism for driving the drill at different speeds and at different feed. A motor is mounted on top of the drill head for this purpose. To adjust the position of drill spindle with respect to the work, the drill head may be made to slide on the guideways of the arm. The drill head can be clamped in position after the spindle is properly adjusted.

4. **Gang drilling machine**

Gang drilling machine has a long common table and a base. Four to six drill heads are placed side by side. The drill heads have separate driving motors. This machine is used for production work.

A series of operations like drilling, reaming, counter boring and tapping may be performed on the work by simply shifting the work from one position to the other on the work table. Each spindle is set with different tools for different operations.

Fig.6 shows a gang drilling machine.

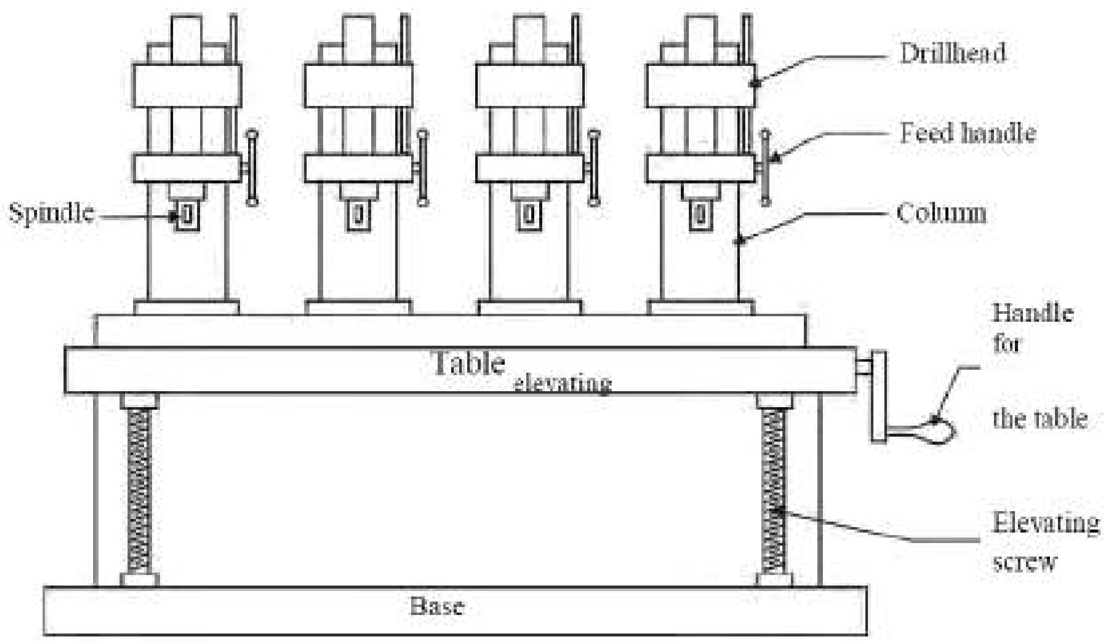


Fig. 6. Gang Drilling Machine

6 Multiple spindle drilling machine

This machine is used for drilling a number of holes in a workpiece simultaneously and for reproducing the same pattern of holes in a number of identical pieces. A multiple spindle drilling machine also has several spindles. A single motor using a set of gears drives all the spindles. All the spindles holding the drills are fed into the work at the same time. The distances between the spindles can be altered according to the locations where holes are to be drilled. Drill jigs are used to guide the drills.

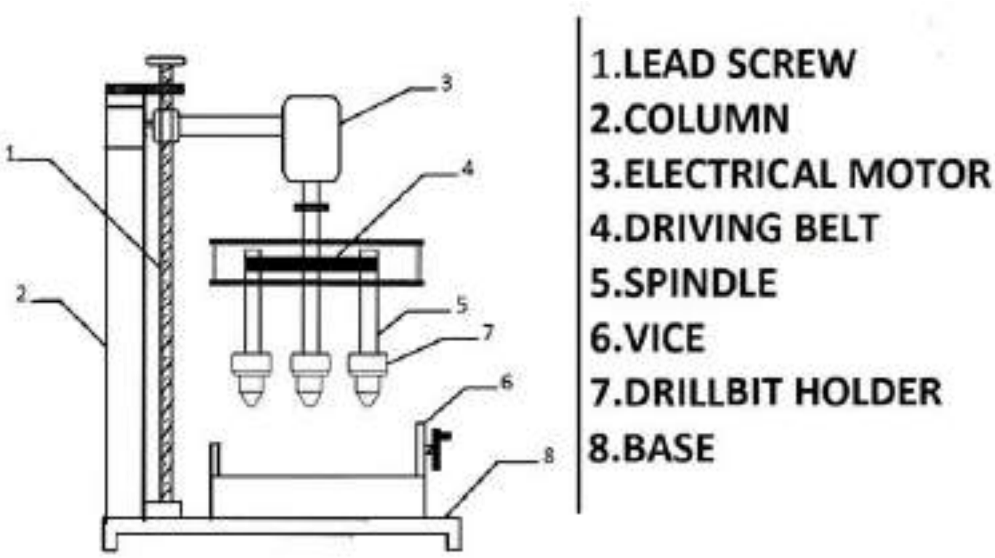


Fig. 7. Multispindle drilling machine

7 Deep hole drilling machine

A special machine and drills are required to drill deeper holes in barrels of gun, spindles and connecting rods. The machine designed for this purpose is known as deep hole drilling machine. High cutting speeds and less feed are necessary to drill deep holes. A non-rotating drill is fed slowly into the rotating work at high speeds. Coolant should be used while drilling in this machine. There are two different types of deep hole drilling machines

1. Vertical type 2. Horizontal type

8. Turret type drilling machine

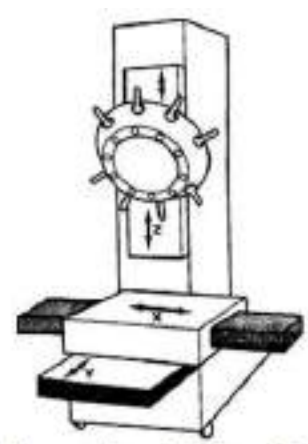


Fig.8. Turret Type Drilling Machine

Fig. 8 schematically shows a typical turret type drilling machine. Turret drilling machine is structurally rigid column type drilling machine but is more productive like gang drilling machine by having a pentagon or hexagon turret. The turret that holds a number of drills and similar tools, is indexed and moved up and down to perform quickly the desired series of operations progressively. These drilling machines are available with varying degree of automation both fixed and flexible type

1.4 Size of a drilling machine

Drilling machines are specified according to their type.

A portable drilling machine is specified by the maximum diameter of the drill that it can handle.

The size of the sensitive and upright drilling machines are specified by the size of the largest workpiece that can be centered under the spindle. It is slightly smaller than twice the distance between the face of the column and the axis of the spindle.

Particulars such as maximum size of the drill that the machine can operate, diameter of the table, maximum travel of the spindle, numbers and range of spindle speeds and feeds available, morse taper number of the drill spindle, floor space required, weight of the machine, power input are also needed to specify the machine completely. The size of the radial drilling machine is specified by the diameter of the column and length of the radical arm.

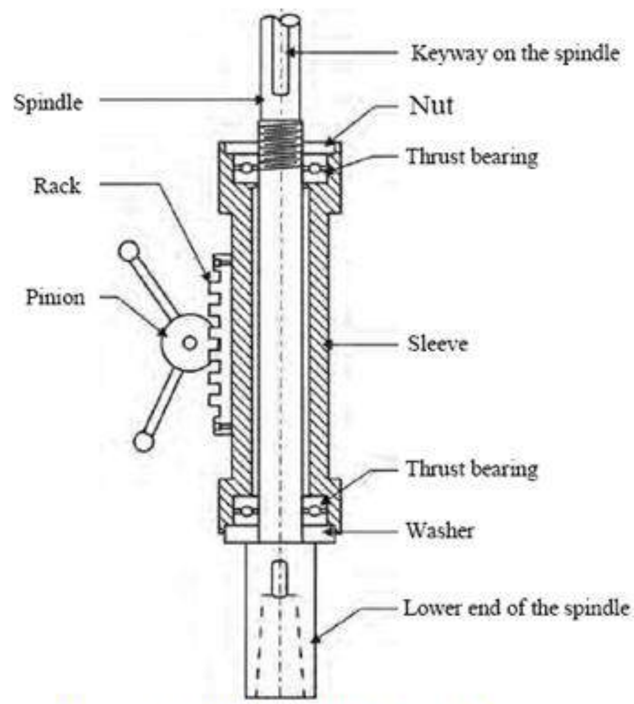


Fig. 9 Drill Spindle Assembly

1.5 DRILL SPINDLE ASSEMBLY

A drill spindle assembly is illustrated in Fig. 9. The drill spindle is a vertical shaft, which holds the drill. A long keyway is cut on the spindle and a sliding key connects it with a bevel gear or a stepped cone pulley. It receives motion from the driving motor. The spindle rotates within a non-rotating sleeve known as quill. The spindle and the sleeve are connected by a thrust bearing.

Rack teeth are cut on the outer surface of the quill. The sleeve (quill) may be moved up and down by rotating a pinion which meshes with the rack. This movement is given to the spindle for providing the required feed. As there is a long keyway on top of the spindle, it is connected to the driving mechanism even during the feed movement.

A morse taper hole is provided at the lower end of the spindle. It is useful in accommodating a taper shank drill. The tang of the drill fits into a slot provided at the end of the taper hole. To remove the drill from the spindle a drift may be pushed through the slot. The spindle drive is obtained in three methods. They are:

- Step cone pulley drive
- Step cone pulley with back gear arrangement
- Gear box drive

1.6 DRILL TOOL NOMENCLATURE

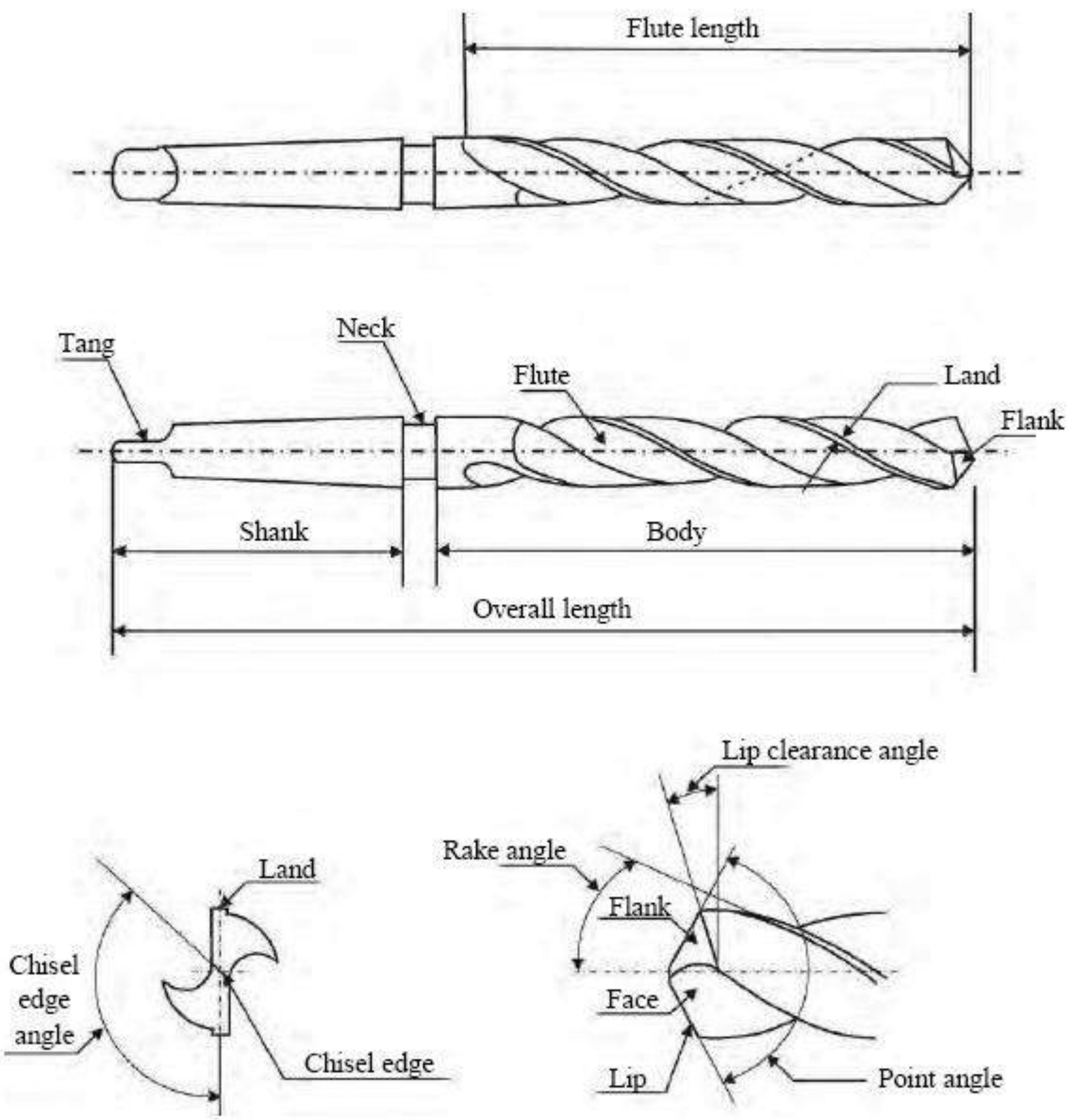


Fig. 10 Twist Drill Nomenclature

Twist drill elements

Axis	:	**The** longitudinal centre line of the drill.
Body	:	That portion of the drill extending from its extreme point to the commencement of the neck, if present, otherwise extending to the commencement of the shank.
Body clearance	:	That portion of the body surface which is reduced in diameter to provide diameteral clearance.
Chisel edge	:	The edge formed by the intersection of the flanks. The chisel edge is also sometimes called dead centre
Chisel edge corner	:	The corner formed by the intersection of a lip and the chisel edge.
Face	:	The portion of the flute surface adjacent to the lip on which the chip impinges as it is cut from the work.
Flank	:	That surface on a drill point which extends behind the lip to the following flute.
Flutes	:	The groove in the body of the drill which provides lip. The functions of the flutes are: 1. To form the cutting edges. 2. To allow the chips to escape. 3. To cause the chips to curl. 4. To permit the cutting fluid to reach the cutting edges.
Heel	:	The edge formed by the intersection of the flute surface and the body clearance.
Lands	:	The cylindrically ground surface on the leading edges of the drill flutes. The width of the land is measured at right angles to the flute helix.
Lip (cutting edge)	:	The edge formed by the intersections of the flank and face. The requirements of the drill lip are: 1. Both lips should be at the same angle of inclination (59^0) with the drill axis. 2. Both lips should be of equal length. 3. Both lips should be provided with the correct clearance
Neck	:	The diametrically undercut portion between the body and the shank of the drill. Diameter and other particulars of the drill are engraved at the neck.
Outer corner	:	The corner formed by the intersection of the flank and face.
Point	:	The sharpened end of the drill, which is shaped to produce lips, faces, flanks and chisel edge
Shank	:	That part of the drill by which it is held and driven. The most common types of shank are the taper shank and the straight shank.
Tang	:	The flattened end of the taper shank intended to fit into a drift slot in the spindle, socket or drill holder. The tang ensures positive drive of the drill from the spindle.
Web	:	The central portion of the drill situated between the roots of the flutes and

		extending from the point toward the shank; the point end of the web or core forms the chisel edge
Linear dimensions Back taper (longitudinal clearance)	:	It is the reduction in diameter of the drill from the point towards the shank. This permits all parts of the drill behind the point to clear and not rub against the sides of the hole being drilled. The taper varies from 1:4000 for small diameter drills to 1:700 for larger diameters.
Body clearance diameter	:	The diameter over the surface of the drill body which is situated behind the lands.
Depth of body clearance	:	The amount of radial reduction on each side to provide body clearance.
Diameter	:	The measurement across the cylindrical lands at the outer corners of the drill.
Flute length	:	The axial length from the extreme end of the point to the termination of the flute at the shank end of the body.
Lead of helix	:	The distance measured parallel to the drill axis between the corresponding points on the leading edge of the flute in one complete turn of the flute.
Lip length	:	The minimum distance between the outer corner and the chisel edge corner of the lip.
Overall length	:	The length over the extreme ends of the point and the shank of the drill.
Web (core) taper	:	The increase in the web or core thickness from the point of the drill to the shank end of the flute. This increasing thickness gives additional rigidity to the drill and reduces the cutting pressure at the point end.
Web thickness	:	The minimum dimension of the web or core measured at the point end of the drill.
Drill angles Chisel edge angle	:	The obtuse angle included between the chisel edge and the lip as viewed from the end of the drill.
Helix angle or rake angle	:	This is the angle formed by the leading edge of the land with a plane having the axis of the drill.
Point angle	:	This is the angle included between the two lips.
Lip clearance angle	:	The angle formed by the flank and a plane at right angles to the drill axis.

1.7 Drilling machine 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. Counterboring
5. Countersinking
6. Spot facing
7. Tapping
8. Trepanning

1. Drilling

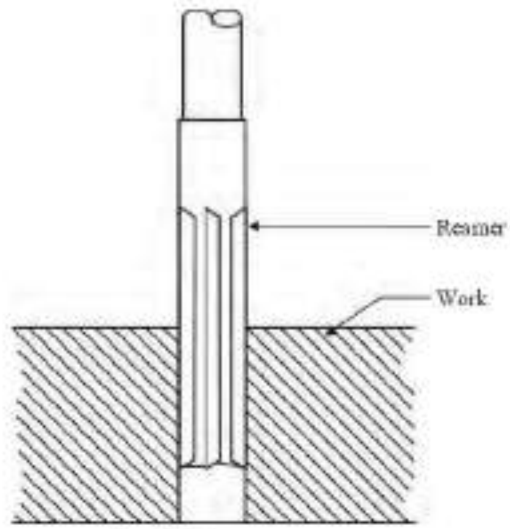


Fig. 12 Reaming

Drilling is the operation of producing a cylindrical hole of required diameter and depth by removing metal by the rotating edge of a cutting tool called drill. Drilling is one of the simplest methods of producing a hole. Drilling does not produce an accurate hole in a workpiece. The internal surface of the hole generated by drilling becomes rough and the hole is always slightly oversize due to vibration of the spindle and the drill. A hole made by a drill of size 12 mm will measure approximately upto 12.125 mm and by a drill of size 22mm will measure upto 22.5 mm. *Fig. 11 illustrates drilling operation.*

2. Reaming

The size of hole made by drilling may not be accurate and the internal surface will not be smooth. Reaming is an accurate way of sizing and finishing a hole which has been previously drilled by a multi point cutting tool known as reamer. The surface obtained by reaming will be smoother and the size accurate. The speed of the spindle is made half that of drilling. Reaming removes very small amount of metal (approx 0.375 mm). In order to finish a hole and bring it to the accurate size, the hole is drilled slightly undersize. *Fig . 12 illustrates reaming operation.*

3 Boring

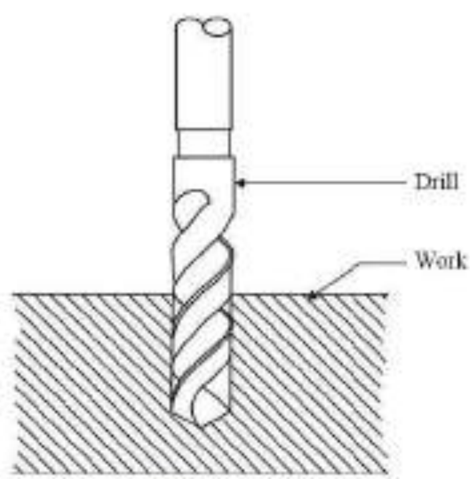


Fig.11. Drilling

Boring is the operation enlarging the diameter of the previously made hole. It is done for the following reasons.

1. To enlarge a hole by means of an adjustable cutting tool. This is done when a suitable sized drill is not available or the hole diameter is so large that is cannot be ordinarily drilled.
2. To finish a hole accurately and bring it to the required size
3. To machine the internal surface of the hole already produced in casting
4. To correct out of roundness of the hole
5. To correct the location of the hole as the boring tool follows independent path with respect to the hole

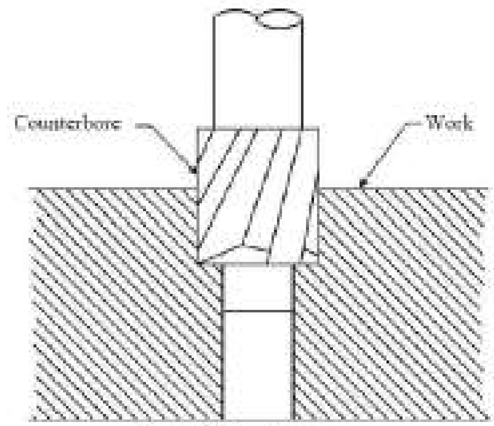


Fig. 14 Counterboring

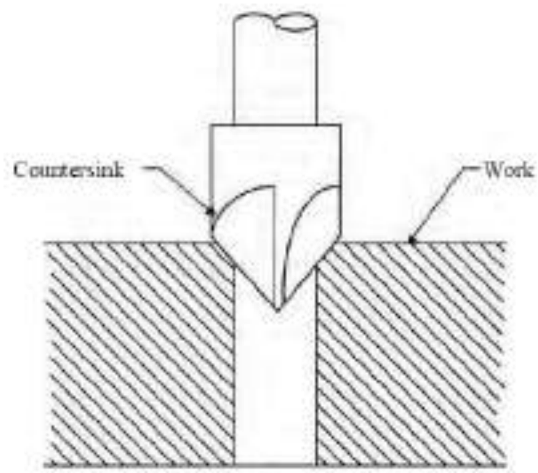


Fig. 15 Countersinking

Boring tool is a tool with only one cutting edge. The tool is held in a boring bar which has a taper shank to fit into the spindle or a socket. For perfectly finishing a hole, the job is drilled undersize slightly. Boring operation in some precise drilling machine is performed to enlarge the holes to an accuracy of 0.00125mm. The spindle speed during boring should be adjusted to be lesser than that of reaming. *Fig. 13 illustrates boring operation.*

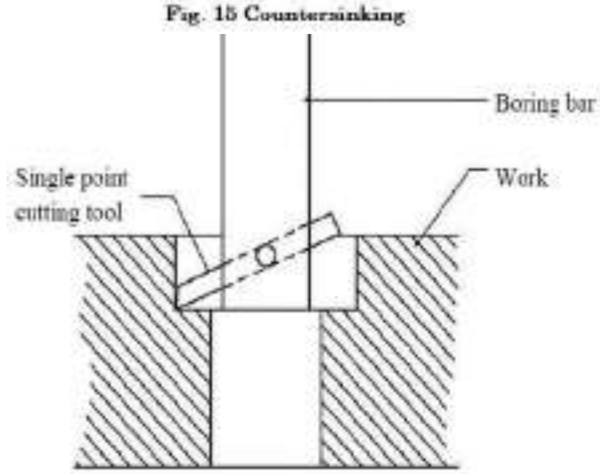


Fig. 13 Boring

4 Counterboring

Counterboring is the operation of enlarging the end of the hole cylindrically. The enlarged hole forms a square shoulder with the original hole. This is necessary in some cases to accommodate the heads of bolts, studs and pins. The tool used for counter boring is known as counter bore. The counterbores are made with cutting edges which may be straight or spiral. The cutting speed for counterboring is atleast 25% lesser than that of drilling.

5 Countersinking

Countersinking is the operation of making a cone shaped enlargement at the end of the hole. The included angle of the conical surface may be in the range of 60° to 90°. It is used to provide recess for a flat headed screw or a counter sunk rivet fitted into the hole. The tool used for counter sinking is known as a countersink. It has multiple cutting edges on its conical surface. The cutting speed for countersinking is 25% lesser than that of drilling. *Fig.14 illustrates countersinking operation.*

6 Spot facing

Spot facing is the operation of smoothing and squaring the surface around a hole. It is done to provide proper seating for a nut or the head of a screw. A counterbore or a special spot facing tool may be employed for this purpose. *Fig. 16 illustrates spot facing operation.*

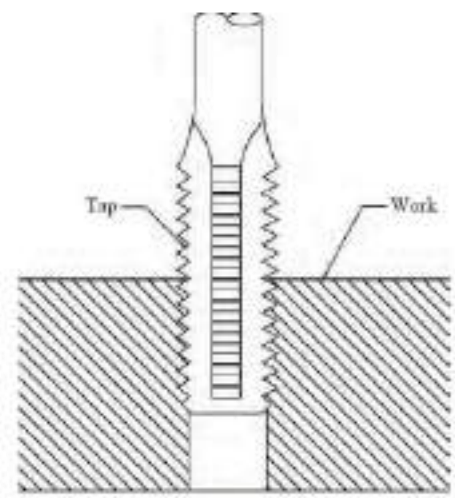


Fig. 17 Tapping

7 Tapping

Tapping is the operation of cutting internal threads by means of a cutting tool called 'tap'. Tapping in a drilling machine may be performed by hand or by power. When the tap is screwed into the hole, it removes metal and cuts internal threads which will fit into external threads of the same size. *Fig. 17 illustrates tapping operation.*

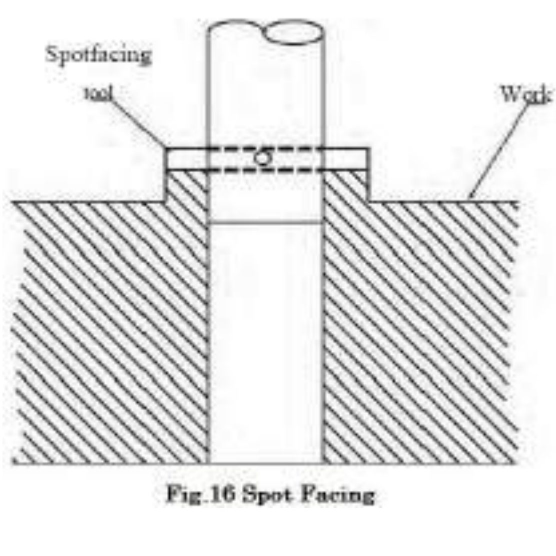


Fig.16 Spot Facing

8 Trepanning

Trepanning is the operation of producing a hole in sheet metal by removing metal along the circumference of a hollow cutting tool. Trepanning operation is performed for producing large holes. Fewer chips are removed and much of the material is saved while the hole is produced. The tool may be operated at higher speeds. The speed depends upon the diameter of the hole to be made. The tool resembles a hollow tube having cutting edges at one end and a solid shank at the other to fit

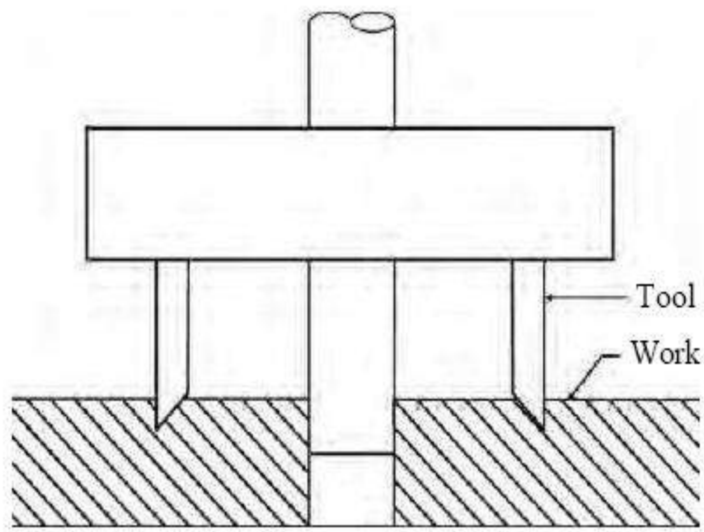


Fig. 18 Trepanning

1.8 Cutting speed, Feed & Depth of cut

1 Cutting speed

Speed in general refers to the distance a point travels in a particular period of time. The cutting speed in a drilling operation refers to the peripheral speed of a point on the cutting edge of the drill. It is usually expressed in meters per minute. The cutting speed (v) may be calculated as

πdn

Cutting speed $V = \frac{\pi DN}{1000}$ m/min

Where

'd' - is the diameter of the drill in mm, and

'N'- is the speed of the drill spindle in r.p.m.

The cutting speed of a drill depends, as in other machining processes, upon several factors like the cutting tool material, the kind of material being drilled, the quality of surface finish desired, the method of holding the work, the size, type and rigidity of the machine.

Example

A drill of diameter 20mm makes a hole on a steel part at a cutting speed of 25m/ min. Find out the spindle speed.

Cutting speed $V = \frac{\pi DN}{1000}$ m/min

$$25 = \frac{\pi * 20 * N}{1000}$$

Spindle speed, N = 398 r.p.m

2 Feed

The feed of a drill is the distance the drill moves into the work at each revolution of the spindle. It is expressed in millimeters. The feed may also be expressed as feed per minute. The feed per minute may be defined as the axial distance moved by the drill into the work per minute. Feed depends upon factors like the material to be drilled, the rigidity of the machine, power, depth of the hole and the type of finish required.

3 Depth of cut

The depth of cut in drilling is equal to one half of the drill diameter. If 'd' is the diameter of the drill, the depth of cut (t) t = d/2 mm.

ABRASIVE PROCESSES

1.1 INTRODUCTION

Grinding is a metal cutting operation performed by means of abrasive particles rigidly mounted on a rotating wheel. Each of the abrasive particles act as a single point cutting tool and grinding wheel acts as a multipoint cutting tool. The grinding operation is used to finish the Workpieces with extremely high quality of surface finish and accuracy of shape and dimension. Grinding is one of the widely accepted finishing operations because it removes material in very small size of chips 0.25 to 0.50 mm. It provides accuracy of the order of 0.000025 mm. Grinding of very hard material is also possible.

1.2 TYPES OF GRINDING MACHINES

According to the accuracy of the work to be done on a grinding machine, they are classified as

1. Rough grinding machines
2. Precision grinding machines

1.2.1 Rough grinding machines

The rough grinding machines are used to remove stock with no reference to the accuracy of results. Excess metal present on the cast parts and welded joints are removed by rough grinders. The main types of rough grinders are:

1. Hand grinding machine
2. Bench grinding machine
3. Floor stands grinding machine
4. Flexible shaft grinding machine
5. Swing frame grinding machine
6. Abrasive belt grinding machine

1.2.2 Precision grinding machines

Precision grinders are used to finish parts to very accurate dimensions. The main types of precision grinders are:

1. Cylindrical grinding machines
2. Internal grinding machines
3. Surface grinding machines
4. Tool and cutter grinding machines
5. Special grinding machines

1.3 Cylindrical grinding machine

Cylindrical grinders are generally used to grind external surfaces like cylinders, taper cylinders, faces and shoulders of work. There are two types of cylindrical grinding machines and they are

1. External cylindrical grinding machines
2. Internal cylindrical grinding machines

1.3.1. External cylindrical grinding machine

Cylindrical centre type grinders are intended primarily for grinding plain cylindrical parts. *Fig.1 illustrates a cylindrical grinder.*

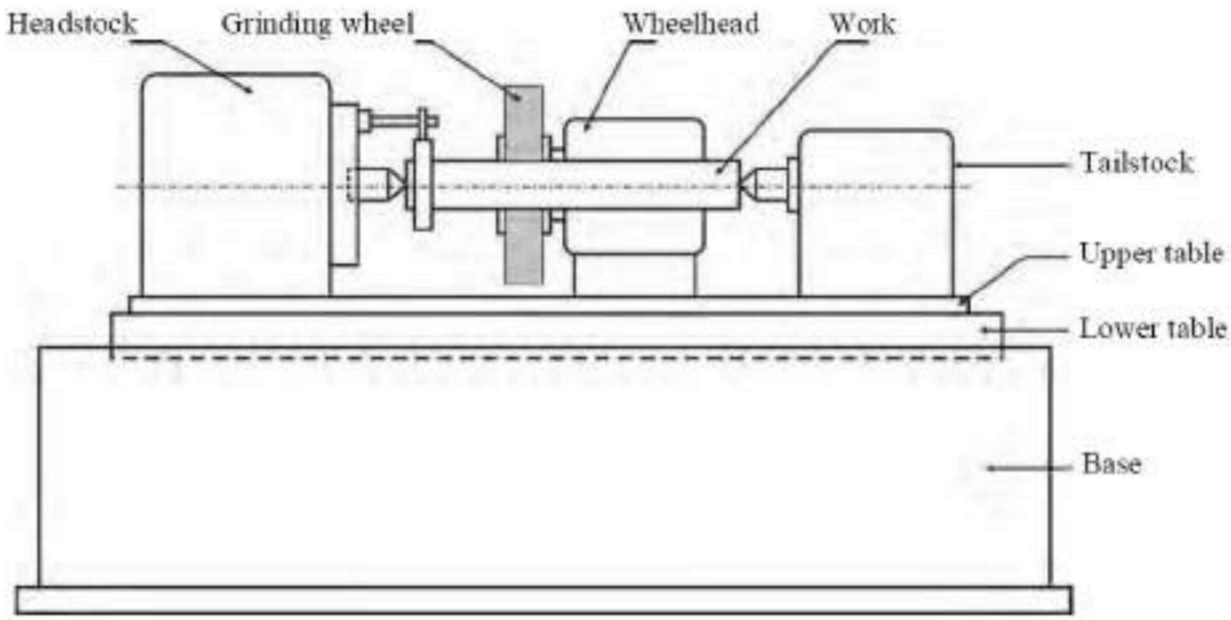
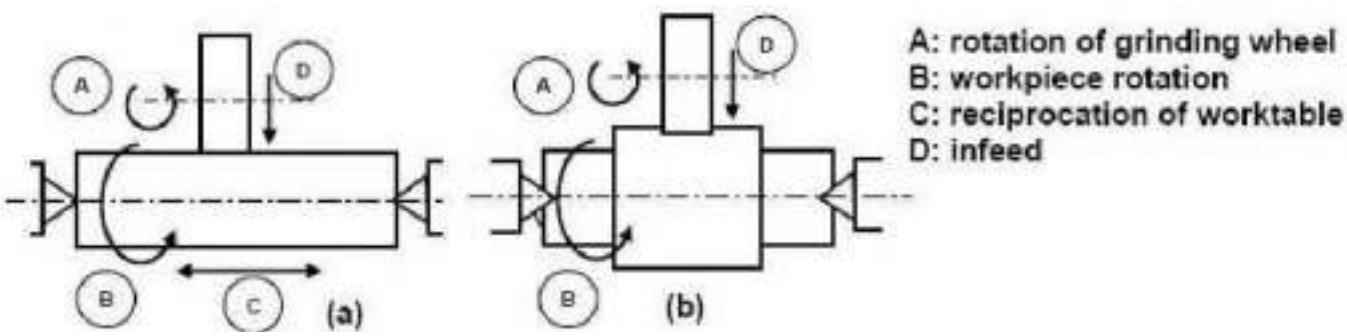


Fig.1 External Cylindrical Grinding machine



Cylindrical grinding process (a) traverse grinding and (b) plunge grinding

Base

The base is made of cast iron and rests on the floor. It supports the parts mounted on it. The top of the base is accurately machined and provides guideways for the table to slide on. The base contains the table driving mechanisms.

Tables

The tables are mounted on top of the base. There are two tables' namely lower table and upper table. The lower table slides on the guideways on the bed. It can be moved by hand or by power within required limits.

The upper table can be swiveled upto $\pm 10^0$ and clamped in position. Adjustable dogs are clamped in longitudinal slots at the side of the lower table. They are set up to reverse the table at the end of the stroke.

Headstock

The headstock is situated at the left side of upper table. It supports the workpiece by means of a centre and drives it by means of a dog. It may hold and drive the workpiece in a chuck. It houses the mechanism meant for driving the work. The headstock of a universal grinding machine can be swiveled to any required angle.

Tailstock

The tailstock is situated at the right side of the table. It can be adjusted and clamped in various positions to accommodate different lengths of Workpieces.

Wheel head

The wheel head may be moved at right angles to the table ways. It is operated by hand or by power to feed the wheel to the work. The wheel head carries a grinding wheel. Its driving motor is mounted on a slide at the top and rear of the base. The grinding wheel rotates at about 1500 to 2000 r.p.m.

1.3.2. Internal cylindrical grinding machine

Internal grinding is employed chiefly for finishing accurate holes in hardened parts, and also when it is impossible to apply other more productive methods of finishing accurate hold, for example, precision boring, honing etc.

There are two general methods of internal grinding:

- With a rotating work piece.
- With the work piece held stationary.

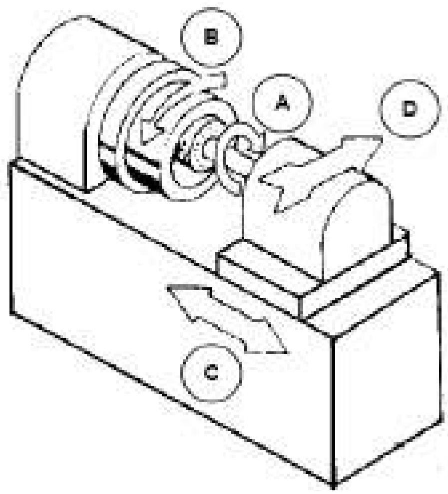
The first method is used in grinding holes in relatively small work pieces, mostly bodies of revolution, for example, the bores of gears and the inner surfaces of ball bearing rings. The work piece is held in a chuck or special fixture and rotated in the same manner as in a lathe. A straight type grinding wheel is rotated and has two feed-longitudinal feed along the wheel axis and is thus reciprocated back and forth through the length of the hole, and intermittent cross feed(radial feed) at the end of each pass, which determines the depth of cut.

The second method of internal grinding is used for grinding holes in large bulky work pieces (housing-type parts) that are inconvenient or even impossible to clamp in a chuck of the grinder. They are mounted on the table of a planetary grinding machine. In addition to rotation about its axis, the wheel spindle of this type of machine also rotates with a planetary motion about the axis of the hole being ground. Axial motion of the wheel provides the longitudinal feed.

a. Chucking type internal grinding machine

Fig. 2 illustrates schematically this machine and various motions required for grinding action. The work piece is usually mounted in a chuck. A magnetic face plate can also be used. A small grinding wheel performs the necessary grinding with its peripheral surface. Both transverse and plunge grinding can be carried out in this machine as shown in Fig. 3.

A: rotation of grinding wheel **B: workpiece rotation**
C: reciprocation of worktable **D: infeed**



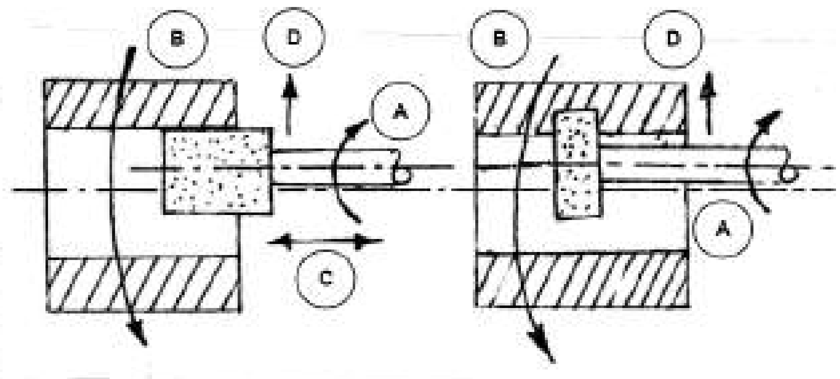


Fig. 2 Internal cylindrical grinding machine

Fig. 3 Internal (a) transverse grinding and (b) plunge grinding

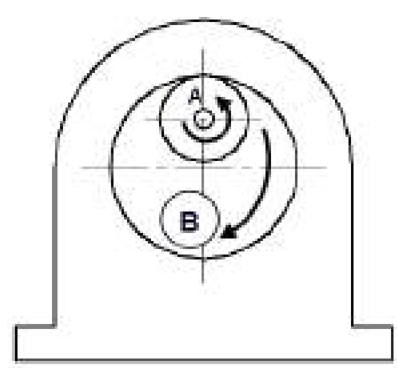
b. Planetary internal grinder

Planetary internal grinder is used where the workpiece is of irregular shape and can not be rotated conveniently as shown in Fig. 4. In this machine the workpiece does not rotate. Instead, the grinding wheel orbits the axis of the hole in the workpiece.

A: rotation of grinding wheel

B: orbiting motion of grinding

Fig. 4 Internal Grinding in Planetary Grinder



1.3.3 Centreless internal grinder

This machine is used for grinding cylindrical and tapered holes in cylindrical parts (e.g. cylindrical liners, various bushings etc). The workpiece is rotated between supporting roll, pressure roll and regulating wheel and is ground by the grinding wheel as illustrated in Fig. 5.

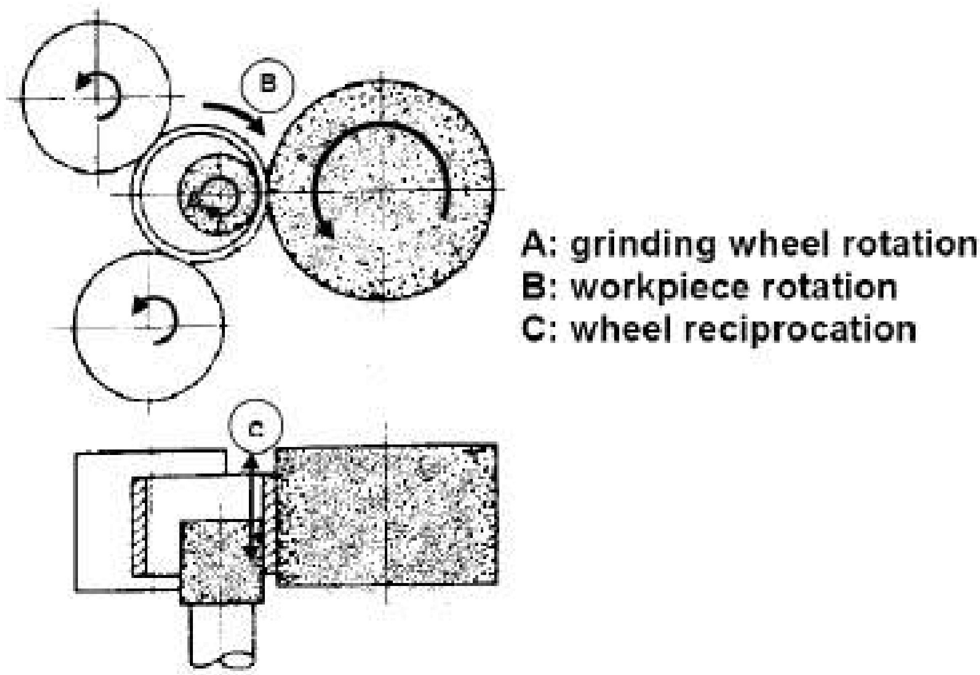


Fig. 5 Internal Centreless Grinding

1.4. SURFACE GRINDING

Surface grinding machines are generally used for generating flat surfaces. These machines are similar to milling machines in construction as well as motion. There are basically four types of machines depending upon the spindle direction and the table motion. *They are,*

1. Horizontal spindle and rotating table grinding machine.
2. Vertical spindle and rotating table grinding machine.
3. Horizontal spindle and reciprocating table grinding machine, and
4. Vertical spindle and reciprocating table grinding machine.

The table in the case of reciprocating machines is generally moved by the hydraulic power. The wheel head is given a cross feed motion at the end of each table motion. In this machine the wheel should over travel the work piece at both the ends to prevent the grinding wheel removing the metal at the same work spot during the table reversal.

Vertical spindle machines are generally of a bigger capacity. The diameter of the wheel is wider than the work piece and as a result no traverse feed is required. The complete machining surface is covered by the grinding wheel face. They are suitable for production grinding of very flat surfaces.

1.4.1 Horizontal spindle and rotating table grinding machine

Surface grinding in this machine is shown in Fig. 6. In principle the operation is same as that for facing on the lathe. This machine has a limitation in accommodation of work piece and therefore does not have wide spread use. However, by swiveling the worktable, concave or convex or tapered surface can be produced on individual part as illustrated in Fig. 7.

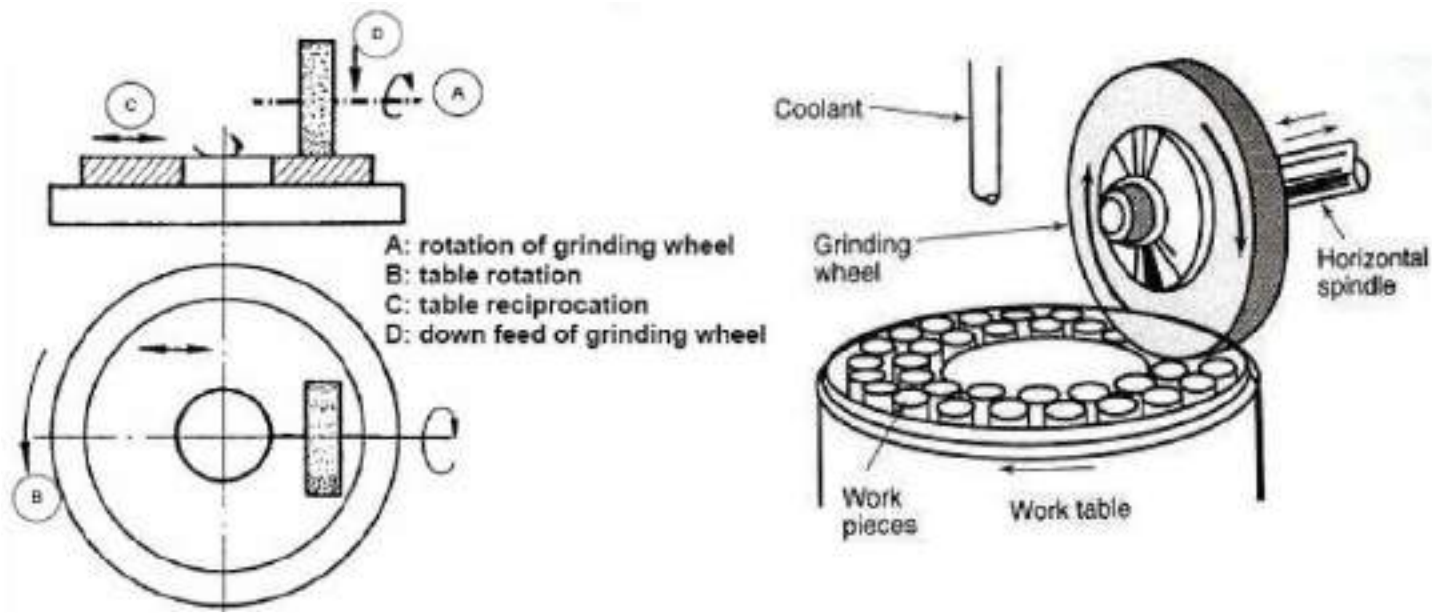


Fig. 6 Surface grinding in horizontal spindle and rotating table grinding machine

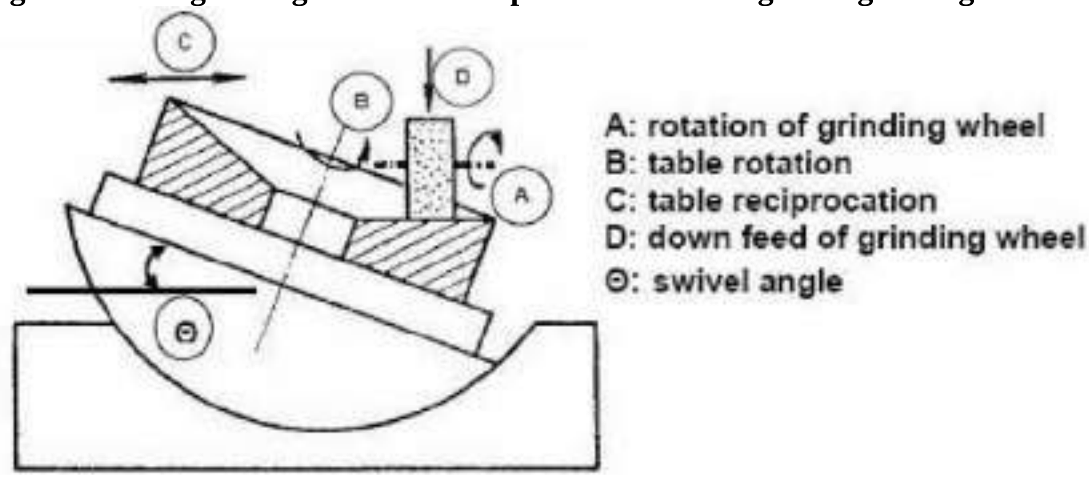


Fig. 7Grinding of a tapered surface in horizontal spindle and rotating table grinding machine

1.4.2 Vertical spindle and rotating table grinding machine

The principle of grinding in this machine is shown in Fig. 8. The machine is mostly suitable for small work pieces in large quantities. This primarily production type machine often uses two or more grinding heads thus enabling both roughing and finishing in one rotation of the work table.

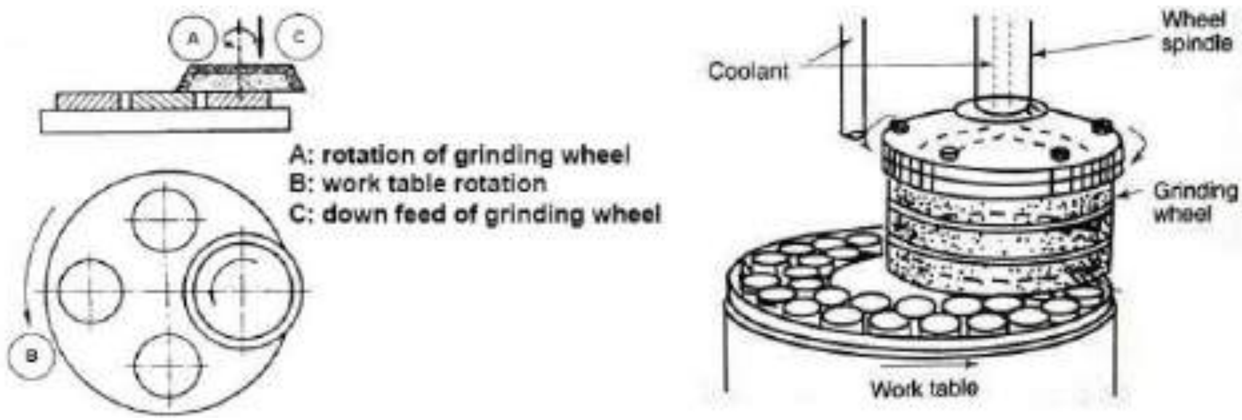


Fig. 8 Surface grinding in vertical spindle and rotating table grinding machine

1.4.3 Horizontal spindle and reciprocating table grinding machine

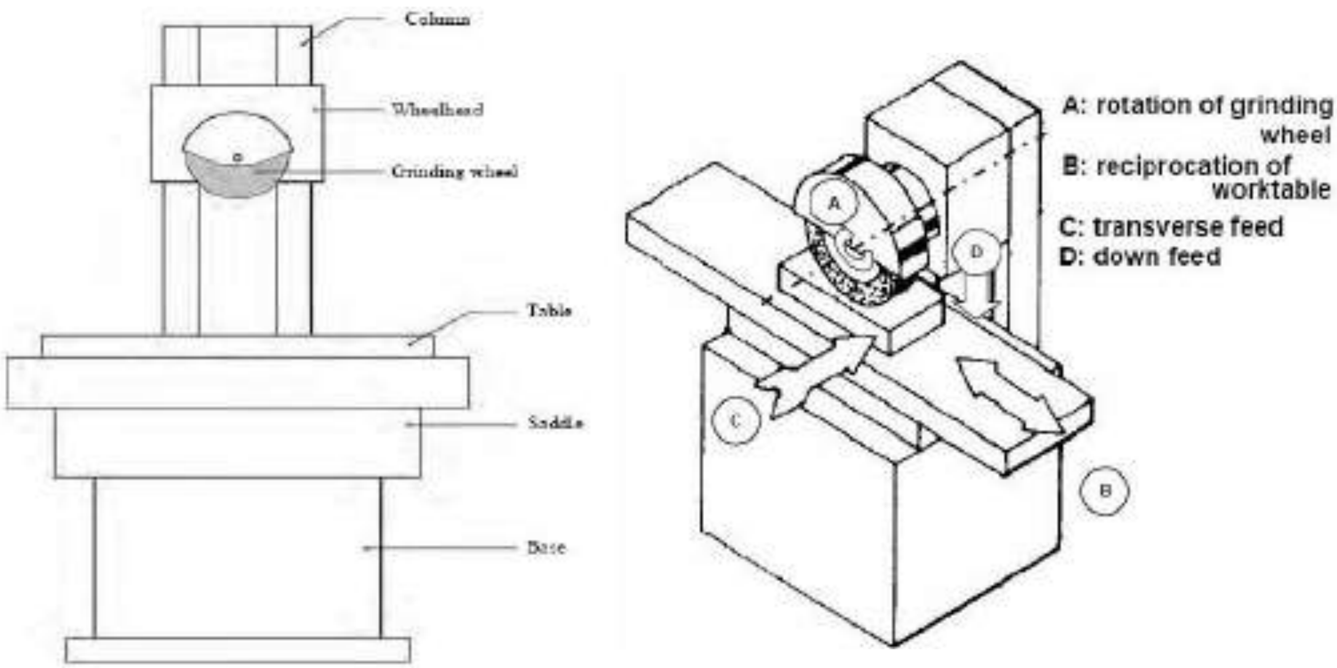


Fig. 9. Horizontal Spindle Reciprocating table surface grinding machine

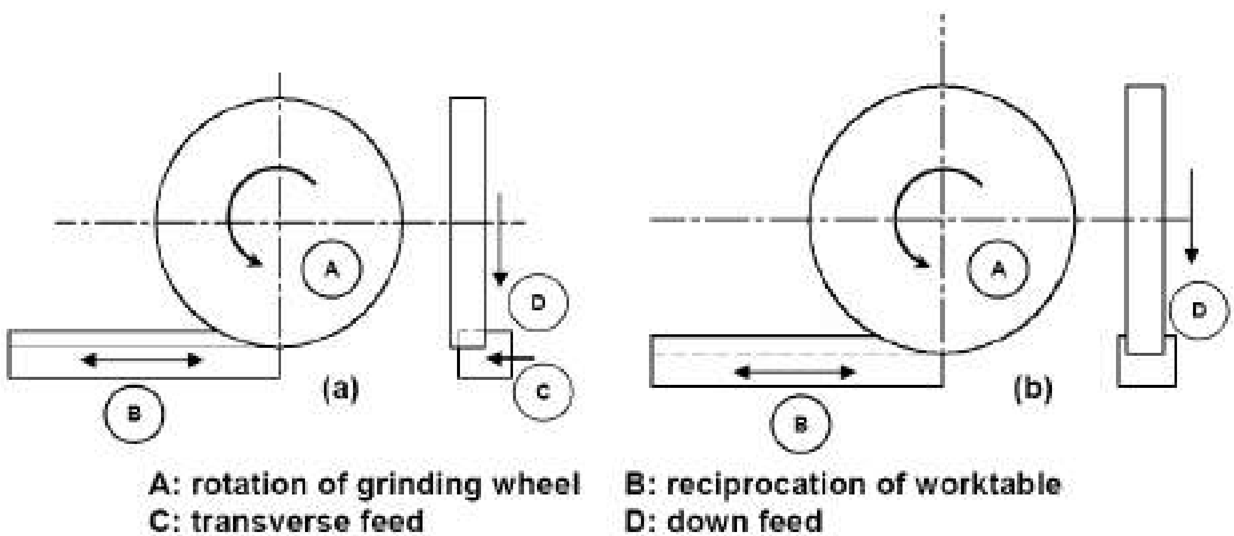


Fig. 10 Surface grinding (a) traverse grinding (b) plunge grinding

The majority of surface grinders are of horizontal spindle type. In the horizontal type of the machine, grinding is performed by the abrasives on the periphery of the wheel. Though the area of contact between the wheel and the work is small, the speed is uniform over the grinding surface and the surface finish is good. The grinding wheel is mounted on a horizontal spindle and the table is reciprocated to perform grinding operation.

Base

The base is made of cast iron. It is a box like casting which houses all the table drive mechanisms. The column is mounted at the back of the base which has guideways for the vertical adjustment of the wheel head.

Saddle

Saddle is mounted on the guideways provided on the top of the base. It can be moved at cross towards or away from the column.

Table

The table is fitted to the carefully machined guideways of the saddle. It reciprocates along the guideways to provide the longitudinal feed. The table is provided with 'T'- slots for clamping Workpieces directly on the table or for clamping grinding fixtures or magnetic chuck.

Wheel head

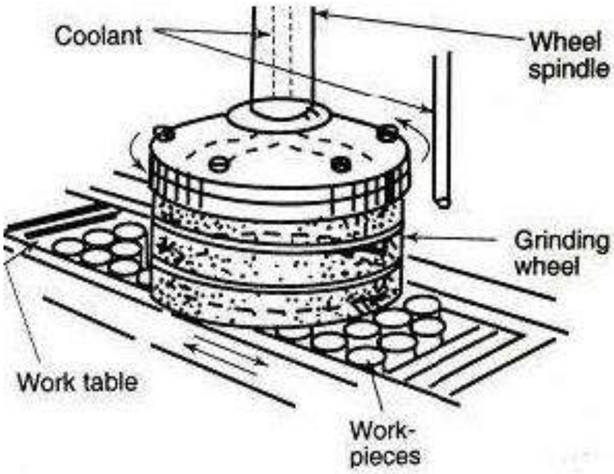
An electric motor is fitted on the wheel head to drive the grinding wheel. The wheel heed is mounted on the guideways of the column, which is secured to the base. It can be raised or lowered with the grinding wheel to accommodate work pieces of different heights and to set the wheel for depth of cut.

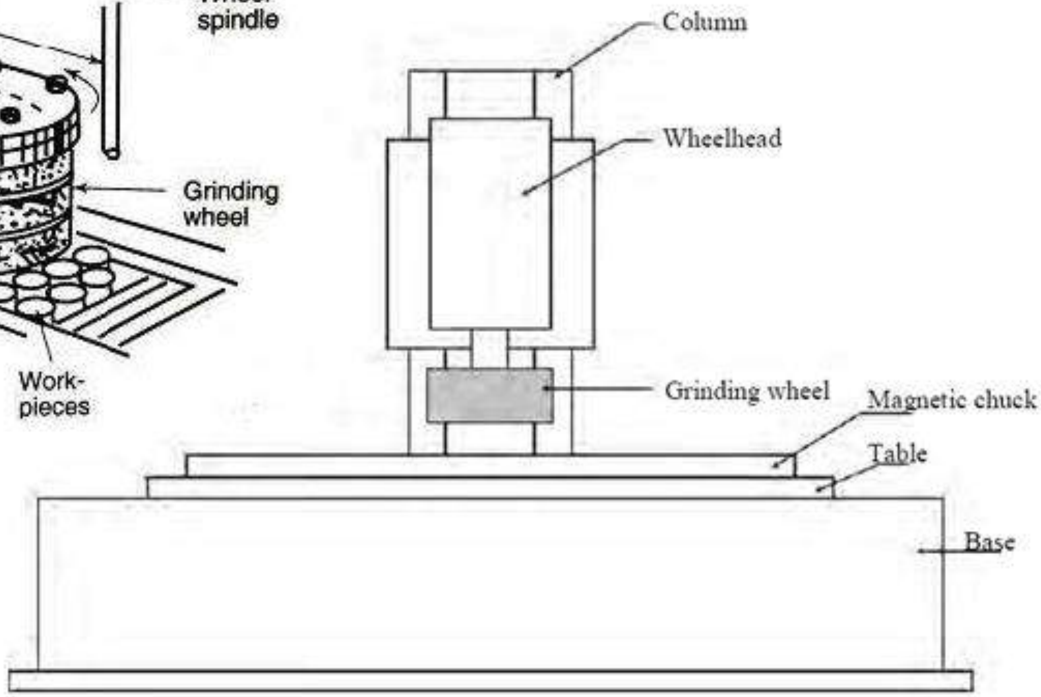
1.4.4 Vertical spindle surface grinding machine

The grinding operation is similar to that of face milling on a vertical milling machine. In this machine a cup shaped wheel grinds the work piece over its full width using end face of the wheel as shown in Fig. 11. This brings more grits in action at the same time and consequently a higher material removal rate may be attained than for grinding with a peripheral wheel. The face or sides of the wheel are used for grinding in the vertical type surface grinders. The area of contact is large and stock can be removed quickly. But criss-cross patterns of grinding scratches are left on the work surface. Considering the quality of surface finish obtained, the horizontal spindle type machines are widely used.

The grinding wheel is mounted on the vertical spindle of the machine. The work is held on the table and grinding is done. The base of the machine is a box like casting. The base is very similar to the one of the horizontal spindle type. It houses all the table drive mechanisms.

The table is mounted on the base on top of which a magnetic chuck is mounted. A grinding wheel is mounted on the wheel head which slides vertically on the column. The table is made to reciprocate or rotate to bring the work surface below the grinding wheel to perform grinding.





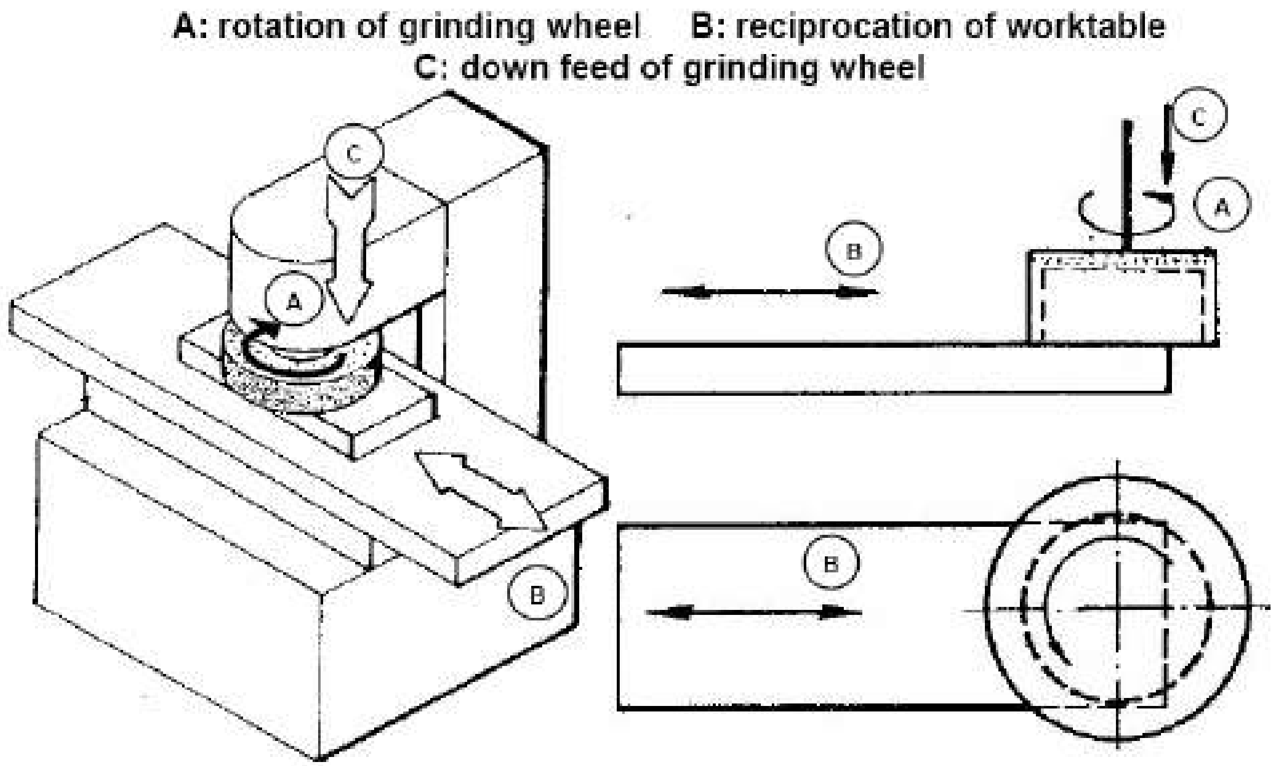


Fig.11. Vertical Spindle Reciprocating Table Surface grinder

1.5 CENTRELESS GRINDING

Centreless grinding makes it possible to grind cylindrical work pieces without actually fixing the work piece using centres of a chuck. As a result no work rotation is separately provided. The process consists of two wheels, one large grinding wheel and another smaller regulating wheel. The work is held on a work rest blade. The regulating wheel is mounted at an angle to the plane of the grinding wheel.

The centre of the work piece is slightly above the centre of the grinding wheel. The work piece is supported by the rest blade and held against the regulating wheel by the grinding force. As a result the work rotates at the same surface speed as that of regulating wheel. The axial feed of the work piece is controlled by the angle of tilt of the regulating wheel. Typical work speeds are about 10 to 50m/min. There are two types of centreless grinding and they are

1. External centreless grinding
2. Internal centreless grinding

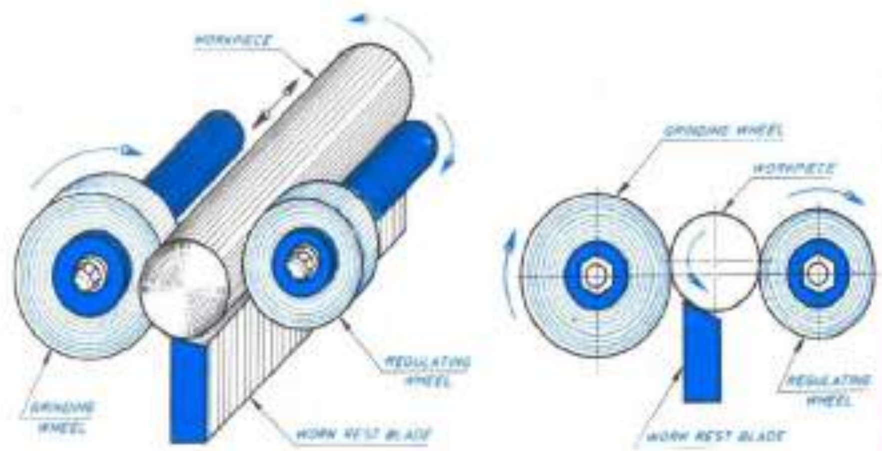


Fig. 12 Principle of Centreless Grinding Machine

1.5.1 External centreless grinding

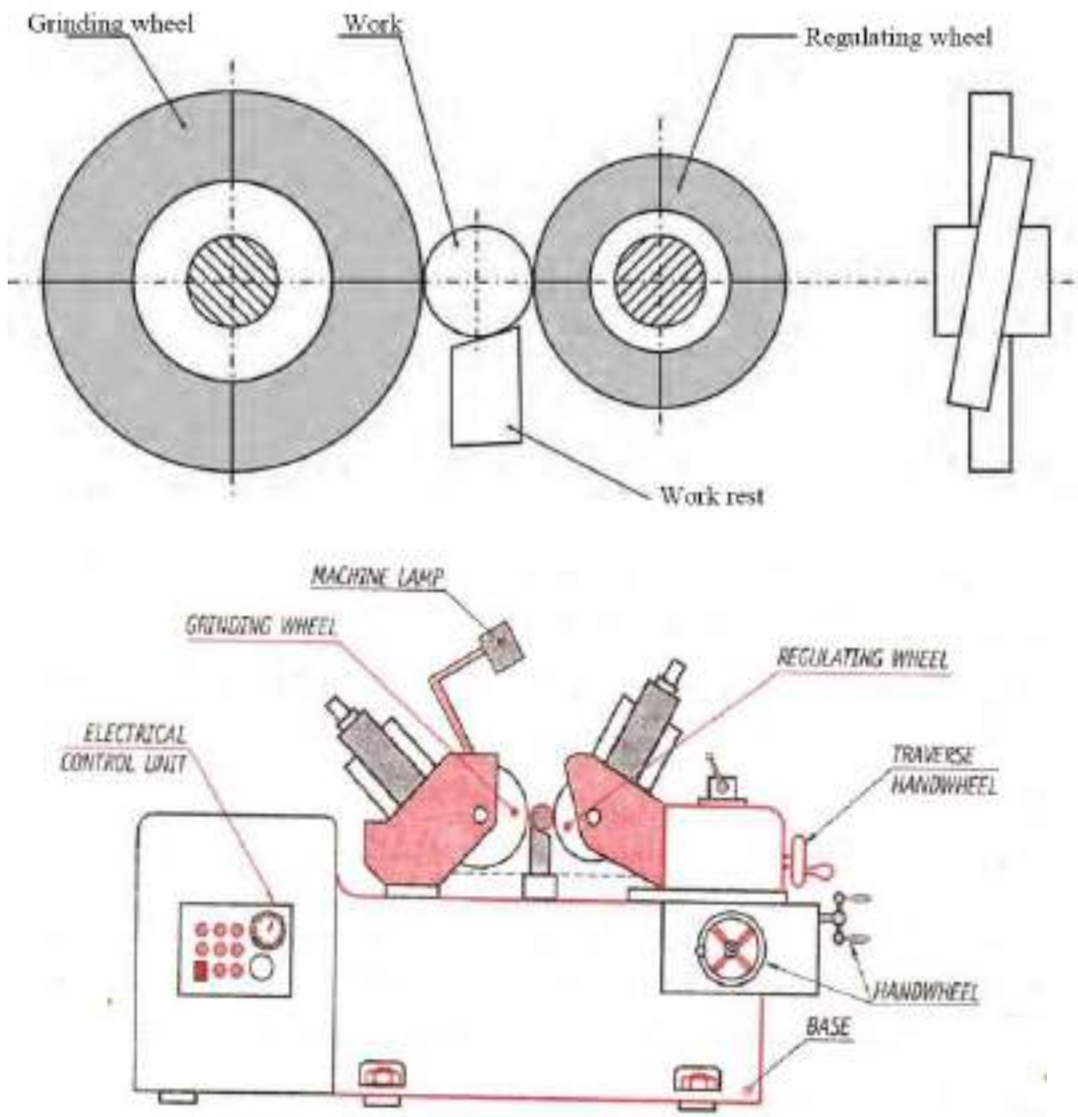


Fig. 13 Centreless Grinding Machine

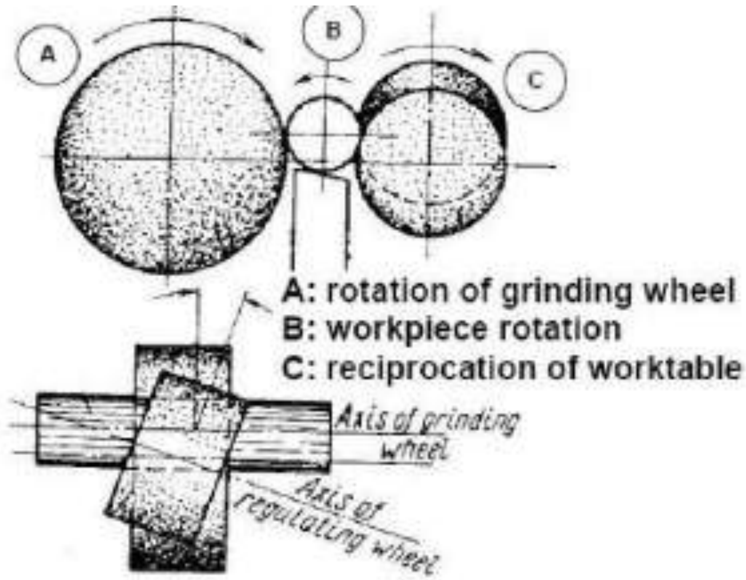


Fig. 14. Centreless grinding through feeding

This grinding machine [shown in Fig. 13] is a production machine in which outside diameter of the work piece is ground. The work piece is not held between centres but by a work support blade. It is rotated by means of a regulating wheel and ground by the grinding wheel. In through-feed centreless grinding, the regulating wheel revolving at a much lower surface speed than grinding wheel controls the rotation and longitudinal motion of the work piece. The regulating wheel is kept slightly inclined to the axis of the grinding wheel and the work piece is fed longitudinally as shown in Fig. 14.

Methods of Centerless Grinding

Basically there are three different methods by which centreless grinding can be done on different types of jobs.

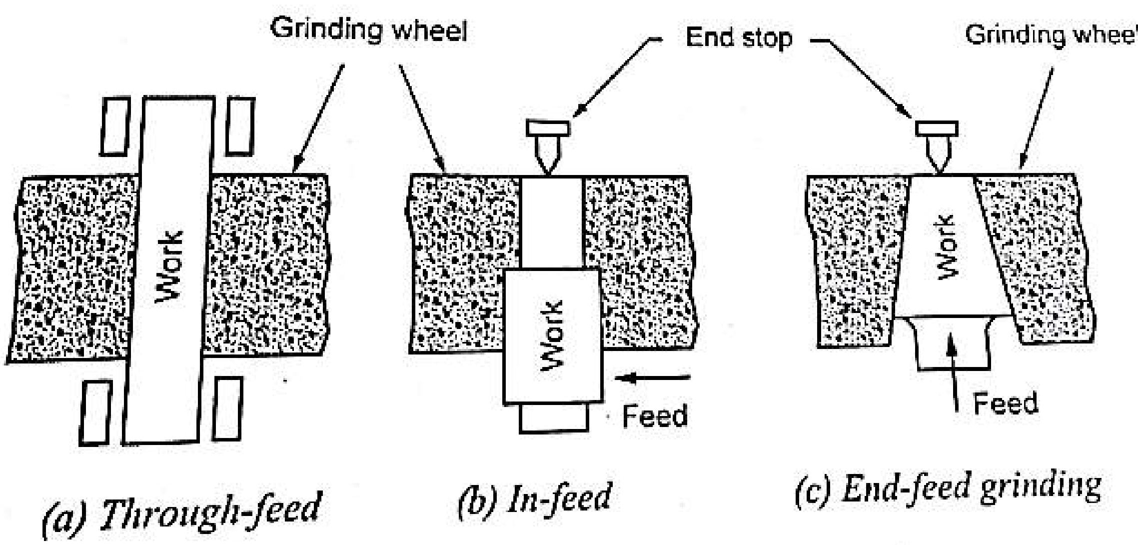
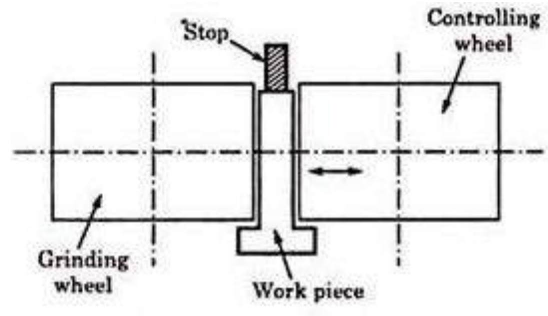


Fig. 15. Methods of centreless Grinding

Method # 1. Through-Feed:

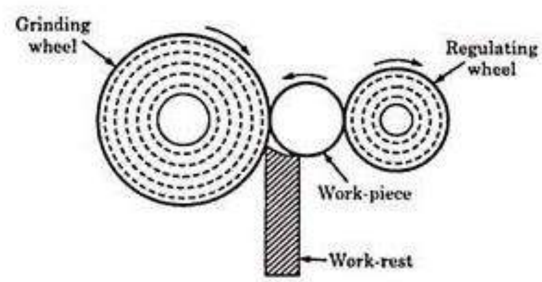
This is simplest method and is applied only to plain parallel parts such as roller pins and straight long bars which are difficult to grind by ordinary cylindrical grinding method. In this case, controlling wheel is first positioned for the proper diameter, i.e. the gap between the regulating wheel and grinding wheel is adjusted equal to the desired diameter of the workpiece and then the job is fed and passed through the wheels.



In-feed grinding.

The machine can remove upto 0.38 mm of stock on the diameter in one pass. The rate of longitudinal feed = π x diameter of regulating wheel x r.p.m. of regulating wheel x sin (angle of inclination of regulating wheel, usually 1° to 6°). This relationship does not consider effect of slip. For finish grinding, high speed of regulating wheel combined with less inclination are used and vice versa.

Advantages:



(a) The operation becomes automatic by employing continuous magazine feed for longer bars and hopper for small jobs.

(b) No holding of the workpiece is required except the work support and hence the long bars can be ground easily without any deflection being produced.

Disadvantages:

(a) This process is only used in case of straight cylindrical parts. If there is a head on the workpiece or it is tapered one, then this process cannot be used.

(b) Form grinding operation cannot be carried out by this process.

Method # 2. In-Feed (Plunge-Cut-Grinding):

This method of grinding is used when the workpiece is of headed, stepped or taper form. In this case there is no axial movement of the workpiece as the length of grinding has to be controlled. The only movement occurring during the process is the rotating movement. During the process the workpiece is placed on the work-rest against an end stop and then the control wheel is advanced towards grinding wheel by some lever arrangement, and grinding continued till the workpiece is reduced to the required diameter.

The regulating wheel is given a slight inclination of the order of 1/2° in order that the workpiece may remain tight against the end stop. If this inclination is not given to the controlling wheel, then there are chances that workpiece may fall down during the process. The length of the workpiece that can be ground is limited to 30 cm. By this process form grinding can also be done.

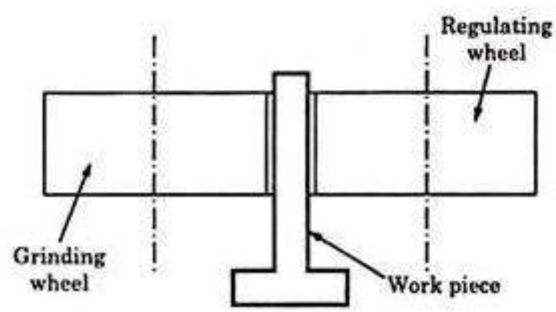
Method # 3. End-Feed:

This process of centreless grinding is used for headed components which are too long to be ground by the infeed method, i.e. when the length of workpiece is greater than the width of the grinding wheel.

The work is fed as in case of in-feed method and after a certain portion of length of workpiece has been ground, the axial movement takes place until the whole of length has been ground. It is also used for tapered work. Usually both grinding wheel and regulating wheel are trued to obtain the required taper.

1.5.2 Internal centreless grinding

The principle of external centreless grinding is applied to internal centreless grinding also. Grinding is done on the inner surfaces of the holes. In internal centreless grinding, the work is supported by three rolls - a regulating roll, a supporting roll and a pressure roll. The grinding wheel contacts the inside surface of the workpiece directly opposite the regulating roll. The distance between the contours of these two wheels is the wall thickness of the work. *Fig. 16 shows internal centreless grinding operation.*



End-feed grinding.

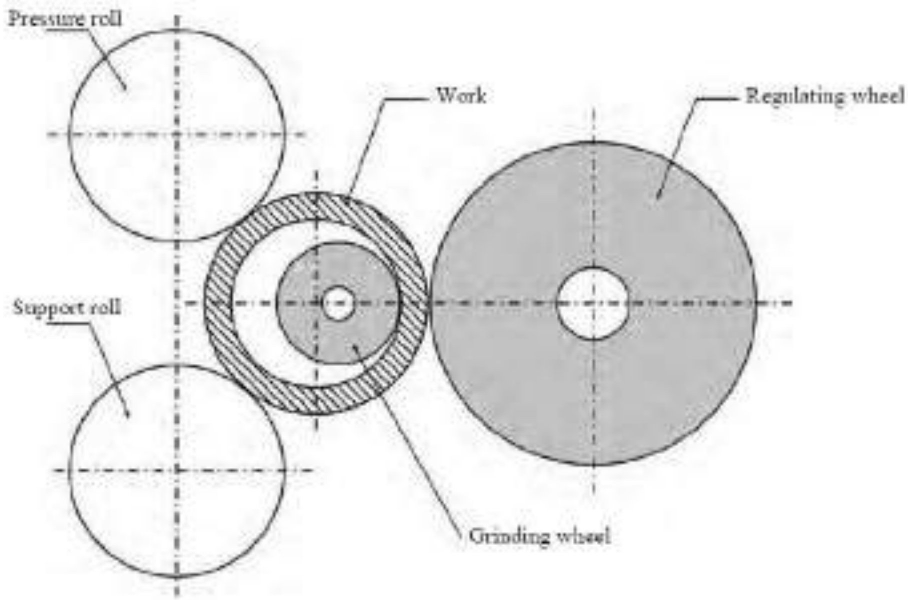


Fig. 16 Internal Centreless Grinding Machine

1.6 Special application of cylindrical grinder

Principle of cylindrical grinding is being used for thread grinding with specially formed wheel that matches the thread profile. A single ribbed wheel or a multi ribbed wheel can be used as shown in Fig. 17.

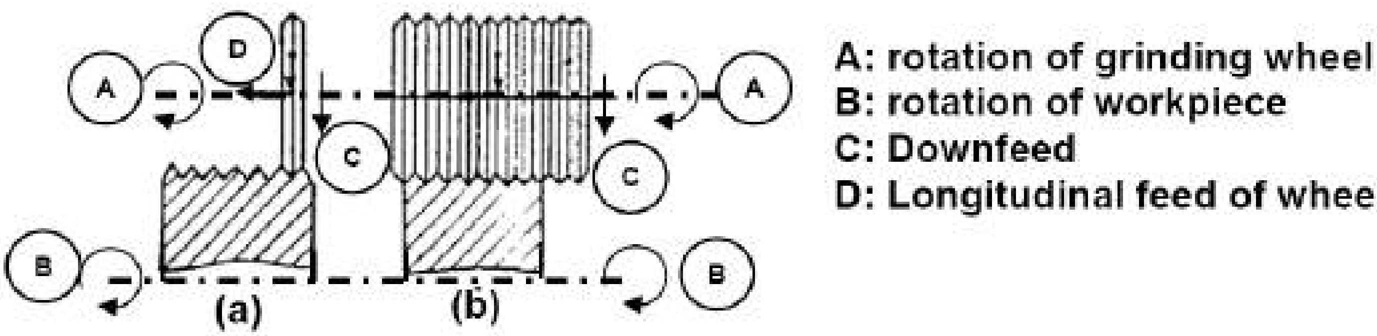
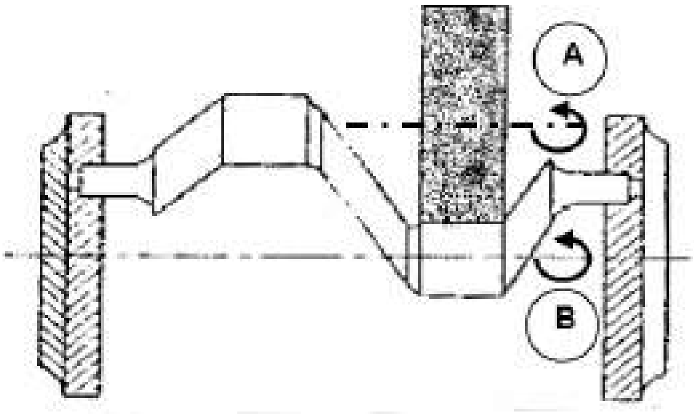


Fig. 17 *Thread grinding with (a) single rib (b) multi-ribbed wheel*

Roll grinding is a specific case of cylindrical grinding wherein large workpieces such as shafts, spindles and rolls are ground.

Crankshaft or crank pin grinders also resemble cylindrical grinder but are engaged to grind crank pins which are eccentric from the centre line of the shaft as shown in Fig. 18. The eccentricity is obtained by the use of special chuck.

Cam and camshaft grinders are essentially subsets of cylindrical grinding machine dedicated to finish various profiles on disc cams and cam shafts. The desired contour on the workpiece is generated by varying the distance between wheel and workpiece axes. The cradle carrying the head stock and tail stock is provided with rocking motion derived from the rotation of a master cam that rotates in synchronisation with the workpiece. Newer machines however, use CNC in place of master cam to generate cam on the workpiece.



A: rotation of wheel
B: rotation of crank pin

Fig. 18 *Grinding of crank pin*

1.7 ADVANTAGES OF GRINDING OPERATION:

- This can produce a high surface finish with accurate can obtain.
- This can machine hard materials.
- This operation can be done with less pressure applied on work.
- It can obtain highly accurate dimensions.
- It can work at high temperature also.
- Speed of cutting can be done by this process.
- In grinding abrasive particles, they are self-sharpened action.
- This can operate for complex things also.
- Smooth surface can obtain.

1.8 DISADVANTAGES OF GRINDING OPERATIONS:

- Required tool is high cost.
- Process is also a costly one.
- It cannot remove the high amount of material, it only removes a little amount.
- For removing the required amount from work it consumes more time.
- You should work carefully, because imperfect contact may lead to damages.

1.9 ADVANTAGES OF CENTRELESS GRINDING:

- Centreless grinding operation requires less grinding stock.
- Centreless grinding has a high rate of production.
- This process helps in high rigid support to work.

- This operation is easy to control.
- This operation is done with quality output.
- This type of work can be done by less skilled persons.
- This centreless grinding process occurs an exact floating condition.

1.10 DISADVANTAGES OF CENTRELESS GRINDING:

- This operation is not so easy to handle at different working diameters.
- This type of operations is not useful at less production.
- Changing the tool of grinding wheels takes a long time.
- This cannot be highly useful to long Flat and key ways.

1.11 Grinding wheel

A grinding wheel is a multi-tooth cutter made up of many hard particles known as abrasives having sharp edges. The abrasive grains are mixed with a suitable bond, which acts as a matrix to manufacture grinding wheels.

According to construction, grinding wheels are classified under three categories.

1. Solid grinding wheels
2. Segmented grinding wheels
3. Mounted grinding wheels

1.11.1 Abrasives

Abrasives are used for grinding and polishing operations. It should have uniform physical properties of hardness, toughness and resistance to fracture. Abrasive may be classified into two principal groups.

1. Natural abrasives
2. Artificial abrasives

1.11.1.1 Natural abrasives

The natural abrasives are obtained from the Earth’s crust. They include sandstone, emery, corundum and diamond.

Sandstone is used as abrasive to grind softer materials only.

Emery is natural alumuna. It contains aluminium oxide and iron oxide.

Corundum is also a natural aluminium oxide. It contains greater percentage of aluminium oxide than emery. Both emery and corundum have a greater hardness and abrasive action than sandstone.

Diamond is the hardest available natural abrasive. It is used in making grinding wheels to grind cemented carbide tools.

1.11.1.2 Artificial abrasives

Artificial abrasives are of two types.

1 Silicon carbide abrasives

2 Aluminium oxide abrasives

1. Silicon carbide

Silicon carbide is manufactured from 56 parts of silica, 34 parts of powdered coke, 2 parts of salt and 12 parts of sawdust in a long rectangular electric furnace of the resistance type that is built of loose brick work. Silicon carbide is next to diamond in the order of hardness. But it is not tough enough as aluminium oxide. It is used for grinding materials of Iow tensile strength such as cemented carbides, ceramic materials, grey brass, bronze, copper, aluminium, vulcanized rubber etc. This is manufactured under trade names of carborundum. It is denoted by the letter 'S'.

2. Aluminium oxide

Aluminium oxide is manufactured by heating mineral bauxite, silica, iron oxide, titanium oxide, etc., mixed with ground coke and iron borings in arc type electric furnace. Aluminium oxide is tough and not easily fractured, so it is better adapted to grinding materials of high tensile strength such as most steels, carbon steels, high speed steels, and tough bronzes. This is denoted by the letter 'A'.

1.11.2 Bonds

A bond is an adhesive material used to held abrasive particles together; relatively stable that constitute a grinding wheel. Different types of bonds are :

(a) Vitrified bond (V),

(b) Silicate bond (S),

(c) Shellac bond (E),

(d) Resinoid bond (B),

(e) Rubber bond (R)

(f) Oxychloride bond(O)

These bonds are being explained here in brief.

(a) Vitrified Bond

This bond consists of mixture of clay and water. Clay and abrasives are thoroughly mixed with water to make a uniform mixture. The mixture is moulded to shape of a grinding wheel and dried up to take it out from mould. This bond is denoted by symbol 'V' in specification.

(b) Silicate Bond

Silicate bonds are made by mixing abrasive particals with silicate and soda or water glass. It is moulded to required shape, allowed to dried up and then taken out of mould. The raw moulded wheel is baked in a furnace at more than 200^0 C for several days. These wheels are denoted by 'S' in specification.

(c) Shellac Bond

These are prepared by mixing abrasive with shellac then moulded by rolling and pressing and then by heating upto 150^0 C for several hours. This bond exhibit greater elasticity than other bonds with appreciable strength. This bond is denoted by 'E' in specifications.

(d) Resinoid Bond

These bonds are prepared by mixing abrasives with synthetic resins like backelite and redmanol and other compounds. Mixture is moulded to required shape and baked upto 200^0 C to give a perfect grinding wheel. A resinoid bond is denoted by the letter 'B'.

(e) Rubber Bond

Rubber bonded wheels are made by mixing abrasives with pure rubber and sulphur. After that the mixture is rolled into sheet and wheels are prepared by punching using die and punch. The wheels are vulcanized by heating then in furnace for short time. A rubber wheel bonded wheel is denoted by the letter 'R'.

(f) Oxychloride Bond

These bonds are processed by mixing abrasives with oxides and chlorides of magnesium. The mixture is moulded and baked in a furnace to give shape of a grinding wheel. These grinding wheels are used for disc grinding operations. An oxychloride bonded wheel is specified the letter 'O'.

1.11.3 Grain size, Grade and Structure

a. Grain size (Grit)

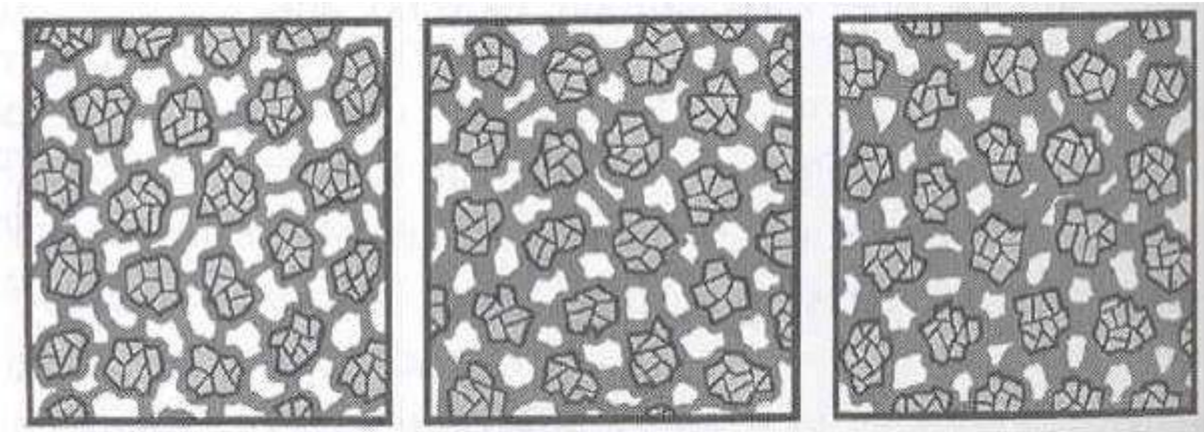
The grinding wheel is made up of thousands of abrasive grains. The grain size or grit number indicates the size of the abrasive grains used in making a wheel, or the size of the cutting teeth. Grain size is denoted by a number indicating the number of meshes per linear inch of the screen through which the grains pass when they are graded. There are four different groups of the grain size namely coarse, medium, fine and very fine. If the grit number is large, the size of the abrasive is fine and a small grit number indicates a large grain of abrasive. Grain size depends upon quantity of material to the ground required quality of surface finish; and hardness of workpiece material. **Find and very fine grain size is used for precision grinding, however, coarse and medium grain size is used for rough grinding**.

Class	Grain Size of Abrasive (Grit)						
Coarse	10	12	14	16	20	24	
Medium	30	36	46	54	60		
Fine	80	100	120	150	180		
Very fine	220	240	280	320	400	500	600

b. Grade

The grade of a grinding wheel refers to the hardness with which the wheel holds the abrasive grains in place. It does not refer to the hardness of the abrasive grains. The grade is indicated by a letter of the English alphabet. The term 'soft' or 'hard' refers to the resistance a bond offers to disruption of the abrasives. A wheel from which the abrasive grains can easily be dislodged is called soft whereas the one, which holds the grains more securely, is called hard. The grade of the bond can be classified in three categories.

Class	Coding for Grade									
Soft	A	B	C	D	E	F	G	H		
Medium	I	J	K	L	M	N	O	P		
Hard	Q	R	S	T	U	V	W	X	Y	Z

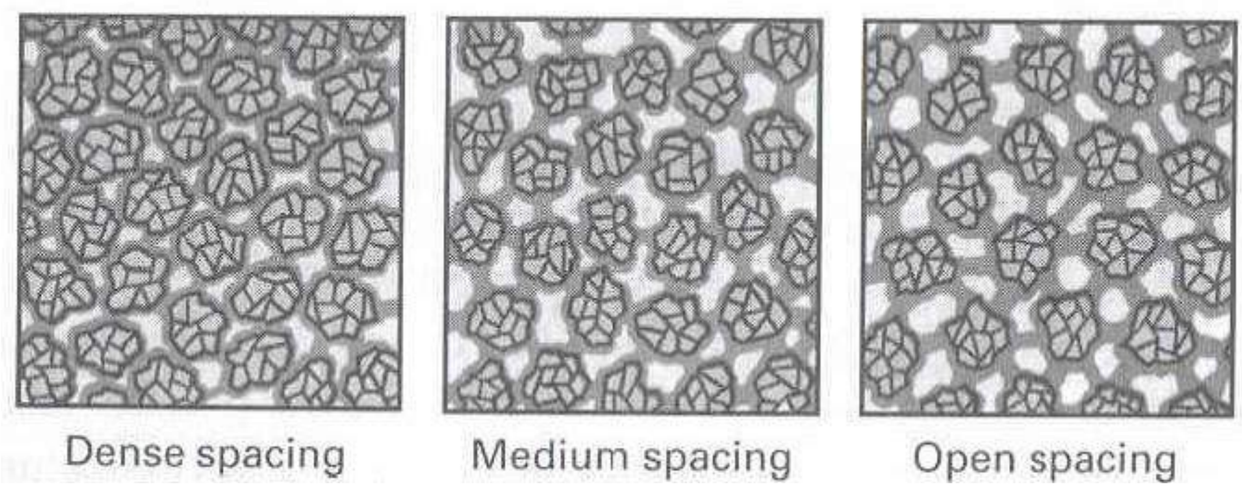


The worn out grit must pull out from the bond and make room for fresh sharp grit in order to avoid excessive rise of grinding force and temperature. Therefore, a soft grade should be chosen for grinding hard material. On the other hand, during grinding of low strength soft material grit does not wear out so quickly. Therefore, the grit can be held with strong bond so that premature grit dislodgement can be avoided.

c. Sturcture

` The relative spacing occupied by the abrasives and the bond is referred to as structure. It is denoted by the number and size of void spaces between grains. It may be 'dense' or open'. The structure should be open for grinding wheels engaged in high material removal to provide chip accommodation space. The space between the grits also serves as pocket for holding grinding fluid. Open structured wheels are used to grind soft and ductile materials. Dense wheels are useful in grinding brittle materials.

Type of Structure	Structure Code							
Dense structure	1	2	3	4	5	6	7	8
Open structure	9	10	11	12	13	14	15	



1.12 Coding of a Grinding Wheel

The Indian Standard Coding system of grinding wheel is IS : 551-1954. It provides uniform system of coding of grinding wheels to designate their various characteristics. It gives a general indication of the hardness and grit size of any wheel as compared with another. Coding of a grinding wheel consists of six symbols as described below.

W : Symbol for Manufacturer's Abrasive Type (Prefixed)

C : Name of Abrasive

30 : Grain Size

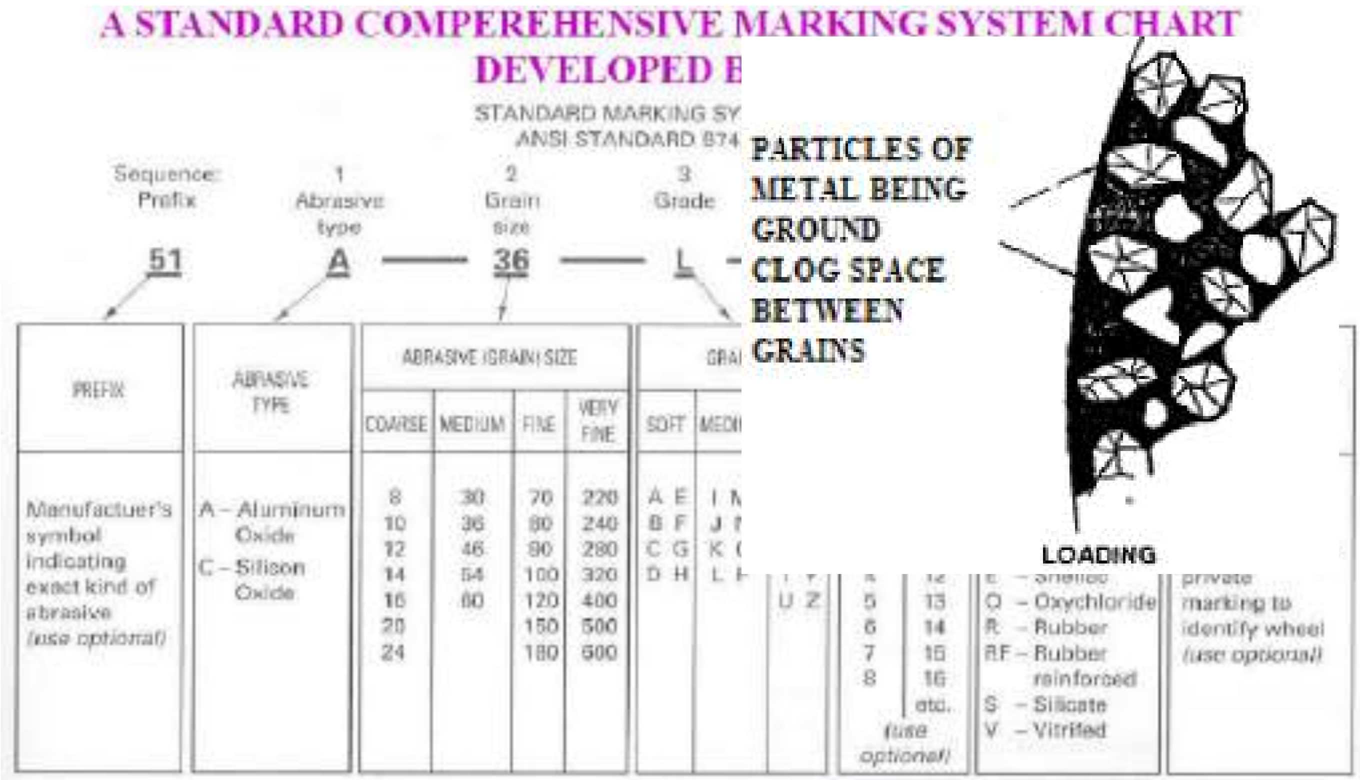
L : Grade

5 : Structure Type

R : Bond Type

17 : Manufacturer Symbol for Record (Suffix)

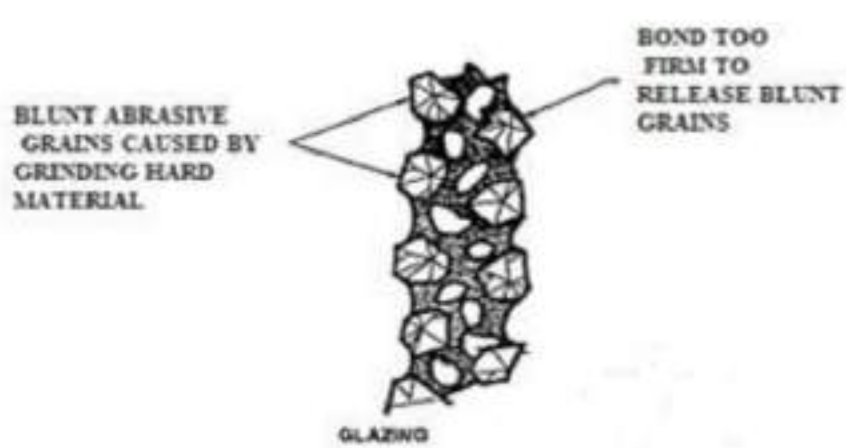
The sequence of codes of a grinding should be followed in the same sequence as described above. There are six symbols and first one which is seventh, is optional. Their brief description is given below.



1.13 Glazing, Loading and Chattering

Glazing

After the continuous use grinding wheel becomes dull or glazed. Glazing of the wheel is a condition in which the face or cutting edge acquires a glass like appearance. That is, the cutting points of the abrasives have become dull and worn down to bond. Glazing makes the grinding face of the wheel smoother and that stops the process of grinding. Glazing takes place if the wheel is rotated at very high speeds and is made with harder bonds. Rotating the wheel at lesser speeds and using soft bonds are the remedies. The glazed wheels are dressed to have fresh, sharp cutting edges.



Loading

The wheel is loaded if the particles of the metal being ground adhere to the wheel. The openings or pores of the wheel face are filled up with the metal preventing the grinding action. It is caused by grinding softer materials (aluminium, copper or lead) or by using a very hard bonded wheels and running it very slowly. It may also take place if very deep cuts are taken by not using the right type of coolant.

The effects of a glazed or a loaded grinding wheel are almost the same.

Chattering

The wavy patterns of crisscross lines are visible on the ground surface some times. This condition is known as chattering. It takes place when the spindle bearings are not fitted correctly and because of the imbalance of the grinding wheel.

1.14 Dressing and truing of grinding wheels

a. Dressing

If the grinding wheels are loaded or gone out of shape, they can be corrected by dressing or truing of the wheels. Dressing is the process of breaking away the glazed surface so that sharp particles are again presented to the work. The common types of wheel dressers known as "Star" –dressers or diamond tool dressers are used for this purpose.

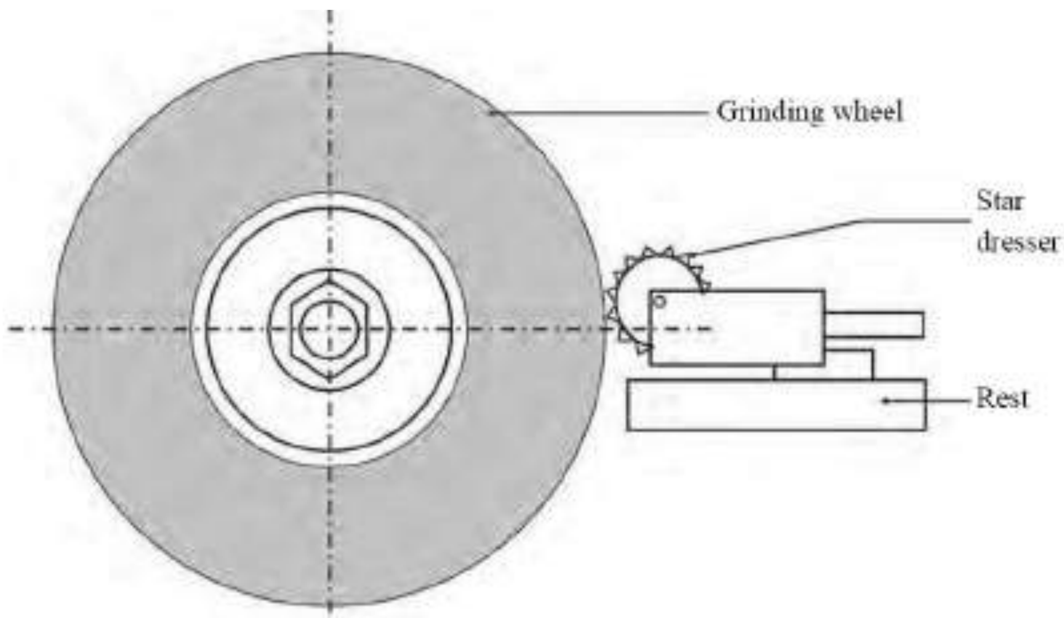


Fig 19 Dressing of a grinding wheel (Star wheel method)

A star dresser consists of a number of hardened steel wheels on its periphery. The dresser is held against the face of the revolving wheel and moved across the face to dress the wheel surface. This type of dresser is used particularly for coarse and rough grinding wheels. *Fig. 19 shows dressing by a star wheel dresser*.

For precision and high finish grinding, small industrial diamonds known as 'bors' are used. The diamonds are mounted in a holder. The diamond should be kept pointed down at an angle of 15° and a good amount of coolant is applied while dressing. Very light cuts only may be taken with diamond tools. Good supply of coolant should be used when dressing with a diamond, as overheating can cause the diamond to fracture or drop out of its setting. Very light cuts only may be taken with diamond tools

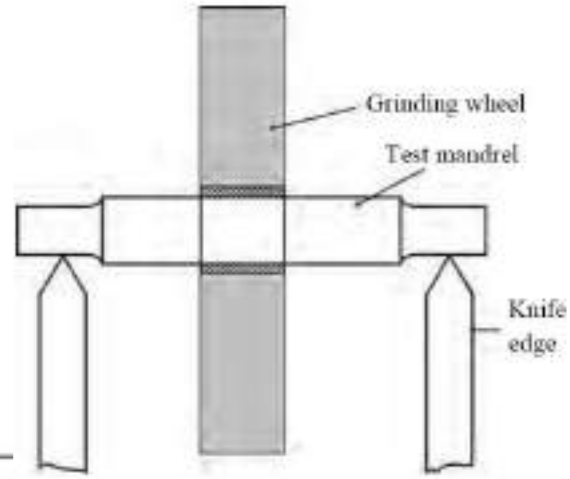


Fig. 20 shows dressing by a diamond tool dresser.

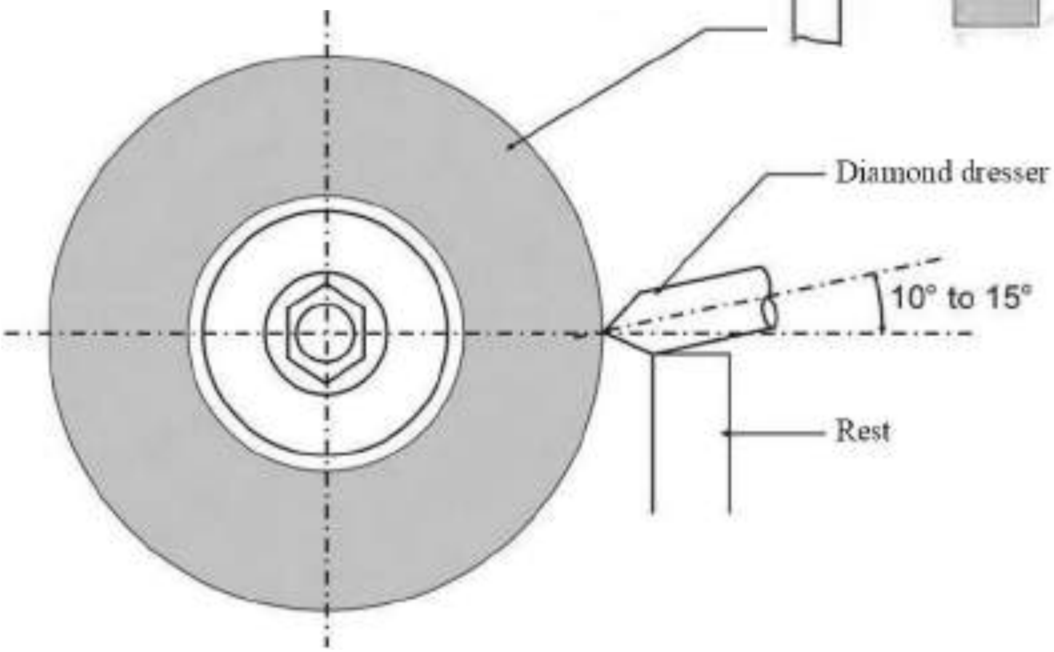


Fig 20 Dressing of a grinding wheel (Diamond dresser method)

b. Truing

The grinding wheel becomes worn from its original shape because of breaking away of the abrasive and bond. Sometimes the shape of the wheel is required to be changed for form grinding. For these purposes the shape of the wheel is corrected by means of diamond tool dressers. This is done to make the wheel true and concentric with the bore or to change the face contour of the wheel. This is known as truing of grinding wheels.

Diamond tool dressers are set on the wheels at 15° and moved across with a feed rate of less than 0.02mm. A good amount of coolant is applied during truing.

1.15 Balancing of grinding wheels

Grinding wheels rotate at high speeds. The density and weight should be evenly distributed throughout the body of the wheel. If it is not so, the wheel will not rotate with correct balance. The grinding wheels are balanced by mounting them on test mandrels. The wheel along with the mandrel is rolled on knife edges to test the balance and corrected.

1.16 PARAMETERS OF GRINDING OPERATION

Parameters used in grinding operation are cutting speed, feed rate and depth of cut. These parameters are described below.

1.16.1 Cutting Speed

Cutting speed is grinding wheel is the relative peripheral speed of the wheel with respect to the workpiece. It is expressed in meter per minute (mpm) or meter per second (mps). The cutting speed of grinding wheel can be calculated as $V = \frac{\pi D N}{1000}$ mpm

where, *D* is diameter of grinding wheel in mm. *N* are the number of revolution of grinding wheel if *N* is expressed in number of revolutions per minute, *V* will be in mpm, if *N* is expressed in number of revolution per second, *V* will be in mps.

1.16.2 Feed Rate

Feed rate is a significant parameter in case of cylindrical grinding and surface grinding. Feed rate is defined as longitudinal movement of the workpiece relative to axis of grinding wheel per revolution of grinding wheel. Maximum feed rate should be upto 0.9 time of face width of grinding wheel for rough grinding and upto 0.6 times of face width of grinding wheel for finish grinding. Feed cannot be equal to or more than the width of grinding wheel. Feed is used to calculate the total grinding time as given below.

$$T = \frac{L \ x \ i}{S \ x \ N} \ x \ K$$

where *T* is the grinding time (min) *L* is the required longitudinal travel in mm. *i* is the number of passes required to cover whole width *S* is the longitudinal feed rate (mm/rev.). *N* is the rpm and *K* is the coefficient depending on the specified grade of accuracy and class of surface finish for rough grinding *K* = 1 to 1.2 and for finish grinding *K* = 1.3 to 1.5.

1.16.3 Depth of Cut

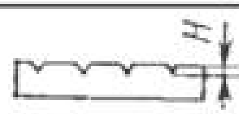
Depth of cut is the thickness of the layer of the metal removal in one pass. It is measured in mm. normally depth of cut is kept ranging 0.005 to 0.04 mm. Smaller depth of cuts are set for finish and precision grinding.

1.17 SUPERFINISHING PROCESS

To ensure reliable performance and prolonged service life of modern machinery, its components require to be manufactured not only with high dimensional and geometrical accuracy but also with high surface finish. The surface finish has a vital role in influencing functional characteristics like wear resistance, fatigue strength, corrosion resistance and power loss due to friction. Unfortunately, normal machining methods like turning, milling or even classical grinding cannot meet this stringent requirement.

Table 1 illustrates gradual improvement of surface roughness produced by various processes ranging from precision turning to superfinishing including lapping and honing.

Table 1. Surface Roughness of various Manufacturing Processes

Process	Diagram of resulting surface	Height of micro irregularity (µm)
Precision Turning		1.25-12.50
Grinding		0.90-5.00
Honing		0.13-1.25
Lapping		0.08-0.25
Super Finishing		0.01-0.25

Therefore, superfinishing processes like lapping, honing, polishing, burnishing are being employed to achieve and improve the above-mentioned functional properties in the machine component.

1.17.1 OBJECTIVES OF SURFACE PREPARATION

Surfaces are very important due to various commercial and technological reasons. These reasons may be different depending on different applications of the product. The main objectives are described below.

a) All smooth surfaces which are free from scratches and blemishes provide good aesthetic appearance. This all add value to product and give a favourable impression to the customers.
b) Smooth surfaces free from scratches and sharp corners and edges give safety to users.
c) Friction and wear also decided by surface conditions. In case of mating parts, the mating surfaces should be perfectly finished to avoid wear and energy loss due to friction.
d) Good quality surfaces improve mechanical and physical properties. Any surface flow can act as a point of stress concentration.
e) A slightly rough surface having uniform and constantly maintained value of surface roughness provides anti-glazed property to the same.
f) Smooth surfaces improve capability to make good electrical contacts.

1.17.2 TYPES OF SURFACE FINISHING PROCESSES

Manufacturing process employed determines surface finish level. Some processes are inherently capable of producing better surfaces than others. The processes recognized for good surface finish are honing, lapping, polishing and surface finishing.

1.17.2.1 HONING

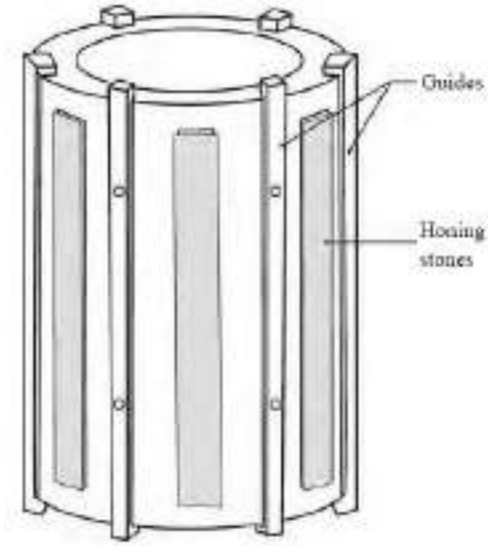
Honing is a surface finishing operation based on abrasive action performed by a set of bonded abrasive sticks. It is generally used to finish bores of cylinders of IC engine, hydraulic cylinders, gas barrels, bearings, etc. It can reduce the level of surface roughness below 32 μm. It produces a characteristics surface pattern as cross hatched which is a fit case to retain lubrication layer to facilitate motion to moving parts, their best example is IC engine. The honing tool used to finish internal surface is shown in Figure 21. The honing tool consists of a set of bonded abrasive sticks. The number of sticks mounted on a tool depends on its circumferential area. Number of sticks may be more than a dozen.

The motion of a honing tool a combination of rotation and reciprocation (linear). The motion is managed in such a way that *a given point on the abrasive stick does not trace the same path repeatedly*. The honing speed may be kept upto 10 cms per sec. Lower speeds are recommended for better surface finish. Manufacturing defects like slight eccentricity a way surface, light tapper, less of circulating can also be corrected by honing process.

The process of honing is always supported by flow of coolants. It flashes away the small chips and maintains a low and uniform temperature of tool and work.

Honing Machines

Honing machines resembles with vertical drilling machines in their construction. Reciprocating motion of spindle is obtained by hydraulic means. The rotary motion may be by hydraulic motor or by a gear train.



Depending upon the movement of spindle or hones a machine may be vertical honing machine or horizontal honing machine. Generally honing vertical honing machines are used. Horizontal honing machines are recommended for finishing internal of long gun barrels.

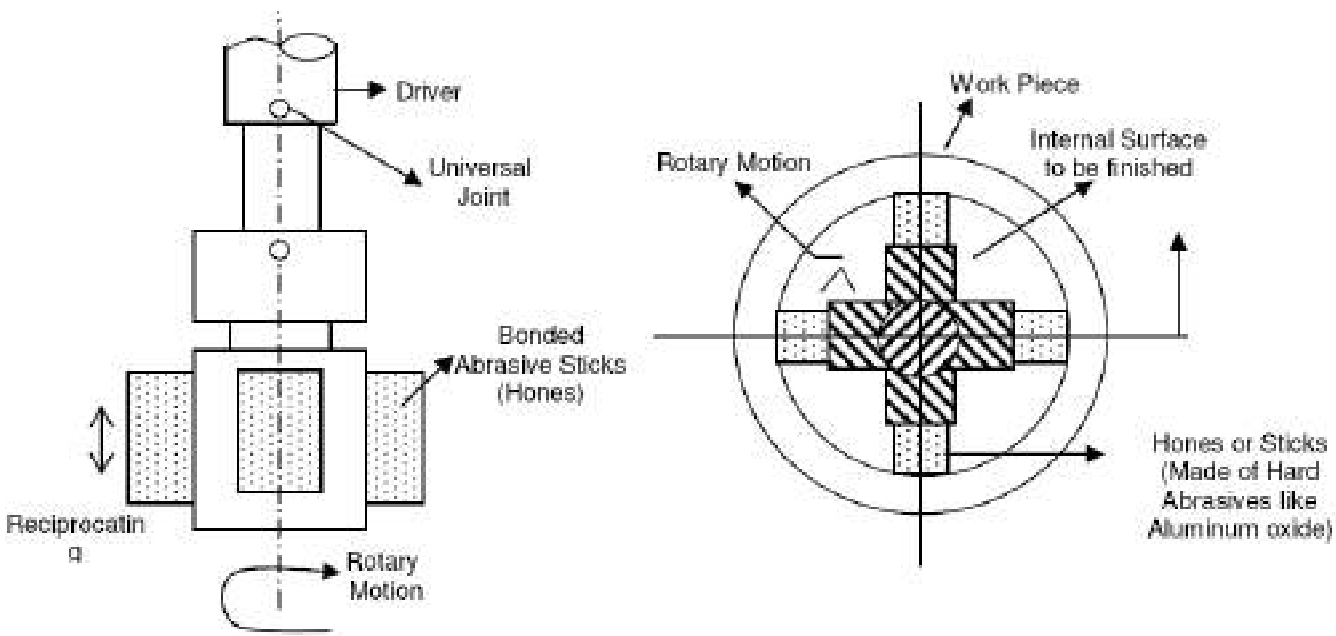


Fig. 21 Honing Tool and its Operations

1.17.2.2 LAPPING

Lapping is also one of the abrasive processes used to produce finished (smoothly accurate) surfaces. It gives a very high degree of accuracy and smoothness so it is used in production of optical lenses, metallic bearing surfaces, measuring gauges, surface plates and other measuring instruments. All the metal parts that are subjected to fatigue loading or those surfaces that must be used to establish a seal with a mating part are often lapped. The process of lapping uses a bonded abrasive tool and a fluid suspension having very small sized abrasive particles vibrating between the workpiece and the lapping tool. The process of lapping is shown in Figure 22. The fluid with abrasive particles is referred as lapping compound. It appears as a chalky paste. Normally the fluid used in lapping compound is oil or kerosene. The fluid should have slightly lubricating properties to make the action of abrasive mild in nature. Abrasives used in lapping compound are aluminium oxide and silicon carbide. Their girt size is kept 300 to 600 μm. It is hypothesized that two alternative cutting mechanisms are working in the process of lapping.

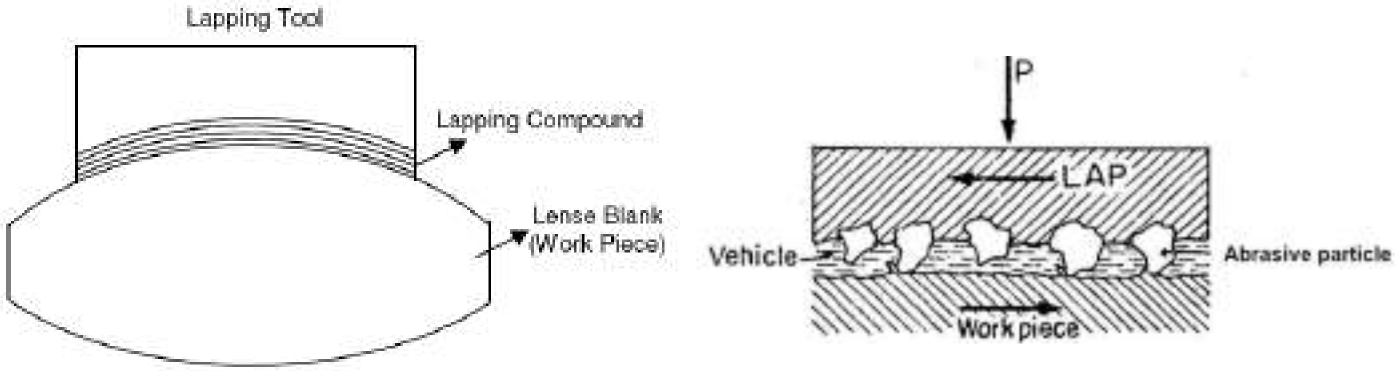


Figure 22 : Lapping Process

In first mechanism the abrasive particles roll and slide between the lapping tool and workpiece. These particles produce small cuts on both surfaces. Another mechanism suppose to work in lapping is that the abrasives become imbedded in the lap surface to give cutting action like in case of grinding. It is assumed that lapping is due to the combination of these two above mentioned mechanism. Lapping can be done manually but use of lapping machine makes the process accurate, consistent and efficient.

Machine Lapping

Machine lapping is recognized as fast lapping process. Gudgeon pins with 25 mm diameter and 75 mm long can be lapped at the rate of 500 units per hour. Mechanical lapping machines have vertical construction with the work holder mounted on the lower table which is given oscillatory motion. The upper lap is stationery and floating while lower one revolves at 60 rpm. Some special purpose lapping machines are available for lapping of small parts such as piston pins ball bearing races, etc. in machine lapping a pressure upto 0.02 N/mm^2 for soft material and 0.5 N/mm^2 for hard material is applied.

Lapping Applications

Materials processed by lapping range from steel, cast iron to non-ferrous metal like copper, brass and lead. Wooden parts, made of hard wood, can also be finished using wood laps. Lapping removes material at a very slow rate. So lapping is generally followed by accurate machining of workpieces. Lapping is a costlier process so its applications are justified only when high grade of surface finishing is required. Lapped surfaces are well resistant to corrosion and wear, used in manufacturing of high precision parts. Work Piece)

1.17.2.3 **Buffing**

Metal polishing, also termed buffing, it is the smoothing and brightening process of a surface by the rubbing action of fine abrasive in a lubricating binder applied intermittently to a moving wheel of wood cotton fabric felt or a cloth or a felt belt. Buffing is used to give a much more lustrous reflective finish that cannot be obtained by polishing. This gives a smooth finish by forming very thin lines that are not visible to the naked eye.

Buffing wheels are made of felt, pressed and glued layers of duck, or some select cloth and also of leather. The abrasive is mixed with a binder and is applied on either the buffing wheel or on the work. The buffing wheel rotates with a high peripheral speed up to 40 m/sec. The abrasive may consist of iron oxide, chromium oxide etc. The binder is a paste consisting of wax mixed with grease, paraffin and kerosene or turpentine and other liquids. The stone is given as oscillating motion in the axial direction and simultaneously the job is given a rotary motion about the axis.

Buffing is commonly used in metal polishing of pressure cookers, cookware, and kitchenware. Pipes which are used in pharmaceutical, dairy and water industries are buffed to maintain hygienic conditions and prevent corrosion.

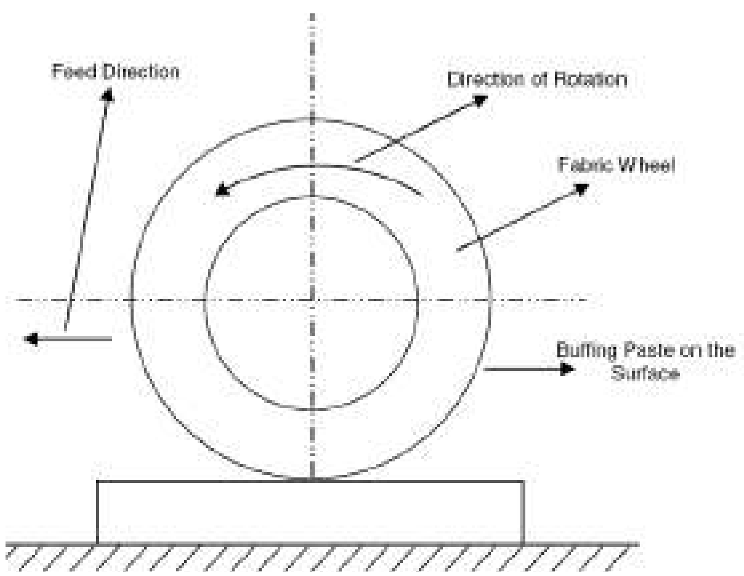


Fig. 23. Buffing Process

1.17.2.4 Super finishing

Super finishing is a type of grinding also known as Micro finishing. An abrasive stick of very fine girt size (400-600 mesh) is retained in a suitable holder and held against the workplace surface under a high spring pressure. The stick is given a feeding and oscillatory motion. The work piece is rotated or reciprocated accordingly to the requirements of the shape being super finished. A lubricant is also fed into the contact surface. An extremely high quality surface with almost no defects is obtained. Holes can also be super-finished.

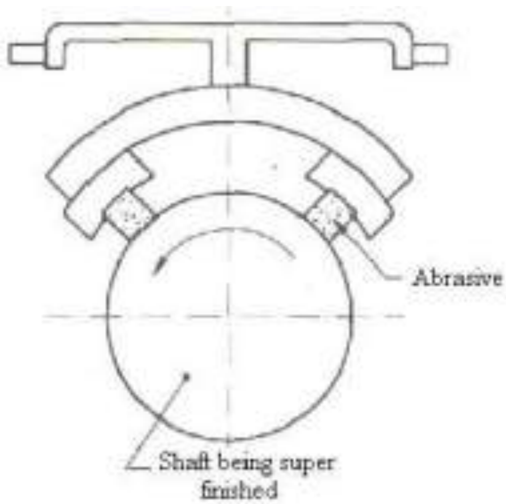


Fig. 24 Superfinishing of a shaft

1.17.2.5 Burnishing

The burnishing process consists of pressing hardened steel rolls or balls into the surface of the workpiece and imparting a feed motion to the same. Ball burnishing of a cylindrical surface is illustrated in Fig. 25.

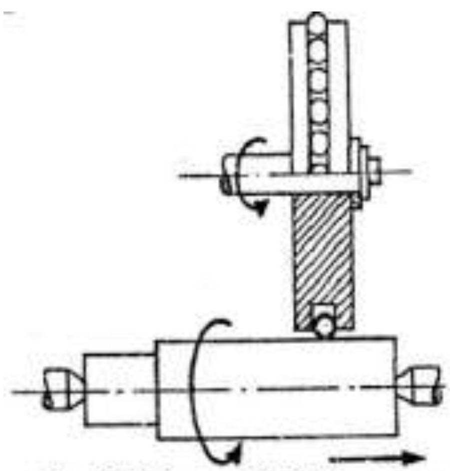


Fig. 25. Scheme of Ball Burnishing

During burnishing considerable residual compressive stress is induced in the surface of the workpiece and thereby fatigue strength and wear resistance of the surface layer increase.

1.17.2.6 Magnetic float polishing

Magnetic float polishing (Fig.26) finds use in precision polishing of ceramic balls. A magnetic fluid is used for this purpose. The fluid is composed of water or kerosene carrying fine ferro-magnetic particles along with the abrasive grains. Ceramic balls are confined between a rotating shaft and a floating platform. Abrasive grains ceramic ball and the floating platform can remain in suspension under the action of magnetic force. The balls are pressed against the rotating shaft by the float and are polished by their abrasive action. Fine polishing action can be made possible through precise control of the force exerted by the abrasive particles on the ceramic ball.

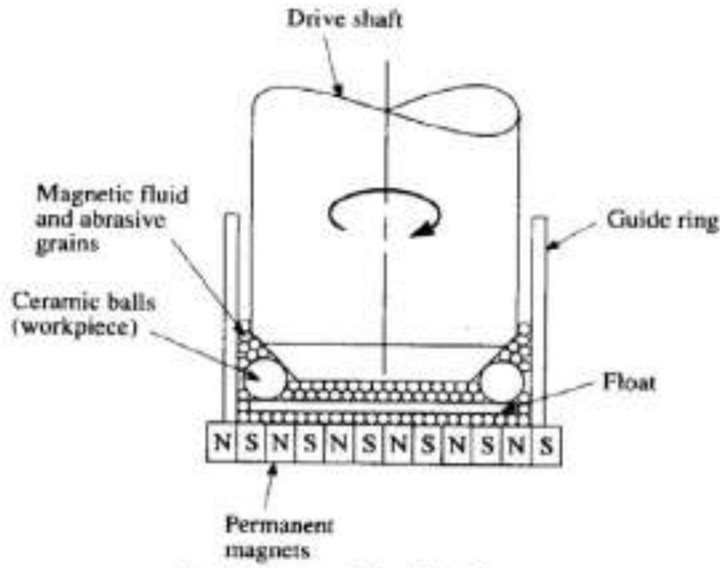


Fig. 26 Magnetic Float Polishing

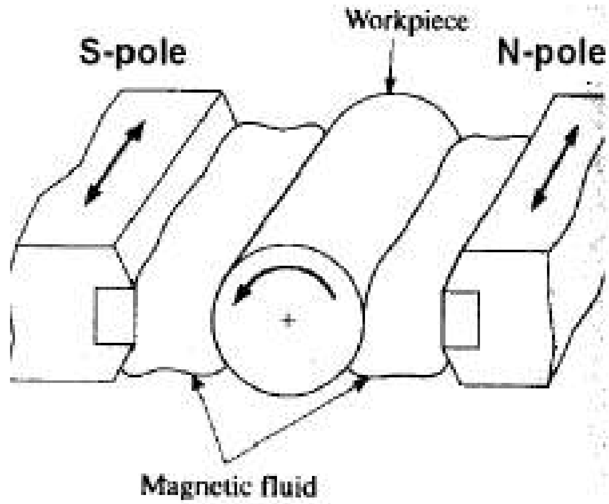


Fig. 27 Magnetic Field Assisted Polishing

1.17.2.7 Magnetic field assisted polishing

Magnetic field assisted polishing is particularly suitable for polishing of steel or ceramic roller. The process is illustrated schematically in Fig. 27. A ceramic or a steel roller is mounted on a rotating spindle. Magnetic poles are subjected to oscillation, thereby, introducing a vibratory motion to the magnetic fluid containing this magnetic and abrasive particles. This action causes polishing of the cylindrical roller surface. In this technique, the material removal rate increases with the field strength, rotational speed of the shaft and mesh number of the abrasive. But the surface finish decreases with the increase of material removal rate.

BROACHING

1.1. BASIC PRINCIPLES OF BROACHINING

Broaching is a **method** of removing metal by pushing or pulling a cutting tool called a **broach** which cuts in a fixed path. 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 Fig. 1. 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 (Fig. 4.10.1).. 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.

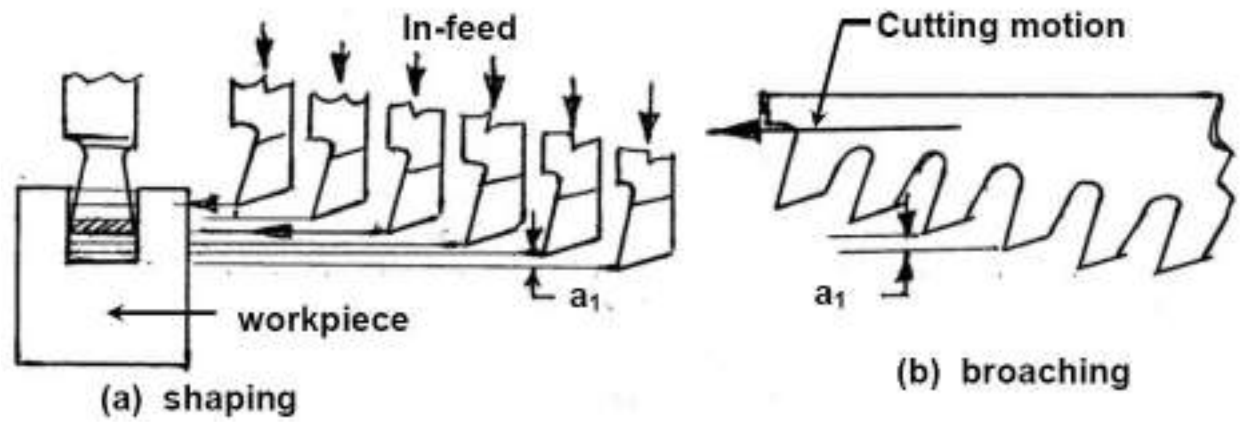


Fig. .1 Basic principle of broaching.

Machining by broaching is preferably used for making straight through holes of various forms and sizes of section, internal and external through straight or helical slots or grooves, external surfaces of different shapes, teeth of external and internal splines and small spur gears etc.

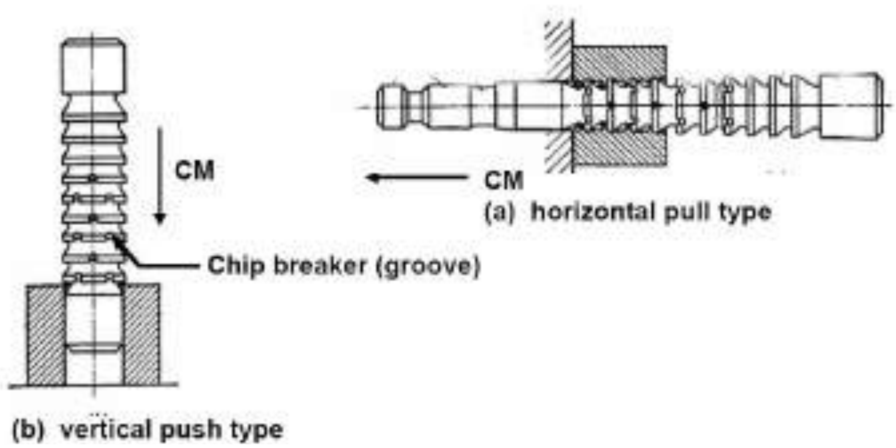


Fig. 2 Schematic views of finishing hole by broaching.

Fig. 2 schematically shows how a through hole is enlarged and finished by broaching. The cutting tool is called a broach, and the machine tool is called a broaching machine. The shape of the machined surface is determined by the contour of the cutting edges on the broach, particularly the shape of final cutting teeth. Broaching is a highly productive method of machining.

The cutting tool is called a broach, and the machine tool is called a broaching machine. The shape of the machined surface is determined by the contour of the cutting edges on the broach, particularly the shape of final cutting teeth. Broaching is a highly productive method of

machining. Advantages include good surface finish, close tolerances, and the variety of possible machined surface shapes, some of them can be produced only by broaching. Owing to the complicated geometry of the broach, tooling is expensive. Broaching is a typical mass production operation.

Productivity improvement to ten times or even more be not uncommon, as the metal removal rate by broaching is vastly greater. Roughing, semi finishing and finishing of the component is done just in one pass by broaching, and this pass is generally accomplished in seconds. Broaching can be used for machining of various integrate shapes which can not be otherwise machined with other operations. Some of the typical examples of shapes produced by internal broaching are shown in Fig. 3..

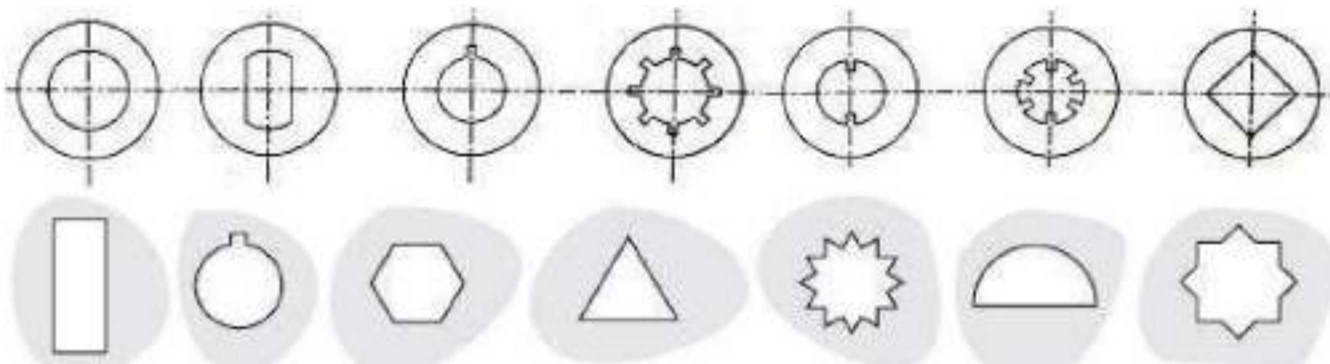


Fig. 3 Typical Shapes produced by Internal Broach

1.2 CLASSIFICATION OF BROACHES

Large variety of broaches are used in the Industries. They are categorized as follows:

1. Based on the method of operation
   a. Push type broach b. Pull type broach
2. Based on the kind of operation performed
   a. Internal broach b. External Broach
3. Based on construction
   a. Solid broach
   b. Built up broach
   c. Rotor cut broach
   d. Inserted tooth broach
   e. Progressive cut broach
   f. Overlapping tooth broach
4. Based on use

a. Single purpose broach b. Multi purpose broach

5. Based on functions
   a. Keyway broach
   b. Surface broach
   c. Spline broach
   d. Burnishing broach
   e. Roughing broach

1.3 BROACH CONSTRUCTION / BROACH TOOL GEOMETRY / BROACH TOOL NOMENCLATURE

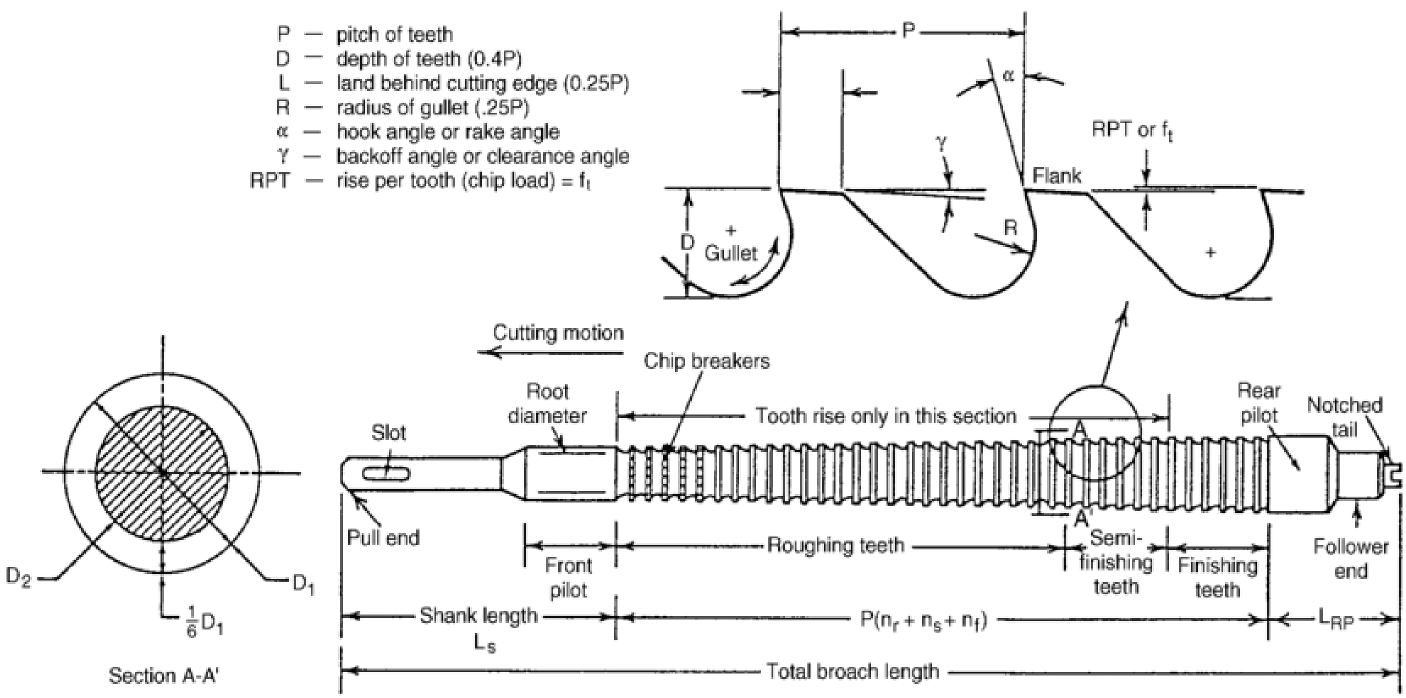


Fig.4. Broach Tool Nomenclature

Front pilot:

When an internal pull broach is used, the pull end and front pilot are passed through the starting hole. Then the pull end is locked to the pull head of the broaching machine. The front pilot assures correct axial alignment of the tool with the starting hole, and serves as a check on the starting hole size.

Length:

The length of a broach tool, or string of tools, is determined by the amount of stock to be removed, and limited by the machine stroke.

Rear pilot:

The rear pilot maintains tool alignment as the final finish teeth pass through the workpiece hole. On round tools the diameter of the rear pilot is slightly less than the diameter of the finish teeth.

Cutting teeth:

Broach teeth are usually divided into three separate sections along the length of the tool: the roughing teeth, semi-finishing teeth and finishing teeth.

The first roughing tooth is proportionately the smallest tooth on the tool. The subsequent teeth progressively increase in size up to and including the first finishing tooth. The difference in height between each tooth, or the rise per tooth, is usually greater along the roughing section and less along the semi-finishing section. All finishing teeth are the same size. The face is ground with a hook or face angle that is determined by the workpiece material. For instance, soft steel workpieces usually require greater hook angles; hard or brittle steel pieces require smaller hook angles.

Tooth land:

The land supports the cutting edge against stresses. A slight clearance or back-off angle is ground onto the lands to reduce friction. On roughing and semi-finishing teeth, the entire land is relieved with a back-off angle. On finishing teeth, part of the land immediately behind the cutting edge is often left straight, so that repeated sharpening (by grinding the face of the tooth) will not alter the tooth size.

Tooth pitch:

The distance between teeth, or pitch, is determined by the length of cut and influenced by type of workpiece material. A relatively large pitch may be required for roughing teeth to accommodate a greater chip load. Tooth pitch may be smaller on semi-finishing teeth to reduce the overall length of the broach tool. Pitch is calculated so that, preferably, two or more teeth cut simultaneously. This prevents the tool from drifting or chattering.

Tooth gullet:

The depth of the tooth gullet is related to the tooth rise, pitch and workpiece material. The tooth root radius is usually designed so that chips curl tightly within themselves, occupying as little space as possible.

1.4 MATERIAL OF BROACH

Common broach material is 18-4-1 stainless steel. As its name indicates, it has 4% chromium, 1% vanadium and 18% tungsten. This is corrosion and wear resistant steel. Carbide is also recommended for broach making, these broaches are used for broaching brittle material like cast iron in automobile industry. Inserted bit type and cemented carbide type broaches are also preferred to reduce the cost of broaches.

1.5 DIFFERENT TYPES OF BROACHES

A broach is normally made in one piece and it is called a solid broach. Different types of the broaches are described below.

Progressive Cut Broaches

These broaches have cutting edges, a part of which are of the same height along the broach but have different widths. In progressive cut broaching, metal is removed in thick layers by each tooth from only part of the work surface. The last tooth of a progressive cut broach remove a very thin layer over the entire profile of the work surface as in ordinary cut broaching.

Rotor Cut Broaching

Rotor cut broaches are used for removing large amount of material in holes in forgings or castings where a primary cutting operation is not desired. Their teeth are staggered around the periphery at different sections so as to shear the work and allow chip clearance. This is recommended for making square holes from a round cost one.

Burnishing Broaches

Burnishing broaches are used for making glazed and finished surface on a steel workpiece. These are used for finish the hole too. Burnishing teeth are rounded and not cut but compress and rub the surface of metal. The amount of stock is intentionally left for burnishing. Its thickness should not be more than 0.025 mm.

1.6 BROACHING MACHINES CLASSIFICATIONS

Broaching machines are generally specified by

- o Type; horizontal, vertical etc.
- o Maximum stroke length
- o Maximum working force (pull or push)
- o Maximum cutting velocity possible
- o Power
- o Foot print

Most of the broaching machines have hydraulic drive for the cutting motion. Electro-mechanical drives are also used preferably for high speed of work but light cuts.

There are different types of broaching machines which are broadly classified

- According to purpose of use
  - ✓ general purpose
  - ✓ single purpose
  - ✓ special purpose
- According to nature of work
  - ✓ internal broaching
  - ✓ external (surface) broaching
- According to configuration
  - ✓ horizontal
  - ✓ vertical
- According to number of slides or stations
  - ✓ single station type
  - ✓ multiple station type
  - ✓ indexing type
- According to tool / work motion
  - ✓ intermittent (one job at a time) type
  - ✓ continuous type

Some of the broaching machines of common use have been discussed here.

1.7 PUSH BROACHING MACHINES

In these machines the broach movement is guided by a ram. These machines are simple, since the broach only needs to be pushed through the component for cutting and then retracted. The work piece is fixed into a boring fixture on the table. Even simple arbor presses can be used for push broaching.

1.7.1 Push down type vertical surface broaching machine

Fig. 5 shows the push down type vertical surface broaching machine. It consists of a box shape column, slide and drive mechanism. Broach is mounted on the slide which is hydraulically operated and accurately guided on the column ways. Slide with the broach travels at various speeds. The slide is provided with quick return mechanism. The worktable is mounted on the base in front of the column. The fixture is clamped to the table. The work piece is held in the fixture.

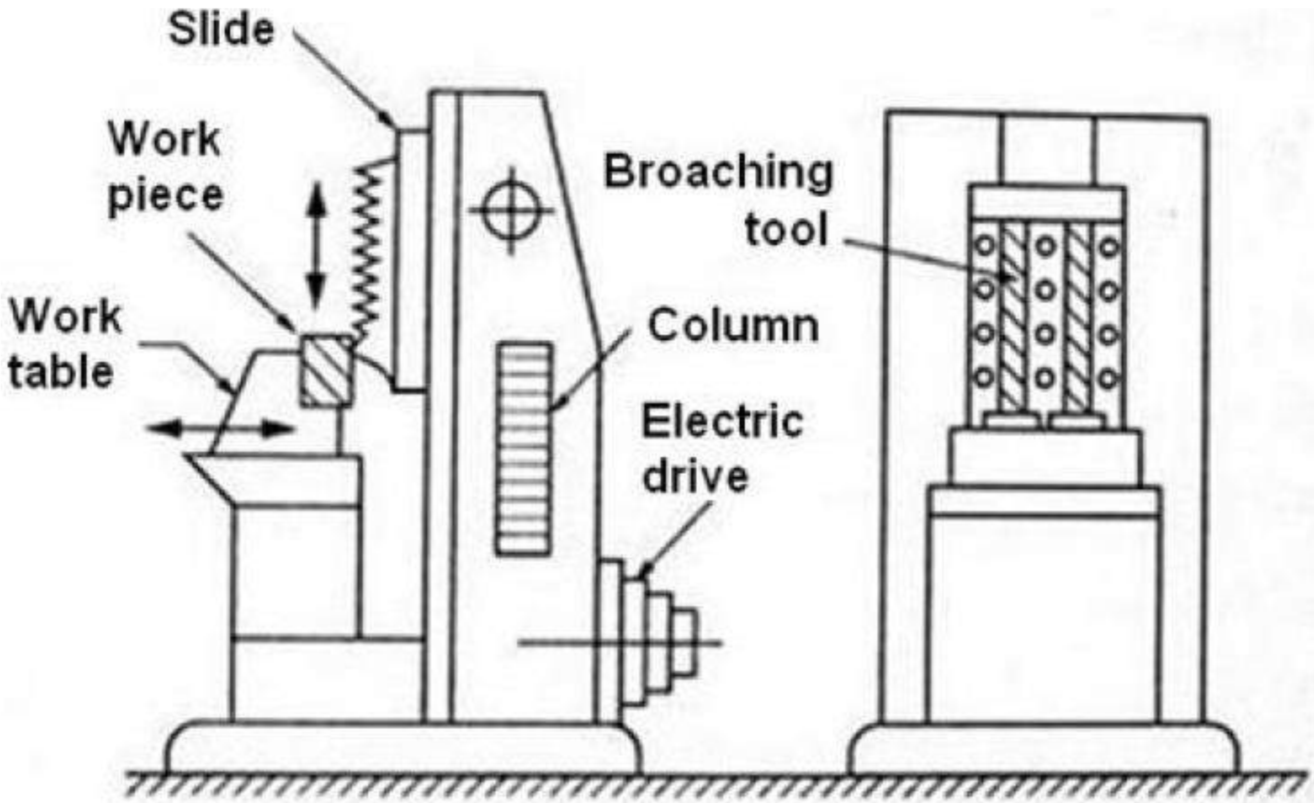


Fig. 5. Push down type vertical surface broaching machine

After advancing the table to the broaching position, it is clamped and the slide with the broach travel downwards for machining the workpiece. Then the table recedes to load a new work piece and the slide returns to its upper position. The same cycle is then repeated.

Vertical broaching machines occupy less floor space and are more rigid as the ram is supported by the base. They are mostly used for external or surface broaching though internal broaching is also possible and occasionally done.

1.8 PULL BROACHING MACHINES

These machines consist of a work holding mechanism, and a broach pulling mechanism along with a broach elevator to help in the removal and threading of the broach through the work piece. The work piece is mounted in the broaching fixture and the broach is inserted through the hole present in the work piece.

Then the broach is pulled through the work piece completely and the work piece is then removed from the table. Afterwards the broach is brought back to the starting point before a new work piece is located on the table. The same cycle is then repeated.

1.8.1. Pull type horizontal internal broaching machine

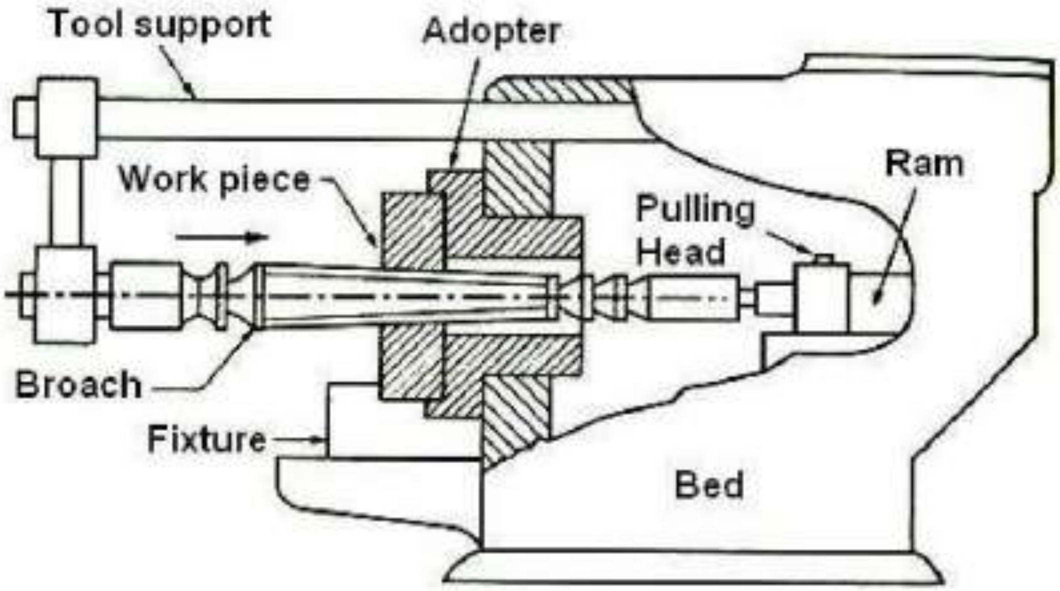


Fig. 6 Pull type horizontal internal broaching machine

Fig..1 shows the pull type horizontal internal broaching machine. This machine has a box type bed. The length of bed is twice the length of stroke. Most of the modern horizontal broaching machines are provided with hydraulic or electric drive. It is housed in the bed. The job is located in the adopter. The adopter is fitted in the front vertical face of the machine. The small end of the broach is inserted through the hole of the job and connected to the pulling head.

The pulling head is mounted in the front end of the ram. The ram is connected to the hydraulic drive mechanism. The rear end of the broach is supported by a guide. The broach is moved along the guide ways. It is used for small and medium sized works. It is used for machining keyways, splines, serrations, internal gears, etc.

Horizontal broaching machines are the most versatile in application and performance and hence are most widely employed for various types of production. These are used for internal broaching but external broaching work is also possible. The horizontal broaching machines are usually hydraulically driven and occupy large floor space.

1.8.2 Pull down type vertical internal broaching machine

This machine has an elevator at the top. The pulling mechanism is enclosed in the base of the machine. The workpiece is mounted on the table by means of fixture. The tail end of the broach is gripped in the elevator. The broach is lowered through the work piece.

The broach is automatically engaged by the pulling mechanism and is pulled down through the job. After the operation is completed, the broach is raised and gripped by the elevator. The elevator returns to its initial position. *This is illustrated in Fig. 7 (a).*

1.8.3 Pull up type vertical internal broaching machine

In this type, the ram slides on the vertical column of the machine. The ram carries the pulling head at its bottom. The pulling mechanism is above the worktable and the broach is in the base of the machine. The broach enters the job held against the underside of the table and is pulled upward. At the end of the operation, the work is free and falls down into a container. *This is illustrated in Fig. 7 (b).*

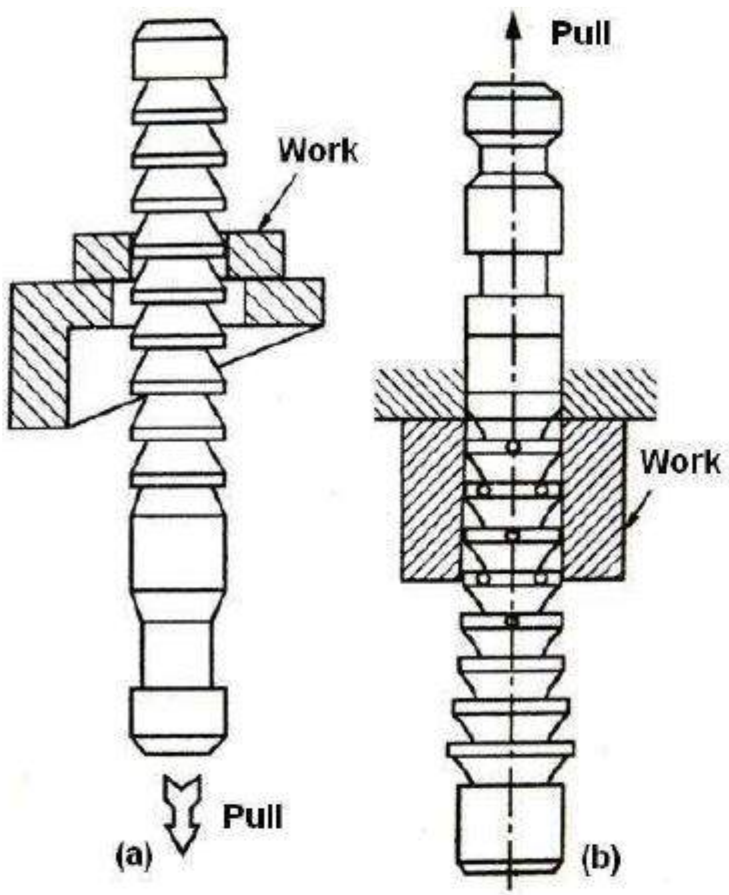


Fig. 7 Vertical internal broaching operation (a) pull down type (b) pull up type

1.9. SURFACE BROACHING MACHINES

In horizontal surface broaching machines, the broach is pulled over the top surface of the work piece held in the fixture on the worktable as shown in Fig. 8. The cutting speed ranges from 3 to 12 *mpm* with a return speed up to 30 *mpm*. The construction and working principle of horizontal surface broaching machine is similar to that of pull type horizontal internal broaching machine.

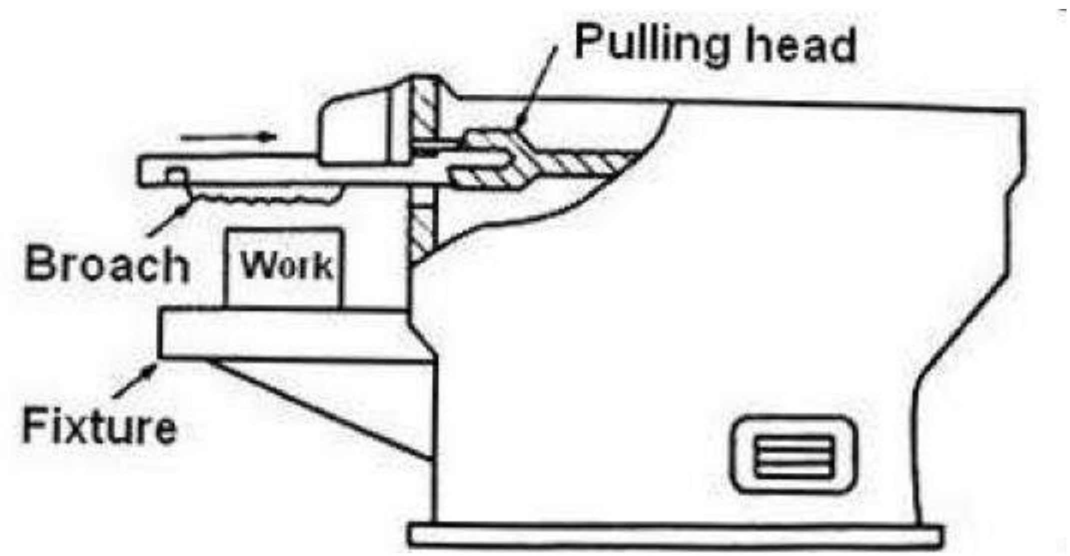


Fig. 8. Horizontal surface broaching machine

In vertical surface broaching machines, the work piece is held in the fixture while the surface broach is reciprocated with the ram on the vertical guide ways on the column as shown in Fig. 9.

Surface broaching is relatively simple since the broach can be continuously held and then it will carry out only a reciprocating action. The construction and working principle of vertical surface broaching machine is already discussed in the article no. 1.7.1 at page no.6

Instead of using simple broach some times the progressive cut type broach with the teeth segments distributed into the three areas as shown in Fig. 10 (b) is used in surface broaching. The progressive action reduces the maximum broaching force, but results in a longer broach.

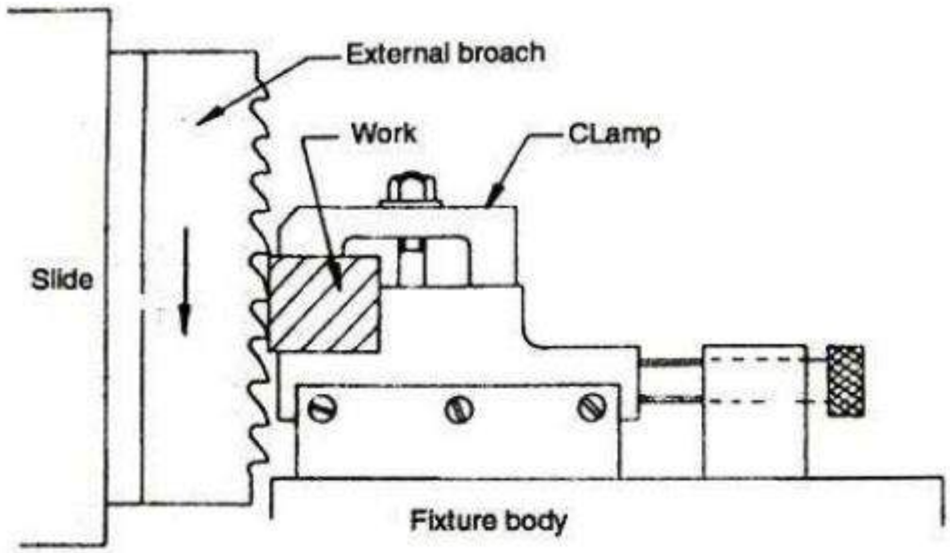


Fig. 9 Vertical surface broaching machine

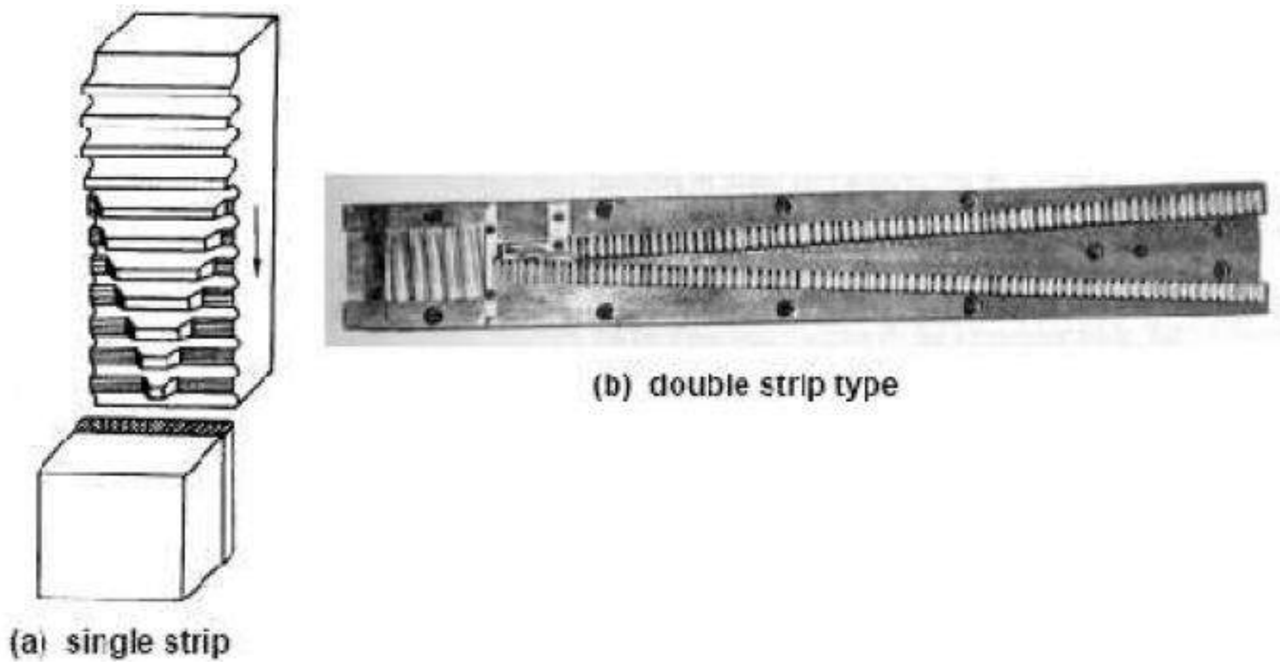


Fig. 10. Progressive Cut Broaches

1.10 CONTINUOUS BROACHING MACHINES

These broaching machines are also known as high production broaching machines. The reciprocation of the broach always involves an unproductive return stroke, which is eliminated in a continuous surface broaching machine. These machines are used for fast production of large number of pieces by surface broaching.

1.10.1 Horizontal continuous broaching machine

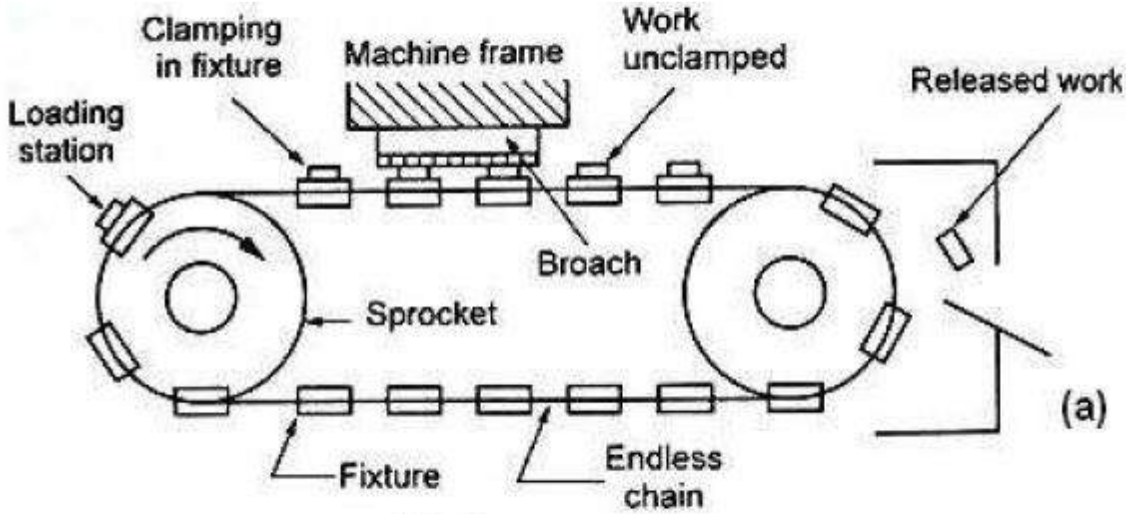


Fig. 11. Continuous broaching machine - Horizontal type

In this the small work pieces are mounted on the broaching fixtures which are in turn fixed to an endless chain continuously moving in between two sprockets. Broaches which are normally stationary are kept above the work pieces. The work pieces are pushed past the stationary broaches by means of the conveyor for cutting. The work pieces are loaded and unloaded onto the conveyor manually or automatically.

1.10.2 Rotary continuous broaching machine

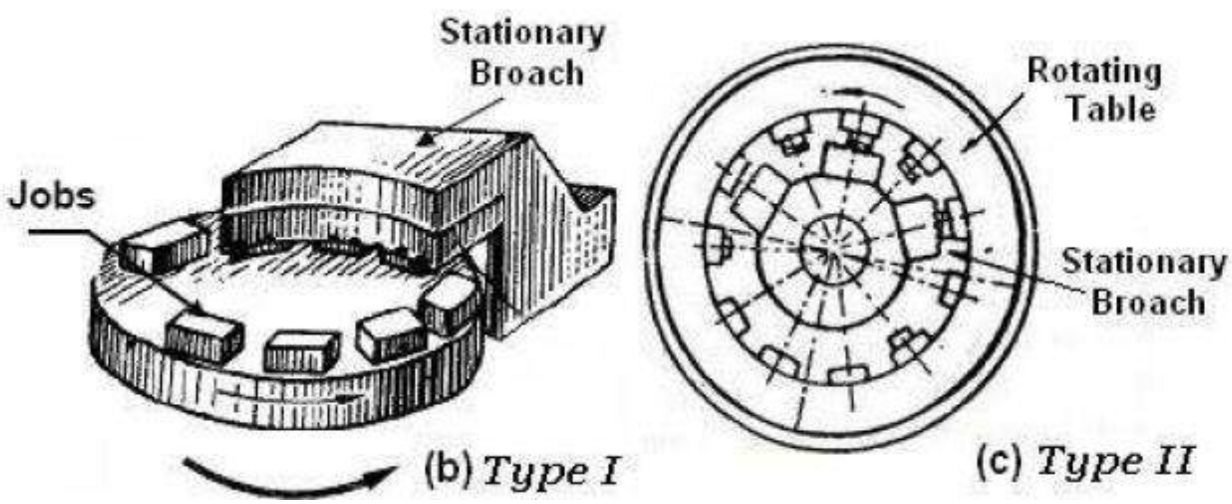


Fig. 11. Continuous broaching machine – Rotary type

Type I:

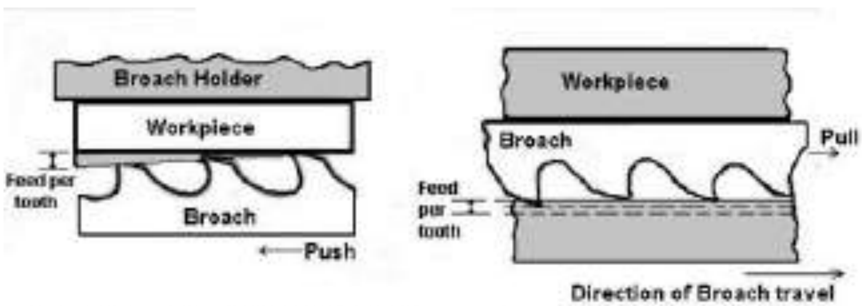
This machine has a rotary table and a vertical column. The vertical column has a guide way. An arm is fixed in the vertical column and it moves up and down in the guide way. Work pieces are clamped in the fixtures horizontally above the work table. The broach is fixed underside of the arm. Now the work table is rotated and the broaching operation is carried out.

Depth of cut is given by moving the work table in upward direction. *This is illustrated in Fig. 12 (b).*

Type II:

This machine has a ring shaped rotating work table. Work pieces are clamped in the fixtures in the inner periphery of the work table. The stationary broaches are fixed in the outer periphery of the vertical column located inside the work table. Now the table is rotated and the broaching operation is carried out. *This is illustrated in Fig. 12 (c).*

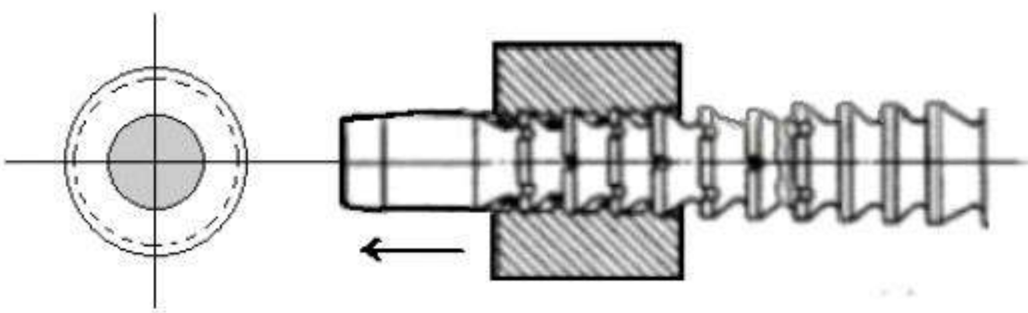
1.11 BROACHING MACHINE OPERATIONS



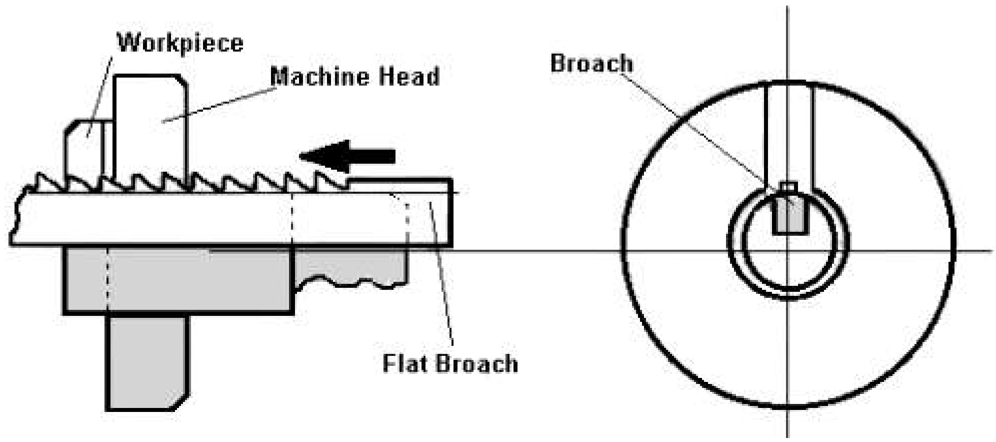
A push type broach in use for machining an external surface **A pull type broach in use for machining internal surface**

Broaching is applied for machining various internal and external surfaces, for a round or irregular shaped holes from 6 to 100mm in diameter, for external flat and contoured surfaces. Certain types of surfaces, for example, spline holes, are machined at the present time only by broaching due to the exceptional difficulties in machining such surfaces by other methods.

A number of important broaching operations are illustrated below. Some operations are done in one pass, but some operation arranged for repeated cuts to the design of the broach.



Broaching hole with a round back



Broaching a keyway in a hole with a keyway broach

The teeth of a gear or spline may be broached all together or one or a few at a time. A comparatively simple broach can be made to cut one or a few tooth spaces. After one pass, the gear blank is indexed and more of its teeth are cut. Successive passes are made until all the teeth are finished.

1.12 ADVANTAGES AND LIMITATIONS OF BROACHING

Following are the features and **advantages** of the broaching.

1. The rate of production is very high.
2. Skill is required from the operator to perform a broaching operation. In most cases, the operator merely loads and unloads the work-piece.
3. High accuracy and a high class of surface finish are possible. A tolerance of ±0.0075mm and a surface finish about 0.8 microns (1 micron=0.001mm) can be easily obtained in broaching.
4. Both roughing finishing cuts are completed in one pass of the tool.
5. The broaching process is used for internal and external surface finishing.
6. Any form or shape that can be reproduced on a broaching can be machined.
7. Cutting fluid may be readily applied where it is most effective because a broach tends to draw the fluid into the cut.

For specific reasons, however, **limit** the utilisation of the broaching process. They are:

1. High tool cost. A broach normally makes only one job and is expensive to make and sharpen.
2. Very large work-piece cannot be broached.
3. The surface to be broached should not have an obstruction.
4. Broaching cannot be used to remove a large amount of stock.

5. Parts to be broached must be capable of being rigidly supported and must be able to withstand the force that set up during cutting.
6. Only through holes and surfaces can be machined.
7. Economic only when the production volume is large.

GEAR MANUFACTURING PROCESS

1. 1. INTRODUCTION

Gears are used extensively for transmission of power. They find application in: Automobiles, gear boxes, oil engines, machine tools, industrial machinery, agricultural machinery, geared motors etc.

A gear is a round blank having teeth along its periphery. Gears are used to transfer power or torque from prime mover to the place where it is to be used. Along with the transmission of power gears also transfer the accurate velocity ratio between two shafts.

Velocity ratio is defined as the ratio of rpm (revolution per minute) of driven shaft to the rpm of driver shaft.

$$\text{Velocity Ratio} = \frac{\text{RPM of driven shaft or driven gear}}{\text{RPM of gear driver shaft or driver}}$$

Power is normally transferred with the help of pair of gears in mesh together, each of these two are mount on driven shaft and driver shaft. To meet the laborious service conditions the gears should have: robust construction, reliable performance, high efficiency, economy and long life. Also, the gears should be fatigue free and free from high stresses to avoid their frequent failures. The gear drives should be free form noise, chatter and should ensure high load carrying capacity at constant velocity ratio. To meet all the above conditions, the gear manufacture has become a highly specialized field. Below, we shall discuss the various materials and manufacturing processes to produce gears.

1.2 MATERIALS USED IN GEAR MANUFACTURING PROCESS

The various materials used for gears include a wide variety of cast irons, non ferrous material &non – material materials the selection of the gear material depends upon:-

i) Type of service
ii) Peripheral speed
iii) Degree of accuracy required

iv) Method of manufacture

v) Required dimensions & weight of the drive

vi) Allowable stress

vii) Shock resistance

viii) Wear resistance.

1) **Cast iron** is popular due to its good wearing properties, excellent machinability & Ease of producing complicated shapes by the casting method. It is suitable where large gears of complicated shapes are needed.
2) **Steel** is sufficiently strong & highly resistant to wear by abrasion.
3) **Cast steel** is used where stress on gear is nigh & it is difficult to fabricate the gears.
4) **Plain carbon steels** find application for industrial gears where high toughness combined with high strength.
5) **Alloy steels** are used where high tooth strength & low tooth wear are required.
6) **Aluminum** is used where low inertia of rotating mass is desired.
7) Gears made of **non – Metallic materials** give noiseless operation at high peripheral speeds.

1. 3 GEAR TERMINOLOGY

The gear terminology is explained below with reference to a spur gear which is a particular type of a gear. The detail of gear terminology is also indicated in Figure 1.1.

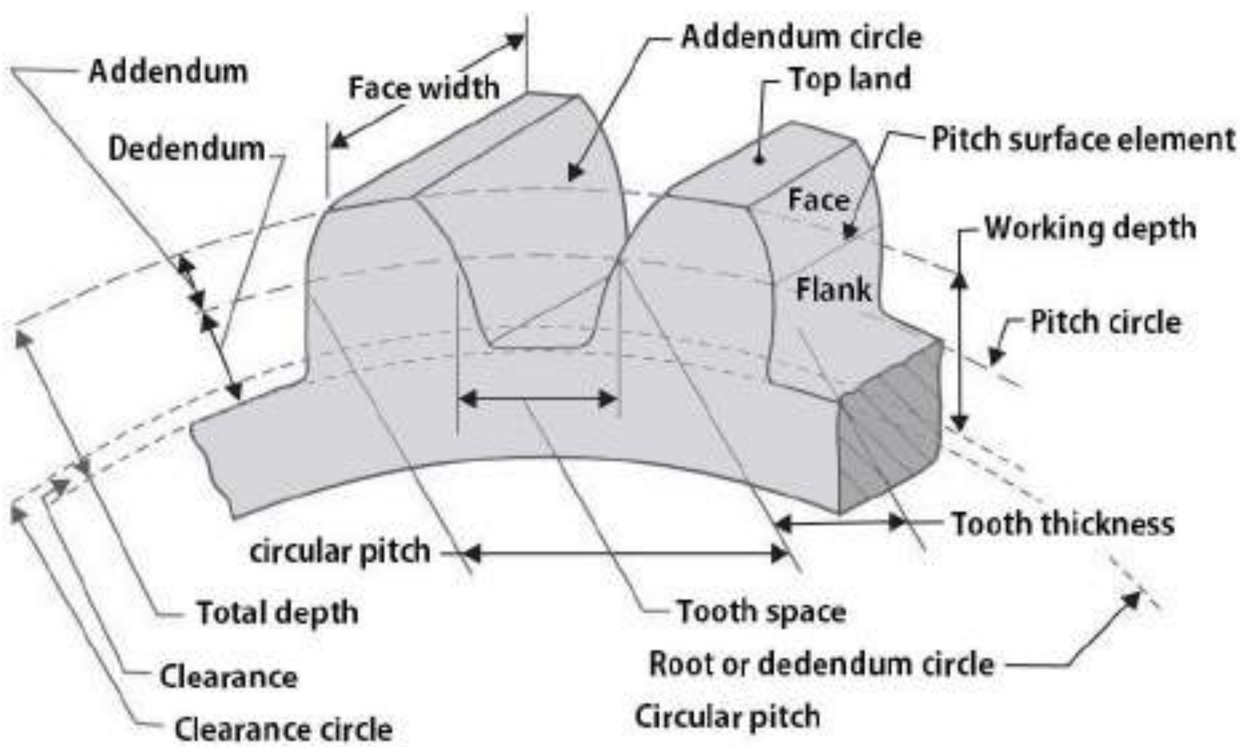


Fig. 1.1 Gear tooth terminology

Gear Blank

The metallic workpiece accurately sized and shaped which is used as workpiece for gear cutting is called gear blank. The diameter of gear blank is called gear blank diameter.

Addendum Circle

It is an imaginary circle which passes through top of all gear teeth and represents maximum diameter of a gear. This maximum diameter is equal to gear blank diameter.

Addendum

Addendum of a gear is the radial distance between addendum circle and pitch circle of the gear.

Pitch Circle

This is an imaginary circle along which thickness of a gear tooth becomes equal to spacing between them.

Dedendum

It is the radial distance between pitch circle and root circle of a gear.

Root Circle

Root circle is an imaginary circle which is supposes to pass through root of all gear teeth.

Tooth Clearance

This is the distance between the top of a tooth of one gear and the bottom of the corresponding tooth of other mating gear is known as clearance or tooth clearance.

Pressure Angle

The angle made by the line of action with the common tangent to the pitch circle is called pressure angle.

Face

It is the portion of the tooth lying between top of the tooth and pitch circle.

Thickness of a Gear Tooth

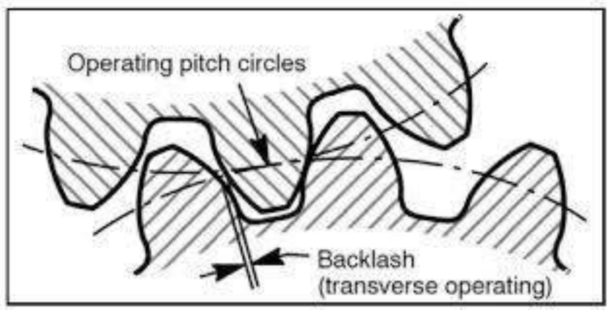
It is also called chordal thickness of gear tooth. It is width of two gear tooth measured along the pitch circle. At the pitch circle width of gear tooth becomes equal to the width of spacing between two consecutive gear teeth.

Flank

This is portion of the gear tooth between its pitch circle and root circle.

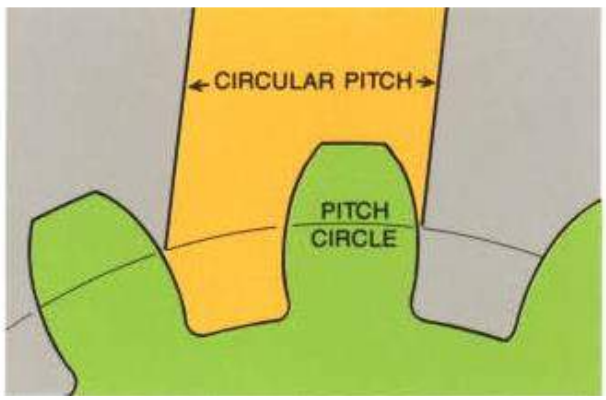
Backlash

It is difference between actual tooth thickness and the width of space at which it meshes with other gear.



Circular Pitch

It is the distance between corresponding points of adjacent teeth measured along the pitch circle.



Diametral Pitch

It is number of teeth of a gear per unit of pitch circle diameter.

Diametal pitch (p) = N/d

where '*N*' is the number of teeth '*d*' is the pitch circle diameter.

Module

It is reciprocal of diameteral pitch. It is linear distance in mm that each tooth of the gear would occupy if the gear teeth were spaced along the pitch diameter.

m = d/N

with usual notations; m is module of gear.

1.4 GEAR MANUFACTURING METHODS

Gear manufacturing can be divided into two categories namely forming and machining as shown in flow chart in Fig 1.1. Forming consists of direct casting, molding, drawing, or extrusion of tooth forms in molten, powdered, or heat softened materials and machining involves roughing and finishing operations. They are discussed in the different sections of this chapter.

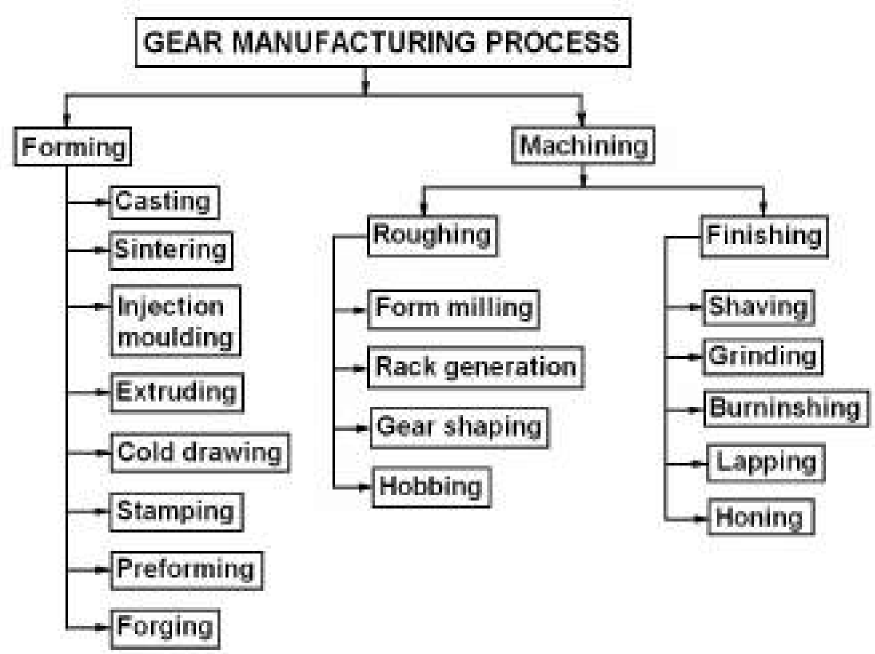


Fig. 1.1. Categories of gear manufacturing process

1.5 FORMING GEAR TEETH

Characteristics: In all tooth-forming operations, the teeth on the gear are formed all at once from a mold or die into which the tooth shapes have been machined. The accuracy of the teeth is entirely dependent on the quality of the die or mold and in general is much less than that can be obtained from roughing or finishing methods. Most of these methods have high tooling costs making them suitable only for high production quantities. The various forming techniques are discussed below in detail:

1.5.1. Casting

Gears can be produced by the various casting processes. Sand casting is economical and can take up large size and module, but the gears have rough surfaces and are inaccurate dimensionally. These gears are used in machinery where operating speed is low and where noise and accuracy of motion can be tolerated, for example, farm machinery and some hand operated devices.

Sand casting is suitable for one off or small batches. Large quantities of small gears are made by "Die – Casting". These gears are fairly accurate and need little finishing. However the materials used are low melting ones, such as alloys of zinc, aluminum and copper so, there gears are suitable for light duty applications only (light loads at moderate speeds), for example, gears used in toys. Cameras and counters and counters etc.

Gears made by **“Investment Casting”** may be accurate with good surface finish. These can be made of strong materials to withstand heavy loads. Moderate – size gears are currently being steel cast in metal moulds to produce performs which are later forged to size. Light gears of thermoplastics are made by “Injection Moulding”. This method is satiable for large volume production. However, gear tooth accuracy is no high and initial tool cost is high. These gears find use in instruments, household appliances etc.

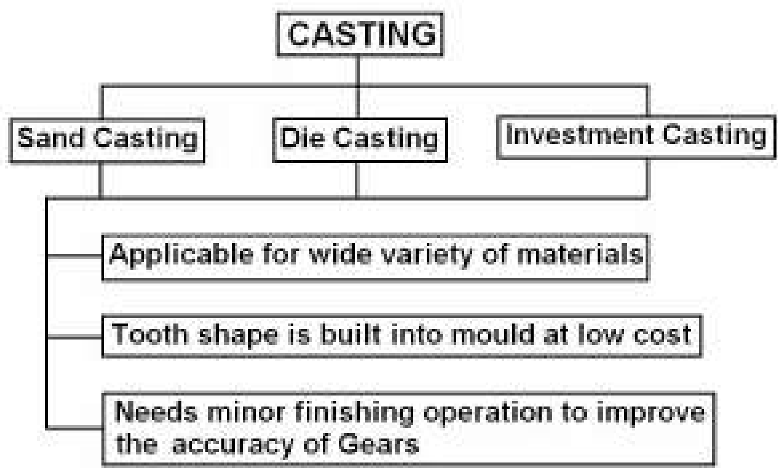


Fig.1.2 Casting processes

For phosphor bronze worm wheel rims, **“centrifugal casting”** is used far more extensively than any other method. Centrifugal casting is also applied to the manufacture of steel gears. Both vertical and horizontal axis spinners are used. After casting, the gears are annealed or nomalized to remove cooling stresses. They may then be heat treated, if required, to provide the needed properties. Centrifugally cast gears perform as well as rolled (discussed ahead) gears and are usually less expensive. **“Shell moulding**” is also sometimes used to produce small gears and the product is a good cast gear of somewhat lower accuracy than one made by investment casting but much superior to sand casting.

Sand casting, die casting and investment casting are the casting processes that are best suited for gears and are shown in fig 1.2. They are explained in the following sections:

1.5.2 Sintering or P/M process:

The powder metallurgy technique used for gear manufacture is shown in fig. 1.3

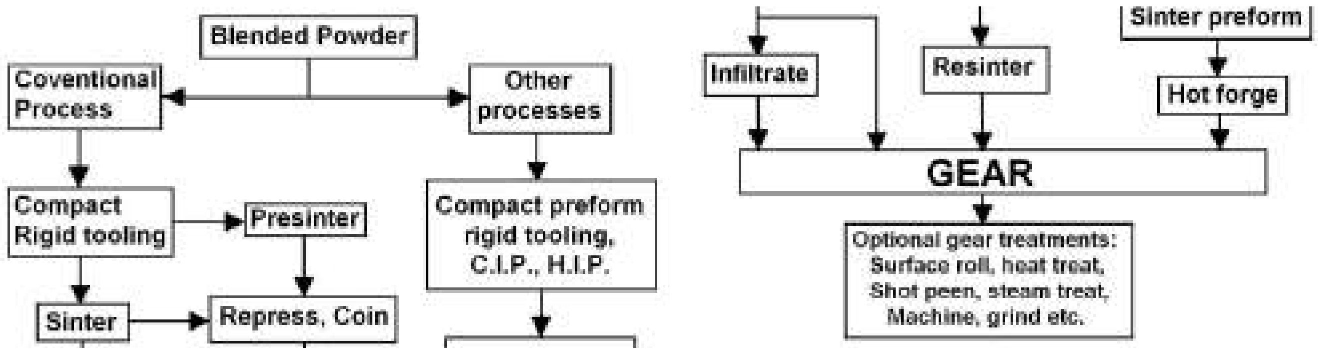


Fig 1.3 Process chart for P/M gear manufacture

Fig. 1.4. Components manufactured by sintering

Characteristics:

- Accuracy similar to die-cast gears
- Material properties can be Tailor made
- Typically suited for small sized gears
- Economical for large lot size only

As shown in Fig 1.4, for the components manufactured by P/M technique, secondary machining is not required. Fig 1.5 shows cluster gears, different types of gears that can be combined and keyways can be built-in.

Fig 1.5 Cluster gears, combination of gears and gears with key ways

1.5.3 Injection Molding:

Injection molding is used to make nonmetallic gears in various thermoplastics such as nylon and acetal. These are low precision gears in small sizes but have the advantages of low cost and the ability to be run without lubricant at light loads.

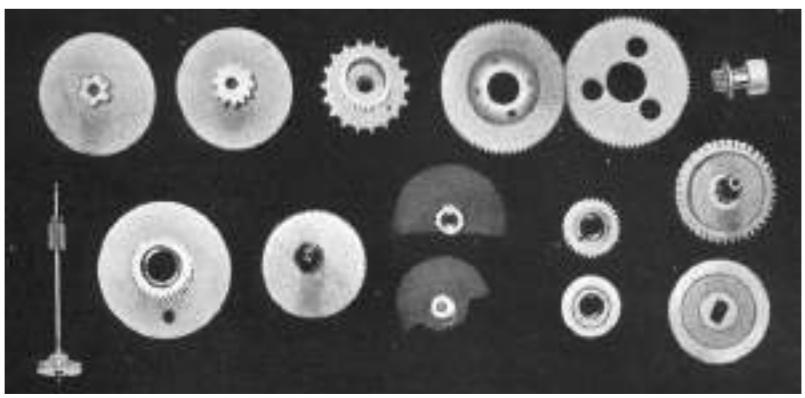


Fig. 1.6 camera gears

Applications:

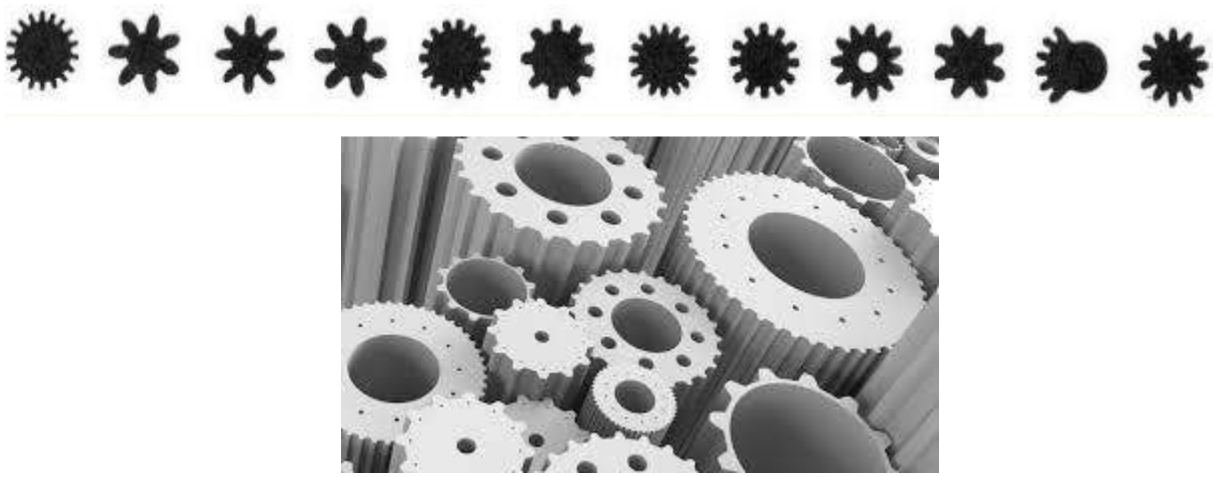
Injection molded gears are used in cameras, projectors, wind shield wipers, speedometer, lawn sprinklers, washing machine. They are shown in fig.1.6.

Materials:

The materials for injection molding components are Nylon, cellulose acetate, polystyrene, polyimide, phenolics.

1. 5 .4 Extruding

Extruding is used to form teeth on long rods, which are then cut into usable lengths and machined for bores and keyways etc. Nonferrous materials such as aluminum and copper alloys are commonly extruded rather than steels. This result in good surface finishes with clean edges and pore free dense structure with higher strength.



1.7 Extruded gears

Materials:

Aluminum, copper, naval brass, architectural bronze and phosphor bronze are the materials that are commonly extruded.

1.5.5 Cold Drawing:

Cold drawing forms teeth on steel rods by drawing them through hardened dies. The cold working increases strength and reduces ductility. The rods are then cut into usable lengths and machined for bores and keyways, etc

1.5.6 Stamping:

Sheet metal can be stamped with tooth shapes to form low precision gears at low cost in high quantities. The surface finish and accuracy of these gears are poor.

Applications:

Stamped gears are used as toy gears, hand operated machine gears for slow speed mechanism.

1.5.7 Precision stamping:

In precision stamping, the dies are made of higher precision with close tolerances wherein the stamped gears will not have burrs.

Applications:

Clock gears, watch gears etc.

1.5.8 Preforming

For close die forging the feed stock has to be very near to the net shape and this is obtained by performing.

1.6 GEAR MANUFACTURING BY MACHINING

This is most widely used gear manufacturing method. Gear blank of accurate size and shape is first prepared by cutting it from metal stock. The gear blank can also be the metal casting. This method lies under the category of chip forming process. Gear is prepared by cutting one by one tooth in the gear blank of desired shape and size along it periphery. Different gear cutting methods are used in this category. These methods are described in details.

Two principal methods of gear manufacturing include

1. Gear Cutting or forming and
2. Gear generation.

1.6.1 **Gear Forming / Cutting:**

The cutting edge of the cutting tool has a shape identical with the shape of the space between the gear teeth.

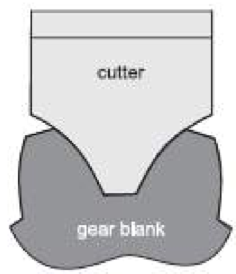


Fig 1.8 The principle of gear forming.

1.6.2 GEAR GENERATION PROCESS

In gear generation, the generating action and the forming of the involute profile is achieved by the successive tangential cuts taken by the tooth one after the other on the gear blank

In *gear generating*, the tooth flanks are obtained (generated) as an outline of the subsequent positions of the cutter, which resembles in shape the mating gear in the gear pair: This technique is based on the fact that two involute gears of the same module and pitch mesh together—the WP blank and the cutter. So this method makes it possible to use one cutting gear for machining gears of the same module with a varying number of teeth. Gear generation methods are characterized by their higher accuracy and machining productivity than gear forming.

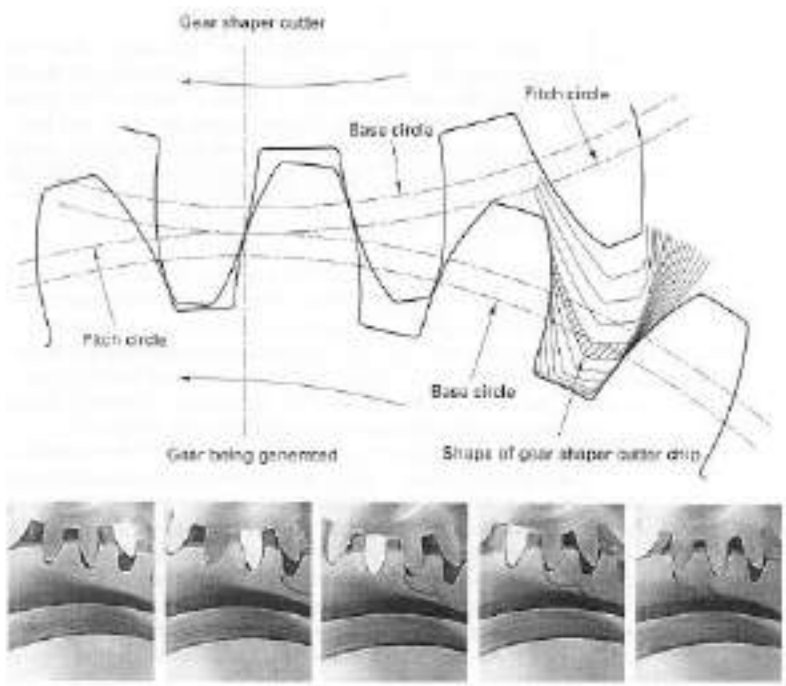


Fig. 1. 9 Generating action of a gear-shaper cutter; (*Bottom*) series of photographs showing various stages in generating one tooth in a gear by means of a gear shaper, action taking place from right to left, corresponding to a diagram above. One tooth of the cutter was painted white.

1.6.3 Difference between gear forming and gear Generation

S.No.	Gear Forming / Cutting	Gear generation
1.	Form tool of exact shape is used	The tangential cutting edge finishes the profile
2.	Cutting is done using conventional machines such as milling, shaping, broaching etc	Cutting is done on special machine which have a generating motion by simulation of gear pairs
3.	Intermittent indexing is used	Continuous indexing
5.	Only one gear with specified number of tooth can be cut with a given cutter	The gears can be cut irrespective of the number of tooth
6.	Low accuracy	High accuracy
7.	Machining time is more	Machining time is less
8.	Meant for odd / Low volume production	Meant for Batch / mass production
9.	Initial setting time is less	Initial setting time is more due to setting of various change gears such as speed, feed and indexing change gears
10.	Low productivity	High Productivity

1.7 INDEXING

Indexing is the operation of dividing the periphery of a workpiece into any number of equal parts. For example if we want to make a hexagonal bolt. Head of the bolt is given hexagonal shape. We do indexing to divide circular workpiece into six equal parts and then all the six parts are milled to an identical flat surface. If we want to cut “n” number of teeth in a gear blank, the circumference of gear blank is divided into “n” number of equal parts and teeth are made by milling operation one by one. The main component used in indexing operation is universal dividing head.

1.7.1 UNIVERSAL DIVIDING HEAD

It is most popular and common type of indexing arrangement. As indicated by its name “universal”, it can be used to do all types of indexing on a milling machine. Universal dividing head can set the workpiece in vertical, horizontal, or in inclined position relative to the worktable in addition to working principle is explained below with the help of illustration in Figure 1.11. The worm gear has 40 teeth and the worm has simple thread. Crank is directly attached with the worm. If we revolve crank by 40 revolutions the spindle attached with worm gear will revolve by only one revolution and one complete turn of the crank will revolve the spindle only by 1/40th revolution (turn). In order to turn the crank precisely a fraction of a revolution, an indexing plate is used. **An indexing plate (Fig. 1.10)is like a circular disc having concentric rings of different number of equally spaced holes**. Normally indexing plate is kept stationary by a lock pin. A spring loaded pin is fixed to the crank which can be fixed into any hole of indexing plate. The turning movement of the workpiece is stably controlled by the movement of crank as explained below.

Fig. 1.10 Index Plate

If the pin is moved by one hole on the indexing plate in the circle of 20 holes, the spindle will revolve by $\frac{1}{40} X \frac{1}{20} = \frac{1}{800}$ th turn of one revolution.

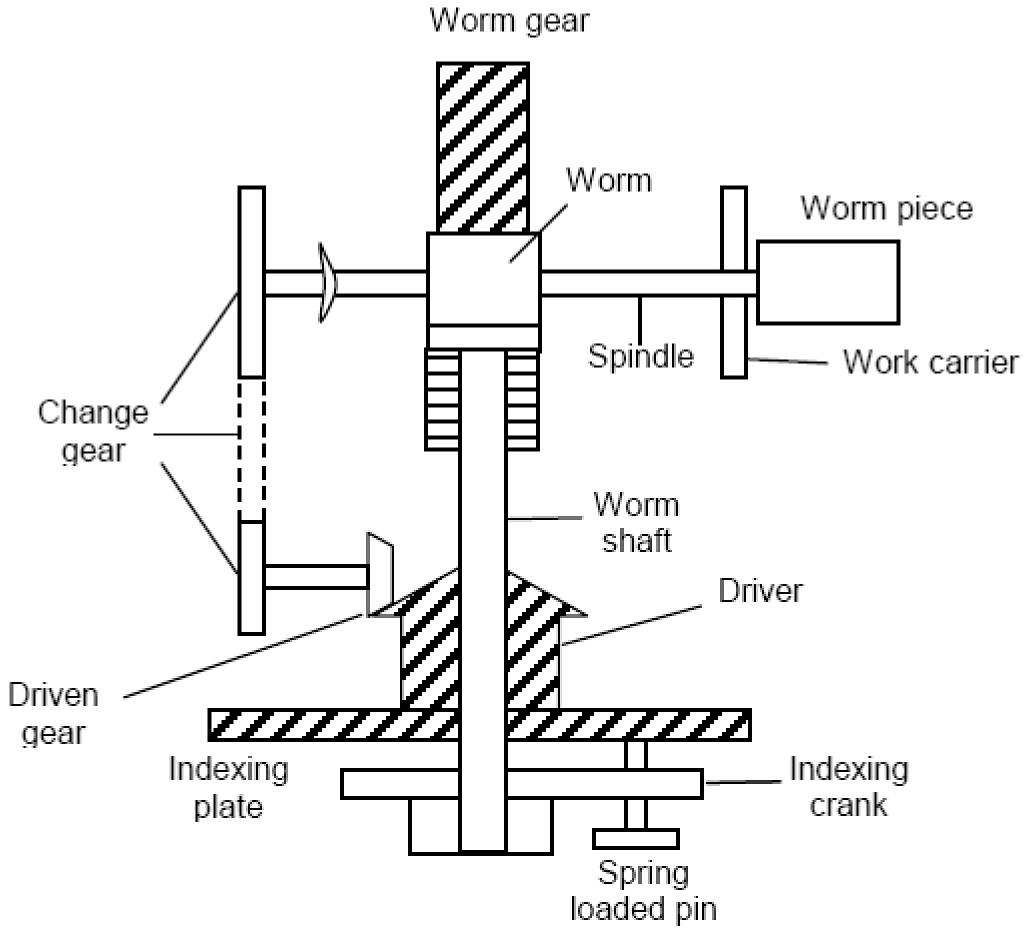


Fig. 1.11. Working Principle of Indexing Mechanism

1.8 INDEXING PLATE

Indexing plate is a graduated circular plate or one with circular rows of holes differently spaced that is used in machines. **The following types of index-plates having the holes given against them are available.**

Brown and Sharp:

Plate 1 : 15, 16, 17, 18, 19 and 20.

Plate 2 : 21, 23, 27, 29, 31 and 33.

Plate 3 : 37, 39, 41, 43, 47 and 49

Parkinson

Plate 1 : 24, 25, 28, 30, 34, 37, 38, 39, 41, 42 and 43

Plate 2 : 46, 47, 49, 51, 53, 54, 57, 58, 59, 62 and 66

German Type Index plate

Plate 1	One side	13,16,18,20,23
	Other side	15,17,19,21,24

Plate 2	One side	27,28,31,37,41,47
	Other side	29,33,39,43,44,49
Plate 3	One side	18,19,20,23,29,33,39,43,49
	Other side	15,17,19,21,27,31,37,41,47

1.8 INDEXING METHOD

There are different indexing methods in popularity. These are :

(a) Direct indexing

(b) Simple indexing

(c) Compound indexing

(d) Differential indexing

1.8.1 Direct Indexing

It is also named as rapid indexing. For this **direct indexing plate is used which has 24 equally spaced holes in a circle.** It is possible to divide the surface of workpiece into any number of equal divisions out of 2, 3, 4, 56, 8, 12, 24 parts. These all numbers are the factors of 24.

In this case, **first of all worm and worm wheel is disengaged**. We should find number of holes by which spring loaded pin is to be moved. If we want to divide the surface into 6 parts then number of holes by which pin is to be moved = $\frac{24}{N}$ (N = 6)

So number of hole space to be moved by the index crank pin after completing one pair of milling is $\frac{24}{6}$ = 4 hole space. Hence indexing has to be done for (N-1) number of times i.e 5 number of times for making a hexagonal bolt.

1.8.2 Simple Indexing

It is also named as plain indexing. It over comes the major limitation of direct indexing that is possibility of dividing circumference of workpiece into some fixed number of divisions. In **this case worm and worm gear is first engaged**. So one complete turn of indexing crank revolves the workpiece by $\frac{1}{40}$th revolution.

Indexing Procedure

(a) Divide 40 by the number of divisions to be done on the circumference of workpiece. This gives movement of indexing crank.

Indexing crank movement = $\frac{40}{N}$

N is the number of divisions to be made on the circumference of workpiece.

(b) If the above number is a whole number, then crank is rotated by that much number of revolutions after each milling operations, till the completion of the work.

For example, if we want to divide the circumference into 10 number of parts.

Indexing crank movement = $\frac{40}{10} = 4$.

That is the indexing crank is given 4 revolutions after each of milling operation for 9 more milling operations.

(c) If indexing crank movement calculated by $\frac{40}{N}$ is not whole number, it is simplified and then expressed as a whole number and a fraction.

(d) The fractional part of the above number is further processed by multiplying its denominator and numerator by a suitable common number so that the denominator will turn to a number equal to any number of holes available on the any of indexing plates.

(e) That particular holes circle is selected for the movement of crank pin.

(f) The numerator of the process fraction stands for the number of holes to be moved by the indexing crank in the selected hole circle in addition to complete turns of indexing crank equal to whole number part of $\frac{40}{N}$

Example

Let us do the indexing to cut 30 teeth on a spur gear blank that means we need to divide the circumference of gear blank into 30 identical parts. Crank movement is calculated s given below.

Index Crank movement = $\frac{40}{30} = 1\frac{10}{30}$

Here, N = 30.

Since 30 hole circle is not available, simplify the index crank movement. Multiply and divide by a common number so that the denominator value equals any one hole circle

Index Crank movement = $1\frac{10}{30} = 1\frac{1}{3} = 1\frac{1 X 5}{3 x 5} = 1\frac{5}{15}$

Limitations

This method can used for indexing upto 50 for any number of divisions. After 50, this method is not capable for some numbers like 96, etc. Compound indexing overcomes the limitations.

1.8.3 Compound Indexing

The word compound indexing is an indicative of compound movements of indexing crank and then plate along with crank. In this case indexing plate is normally held stationary by a lock pin, first we rotate the indexing crank through a required number of holes in a selected hole circle, then crank is fixed through pin. It is followed by another movement by disengaging the rear lock pin, the indexing plate along with indexing crank is rotated in forward or backward direction through predetermined holes in a selected hole circle, then lock pin is reengaged.

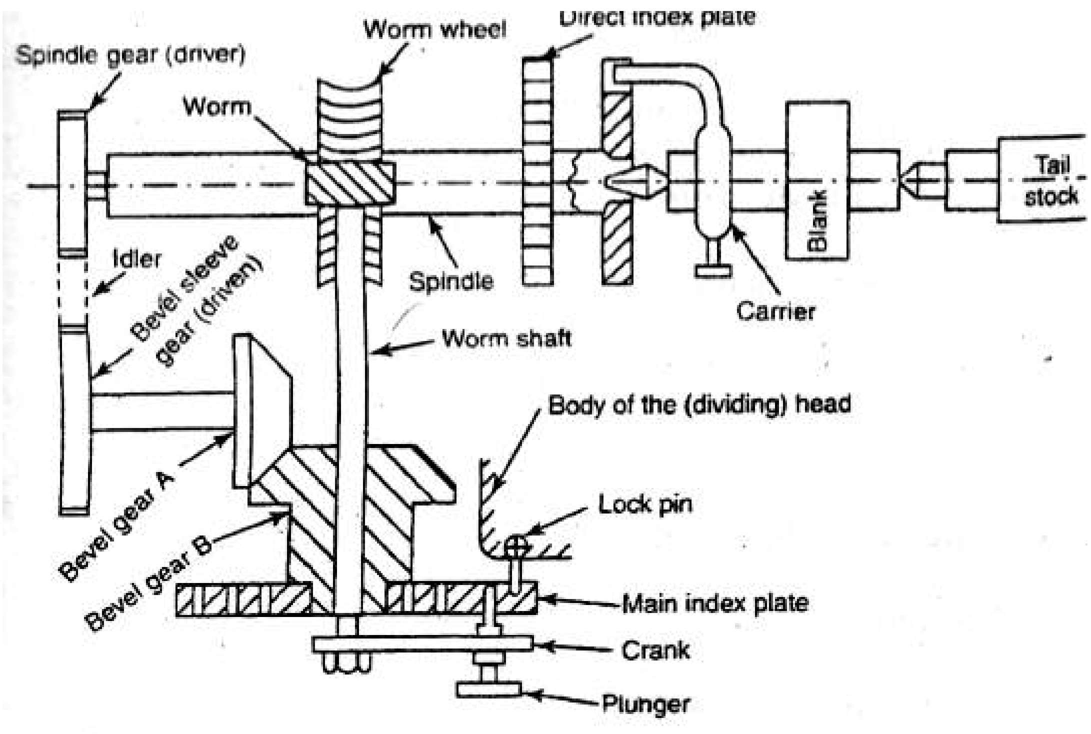


Fig. 1.12 Compound Indexing

Following steps are to be followed for compound indexing operation. The procedure is explained with the help of numerical example.

Indexing rule in compound indexing $\frac{40}{N} = \frac{n_1}{N_1} \pm \frac{n_2}{N_2}$

N = *Number of divisions required*

N_1 = *Number of hole circles used by index crank pin*

n_1 = *Hole space moved by index crank pin within N_1*

N_2 = *Number of hole circles used by lock pin*

n_2 = *Hole space moved by lock pin within N_2*

± *indicates whether we move the index crank pin and lock pin in the same direction or opposite direction.*

Example 1.1

Let us make 69 divisions of workpiece circumference by indexing method. (Using compound indexing)

Solution

Follow the steps given below:

(a) Factor the divisions to be make = 69 = 3 x 23.

(b) Select two hole circles at random (These are 27 and 33 in this case, both of the hole circles should be from same plate).

(c) Subtract smaller number of holes from larger number and factorize the difference

(33 – 27 = 6 = 2 x 3

(d) Factor the number of turns of the crank required for one revolution of the spindle (40). Also factorize the selected hole circles.

(e) Place the factors of *N* and factors of difference of the hole circle above the horizontal line and factors of 40 and both the hole circles below the horizontal line as given below. Cancel the common values on the right hand side

$$69 = 3 \, x \; 23$$
$$6 = 2 \; x \; 3$$

$$40 = 2x\,2x2x5$$
$$27 = 3x3x3$$
$$33 = 3x11$$

(f) If all the factors above the line are cancelled by those which are below the line, then the selected hole circles can be used for indexing, otherwise select another two hole circles at random and repeat the above mentioned steps. In this case there is a need to select another two hole circles, since the number 23 is not cancelled. Let us select 23 and 33 and repeat the step (e) as indicated below.

$$69 = 23 \times 3$$
$$10 = 2 \times 5$$

$$40 = 2 \times \bigcirc\!\!\!\!2 \times \bigcirc\!\!\!\!2 \times 5$$
$$22 = 23 \times 1$$
$$33 = \bigcirc\!\!\!\!\!11 \times 3$$

$69 = 3\ x\ 23$

$6 = 2\ x\ 3$

____

$40 = 2\ x\ 2\ x\ 2\ x\ 5$

$23 = 3\ x\ 3\ x\ 3$

$33 = 3\ x\ 11$

(Difference of hole circle values)

Encircled numbers below the line are the left out numbers after canceling the common factors. All the factors above the horizontal line are cancelled so selected hole circles with 22 and 33 holes can used for indexing.

(g) Following formula is used for indexing :

$$\frac{40}{N} = \frac{n_1}{N_1} \pm \frac{n_2}{N_2}$$

In this formula N_1 = 23 and N_2 = 33 ($N1$ is always given smaller value out of two).

(h) Multiply all the remaining factors below the line as 2 x 2 x 11 = 44

$$\frac{40}{69} = \frac{44}{23} - \frac{44}{33}$$

We will neglect the +ve sign.

$$\frac{40}{69} = 1\frac{21}{23} - \frac{11}{33}$$

The –ve sign indicates backward movement.

Action

For indexing of 69 divisions, the indexing crank should be moved by 21 holes circle in forward direction and then crank along with the plate are moved by 11 holes in 33 hole circle is reversed (backward) direction.

1.8.4. Differential Indexing

Available number of index plates with different hole circles, sometimes confine the range of plain indexing. In such cases, differential indexing is found to be more suitable. Between the indexing plate and spindle of dividing head, a certain set of the gears is incorporated extra. Dividing heads are provided with such standard set of gears. Differential indexing is an automatic method of compound indexing

Differential indexing consists of assuming a little different number of teeth nearer to the actual number of teeth so that simple indexing is possible. To compensate for the difference, change gears are calculated and fixed so that the hole circle plate is moved differentially, to achieve the required number of teeth.

During the differential indexing, the index-plate is unlocked and connected to a train of gears which receive their motion from the worm gear spindle. As the handle is turned, the index plate also turns, but at a different rate and perhaps in the opposite direction. Differential indexing makes it possible to rotate the work by any fraction of revolution with the usual index plates furnished with the equipment.

Differential indexing is carried out in two stages, i.e

1. Crank is moved in certain direction similar to simple indexing
2. Some movement is added to the crank or subtracted from it

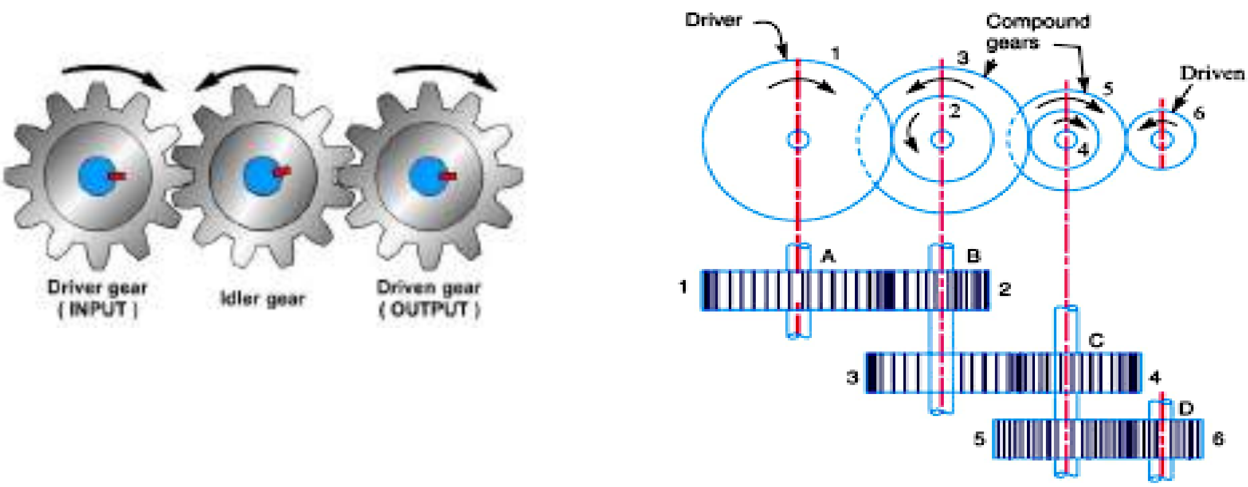


Fig. 1.13 (a). Simple gear train (b) Compound gear train

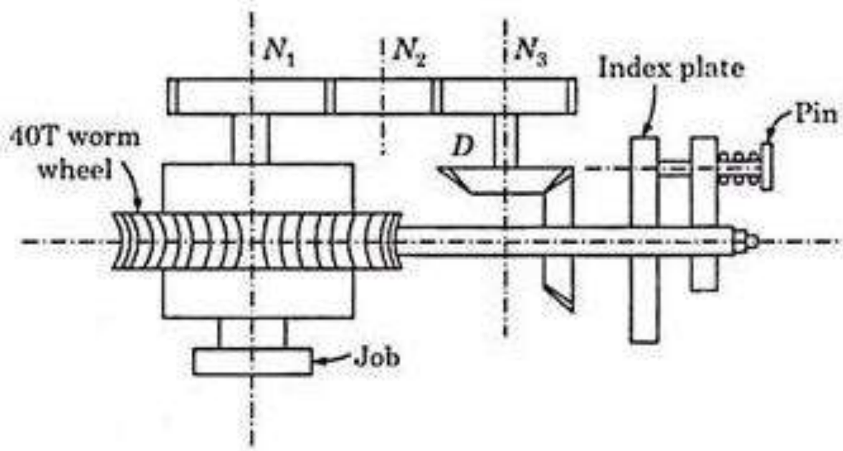


Figure 1.14 Differential Indexing

The dividing heads are having standard sets of change gears, which are as follows

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

In addition to this, some dividing heads are having following gears also :

46, 47, 52, 58, 68, 76, 84

In differential indexing, both simple and compound gear trains are used. In both the gear trains, the first driver is mounted on the main spindle of the dividing head and on the same spindle worm wheel is mounted inside and workpiece at the other end.

Also, the last driven is mounted on the worm spindle which drives the index plate.

The simple gear train consists of one driver and one driven wheel, which are connected together through one or two idle gears.

The compound gear train consists of two driver and two driven wheels and the idle gear may or may not be used.

The direction of rotation of the index plate depends on the type of gear train and number of idle gears.

The use of idle gears and consequent loss or gain of motion is described as follows:

- ✓ Using simple gear train, motion is gained by using 1 idler and motion is lost using 2 or no idler
- ✓ Using compound gear train, motion is gained by using no idler and is lost by using 1 idler.

Examples

1. Find the gear calculations and indexing movement for 139 divisions

Solution : Select a number slightly smaller or greater than the given number, such that the selected number can be easily indexed through simple indexing

Let us select a new number as 140

For 140 divisions, simple indexing = $\frac{40}{n} = \frac{40}{140} = \frac{2}{7} = \frac{2 \ x \ 3}{7 \ x \ 3} = \frac{6}{21}$

It shows 6 holes on 21 hole circle

If the index crank is turned $\frac{6}{21}$ of a revolution 139 times, it will make

$\frac{6}{21} \ x \ 139 = 39\frac{15}{21}$ revolutions

For one complete turn of the workpiece, it should make 50 complete revolutions. Hence, the workpiece would not be indexed through exactly 139 equal divisions. The movement done by the crank is short of the required 40 turns by:

$40 - 39\frac{15}{21} = \frac{6}{21}$ of a revolution

This fraction is to be gained by the plate movement, so the plate is turned in the same direction as the crank

Gearing ratio = $\frac{6}{21} = \frac{2\ x\ 3}{3\ x\ 7} = \frac{32\ \ x\ \ 24\ (Driver)}{48\ \ x\ 56\ \ (Driven)}$

i.e. first driver 32 teeth, 1st follower 48 teeth and

second driver 24 teeth, 2nd follower 56 teeth.

As, it is a compound gear train, no idler is required.

Example 2

Find indexing movement for 119 divisions

Solution : Select a number slightly smaller or greater than the given number, such that the selected number can be easily indexed through simple indexing

Let us select a new number as 120

For 120 divisions, simple indexing = $\frac{40}{n} = \frac{40}{120} = \frac{1}{3} = \frac{1\ \ x6}{3\ x\ 6} = \frac{6}{18}$

It shows 6 holes on 18hole circle

If the index crank is turned $\frac{6}{18}$ of a revolution 119 times, it will make

$\frac{1}{3}\ x119\ \ = 39\frac{2}{3}$ revolutions

For one complete turn of the workpiece, it should make 40 complete revolutions. Hence, the workpiece would not be indexed through exactly 119 equal divisions. The movement done by the crank is less than the required 40 turns by:

$40\ \ - 39\frac{2}{3} = \frac{1}{3}$ of a revolution

This fraction is to be gained by the plate movement,

Gearing ratio = $\frac{1}{3} = \frac{1\ x\ 24}{3\ x\ 24} = \frac{24\ (Driver)}{72\ \ (Driven)}$

Result

Driver 24 teeth and driven 72 teeth

Simple train and as motion is gained, 1 idler is used

Crank motion : 6 holes on 18 hole circle..

1.8.5 Angular Indexing:

Instead of rotating the job through certain division on its periphery, sometimes it may be needed to rotate the job through certain angle. Angular indexing is used for this purpose.

Since the crank and spindle ratio is 40 : 1 and hence when the crank moves through one revolution, the spindle or the job moves through 1/40 of revolution, i.e., the job will revolve through an angular movement of 9° ($\frac{360^0}{40} = 9^0$)

If it is desired to index a job by 35 degree, then the index head movement required to perform the operation will be = $\frac{35}{9} = 3\frac{8}{9} = 3\ \frac{8\ x\ 3}{9\ \ x\ 3} = \ 3\frac{24}{27}$

i.e., the crank must be turned three complete revolutions plus 24 holes in the 27-holes circle.

Note :- When angle does not include half, one third or two third of a degree, then convert it into minutes or seconds. As one turn of the crank rotate through 9^0, i.e. 9 x 60 = 540 minutes or 540 x 60 = 32400 seconds

Example – 4

Index for $3^0\ 30'$

Solution :

$30'$ means 30 minutes = $\frac{1}{2}\ hr$

Crank Movement = $\frac{AngleTurned}{9} = \frac{3\frac{1}{2}}{9} = \frac{7}{18}$

It shows 7 holes on 18 hole circle.

Example – 5

Index for $31^0\ 20'$

$20'$ means 20 minutes = $\frac{20}{60} = \frac{1}{3} hr$

Crank Movement = $\frac{AngleTurned}{9} = \frac{31\frac{1}{3}}{9} = \frac{94}{27} = 3\frac{13}{27}$

It shows 3 full turn and 13 holes on 27 hole circle.

Example – 6

Index for $14^0\ 40'$

$40'$ means 40 minutes = $\frac{40}{60} = \frac{2}{3} hr$

Crank Movement = $\frac{Angle Turned}{9} = \frac{14\frac{2}{3}}{9} = \frac{44}{27} = 1\frac{17}{27}$

It shows 1 full turn and 17 holes on 27 hole circle.

Example – 7

Index for 60^0

Crank Movement = $\frac{Angle Turned}{9} = \frac{60}{9} = 6\frac{6}{9} = 6\frac{6 \ x \ 2}{9 \ x \ 2} = 6\frac{12}{18}$

It shows 6 full turns and 12 holes on 18 hole circle.

Example – 8

Index for 34^0 12'

34^0 = 34 x 60 x 60 = 122400 seconds

12' = 12 x 60 = 720 seconds

34^0 12' = 122400 + 720 = 123120 seconds

And 9^0 = 32400 seconds

Crank Movement = $\frac{Angle Turned}{9} = \frac{123120}{32400} = 3\frac{16}{20} \ turns$

It shows 3 full turns and 16 holes on 20 hole circle.

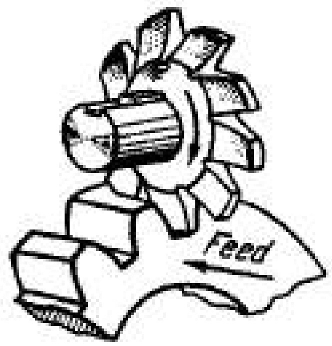
1.9 Gear cutting by milling

1.9.1 Disc type cutter

For cutting a gear on a milling m/c, the gear blank is mounted on an arbor which is supported between a dead centre & a live centre in the in dividing head. The cutter is mounted on the arbor of the machine and must be aligned exactly vertically with the centre line of the indexing head spindle. The table of the machine is moved upward until the cutter just touches the periphery of gear blank. The vertical feed dial is set to zero. The table is then moved horizontally until the cutter clears the gear blank. The table is then moved upwards in steps by an amount equal to the full depth of the gear tooth.

The vertical movement may be less if the gear is to be cut in two or more passes. After this, the longitudinal feed of the table is engaged. The gear blank moves under the rotating cutter & a tooth space is cut. After this, the movement of the table is reversed so that the cutter again clears the gear blank. The gear blank is then indexed to the next position for cutting the second tooth space. This procedure is repeated until all the teeth have been milled.

There is a flat circular disc type cutter and the plane of rotation of the cutter is radial with respect to the blank.



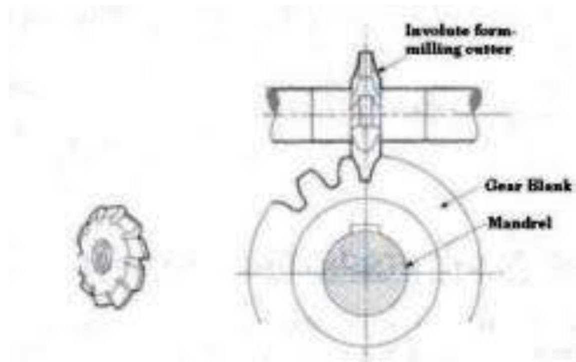


Fig. 1.15 Form Milling by Disc cutter

a. PROCEDURE FOR CUTTING SPUR GEAR IN MILLING MACHINE USING DISC CUTTER

1. The gear blank is mounted on a mandrel which is supported between the center of the dividing head and one more center at the other end, as shown in fig.
2. At a time one tooth space is cut by the milling cutter, and a dividing head is used to index the job to the next required tooth space.
3. The cutter is chosen according to the module (or DP) and number of teeth of the gear to the cut.

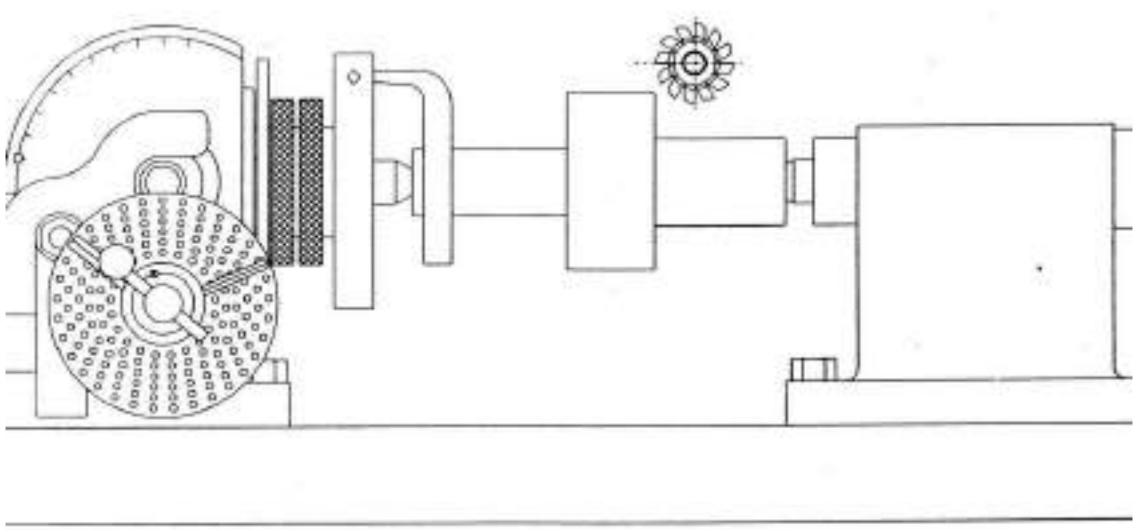


Fig. 1.16 Setup for spur gear cutting in Milling machine

4. This cutter is mounted on the milling arbor.
5. Before the gear can be cut, it is necessary to have the cutter centred accurately relative to the gear holding mandrel.

6. Position the cutter with respect to the cutter of the work piece. This is done by moving the vertical feed lead screw and the cross feed lead screw handle.
7. Do the skins cut by touching the top surface of the work with the plain milling cutter. This is done by placing the wet paper on the top surface of the gear blank and the work piece is lifted against the rotating cutter.

   The movement when the cutter touches the paper, it makes the paper to slide away. This indicates that cutter is having surface contact with the gear blank. Now adjust the micrometer dial of the vertical feed lead screw to zero.
8. Lower the table and by rotating the longitudinal slide hand wheel, the work piece is relived from the cutter.
9. Set the depth of cut by raising the table and the tooth spaced is formed by moving the work piece against the rotating cutter.(Note: - The depth of cut is given in several steps)
10. Remaining tooth is formed by indexing the work piece (N-1) number of items.
11. The above two steps are repeated till the required depth is given in the vertical feed slide.

B. HELICAL GEAR MILLING PROCEDURE IN MILLING MACHINE USING DISC TYPE OF CUTTER

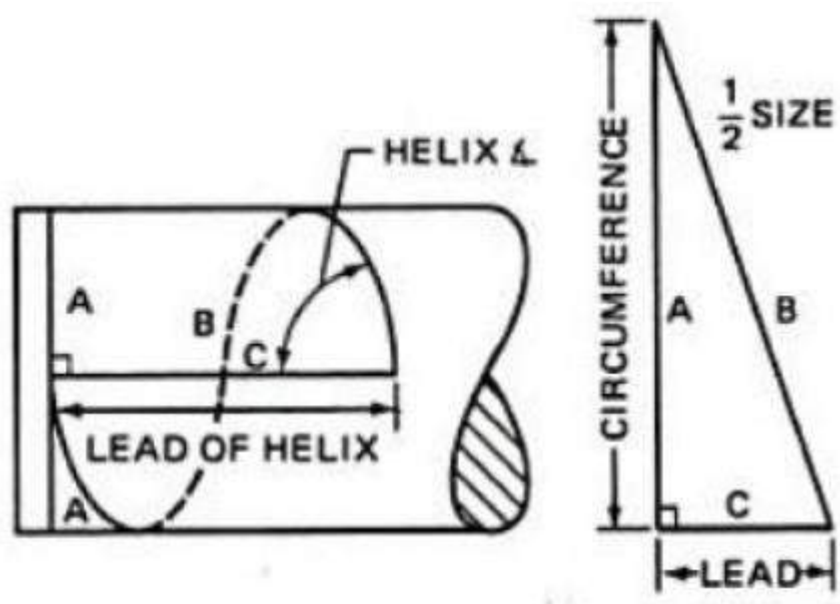


Fig. 1.17 Determining the relation between Circumference and Lead

Setting the table

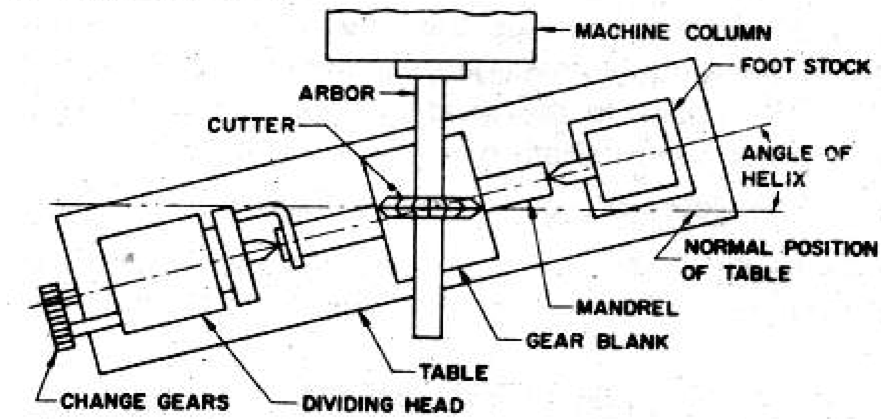
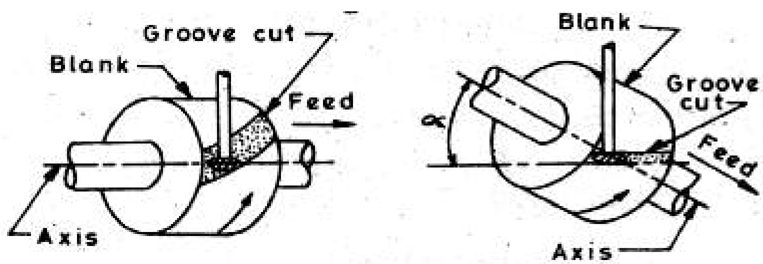
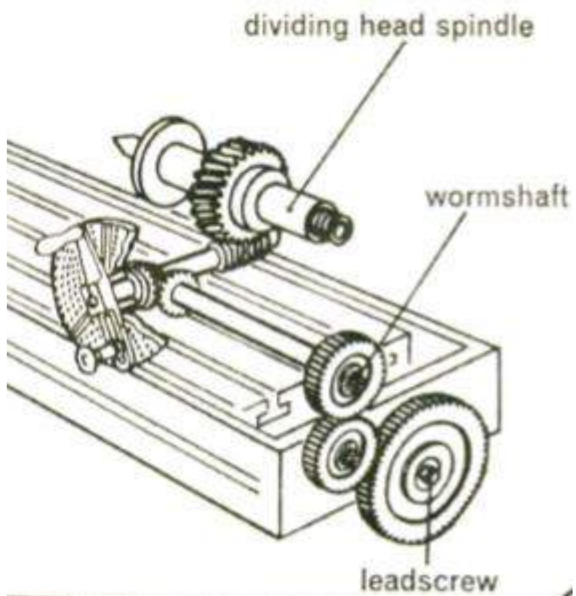


Fig. 1.18 Table setup for helical gear Milling





- A helix is defined as a curve generated by the path of a moving point which moves along the curved surface of a cylinder such that, in each revolution its movement is uniformly parallel to and around the axis of the cylinder
- The operation of milling a helix is generally known as spiral milling.
- A lead of a helix or spiral is defined as the distance travelled (measured) parallel to the axis of rotation in one revolution by a point moving along the curve.
- In fig. 1.17, this distance is represented by Length L.

- If D is the diameter of the workpiece , then the distance travelled by the moving point along the curve in one revolution is equal to the circumference of the workpiece.
- If α is the helix angle, then
    - Tan α = $\frac{\pi D}{L}$
- During the milling operation, there are two simultaneous movements of the workpiece i.e. rotation and at the same time feed movement of the table.
- These movements are accomplished by connecting the worm shaft of the divining head to the feed rod / lead screw of the table, through a set of change gears.
- Lead of the machine is same as the lead of the helix it would cut, if the worm shaft of the dividing head and table feed screw were connected by a gear ratio of 1::
    - Lead of the machine = 40 x P in mm
        - Where, P = Pitch of the lead screw in mm

Setup for milling a Helix

1. Fig. 1.18 shows the general setup for milling a helix.
2. The table has been swung to a new position which is at an angle α to its normal position
3. Now the table and the workpiece will be fed at an inclination α to their normal position
4. While setting a workpiece for milling a helix, following points should be considered:
5. The table has been swung correctly through the required angle and in the correct direction, to enable cutting of right hand or left hand helix.
6. On the table feed lead screw, the dividing head is placed directly
7. Calculated gears are placed correctly on the respective spindles and they have correct number on them
8. Before starting the operation, the lockpin has been withdrawn from the index plate, so that the plate is free to revolve as the table moves.

Change gears calculation:

Gearing ratio = $\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of the Machine in mm}}{\text{Lead of the helix to be cut in mm}} = \frac{40 \ x \ P}{\text{Lead of the helix to be cut in mm}}$

The above ratio will give either simple or compound gear train of gears.

1.9.2 End mill cutter:-

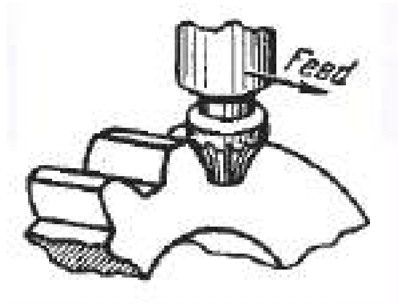


Fig. 1.16 Form milling by end mill cutter

In this method the cutter rotates about an axis which is set radially with respect to the blank & at the same time the cutter is traversed parallel to the axes of the blank.

The cutting edge lie on a surface of revolution, so that any axial cross- section of the cutter corresponds to the shape required for the space b/w two adjacent teeth on the finished wheel. The milling m/c used in this method is vertical milling m/c.

The End mill cutter is mounted straight on the milling m/c spindle through a chuck.

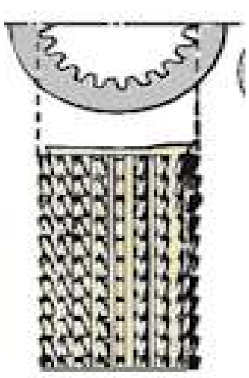
1) The disc type of cutter is used to cut big spur gear of

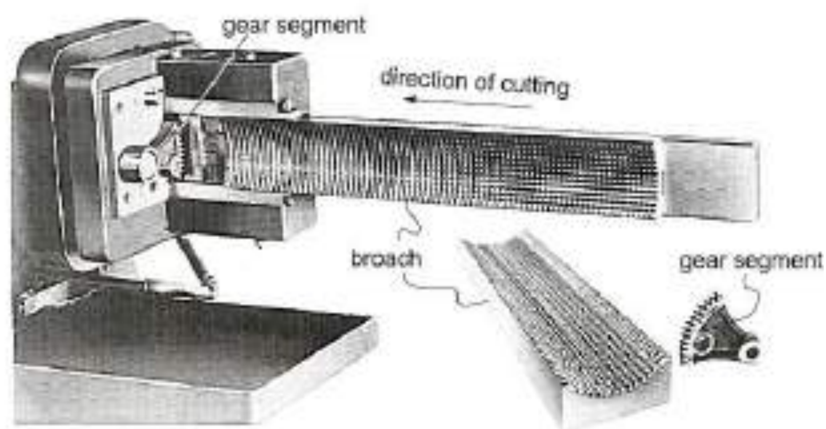
2) This method is very slow since only one tooth is cut at a time. To overcome these drawbacks, "multiple tools shaping cutter head" is used to cut all the tooth spaces of the gear at the same time (Shear speed Process).

Advantage

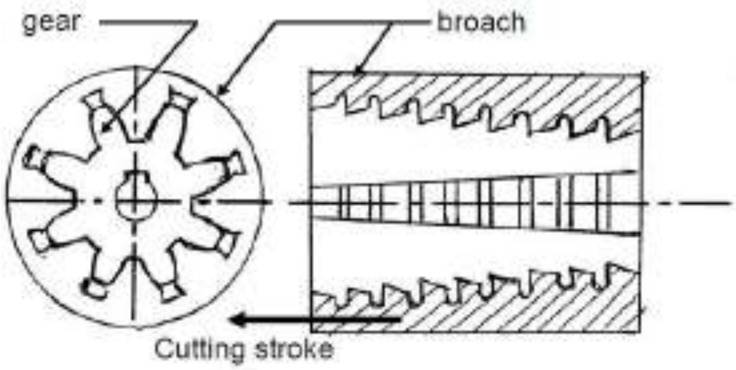
1) Gear milling is a simple, Economical & flexible method of gear making.

2) Spur, helical, bevel gears and racks can be produced by this method.





1.9.3 Broaching



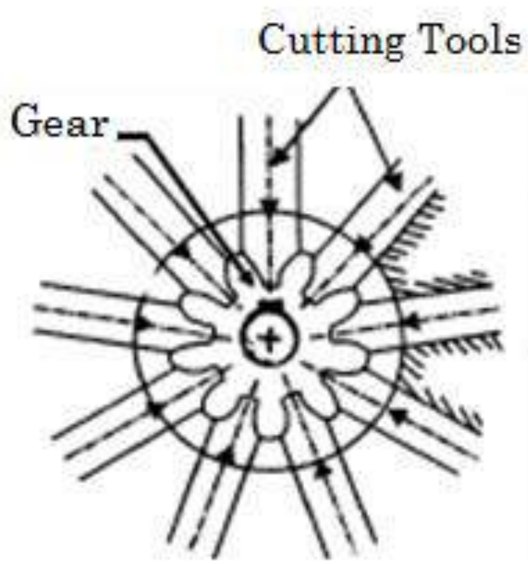
Broaching can also be used to produce gear teeth and is particularly applicable to one pass.

Fig. 1.17 Broaching the teeth of a gear segment by horizontal external broaching in internal teeth. The process is rapid and produces fine surface finish with high dimensional accuracy. However, because broaches are expensive-and a separate broach is required for each size of gear-this method is suitable mainly for high-quantity production (Fig. 1.17)

1.9.4 FAST PRODUCTION OF TEETH OF SPUR GEARS

Parallel multiple teeth shaping

In principle, it is similar to ordinary shaping but all the tooth gaps are made simultaneously, without requiring indexing, by a set of radially infeeding single point form tools as indicated in Fig. 7.2.12(a). This old process was highly productive but became almost obsolete for very high initial and running costs.



1.10 GEAR GENERATION PROCESS

The various gear generation process are listed below:

1. Gear hobbing
    - Axial hobbing
    - Radial hobbing
    - Tangential hobbing
2. Gear shapers
    - Rack – type cutter generating process
    - Pinion type cutter generating process
3. Gear planning process
    - The Sunderland process
    - The Maag process
4. Bevel gear generating
    - Straight Bevel – gear generator
    - Spiral bevel –gear Generator

1.11. GEAR HOBBING:-

Gear hobbing is an important and widely used gear cutting method, where a hobbing machine and a hob cutters are used. The involute profile is cut by a generation processes. The generating action and the relative motion between the cutter and work is similar to that of the worm and worm wheel pair respectively. It is a continuous indexing process. As the hob is fed across the work, all the teeth in the blank are completely finished.

Applications

1. Spur, helical, double helical gears and worm wheel are cut very easily.
2. We can cut worms with tangential hobbing attachement
3. Other products like chain sprockets, spline shafts, ratchets can also be cut.

Advantages:

1. With a single hobbing cutter of given module and pressure angle, gears with any number of teeth or helix angle, left hand or right hand can be cut. This is not possible in other methods like milling and shaping
2. The continuous indexing makes the hobbing process more productive and accurate
3. Gears from 20 mm to 600 mm diameter, modules ranging from 0.5 to 6 mm can be normally cut.

Limitations

Internal gears, cluster gears, bevel gears and racks cannot be cut.

1.11.1 Hobbing Machine Description

The hobbing machine (Fig. 1.18) consists of a rigid base, and a column is mounted on one side of it. The vertical sideway on the column carries the cutter head. The cutter is mounted in the front over a swivel head so that it can be tilted to the required angle during helical gear cutting. The vertical slide moves giving feed to the cutter and cutting the gear. A horizontal slide mounted in front carries a circular table. The workpiece can be mounted directly on the circular table or suitable mandrels are used to centre and carry the blanks. A rear column mounted on the circular table provides additional support to the mandrel through a tail stock. The main column and rear column are connected by a overarm, which provides additional rigidity to the machine.

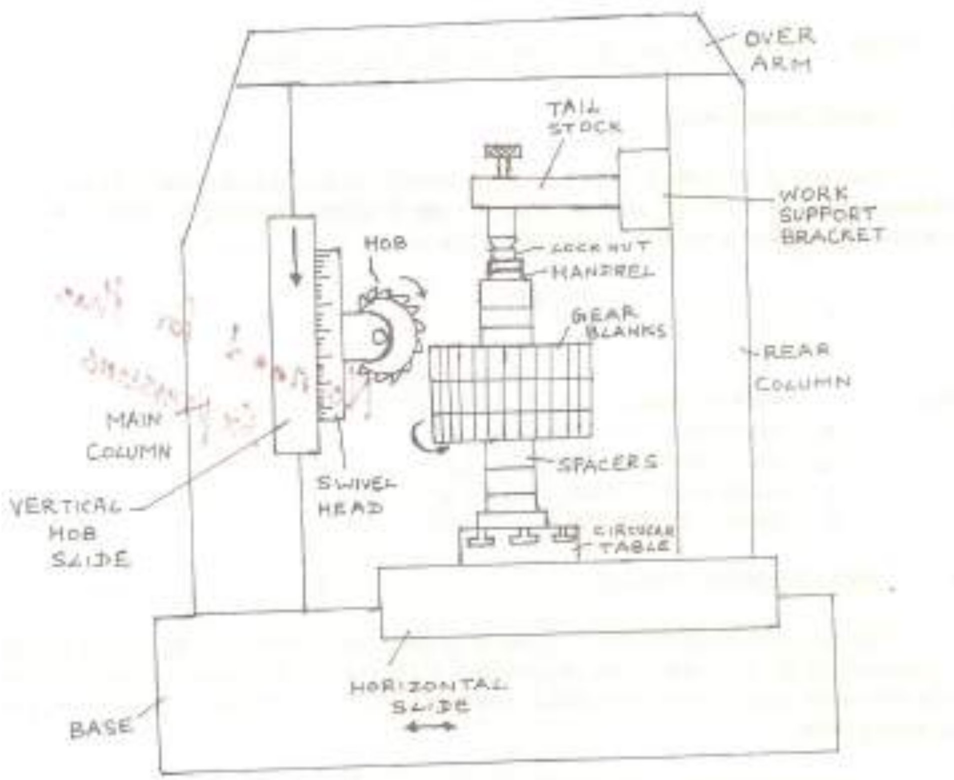


Fig. 1.18. Gear Hobbing Machine

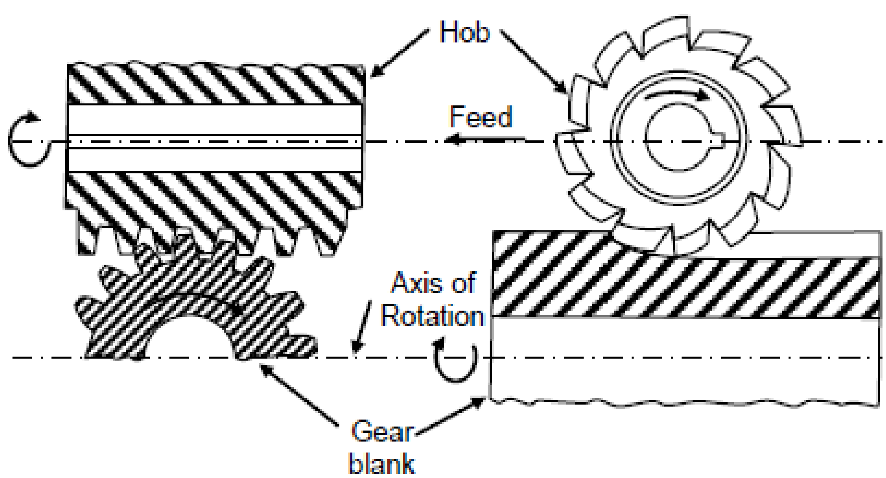


Fig. 1.19 Process of Gear Hobbing

The machine is provided with drives to rotate the cutter and the table in a set ratio. Feed rate of the cutter can also be changed. The cutter speed can be set to get the required cutting velocity. While cutting helical gears, the drive provides a differential motion to the gear blank through a differential gear drive. Depth of cut is setting can be done accurately with horizontal table, with a micrometer distance setting device.

Three important parameters are to be controlled in the process of gear hobbing indexing movement, feed rate and angle between the axis of gear blank and gear hobbing tool (gear hob). A schematic diagram of the setup of a gear hobbing machine is illustrated in Figure 1.20. The axis of hob is set at an inclination equal to the helix angle of the hob with the vertical axis of the blank. If a helical gear is to be cut, the hob axis is set at an inclination equal to the sum of the helix angle of the hob and the helix angle of the helical gear. Proper gear arrangement is used to maintain rpm ratio of gear blank and hob.

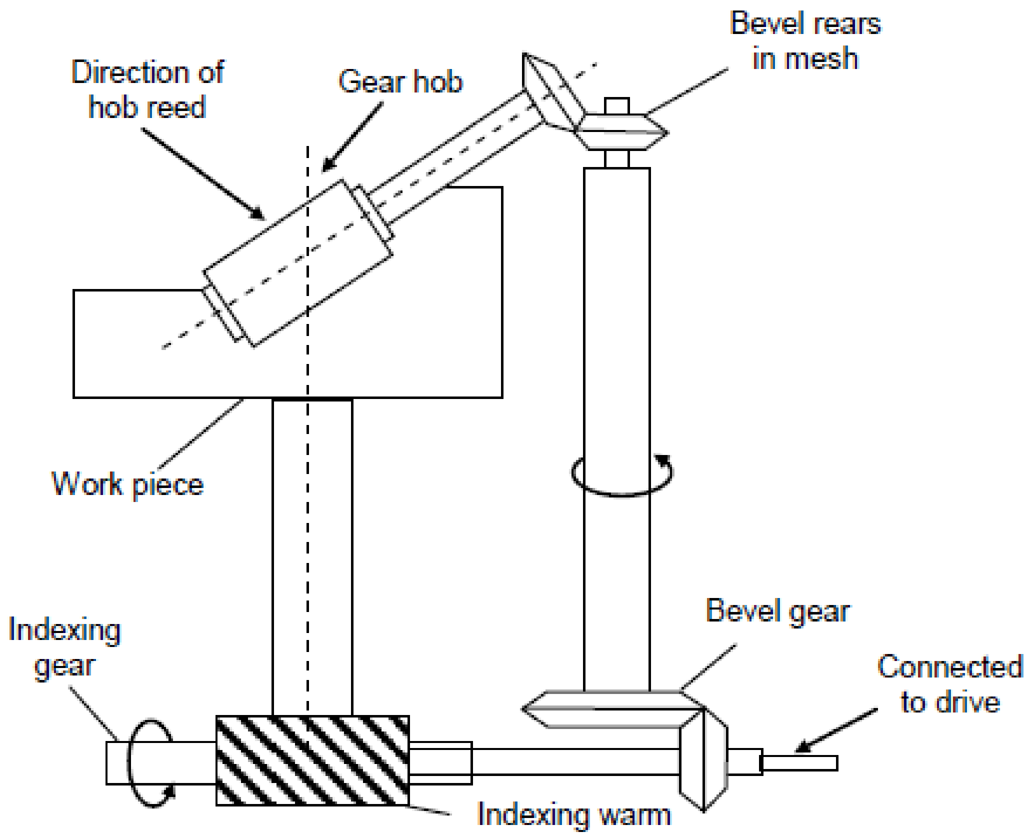


Fig. 1.20 Setup for Gear Hobbing Machine

The operation of gear hobbing involves feeding the revolving hob till it reaches to the required depth of the gear tooth. Simultaneously it is fed in a direction parallel to the axis of rotation. The process of gear hobbing is classified into different types according to the directions of feeding the hob for gear cutting. The classification is described as given below.

Hobbing with Axial Feed

In this process the gear hob is fed against the gear blank along the face of the blank and parallel to its axis. This is used to make spur and helical gears.

Hobbing with Radial Feed

In this method the hob and gear blanks are set with their axis normal to each other. The rotating hob is fed against the gear blank in radial direction or perpendicular to the axis of gear blank. This method is used to make the worm wheels.

Hobbing with Tangential Feed

This is also used for cutting teeth on worm wheel. In this case, the hob is held with its axis horizontal but at right angle to the axis of the blank. The hob is set at full depth of the tooth and then fed forward axially. The hob is fed tangential to the face of gear blank.

1.11.2. Gear Setting and Change Gear Calculations

Various gear settings are to be done in the hobbing machine to set the ratios and cutting parameters. The important settings to be done are

a. Cutting speed change gears (i_v)
b. Index change gears (i_x)
c. Feed change gears (i_s)
d. Differential change gears (i_y)

a. Speed Change Gears

The speed of the hobbing cutter is to be set according to the required cutting velocity. This is determined by the work material, hob material and the type of cutting i.e roughing or finishing. The ratio between the motor and the hob rpm is calculated as follows

$$i_v = C_v \frac{n_H}{n_M} \qquad n_H = \frac{1000\,V}{D_H}$$

C_V = machine Constant
n_H = rpm of hob
n_M = rpm of motor
v = cutting speed in m/min
D_H = Diameter of hob

b. Index change gear ratio (i_x)

The gear tooth generation process is similar to that of worm and worm and wheel cutting action. For one revolution of a single start hobbing cutter one tooth space will be cut. Thus the relation between the rotation of the hob and the workpiece will be in the ratio of number of teeth on the workpiece to number of starts on the hobbing cutter.

1 revolution of hob is 1 x $\frac{k}{z}$ revolutions of gear blank

$$i_x = c_x \; x \; \frac{k}{z}$$

where c_x = Machine cosntant

k = number of starts on the hob

z = number of teeth on workpiece

c. **Feed Change gear ratio (i_s)**

The feed in the hobbing machine is given generally in the axial direction. The feed is defined as the distance travelled in mm by the hobbing cutter in the axial direction per revolution of workpiece. The feed rate varies from 0.5 mm to 5 mm per revolution of workpiece depending upon the material being cut, roughing or finishing operation and the surface finish required.

1 revolution of gear blank is S_v movement of hob slide

$$i_s = C_s \; x \; S_v$$

C_S = machine costant

S_V = feed rate in mm

d. **Differential Change gears (i_y)**

Differential change gears are essential when cutting helical gears, to give a differential motion to the gear blank with respect to the cutter rotation. This is essential so that the successive cut take place in helical path. Differential gear train can also be used for cutting same spur gears with prime number of teeth.

$$i_y = \frac{C_y \; x \sin\beta}{m_n \; x k}$$

C_y = machine constant

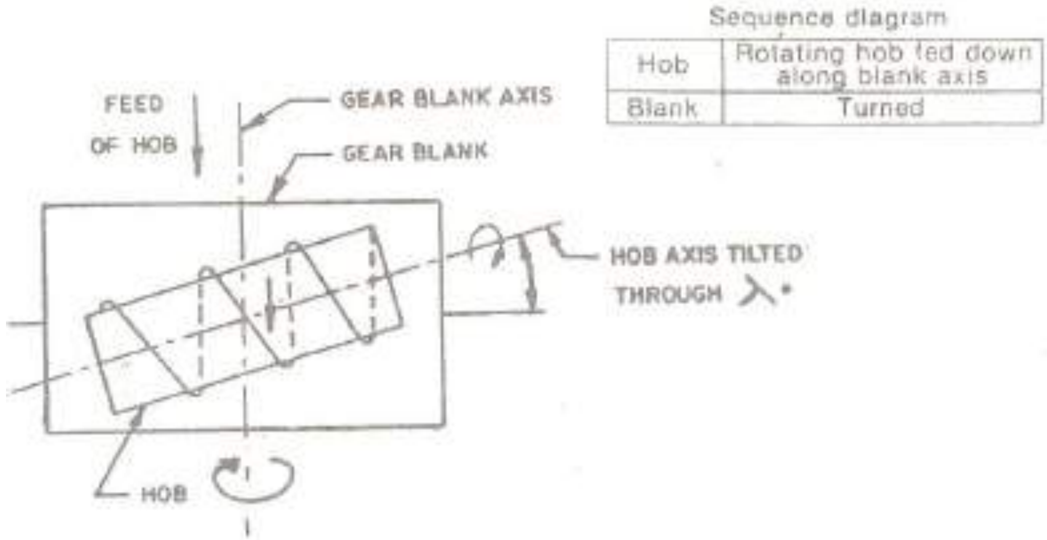
β = helix angle of gear tooth

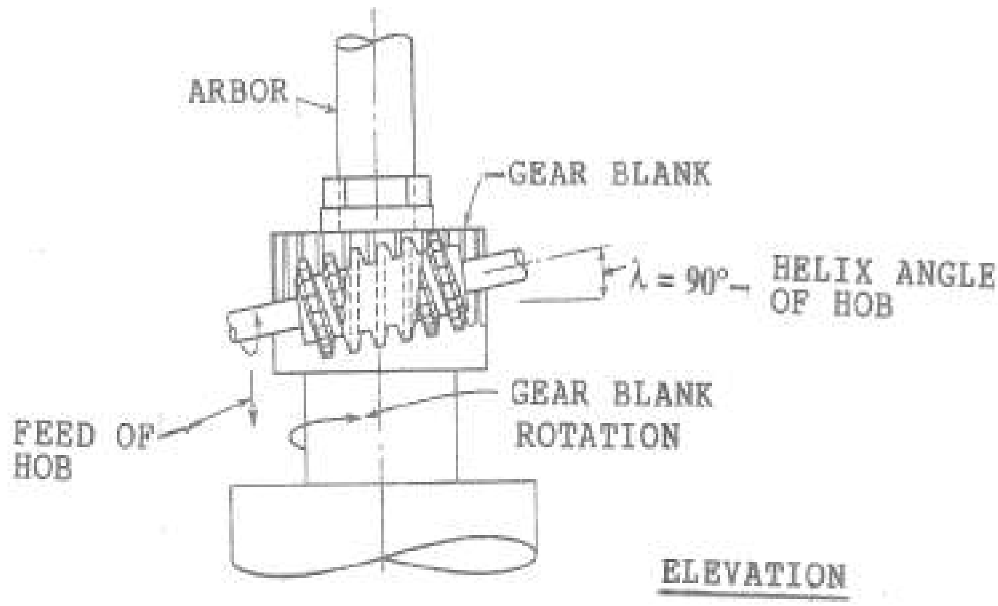
m_n = Normal Module

k = number of starts on hob

1.11.3 Spur gear Generation using Gear hobbing Process

1. Select the hobbing cutter as per the module, pressure angle and the profile required
2. Mount the cutter on the arbor and tilt the swivel head to the lead angle of the hobbing cutter
3. Mount the gear blanks, on the rigid and precision mandrel and set the tail stock support.
4. Set the speed change gears depending upon the cutting velocity required, set the index change gears depending upon the number of tooth required, set the feed change gears depending upon the material being cut. (Note – No need for setting the differential change gears for spur gear cutting)





5. Set the required depth of cut.
6. Start the machine and the coolant.
7. Use automatic machining cycle if available.
8. Measure the base tangent length when finished

1.11.3 Helical gear Generation using Gear hobbing Process

- To cut helical gears, the hob is set up so that the thread of the hob facing the gear blank is directed at the helix angle of the teeth.
- This is done by setting the hob at an angle $\gamma = \beta g \pm \alpha h$, *where* βg is the helix angle of the helical gear being cut and *αh is the helix angle of the hob.*
- *If the hand of* helical gear and that of the hob are different, the positive sign is considered; if the hand is the same, the negative sign should be used.

- Also, the hob attains a continuous feed motion along the axis of the gear blank .
- In cutting helical gears, an incremental motion is imparted to the blank, with an angular velocity that would provide one full additional revolution of the blank during vertical feed of the hob through a distance equal to the lead of the helical teeth on the gear.

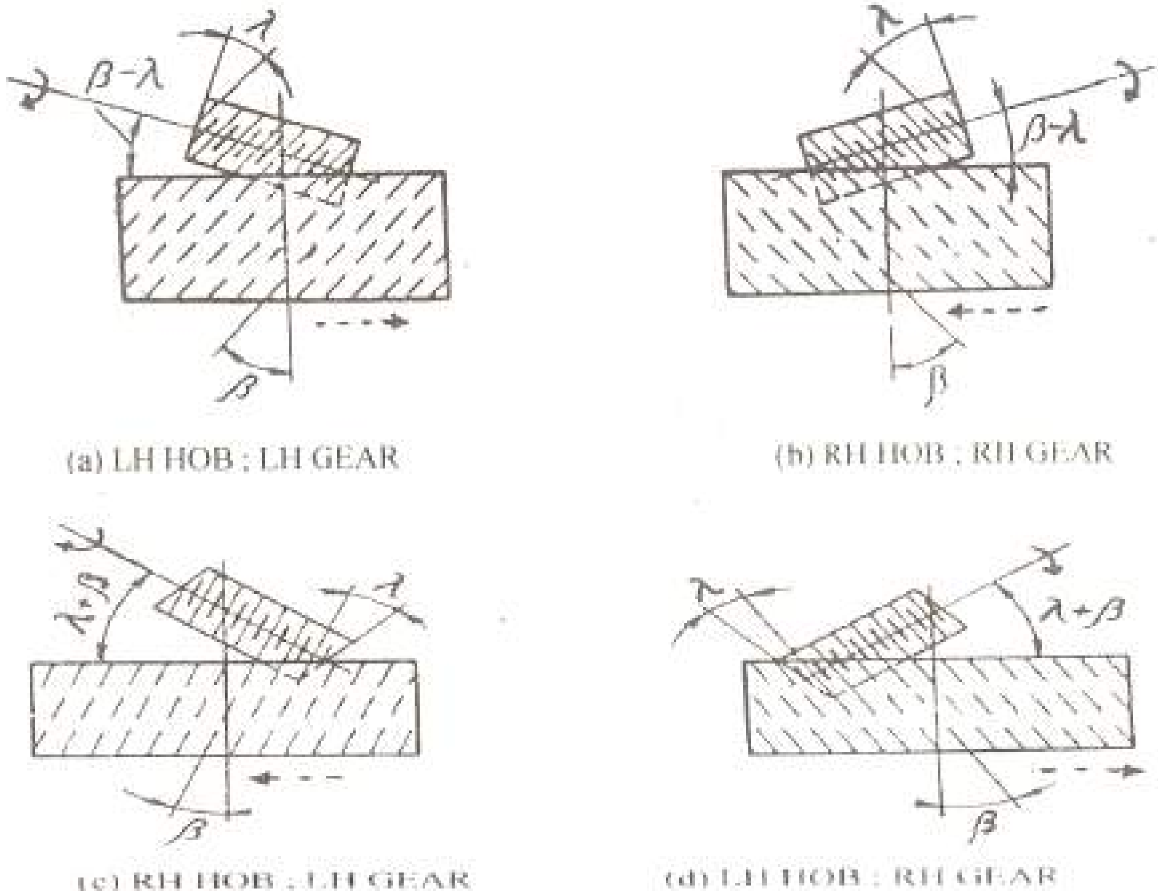


Fig.1.21 Hob setting for Helical gear generation

1.12. GEAR SHAPING

Gear shaping is an important gear cutting method where small, medium and large size gears are cut. A reciprocating cutting tool is used to generate the gear tooth one after the other. It is a generating process and the machine kinematic motion is arranged in such a way that the involute profiles are generated.

Gear shaping process is generally adopted for spur, helical and double helical gears. Internal gears are cut by this method. Mostly the process is adopted for steel gears. Gear shaping is most suitable for mass production of small gears or batcg production of large gears.

The process can be classified as follows

1.	Gear shaping with disc type cutters	Used for spur, helical and internal gear of 15

		to 300 mm
2.	Gear shaping by rack type cutters (MAAG Process)	Used for spur, helical and double helical gears of 150 to 1500 mm
3.	Gear planning by rack type cutter	Large spur, helical and double helical gears upto 5000 mm

1.12.1 Description of a Gear Shaping Machine

The cutter of the disc type gear shaping machine is mounted on the spindle, which reciprocates up and down for cutting motion. The reciprocating motion is by the action of the circular rack on the spindle and the sector gears. As the eccentric rotates, the coupler oscillates the sector gears and the spindle reciprocates for cutting. The number of strokes are controlled by the speed setting belts and the rpm of the eccentric. The stroke length can be controlled by the eccentricity setting in the eccentric. The zone of cutting of the cutter can also be altered by varying the length of the coupler. It is a split one and can be extended or shortened as required.

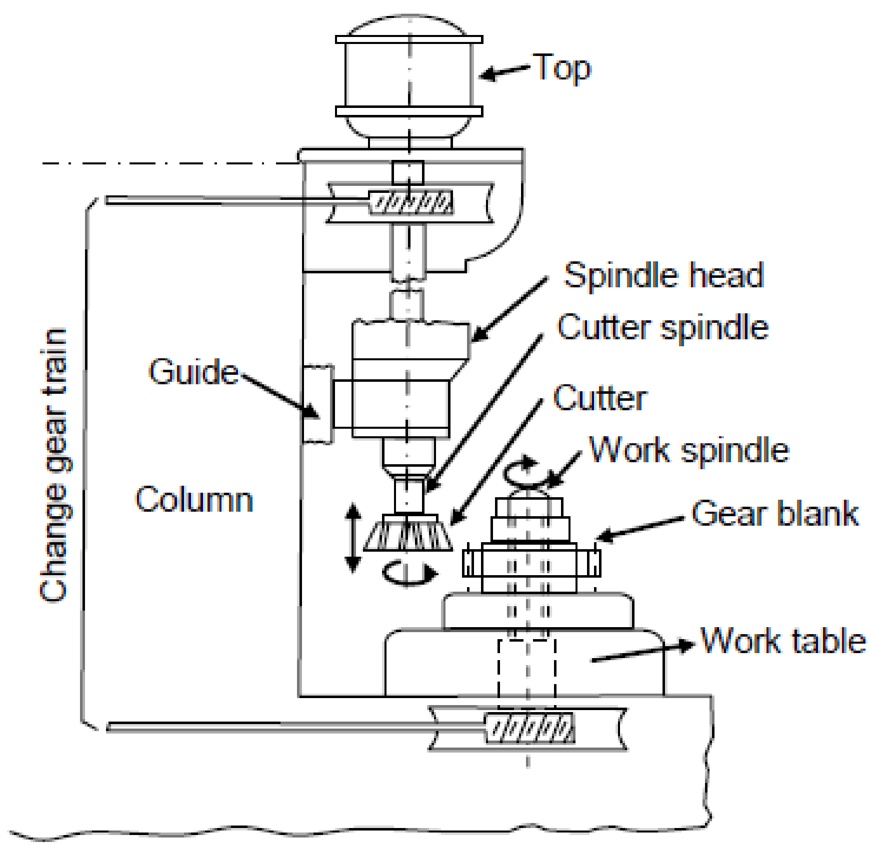
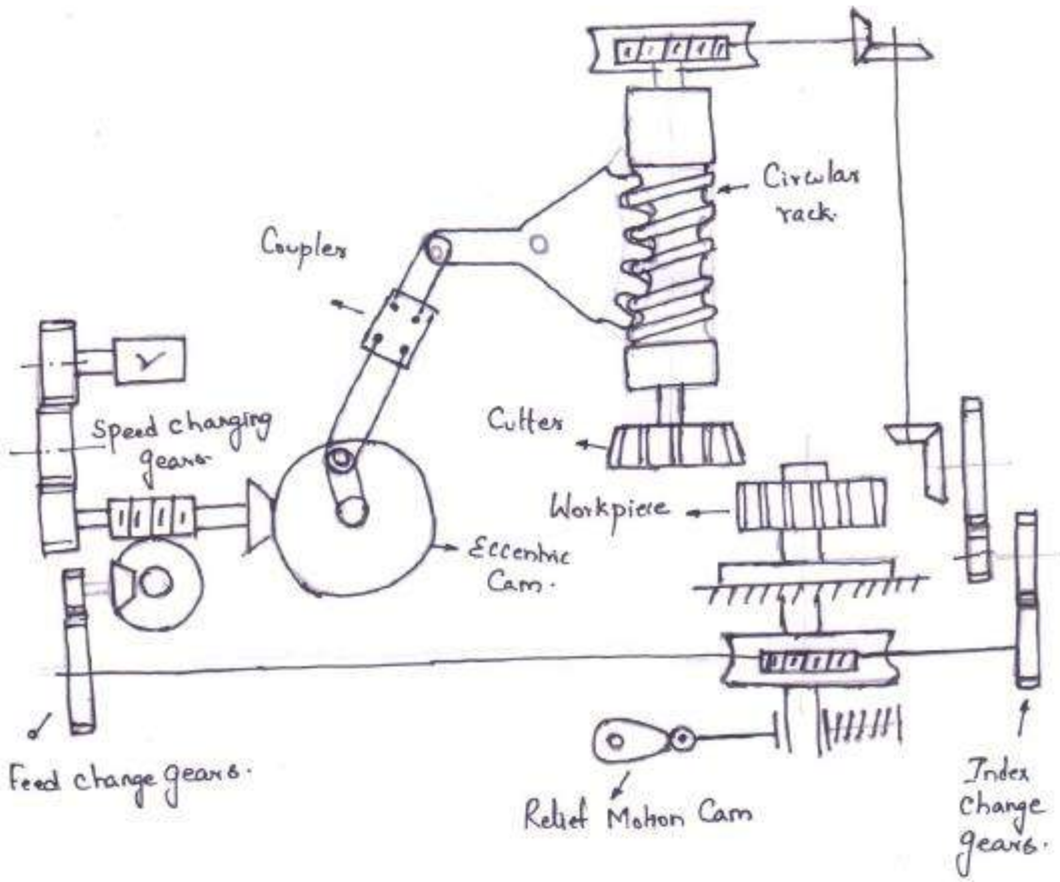


Fig. 1.23 Setup for Gear Shaping Machine

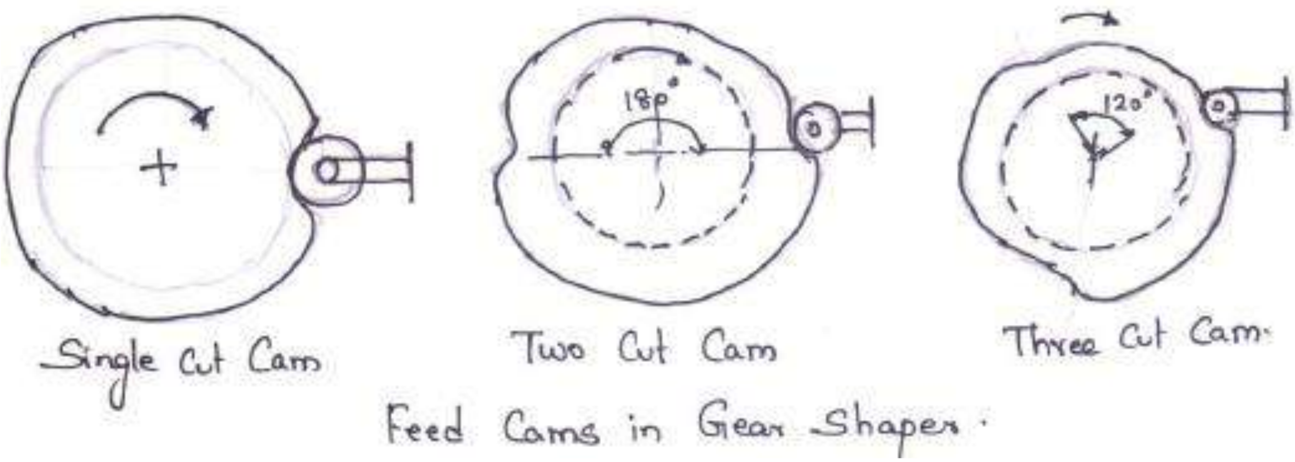


The blank is mounted on a rigid and precision mandrel over the work table. The cutter and workpiece should rotate to generate the tooth one after the other during cutting in a definite relation. The ratio is that of the number of teeth on the work to the number of teeth on the cutter. This is controlled by the index change gears i_x. The circumferential feed rate during cutting is controlled by the feed change gears i_s. The circumference feed rate is the cut taken on the circumference per cut and it will vary from 0.1 to 0.5 mm per stroke.

A relief motion is also given by moving the work away from this cutter when it is in its return stroke. This is done by a cam which is also rotated by the eccentric drive. This avoids the scratch of the cutter during its return stroke.

DESCRIPTION OF FEED CAM

During beginning of the cutting, the cutter must be fed radially towards the work for giving the depth of cut. For gears with longer modules the full depth cannot be given at a single stretch. It has to be given incrementally one round after the other. This is done by a feed cam. The feed cam can be of single, double or triple cut cam as the gear is finished in one, two or three rounds respectively. The raise of the cam is calculated in such a way that during the initial round the depth is more and in the final finishing round the depth is less. A double cut cam has two profiles. As the cam starts rotating the cutter is radially plunged into the job after reaching certain depth, the gear and cutter rotates till one round of the gear is over. The cam rotates only by 180^0 by this time. A change gear between them at 1:2 ratio is set for this purpose. During further rotation the depth is increased to full depth of tooth and the cut is finished at 360^0 of rotation. For three cuts cams the work rotates 3 times for one revolution of the cam and the cam has 3 profiles at 120^0 degree interval.



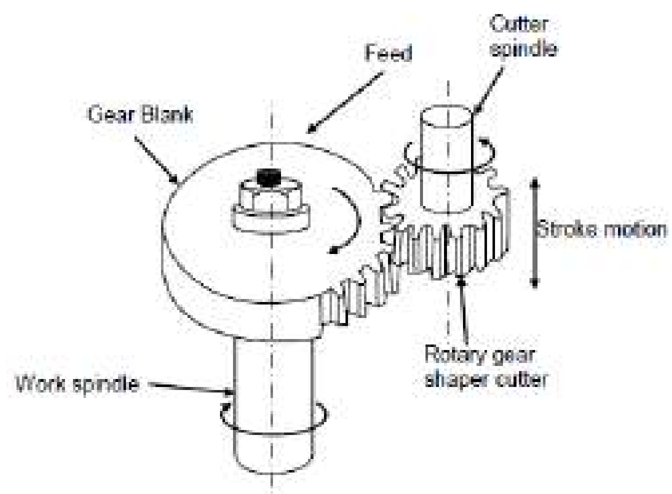


Fig. 1.22 Process of Gear Generation by Gear Shaper

Advantages of Gear Shaping Process

Main advantages of gear shaping process are described below :

I. Shorter product cycle time and suitable for making medium and large sized gears in mass production.
II. Different types of gears can be made except worm and worm wheels.
III. Close tolerance in gear cutting can be maintained.
IV. Accuracy and repeatability of gear tooth profile can be maintained comfortably.

V. For same value of gear tooth module a single type of cutter can be used irrespective of number of teeth in the gear.

Limitations

Main limitations of gear shaping process are described below :

I. It cannot be used to make worm and work wheel which is a particular type of gear.
II. There is no cutting in the return stroke of the gear cutter, so there is a need to make return stroke faster than the cutting stroke.
III. In case of cutting of helical gears, a specially designed guide containing a particular helix and helix angle, corresponding to the teeth to be made, is always needed on urgent basis.

1.12.2 SPUR GEAR GENERATION USING GEAR SHAPER

- Select the cutter of required module and pressure angle. Fit the cutter on the spindle mounting over a shank.
- Fit the work piece on a short, rigid, precision mandrel
- Set the number of strokes, stroke length and zone of cutting
- Set the index change gears for the required number of teeth. Set the feed change gears for the required circumferential feed rate.
- Set the feed cam depending upon the number of cuts and set the ratios.
- Set the head nearer to the work for the required depth and start the machine and coolant.

1.12.3 HELICAL GEAR GENERATION USING GEAR SHAPER

- Helical gear cutting on disc type cutter shaping machine needs special helical gear shaper cutter and a helical guide.
- This is not universal and for each angle and hand of helix they vary. Hence for single piece or small lot of production, this is not adopted.
  - Select the helical gear shaping cutter of required module and pressure angle. The helix angle of the cutter should be same but opposite hand of helix should be used.
  - Fit the helical guide on the top of the spindle as per the required lead of the gear to the cut.

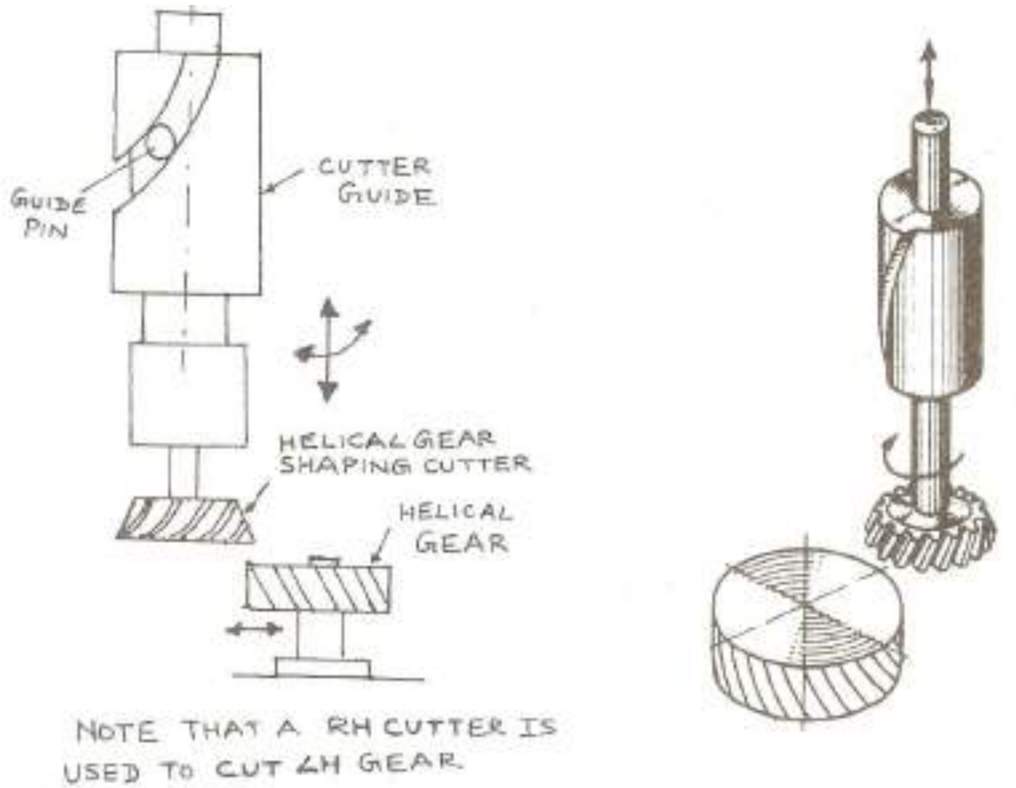


Fig. 1.24 Helical guide for helical gear cutting in gear shaper

1.12.4 CUTTING OF INTERNAL GEARS

Internal gear cutting is done only with shank or disc type cutter. Small, medium size internal gears are cut in this machine. The setting of the blank and selecting the cutter is of importance in this process.

1. Set the blank on a blank holder ring, which is centered with respect to the table. The blank must be firmly held on the ring.
2. Select the gear shaper cutter of shank type, of the correct module and pressure angle. The number of teeth on the cutter with respect to the number of teeth on the blank should be selected to be smaller such that it avoids a. feeding in interference, b. involute interference and c. relief interference.
3. Set the number of strokes, stroke length, cutting zone. Set the index change gears such that both cutter and gear blank rotates in the same direction unlike external gear cutting.
4. Cut the gear and check tooth thickness with a gear tooth vernier.

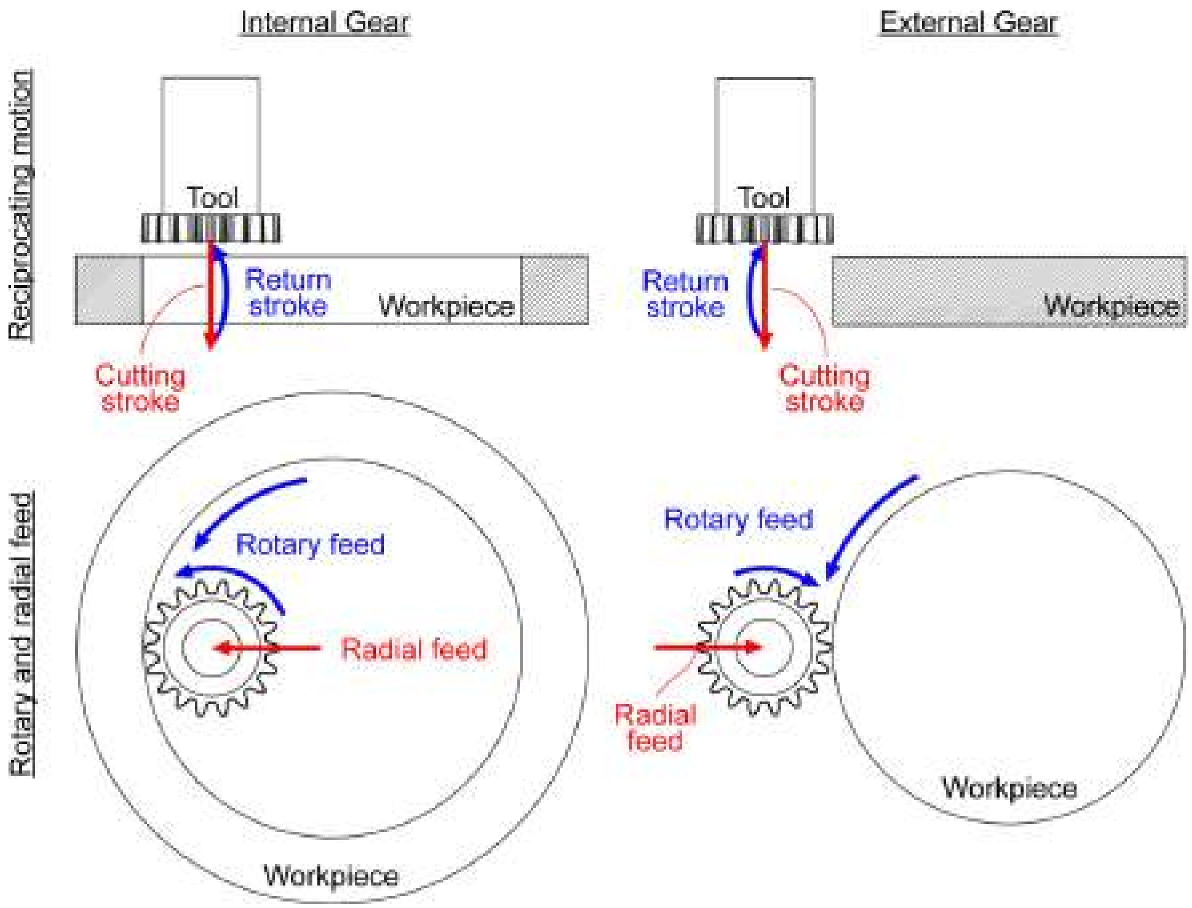


Figure 1.25 Cutting Internal and External Gear using disc cutter

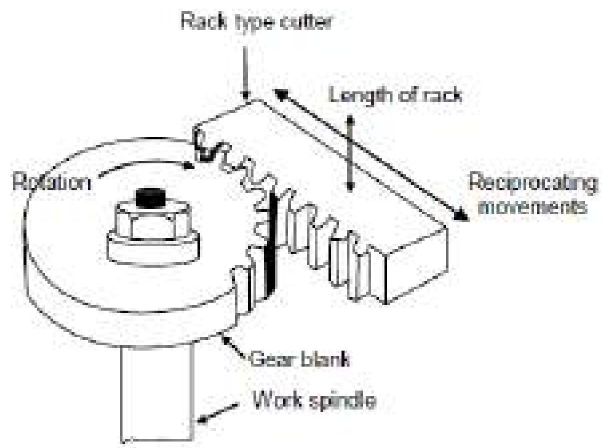
1.13 GEAR SHAPING BY RACK SHAPED CUTTER

In this method, gear cutting is done by a rack shaped cutter called rack type cutter. The principle is illustrated in Figure 1.30. The working is similar to shaping process done by gear type cutter.

Rack type gear shaping machine is adopted for medium to large size spur and helical gear cutting. Herring bone gears can also be cut by this method.

A rack type form relieved high speed steel cutter is mounted on the ram of the machine. The ram can reciprocate up and down for the cutting motion. An eccentric drive is used to set the number of stroke and stroke length of the cutter. The ram can also be tilted towards a side so that helical gear cutting can be achieved. The blank is mounted on a mandrel of a circular table. The circular table is on a linear table. The blank can be rotated or moved sidewards. The combined rotary and linear movement gives a roll motion to the gear blank. The blank can also be moved towards or away from the cutter for setting or for giving the required depth of cut by moving the table.

During operation the ram will be reciprocating with the required number of strokes and stroke length. The blank starts from one end of the cutter and starts entering into the cutter by rolling. As it rolls, the gear tooth is generated slowly and the blank moves to the other end. At this stage the cutter is halted above the work and the table carrying the work is moved to the starting position without rotation. The distance traversed will be equal to the exact number of pitches cut during the preceding working cycle. The cycle is repeated till all the teeth are cut sector by sector. The complete cycle or cutting is automatic and this controlled by a central control drum.



Spur gear is cut with a spur gear rack type cutter with the ram reciprocating vertically. The gear is finished by an automatic cycle. For cutting a helical gear the slide must be tilted through the helix angle and set. The direction depends upon the hand of helix. A spur rack type cutter can be used for cutting but it requires a longer stroke.

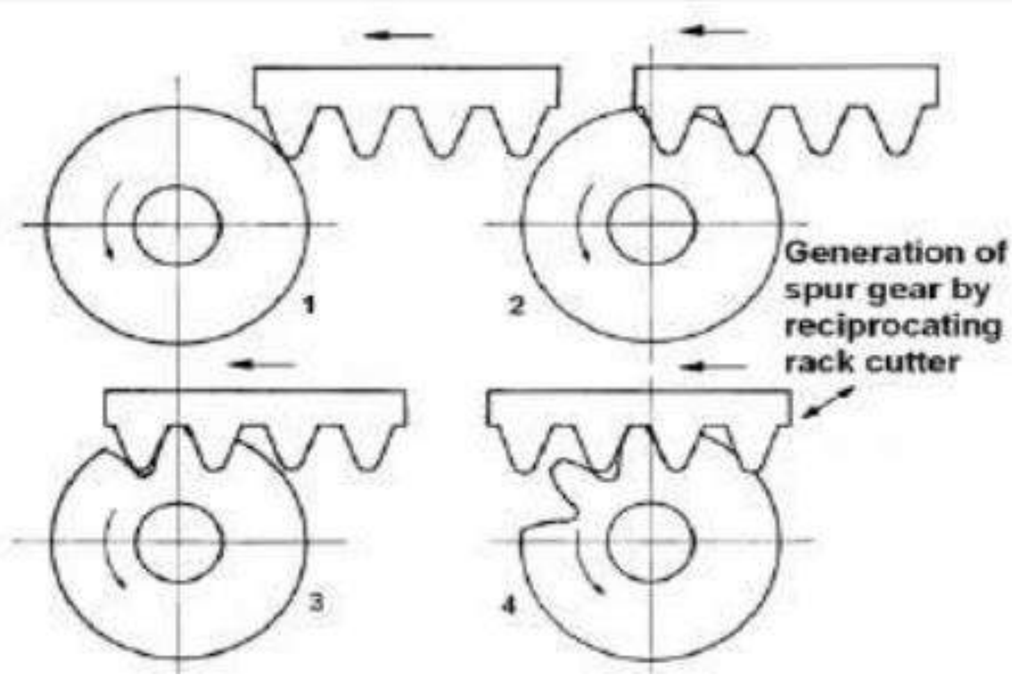


Fig. 1.26 Gear generation by Rack shaped cutter

1.14 Gear Planers (Sunderland Process)

This process is named after the name of its inventor. In this process the cutter reciprocates in a direction towards and away from the gear blank. The process is illustrated in Figure 1.27. Cutter is gradually fed into the gear blank to the required depth. As soon as cutting is completed upto the desired depth, the blank rotates through one pitch distance. The cutter also moves along with the blank and then suddenly withdraws, stepped back by an amount equal to one pitch distance and again made to reciprocate in the normal way. The gear blank does not move till the completion of whole cutting upto the required depth. The whole motion and movement control is basically maintained with the help of synchronous motor and gear train.

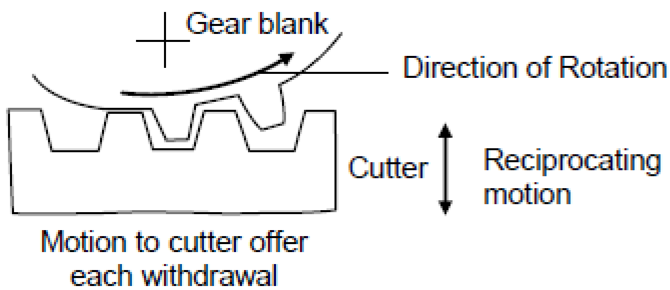


Fig. 1.27 Sunderland Process

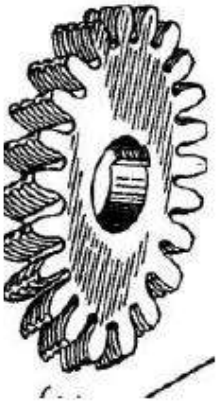
GEAR FINISHING

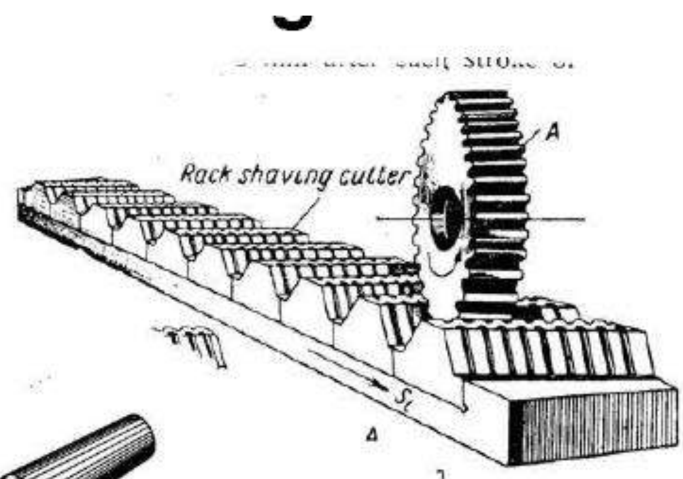
1.1 INTRODUCTION

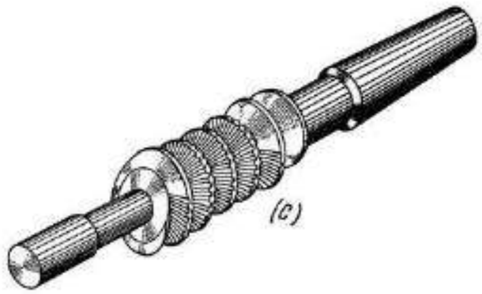
Surface of gear teeth produced by any of the generating process is not accurate and of good quality (smooth). Dimensional inaccuracies and rough surface generated so become the source of lot of noise, excessive wear, play and backlash between the pair of gears in mesh. These all result in loss of power to be transmitted and incorrect velocity ratios. This can be summarized as inefficient power transmission. In order to overcome these problems some finishing operations are recommended for the produced gears. Sometimes poor quality of finish and dimensional inaccuracies occur due to hardening of a produced gear. The prepared (generated) gear is subjected to various hardening processes leading to various problems creating inaccuracies. So finishing operations are to be done at last. Commonly used gear finishing operations are described below.

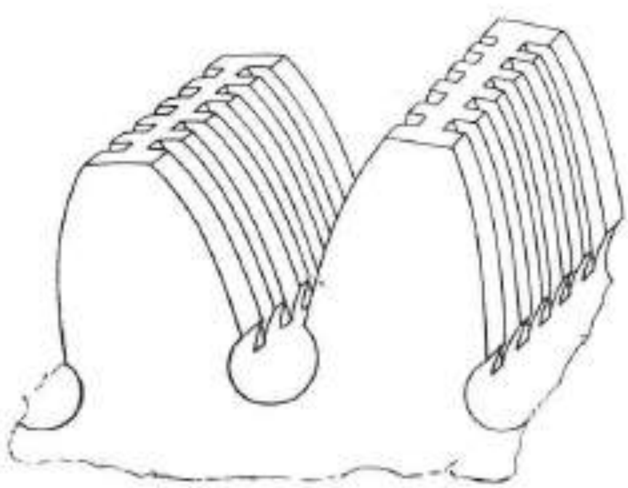
1. Gear shaving or burnishing
2. Gear grinding
3. Gear lapping
4. Shot blasting
5. Phosphate coating

1.2 GEAR SHAVING









Gear shaving is A Process of finishing of gear tooth by running it at very high rpm in mesh with a gear shaving tool. It is used in the **finishing operation** of an **external spur** and **helical gear**. It is also possible to shave an **internal gear**, although generally the work conditions are less favorable.

A gear shaving tool is of a type of rack or pinion having hardened teeth provided with serrations. These serrations serve as cutting edges which do a scrapping operation on the mating faces of gear to be finished. Both gears in mesh are pressed to make proper mating contact. **A gear wheel** being shaved is run in contact with a **shaving cutter. The gear and the cutter** are run in mesh with **their axes crossed** at a small angle. During the rotation the **gear** is reciprocated **longitudinally across the cutter**. This “**shaving“ process** causes very fine chips are cut from the tooth surface(process requires less than 1 minute)

The gear shaving operation is composed by the simultaneous rotation of workpiece and cutter as a pair of gears with crossed axes. The crossed axes generate a reciprocal sliding action between the flank, gear tooth and the cutter teeth.

Soft materials like aluminium alloy, brass, bronze, cast iron etc. and unhardened steels are mostly finished by shaving process**.**

Different types of shaving cutters which while their finishing action work apparently as a spur gear, rack or worm in mesh with the conjugate gears to be finished. All those gear, rack or worm type shaving cutters are of hard steel or HSS and their teeth are uniformly serrated) to generate sharp cutting edges

Most widely used method for the continuous production of large lots, it represents the best cost/performance ratio.

The main limit of the gear shaving process is the lack of the chance to remove the distortion caused by heat treatment.

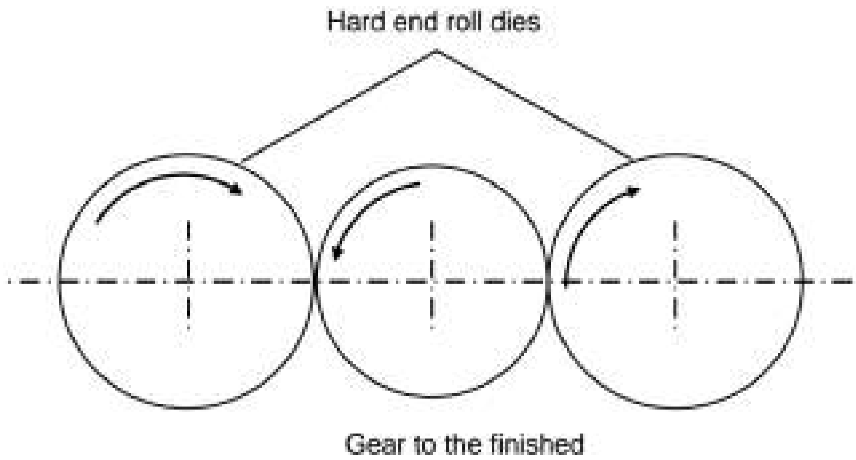
In the automotive industry, the vast majority of gears used in gearboxes are suitable for gear shaving.

The productivity of a gear shaving machine is much higher compared to a gear grinding machine.

Usually, when high noiselessness and strict quality levels are strictly required, the grinding operation is the best solution, although obviously more expensive.

1.3 GEAR ROLLING / ROLL FINISHING

This process involves use of two hardened rolling dies containing very accurate tooth profile of the gear to be finished. The gear to be finished is set in between the two dies and all three are revalued about their axis.



Pressure is exerted by both the rolling dies over the gear to be finished. The material of the die is very hard as compare to the material of gear so there is a plastic deformation of high points and burrs on the profile of gear tooth resulting to smooth surface.

1.4 GEAR BURNISHING

In this method the machined gear is rolled under pressure with three hardened master gears of high accuracy and finish. The minute irregularities of the machined gear teeth are smeared off by cold plastic flow, which also helps in improving the surface integrity of the desired teeth.

The gear to be finished is mounted on a vertical reciprocating shaft and it is kept in mesh with three hardened burnishing compatible gears. The burnishing gears are fed into the cut gear and revalued few revaluations in both the directions. Plastic deformation of irregularities in cold state takes place to give smooth surface of the gear.

The resulting cold Working of the tooth surfaces improves the surface finish, also induces compressive residual stresses thus improving their fatigue life. Process improves only surface finish of teeth and does not correct the tooth profile or pitch of teeth. This process is suitable only for gears which do not require high accuracy.

1.5 GEAR GRINDING

Hardened gears are difficult to finish by shaving and burnishing methods. Since the heat treatment may cause severe distortion and oxide film formation on teeth, therefore there is a necessity for removing considerable stock from the teeth. With the grinding method it is possible to finish the heat treated gears.

Abrasive grinding wheel of a particular shape and geometry are used for finishing of gear teeth.

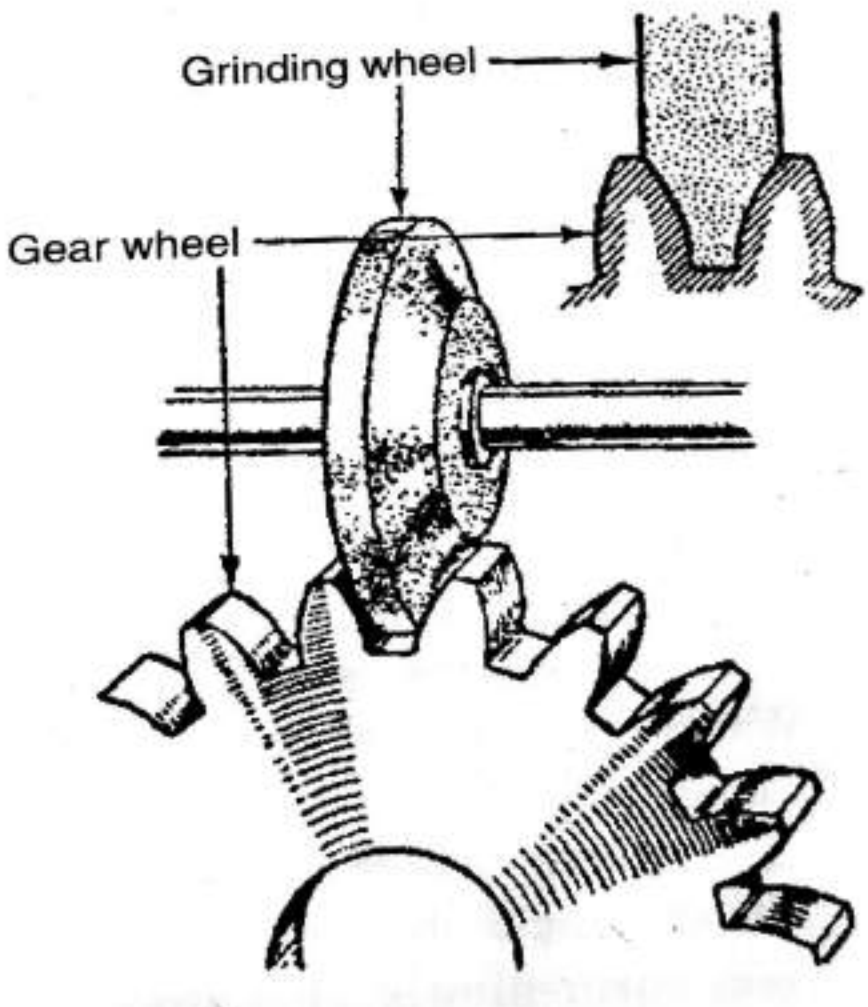
The two basic methods for gear grinding are **form grinding (non-generating) and generation grinding.**

1.5.1 Form Grinding

This is very similar to machining gear teeth by a single disc type form milling cutter where the grinding wheel is dressed to the form that is exactly required on the gear.

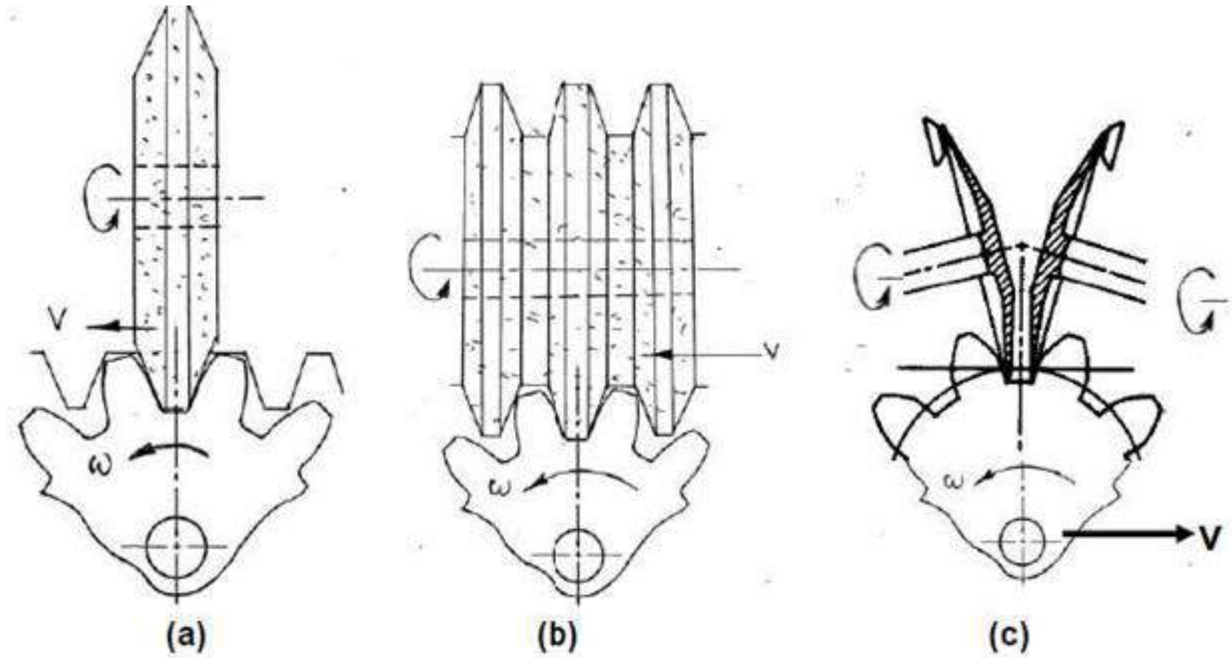
Gear to be finished is mounted and reciprocated under the grinding wheel. The teeth are finished one by one and after one tooth finished, the blank is indexed to the next tooth space as in the form milling operation. Need of indexing makes the process slow and less accurate.
The wheel or dressing has to be changed with change in module, pressure angle and even number of teeth.

Form grinding may be used for finishing straight or single helical spur gears, straight toothed bevel gears as well as worm and worm wheels.



1.5.2 Gear teeth grinding on generation principle

The simplest and most widely used method is very similar to spur gear teeth generation by one or multi-toothed rack cutter. The single or multi-ribbed rotating grinding wheel is reciprocated along the gear teeth as shown. Other tool – work motions remain same as in gear teeth generation by rack type cutter. For finishing large gear teeth, pair of thin dish type grinding wheels are used. Whatsoever, the contacting surfaces of the wheels are made to behave as the two flanks of the virtual rack tooth.



Usually, gear grinding is performed after a gear has been cut and heat-treated to a high hardness. Grinding is necessary for parts above 350 HB (38 HRC), where cutting becomes very difficult. Teeth made by grinding are usually those of fine pitch, where the amount of metal removed is very small.

In addition, grinding of gears becomes the procedure of choice in the case of fully hardened steels, where it may be difficult to keep the heat-treat distortion of a gear within acceptable limits.

In a few cases, medium-hard gears that could be finished by cutting are ground in order to save costs on expensive cutting tools such as hobs, shapers or shaving cutters, or to get a desired surface finish or accuracy on a difficult-to-manufacture gear.

Materials must be removed in small increments when grinding hence costly as compared to cutting. Ground gears attract more inspection than cut gears, and may involve both magnetic particle inspection as well as macroetching with dilute nitric acid.

1.6 Gear Honing

Improving the quality of spur and helical gears is critical in production, and there are many methods to accomplish that. Rotary gear shaving and roll-finishing are done in the green or soft state prior to heat treatment. These processes have the ability to modify the gear geometry in order to compensate for the distortions that occur during heat treatment.

Gear honing is a particularly effective method of removing nicks and burrs from the active profiles of the teeth after heat treatment. Combined with its ability to improve surface finish and make minor form corrections, the honing process is rapidly being accepted as a technique for surface finishing of gears after the heat treatment process.

Benefits of gear honing gear honing:

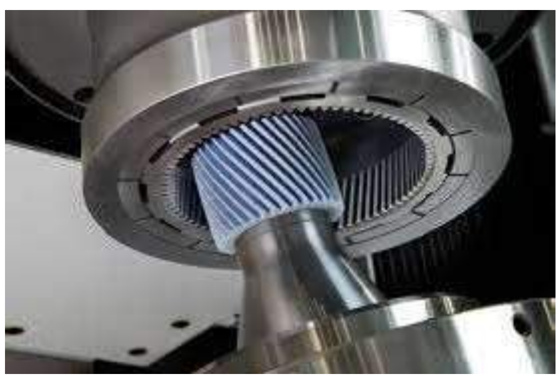
Corrects dimensional errors

Corrects distortions caused by heat treatment

Removes nicks caused by handling

Improves surface finishing

Gear honing is characterized by leaving fine grinding marks due to grinding at an angle. Grinding uses a gear honing wheel in the shape of an internal-tooth gear that meshes with the workpiece to grind the tooth surface. While it is a form of grinding in a way, as it uses a gear honing wheel, it is called “honing” to differentiate it from the normal grinding process. Typical surface finishes for ground tooth surfaces are in the range of 0.8 to 1.6 micron CLA (32 to 63 micro inch). Honing is done to improve the surface finish to the 0.4 to 0.8 micron CLA (16 to 32 micro inch) range.



Increasingly, most gears used in high-load applications are miniaturizing to save weight and space. Improving the surface finish and dimensional accuracy of the gear teeth and bore allow for smaller, more lightweight gears to carry the same load as a lower quality, larger design.

In many of the gear manufacturing shops, grinding is the final processing step for both faces and bores. This post is an effort to expose manufacturing engineers (especially in shops specializing in small and medium production runs) to the idea of cup wheel finishing and honing technology as a potential alternative to grinding.

Economically, gear honing has become an essential part in the production of high-speed transmissions. Gears that have been honed instead of ground offer excellent wear characteristics and are extremely quiet.

Although gear honing is a universal technology, it is mostly used in automotive, aerospace, truck, and heavy equipment industries. This method is suitable for any application where quiet, robust, and reliable gearing is required. As hybrid and electric technology has evolved in the automotive industry, gear noise has become an issue to be addressed. In gears developed using the honing process, engine noise is greatly reduced and people will not be able to hear any gear noise. OEMs are becoming aware of this and have increased the use of noise-eliminating gear honing in these applications. Additionally, in applications such as heavy truck transmissions, where robustness and durability are necessary, honing is the preferred method of gear finishing.

1.7 Gear Lapping

Lapping is done on generally gears having hardness more than 45 RC to remove burrs, abrasions from the surface and to remove small errors caused by heat treatment.
The cutting tools are laps, which are castiron gears engaged with the gear being worked. The la ps are lubricated with a mixture of fine abrasive powder and oil. The gear being worked (Figure) is rolled by three laps.

Abrasive paste is introduced between the teeth under pressure. It is mixed with oil and made to flow through the teeth. Lapping typically improves the wear properties of gear teeth, and corrects the minute errors in involute profile, helix angle, tooth spacing and concentricity created in the forming, cutting or in the heat treatment of the gears.

Therefore, gear lapping is most often applied to sets of hardened gears that must run silently in service.

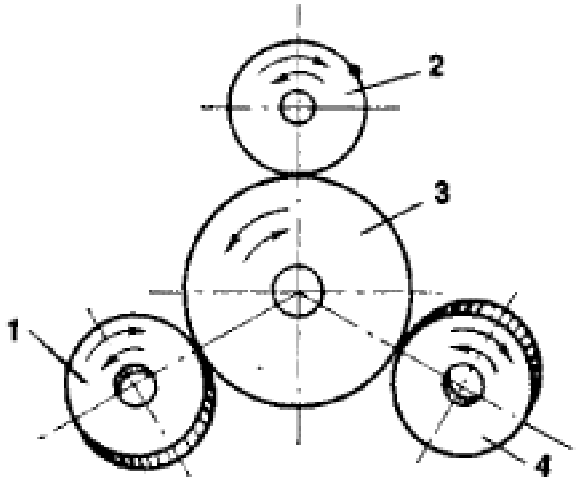


Diagram of lapping of gears: (1), (2), and (4) laps; (3) gear being cut

The axes of the laps with helical or spur teeth are inclined toward the axis of the gear bein g worked; the axis of the third lap is parallel to the axis of the gear being cut and rotates alternat ely in different directions to provide even cutting of the tooth from both sides. The laps also mov e reciprocally in the direction of the axis for a distance of about 25 mm.

1.8 BLAST FINISHING

Blast finishing uses the high-velocity impact of particulate media to clean and finish a surface. The most well known of these methods is sand blasting, which uses grits of sand (SiO_2) as the blasting media. Various other media are also used in blast finishing, including hard abrasives such as aluminum oxide (Al_2O_3) and silicon carbide (SiC), and soft media such as nylon beads and crushed nut shells.

The media is propelled at the target surface by pressurized air or centrifugal force. In some applications, the process is performed wet, in which fine particles in a water slurry are directed under hydraulic pressure at the surface.

1.9 SHOT PEENING

In shot peening, a high-velocity stream of small cast steel pellets (called shot) is directed at a metallic surface with the effect of cold working and inducing compressive stresses into the surface layers.

Used primarily to improve fatigue strength of metal parts. Cleaning is accomplished as a by-product of the operation. Shots (round metallic, glass, or ceramic particles) produce force sufficient to create plastic deformation. It operates by the mechanism of plasticity rather than abrasion: each particle functions as a ball-peen hammer. In practice, this means that less material is removed by the process, and less dust created.

Peening a surface spreads it plastically, causing changes in the mechanical properties of the surface. Its main application is to avoid the propagation of microcracks from a surface. Such cracks do not propagate in a material that is under a compressive stress. Shot peening is often called for in aircraft repairs to relieve tensile stresses built up in the grinding process and replace them with beneficial compressive stresses.

Depending on the part geometry, part material, shot material, shot quality, shot intensity, and shot coverage, shot peening can increase fatigue life up to 1000%.

Plastic deformation induces a residual compressive stress in a peened surface, along with tensile stress in the interior. Surface compressive stresses confer resistance to metal fatigue and to some forms of stress corrosion. The tensile stresses deep in the part are not as problematic as tensile stresses on the surface because cracks are less likely to start in the interior.

1.10 PHOSPHATE COATING

Phosphate coatings are used on steel parts for corrosion resistance, better adherence of lubrication, lubricity, or as a foundation for subsequent coatings or painting. It serves as a conversion coating in which a dilute solution of phosphoric acid and phosphate salts is applied via spraying or immersion and chemically reacts with the surface of the part being coated to form a layer of insoluble, crystalline phosphates.

Phosphate conversion coatings can also be used on aluminum, zinc, cadmium, silver and tin. The main **types of phosphate coatings** are **manganese, iron and zinc.**

Manganese phosphates are used both for corrosion resistance and lubricity and are applied only by immersion.

Iron phosphates are typically used as a base for further coatings or painting and are applied by immersion or by spraying.

Zinc phosphates are used for corrosion resistance (phosphate and oil), a lubricant base layer, and as a paint/coating base and can also be applied by immersion or spraying

Computer Numerical Control of Machine Tools

1.1 INTRODUCTION TO NUMERICAL CONTROL (NC) MACHINE TOOLS

- Numerical control (NC) is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium, as opposed to controlled manually.
- The concept of NC was proposed in the late 1940s by John Parsons who recommended a method of automatic machine control that would guide a milling cutter t produce a curvilinear motion in order to generate smooth profiles on the work-pieces.
- Most NC today is computer numerical control (CNC), in which computers play an integral part of the control.
- In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs.

1.1.1 COMPONENTS OF NC SYSTEMS

The important components of the numerical control or NC system are listed below;

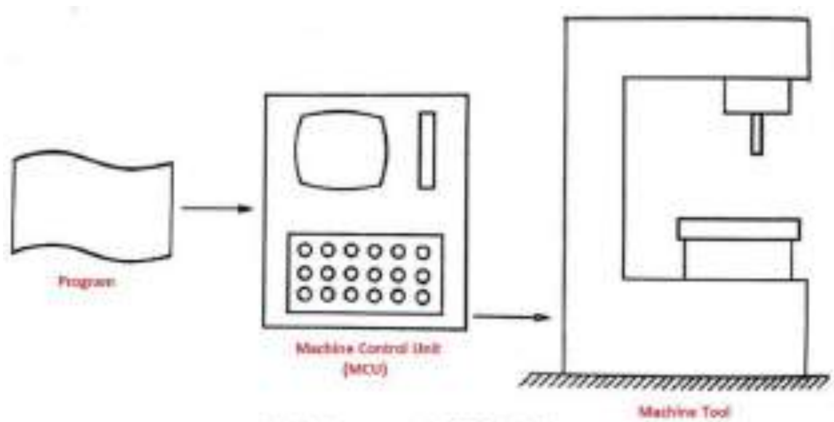
- Program of instructions
- Controller unit, also called as the machine control unit (MCU) and
- Machine tool
- Drive Units (servo motors)
- Feedback Unit
- Magnetic Box
- Control Panel (Manual control)

1. Program of Instructions

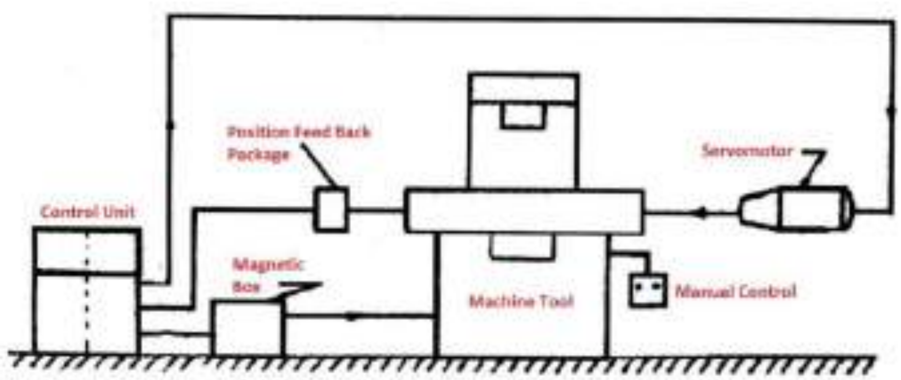
The program of instructions of the NC machine is the step-by-step set of instructions that tells the machines what it has to do. These instructions can tell the machine to turn the piece of metal to certain diameter, drill the hole of certain diameter up to certain length, form certain shape etc. The set of instructions are coded in numerical or symbolic form and written on certain medium that can be interpreted by the controller unit of the NC machine. The mediums commonly used earlier for writing the instructions were punched cards, magnetic tapes and 35mm motion picture film, but now 1 inch wide punched tape is used more commonly.

The program instructions are written by the expert who has programming knowledge as well the machining knowledge. The person should know the various steps of the machining required to manufacture a particular product and should be able to write these steps in the form of the program that can be understood by the control unit of the NC machine, which would eventually direct the machine tool to perform the required machining operations.

One can also input the instructions directly into the controller unit manually, this method is called as manual data input (MDI), which is used for very simple jobs. Then there is direct numerical control method (DNC) in which the machines are controlled by the computers by direct link omitting the tape reader.



Basic Components of NC Machines



Main Components of NC Machines

2. Machine Controller Unit

The controller unit is most vital parts part of the NC and CNC machines. The controller unit is made of the electronics components. It reads and interprets the program of instructions and converts them in the mechanical actions of the machine tool. Thus the controller unit forms an important link between the program and the machine tool. The control unit operates the machines as per the set of instructions given to it.

The typical control unit comprises of tape reader, a date buffer, signal output channels to the machine tools, feedback channel from the machine tool, and the sequence control to coordinate the overall machining operation.

Initially, the set of instructions from the punched tape are read by the tape reader, which is sort of the electromechanical devise. The data from the tape is stored into the data buffer in form of logical blocks of instructions with each block resulting in certain sequence of operations.

The controller sends the instructions to the machine tool via signal output channels that are connected to the servomotors and other controls of the machines. The feedback channels ensure that the instructions have been executed by the machine correctly. The sequence control part of the controller unit ensures that all the operations are executed in the proper sequence.
One important thing to note about the controller unit here is that all the modern NC machines are equipped with the microcomputer that acts as the controller unit. The program is fed into the computer directly and the computer controls the working the machine tool. Such machines are called as Computer Controller Machines (CNC) machines.

3. Machine Tool

It can be any type of machine tool or tool. The third basic component of an NC system is the machine tool or other controlled processes. It is a part of the NC system that performs useful functions. In the most common example of an NC system, one is designed to perform machining operations, with machine tools having a workable and spindle-like motor and controls necessary to drive them.

It also includes the cutting tools, work fixtures, and other auxiliary equipment needed in the machining operation Machine tools which are operated by Numerical Control must have the following special features.

- Structure
- Screw, Nuts and Guideways
- Spindle drives
- Tool changes
- Lubrication system

4. Drive Units (servo motors)

All axes of the machine driven by powerful DC servomotors mounted on preloaded ball bearings. The signals sent by the control unit actuate the servomotors, this cause different slide to move to give the desired length of travel and feed rate. The drive units usually have hydraulic DC motor or stepping motor.

5. Feedback Unit

This unit is also known as position feedback package. Feedback unit feeds back the information about the actual position of the movement to the control unit. The control unit compares the actual movements with the required movements. If there is a difference, the drive units are actuated to do the necessary correction.

6. Magnetic Box

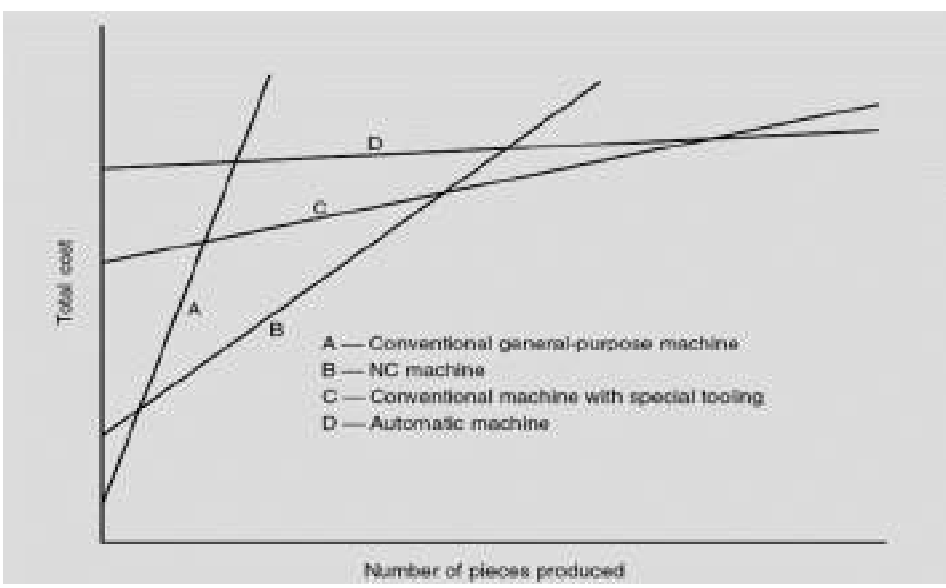
The magnetic box receives electric signals from the control unit for all the other activities except for the servo motor drives. Spindle motor starting and stopping, selecting spindle speeds, making tool changes, control of coolant supply etc, are directed from the magnetic box.

7. Control Panel (Manual control)

Another element of the NC system, which may be actually part of the controller unit or machine tool, is the control panel. The control panel contains the dials and switches by which the machine operator runs the NC system.

It may also include data displays to give information to the operator. Although the NC system is an automatic system, the human operator is all needed to the machine on and off, to change tools, to load and unload the machine.

1.1.2 WHEN TO USE NC



The general characteristics of jobs for which NC is most appropriate are the following:

1. Greater operator safety
2. Greater operator efficiency:
    1. Reduction of scrap
    2. Reduced lead time for production
    3. Fewer chances for human error
    4. Maximal accuracy and interchangeability of parts
3. Lower tooling costs:
    1. Increased productivity
    2. Minimal spare parts inventory
4. Greater machine tool safety:
    1. Fewer man-hours for inspection functions
5. Greater machine utilization:
    1. Parts are often processed in small to medium sizes.

1.1.3 Advantages of NC

1. **Greater flexibility.** With NC, a wide variety of operations can be performed, changeovers from one run to another through tape or program changes can be made rapidly, and design changes to parts can be made rapidly through minor changes to the part program.
2. **Elimination of templates, models, jigs, and fixtures.** The NC control tape takes over the job of locating the cutting tools, which eliminates the design, manufacture, and the use of templates, jigs, and fixtures.
3. **Easier setups.** By using more simple work holding and locating devices, the operator does not have to set table limit stops or dogs, or depend on the feed screw dials when setting up for machining.
4. **Reduced machining time**. Machining with NC allows the use of a wider range of speeds and feeds than conventional machine tools. Optimum selection of feed rates and cutting speeds is ensured. The NC equipment can also move from one cutting operation to the next faster than the operator, which significantly reduces the total machining time.
5. **Greater accuracy and uniformity**. During NC machining, no human errors are possible and machining of the same part is performed in the same way through the stored tape or program, which improves the uniformity and interchangeability of the machined parts. Therefore, inspection time is greatly reduced, and is necessary for the fi rst piece only, in

addition to random checks for critical dimensions. Hence, scrape and rework are greatly reduced or completely eliminated by using NC.

6. **Greater safety.** The operator is not as closely involved with the actual machining operations as with conventional machine tools. As the tape is checked out before actual production runs, there is less chance of machine damage that may cause human injuries.
7. **Conversion to the metric system**. An NC system can be converted to accept either inch or metric inputs.

1.1.4 Problems Associated with the Conventional NC Machines

1) Mistakes related with part programming (programming for the parts to be manufactured):

When the programs of instructions related to the particular part to be manufactured are written on the punched tape, the syntax or numerical mistakes are quite common. The NC tape is not completed correctly in a single pass and at least three passes are required to get the correct program written. Another major problem with the part programming is achieving the best sequence of steps required for the machining the part.

2) Non optimal speed and feeds:

For most economic manufacturing of the object from the raw material it should be given optimum speed and feeds during manufacturing. The conventional numerical control does not provide opportunity to change the speeds and feeds during the cutting operations, so the programmer is compelled to set the speeds and feeds for the worst-case conditions that can result in highly expensive manufacturing due to wastages, and low quality jobs. This also results in manufacturing of the jobs at lower than optimum productivity.

3) Punched tape:

The punched, which is made up of paper and on which the program is written is the problem in itself. This tape is fragile and susceptible to wear and tear so it has short life and cannot be reliable enough for the repeated use. Instead of paper, other media like Mylar can be used for writing the program of instructions, but these materials are quite expensive.

4) Unreliable tape reader:

The tape reader reads the program of instructions from the punched tape, but it is considered to be highly unreliable hardware component of the NC machine. When the NC machine breaks down the first thing the maintenance personnel checks is the tape reader.

5) **The inflexible controller:**

The conventional NC machine has the controller unit which is hard wired and the making the changes in the controls of the machines is a tough task. The controller used in the CNC machines is the computer, which is highly flexible.

6) **Important information:**

The conventional NC machine cannot provide crucial information to the operator and the supervisor like the number of pieces manufactured, tools changes and others.

The problems associated with the NC machines have been solved over the time with the improvement in the NC technology mostly due to advancement in the electronics. The major change obviously came when mini or microcomputers were introduced in the NC system. The computers have had major impact on the NC system and with their introduction the whole technology has come to be known as the CNC (computer numerical control) technology. For the common man and also to the engineers the automatic machine tools are now known by the name CNC machines and not the NC machines.

1.1.5 Applications of NC Machine

NC machines are widely used in the metal cutting industry and are best used to produce the following types of product:

1. Parts with complicated contours.
2. Parts requiring close tolerance and/or good repeatability,
3. Its parts requiring expensive jigs and fixtures if produced on conventional machines.
4. Parts can undergo many engineering changes, such as during the development phase of a prototype.
5. In cases where human errors could be extremely costly Parts that are needed in a hurry.
6. Small batch lots or short production runs.

The NC technique has been employed to a wide range of operations, including drafting assembly, inspection, sheet metal press work, and spot welding. However, numerical control finds its major applications in metal machining processes. But it is more prominently used for various metal machining processes such as turning, drilling, milling, shaping, etc.

Machine Tool Applications (Machining applications)

1. Milling, drilling, turning, boring, grinding
2. Machining centres, turning centres, mill-turn centres

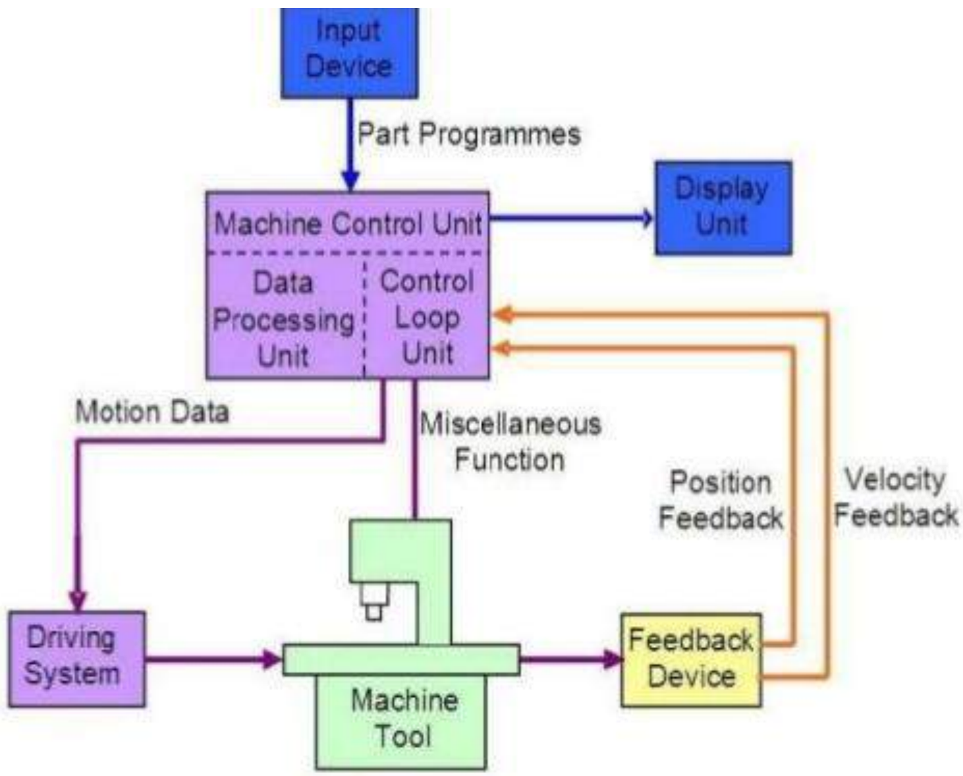
3. Punch presses, thermal cutting machines, etc.

Other NC Applications: (Non-machining applications)

1. Component insertion machines in electronics
2. Drafting machines (x-y plotters)
3. Coordinate measuring machines
4. Tape laying machines for polymer composites
5. Filament winding machines for polymer composites

1.2 COMPUTER NUMERICAL CONTROL (CNC)

A CNC is specifically defined as “The numerical control system where a dedicated, stored program computer is used to perform some or all of the basic numerical control functions in accordance with control programs stored in read & write memory of the computer” by Electronic Industries Association (EIA). CNC controls are also referred to as soft-wired NC systems because most of their control functions are implemented by the control software programs. CNC is a computer assisted process to control general purpose machines from instructions generated by a processor and stored in a memory system.



CNC is a microprocessor based control system that accepts a set of program instructions, processes and sends output control information to a machine tool, accepts feedback information acquired from a transducer placed on the machine tool and based on the instructions and feedback, assures that proper motion, speed and operation occur.

The information stored in the computer can be read by automatic means and converted into electrical signals, which operate the electrically controlled servo systems. Electrically controlled servo systems permits the slides of a machine tool to be driven simultaneously and at the appropriate feeds and direction so that complex shapes can be cut, often with a single operation and without the need to reorient the work piece. The programmer can easily write the codes, and edit the programs as per the requirements. These programs can be used for different parts, and they don't have to be repeated again and again.

Compared to the NC machine, the CNC machine offers greater additional flexibility and computational capability. New systems can be incorporated into the CNC controller simply by reprogramming the unit. Because of its capacity and the flexibility the CNC machines are called as "soft-wired" NC.

Some of the important parts of CNC machines are Machine structure, guide ways, feed drives, spindle and Spindle bearings, measuring systems, controls, software and operator interface, gauging, tool monitoring.

1.2.1 ELEMENTS OF A CNC SYSTEM

A CNC system basically consists of the following:

(a) Central processing unit (CPU)

(b) Servo control unit

(c) Operator control panel

(d) Machine control panel

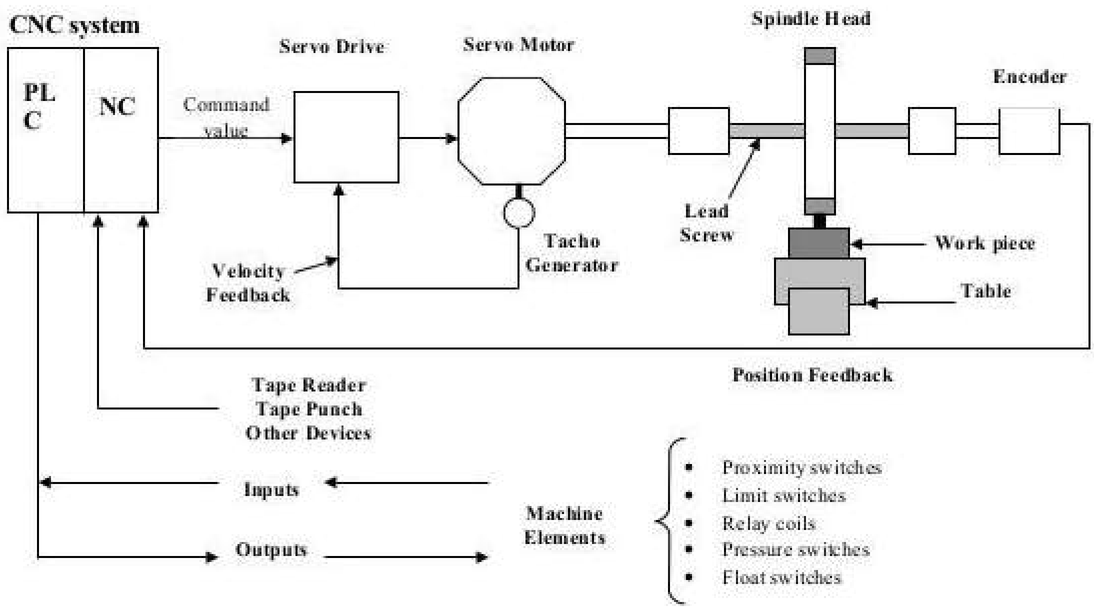
(e) Programmable logic controller

(f) Other peripheral devices.

(a) Central Processing Unit (CPU)

The CPU is the heart of a CNC system. It accepts the information stored in the memory as part program. This data is decoded and transformed into specific position control and velocity signals. It also oversees the movement of the control axis or spindle and whenever this does not match with the programmed values, a corrective action as taken.

All the compensation required for machine acquires (like lead screw pitch error, tool wear out, backlashes.) are calculated by CPU depending upon the corresponding inputs made available to the system. The same will be taken care of during the generation of control signals for the axis movement. Also, some basic safety checks are built into the system through this unit and continuous necessary corrective actions will be provided by CPU unit. Whenever the situation goes beyond control of the CPU, it takes the final action of shutting down the system and in turn the machine.



(b) Servo Control Unit

The decoded position and velocity control signals, generated by the CPU for the axis movement forms the input to the servo control unit. This unit in turn generates suitable signals as command values. The command values are converted by the servo drive units which are interfaced with the axes and the spindle motors. The servo control unit receives the position feedback signals for the actual movement of the machine tool axes from the feedback devices (like linear scales, rotary encoders, revolvers, etc.)

(c) Operator Control Panel

The Operator Control Panel provides the user interface to facilitate a two way communication between the user, CNC system and the machine tool. This consists of two parts: video display unit and Keyboard.

(d) Machine Control Panel

It is the direct interface between the operator and the NC system, enabling the operation of the machine through the CNC system. During program execution, the CNC controls the axis the motion, spindle function or tool function on a machine tool, depending upon the part program stored in the memory. Prior to the starting of the machining process, machine should first be prepared with some specific takes like, establishing a correct reference point, loading the system memory with the required part program, loading and checking of tool offsets, zero offsets, etc.

(e) Programmable Logic Controller (PLC)

PLC's were basically as replacement for hard wired relay control panels. They were basically introduced as replacement for hard wired relay panels. They developed to be re-programmed without hardware changes when requirements were altered and thus are re-usable. PLC's are now available with increased functions, more memory and larger input/output capabilities. In the CPU, all the decisions are made relative to controlling a machine or a process. The CPU receives input data, performs logical decisions based upon stored programs and drives the output connection to a computer for hierarchical control are done through CPU.

(f) Other Peripheral Devices

These include sensor interface, provision for communication equipment, programming units, printer, tape reader interface, etc.

1.2.2 Advantages of the CNC Machines

1. **Part program tape and tape reader:**

   In the older CNC machines the part program tape and the tape reader is still required, but they are used only for feeding the program into the memory of the computer. Once the program is saved into the memory, the tape is no more required and the program stored in the memory can be used repeatedly. Thus the tape and the tape reader that poses the major maintenance problems are done away with. In fact the latest CNC machine don't even require the tape and tape reader, for the program of instructions are fed directly into the mini or microcomputer via the control panel of the computer.

2. **Editing the program:**

   Since the program of instructions is saved in the computer memory, they can be edited and changed as per the requirements. Thus the CNC system is highly flexible. One can also make necessary changes in the program for providing variable speeds and feeds for the manufacture of the jobs resulting in economic manufacturing. Even the NC tape used for the programming in CNC machines can be corrected and optimized since it allows changes in the tool path, speed, feed etc.

3. **Metric conversion:**

   The CNC machine allows the conversion of tapes prepared in the metric system into the SI system of measurements. Thus programmer does not have to re-enter the whole program of instructions merely because of the different units of measurements used in the program.

4. **Highly flexible:**

   The CNC machines are highly flexible. One can easily make the changes in the program and store them as the new program. One can also introduce new control options like the new interpolation scheme quite easily. It is easier to make updates in the CNC machines with lesser cost; hence risk of the obsolescence of the CNC machine is reduced.

5. **Easier programming:**

   The programs are written in the CNC machine using language which has statements similar to the ordinary English language statements. The programmer can easily master the CNC programming language and use it for the wide range of the machining operations of the job. The programmer can set the various dimension of the job, the machining operations to be carried out and their sequence, the amount of metal to be removed in each cutting operation, the speed of cutting, etc. The program of instructions is written as per the available size of the raw materials and also the surface finish required for the final finished job. Some of the programs take the form of the macro subroutines stored in the memory of the CNC machine and the programmer can use them frequently whenever required. Some of the programs are stored in the library and they can be used wherever required completely or as a small part of the big program.

1.2.3 DISADVANTAGES OF CNC SYSTEMS

1. CNC machines are generally more expensive than manually operated machines.
2. The CNC machine operator only needs basic training and skills, enough to supervise several machines.
3. Increase in electrical maintenance, high initial investment and high per hour operating costs than the traditional systems.
4. Fewer workers are required to operate CNC machines compared to manually operated machines. Investment in CNC machines can lead to unemployment.

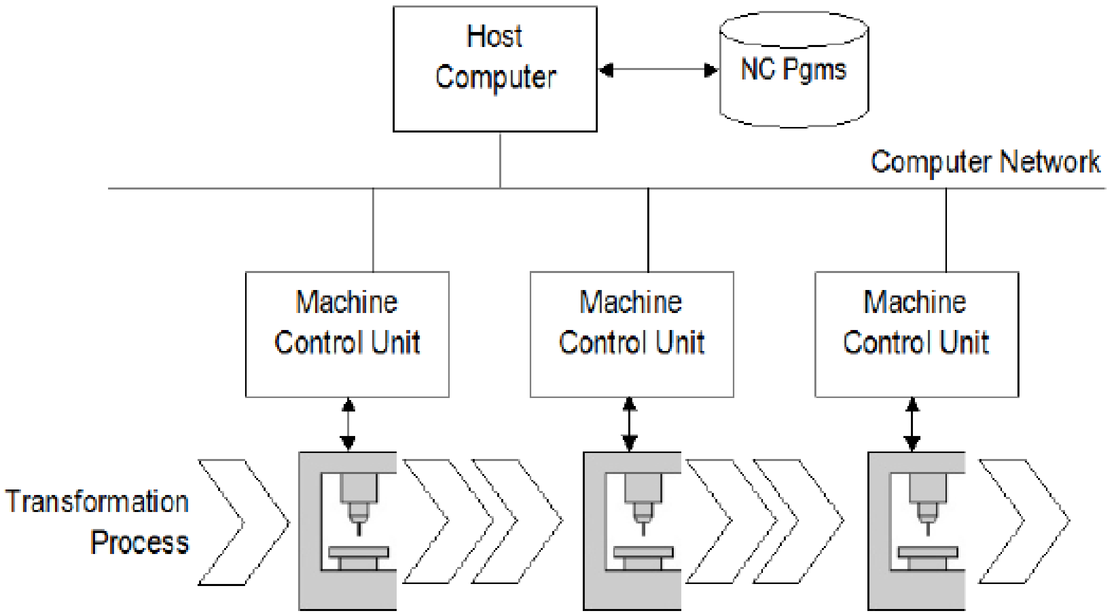
1.2.4 PARTS SUITABLE FOR CNC MACHINES

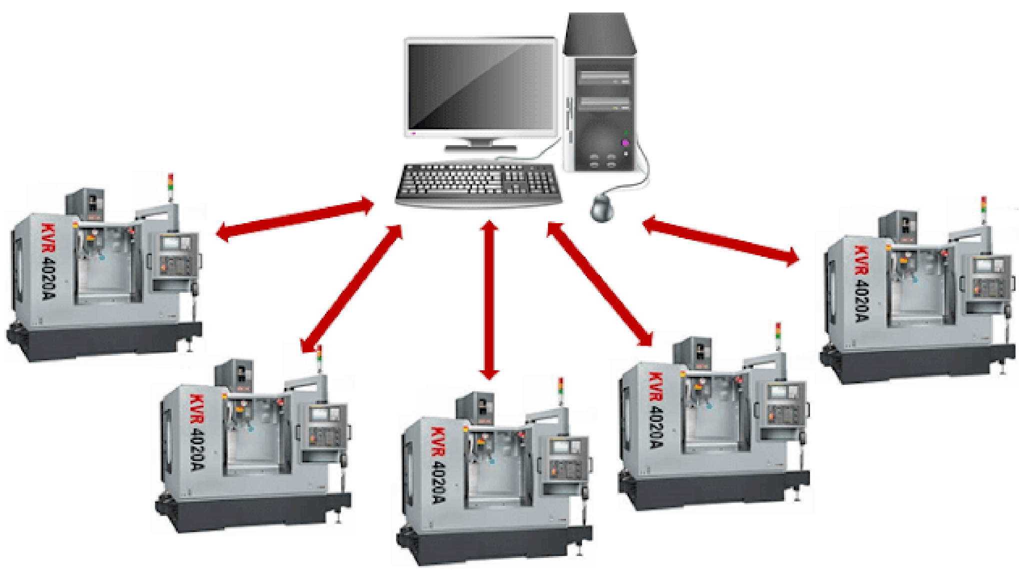
- Aerospace equipments.
- Automobile Parts.
- Complex shapes.
- Electronic industry uses CNC e.g. Printed circuit board.
- Electrical industry uses CNC e.g. Coil winding.
- For small to medium batch quantity.
- Where the set-ups are very large.
- Where the tool storage is a problem.
- Where much metal needs to be removed.
- The operations are very complex.
- For parts subjected to regularly design changes.
- When the inspection is required 100%.
- When lead time does not permit the conventional tooling manufacture.
- When the machining time is very less as compared to down time.
- Where tool storage is a problem.
- Where repetitive operations are required on the work.

1.3 DIRECT NUMERICAL CONTROL (DNC) MACHINES

Direct Numerical Control can be defined as a type of manufacturing system in which several NC or CNC machines are controlled remotely from a Host/Main frame computer or direct numerical control (DNC) – control of multiple machine tools by a single (mainframe) computer through direct connection.

A DNC is specifically defined as "A system connecting a set of numerically controlled machines to a common memory for part program or machine program storage with provision for on-demand distribution of data to machines" by Electronic Industries Association (EIA).





In DNC, several NC machines are directly controlled by a computer, eliminating substantial hardware from the individual controller of each machine tool. The part-program is downloaded to the machines directly (thus omitting the tape reader) from the computer memory.

The basic DNC system requires following basic component are Main frame computer, Memory, Communication network, NC machine tool. The communication network can be done either through connecting the remotely located computer, with lengthy cables to the individual machine control directly or connecting the main frame computer with a small computer at individual operator's station known as satellite computer. DNC system is expensive and is preferably used in large organizations. The combination of DNC/CNC makes possible to eliminate the use of programme as the input media for CNC machines. The DNC computer downloads the program directly to the CNC computer memory. This reduces the amount of communication required between the central computer and each machine tool.

1.3.1 DIFFERENCE BETWEEN CNC AND DNC

s.No	Computer Numerical Control	Direct Numerical Control
1	CNC is an integral part of the machine	DNC is not an integral part of the machine DNC computer can be located at some distances from the machines.
2	CNC transferring the machining instructions	DNC manages information distribution to the number of machines
3	CNC computer control only one NC machine tool	Using DNC, Programmer can control more than one machine as required
4	CNC have low processing power when compared to DNC	CNC have high processing power when compared to CNC
5	CNC software is to increase the capability of a particular machine tool	DNC not only control the equipment but also serve as a management information system.

1.3.2 ELEMENTS OF DIRECT NUMERICAL CONTROL

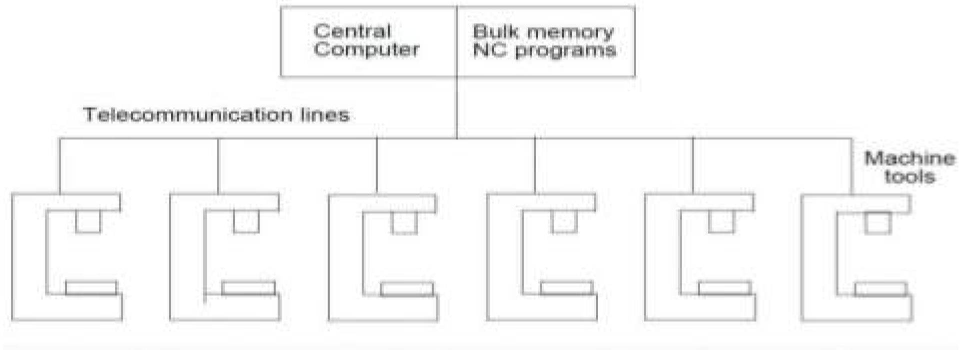
1. Central computer
2. Bulk memory which stores the NC part programs.
3. Telecommunication lines
4. Machine Tools.

The configuration of the DNC system can be divided into:

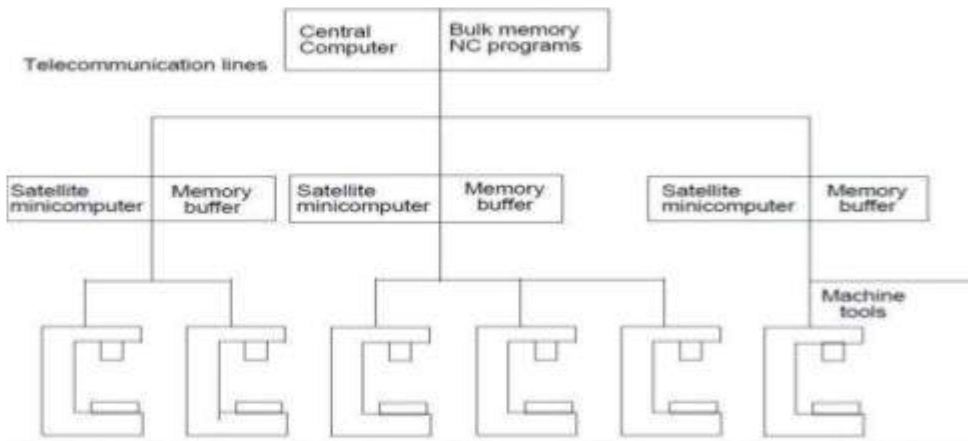
1. DNC system without satellite computer.
2. DNC system with satellite computer.

   Satellite computers are minicomputers and they serve to take some of the burden off central computer. Each satellite controls several machine tools.

1. DNC system without satellite computer



2. DNC system with satellite computer



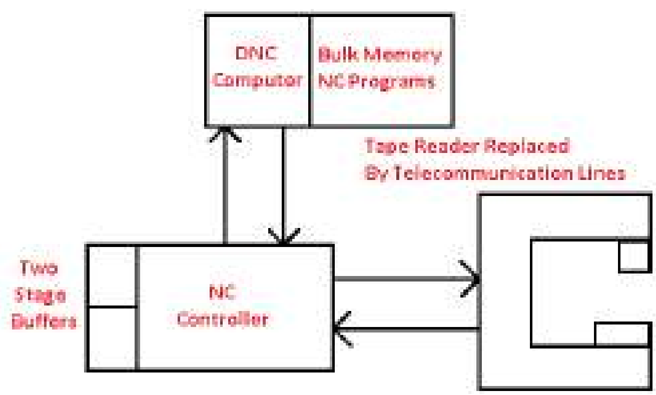
1.3.3 TYPES OF DNC

There are two alternative system configurations by which the communication link is established between the control computer and the machine tool.

1. Behind the Tape Reader (BTR) system.
2. Special Machine Control Unit.

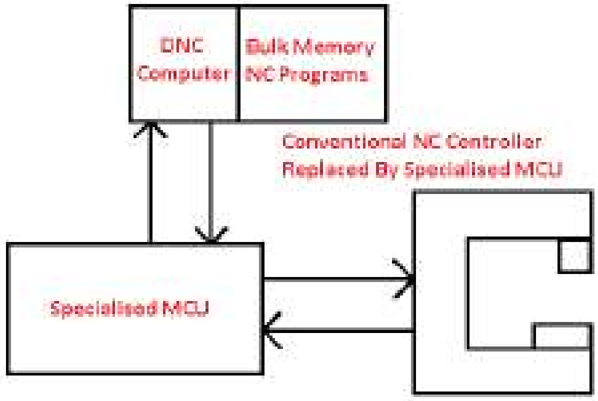
1. Behind the Tape Reader (BTR) system

- The computer is linked directly to the regular NC controller unit
- Except for the source of the command instructions, the operation of the system is very similar to conventional NC
- The controller unit uses two temporary storage buffers to receive blocks of instructions from the DNC computer and convert them into machine actions
- One buffer is receiving a block of data, the other is providing control instructions to machine tool.
- Cost is very less



Behind The Tape Reader (BTR) System

2. Special Machine Control Unit.



Specialised MCU

- Replace the regular controller unit with a special machine control unit
- The special control unit is designed to facilitate communication between the machine tool and the computer
- The special MCU configuration achieve a superior balance between accuracy of the interpolation and fast metal removal rates than is generally possible with the BTR system

1.3.4 Functions of DNC

The functions which a DNC system is designed to perform

1. NC without punched tape.
2. NC part program storage.
3. Data collection, processing, and reporting.
4. Communication

NC part program storage

The program storage subsystem must be structured to satisfy several purposes

- The program must be made available for downloading to the NC machine tools.
- The subsystem must allow for new programs to be entered, old programs to be deleted, and existing programs to be edited
- The storage subsystem must be structured to perform certain data processing and management functions, such as file security, displays of programs, and manipulation of data

Data collection, Processing, and Reporting

- The purpose of this function is to "monitor" production of the factory.
- The data concerned are: Tool usage Machine utilization Production piece counts
- These data must be processed by the DNC computer, and reports are prepared to provide management with information necessary for running the plant.

Communication

A "Communication Network" is required to accomplish the previous functions of DNC
The essential communication links in DNC are between the following components of the system:

- Central computer and machine tools
- Central computer and NC part programmer terminal
- Central computer and bulk memory

1.3.5 Advantages of DNC System

Following are the advantages of the DNC system:

1. The DNC rejects the use of tape readers, which are absolutely the weakest component of the NC system.
2. Time-sharing by central control makes it possible to keep close control over the entire machine shop.
3. The huge memory of DNC allows it to store a large number of part programs for subsequent use. It also receives the memories of NC control unit.
4. Presence of a central bulk memory allows the same program to be run on different machines at the same time without duplicating it at individual places

1.3.6 Disadvantages of DNC System

Following are the disadvantages of the DNC system:

1. DNC uses a central control and in an event of computer failure, the complete activities of the machine shop would come to a standstill.
2. DNC is expensive and its use is practical in areas where high automation is required.

1.3.7 Comparison between NC, CNC and DNC machine tools

Numerical Control System	Computer Numerical System	Direct / Distributed Numerical control system
1. The part program is fed to the machine through the tapes or other such media. 2. In order to modify the program, the tapes have to be changed. 3. In NC machine tool system, tape reader is a part of	1. In CNC machine tool system, the program is fed to the machine through the computer. 2. The programs can be easily modified with the help of computer. 3. The microprocessor or minicomputer forms the	1. The part program is fed to the machine through the Main computer 2. In order to modify the program, single computer is used 3. Large memory of DNC allows it to store a large

machine control unit. 4. System has no memory storage and each time it is run using the tape. 5. It cannot import CAD files. 6. It cannot use feedback system. 7. They are not software driven.	machine control unit. The CNC machine does not need tape reader. 4. It has memory storage ability, in which part program can be stored. 5. System can import CAD files and convert it to part program. 6. The system can use feedback system. 7. The system is software driven.	amount of part program. 4. Same part program can be run on different machines at the same time. 5. The data can be processed using the MIS software so as to effectively carry out the Production planning and scheduling.

1.4 CLASSIFICATION OF NC SYSTEM

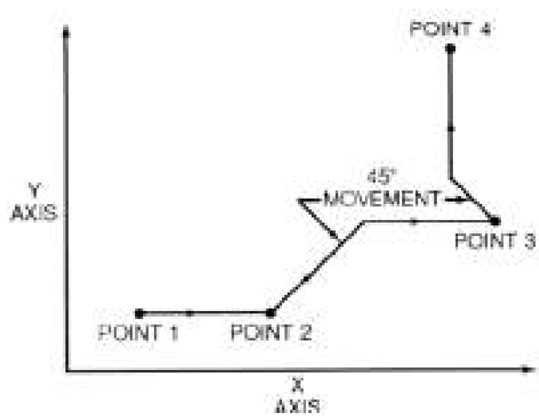
1. Classification based on the motion type.
2. Classification based on the control loops.
3. Classification based on the number of axes.

1.4.1 Classification based on the motion type

a) Point – to- point positioning
b) Straight Cut NC
c) Contouring or continuous path NC

a. Point-to-Point Positioning

Point-to-point positioning is used when it is necessary to accurately locate the spindle, or the workpiece mounted on the machine table, at one or more specific Iodations to perform such operations as drilling, reaming, boring, tapping, and punching **(Fig.9).**



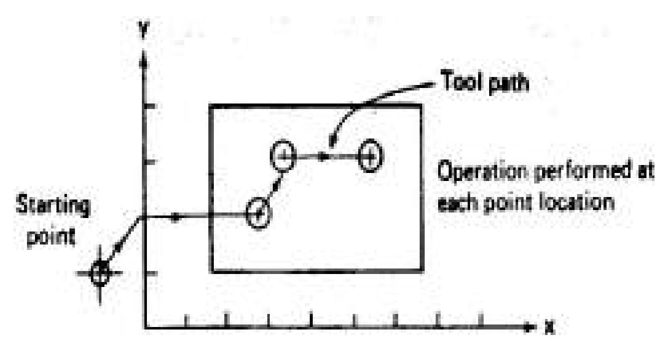


Fig. 9 The path followed by point-to-point positioning to reach various programmed points (machining locations) on the XY axis.

Point to point (PTP) is also sometimes called a positioning system. In PTP, the objective of the machine tool control system is to move the cutting tool to a predefined location. The speed or path by which this movement is accomplished is not important in point-to-point NC. Once the tool reaches the desired location, the machining operation is performed at that position.

NC drill operations are a good example of PTP systems. The spindle must first be positioned at a particular location on the workpiece. This is done under PTP control. Then, the drilling of the hole is performed at that location, the tool is moved to the next hole location, and so on. Since no cutting is performed between holes, there is no need for controlling the relative motion of the tool and workpiece between hole locations. On positioning systems, the speeds and feeds used by the machine tool are often controlled by the operator rather than by the NC tape. Figure 8.5 illustrates the point-to-point type of control.

In **Fig. 9** point 1 to point 2 is a straight line, and the machine moves only along the X axis; but points 2 and 3 require that motion along both the X and Y axes takes place. As the distance in the X direction is greater than in the Y direction, Y will reach its position first, leaving X to travel in a straight line for the remaining distance. A similar motion takes place between points 3 and 4.

Straight cut NC

Straight-cut control systems are capable of moving the cutting tool parallel to one of the major axes at a controlled rate suitable for machining. It is therefore appropriate for performing milling operations to fabricate workpieces of rectangular configurations. With this type of NC system it is not possible to combine movements in more than a single axis direction. Therefore, angular cuts on the workpiece would not be possible. An example of a straight-cut operation is shown in **Figure 8.6.**

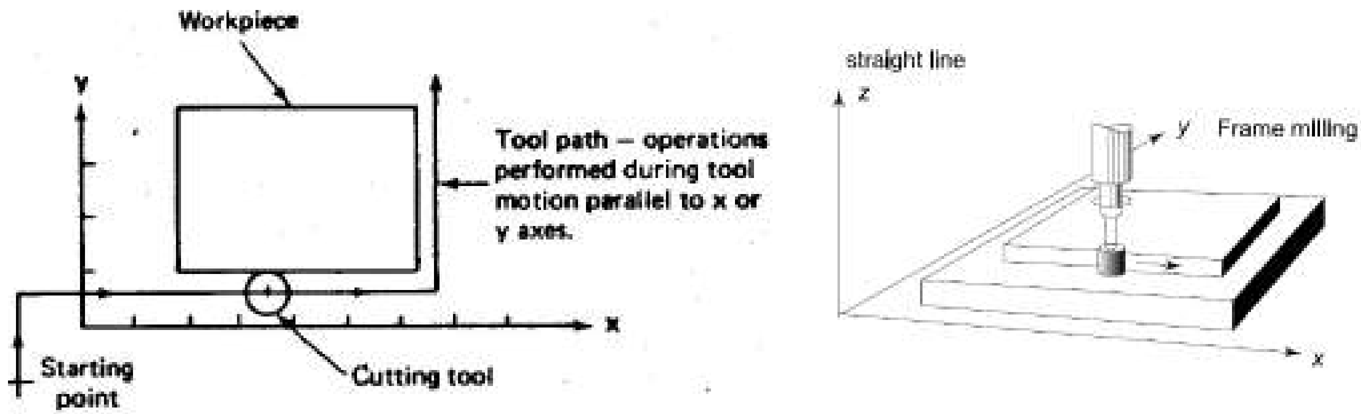


Figure 8.6 Straight Cut in NC

An NC machine tool capable of performing straight-cut movements is also capable of point to point movements.

Continuous Path (Contouring)

Contouring, or continuous path machining, involves work such as that produced on a lathe or milling machine, where the cutting tool is in contact with the workpiece as it travels from one programmed point to the next. Continuous path positioning is the ability to control motions on two or more machine axes simultaneously to keep a constant cutter-workpiece relationship. The programmed information in the CNC program must accurately position the cutting tool from one point to the next and follow a predefined accurate path at a programmed feed rate in order to produce the form or contour required (Fig. 10).

For the mathematically oriented reader, it might be useful to distinguish between PTP, straight-cut, and contouring in the following way. Consider a two-axis control system, where the table is moved in the XY plane. With point-to-point systems, control is achieved over the x and y coordinates. With straight-cut systems, control is provided for either dx/dt or dy/dt, but only one at a time. With contouring systems, both of the rates dx/dt and dy/dt can be controlled simultaneously. In order to cut a straight path at some angle, the relative values of dy/dt and dx/dt must be maintained in proportion to the tangent of the angle. In order to machine along a curved path, the values of dy/dt and dx/dt must continually be changed so as to follow the path.

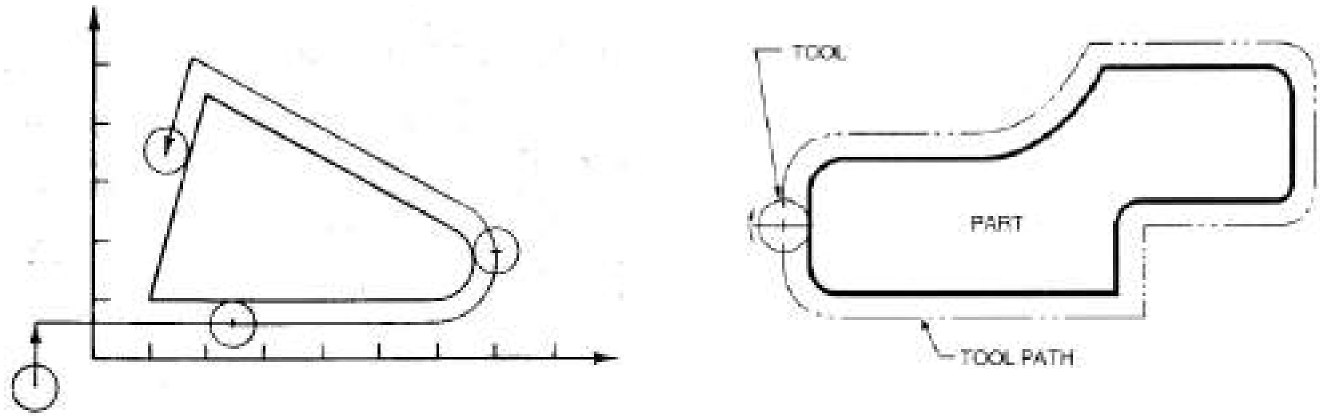
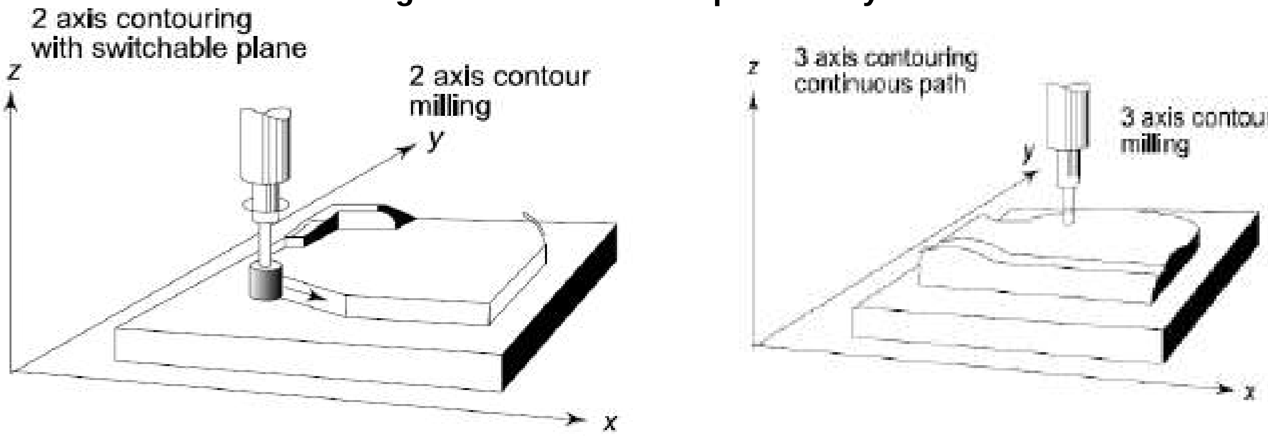


Figure 10. Continuous path NC system



Interpolation

The method by which contouring machine tools move from one programmed point to the next point is called interpolation.

This ability to merge individual axis points into a predefined tool path is built into most of today's MCUs. There are five methods of interpolation: linear, circular, helical, parabolic, and cubic. All contouring controls provide linear interpolation, and most controls are capable of both linear and circular interpolation. Helical, parabolic, and cubic interpolation are used by industries that manufacture parts which have complex shapes, such as aerospace parts and dies for car bodies.

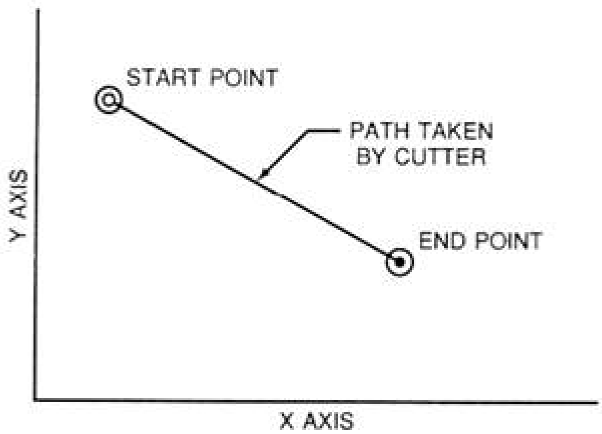
Linear Interpolation

Linear Interpolation consists of any programmed points linked together by straight lines, whether the points are close together or far apart **(Fig. 11).** Curves can be produced with linear interpolation by breaking them into short, straight-line segments. This method has limitations, because a very large number of points would have to be programmed to describe the curve in order to produce a contour shape.

A contour programmed in linear interpolation requires the coordinate positions (XY positions in two-axis work) for the start and finish of each line segment. Therefore, the end point of one line or segment becomes the start point for the next segment, and so on, throughout the entire program.

Circular Interpolation

The development of MCUs capable of circular interpolation has greatly simplified the process of programming arcs and circles. To program an arc **(Fig. 12),** the MCU requires only the coordinate positions (the XY axes) of the circle center, the radius of the circle, the start point and end point of the arc being cut, and the direction in which the arc is to be cut (clockwise or counterclockwise) See **Fig. 12**. The information required may vary with different MCUs.



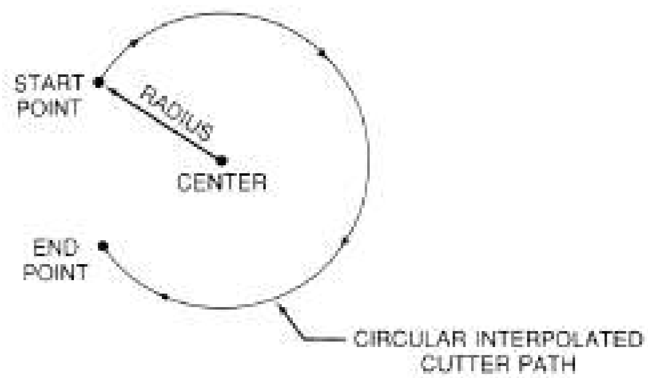


Fig. 11. Linear Interpolation **Fig. 12 Circular Interpolation**

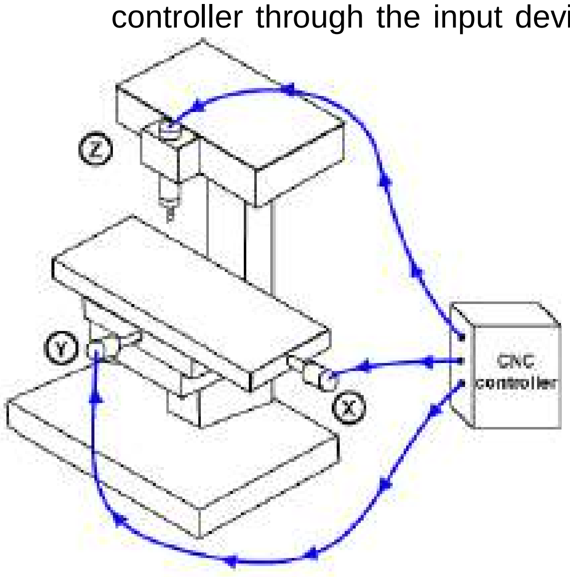
1.4.2 Classification based on the control loops

Based on the looping system the CNC systems are classified into two categories,

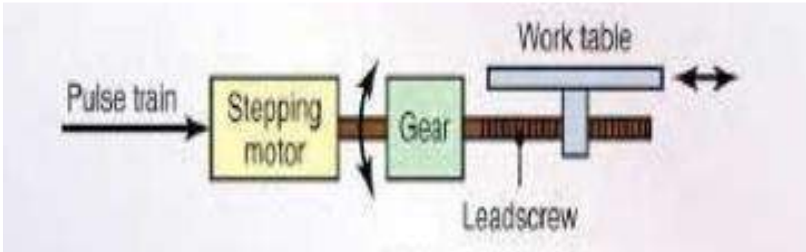
a) Open loop system and

b) Closed loop system.

a. **Open loop system** are the systems in which instructions of program are sent into the controller through the input device and then these instructions are converted to the signals by the controller and sent to the servo amplifiers to energize the servo motors.



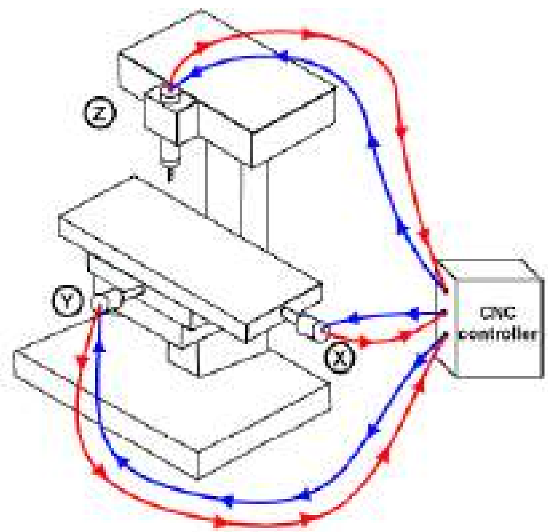
The open loop systems are usually used in the point to point control systems where the accuracy doesn't matter much and in few continuous path control system since there is no feedback from the system the result may deviate from the actual.

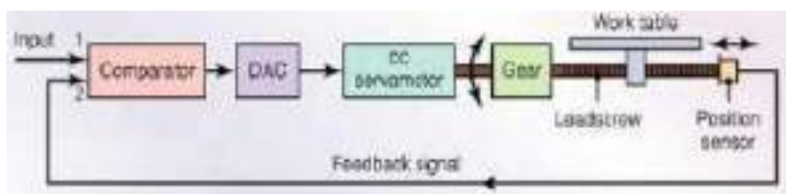


b. **Closed Loop system:**

.

The closed-loop system has a feedback subsystem to monitor the actual output and correct any discrepancy from the programmed input. These systems use position and velocity feed back sensors. The feedback system could be either analog or digital.





The analog systems measure the variation of physical variables such as position and velocity in terms of voltage levels. Digital systems monitor output variations by means of electrical pulses.

Closed-loop systems are very powerful and accurate because they are capable of monitoring operating conditions through feedback subsystems and automatically compensating for any variations in real-time.

1.4.3 CLASSIFICATION BASED ON THE NUMBER OF AXES.

a. Two axis control system

b. Two and half axis control system

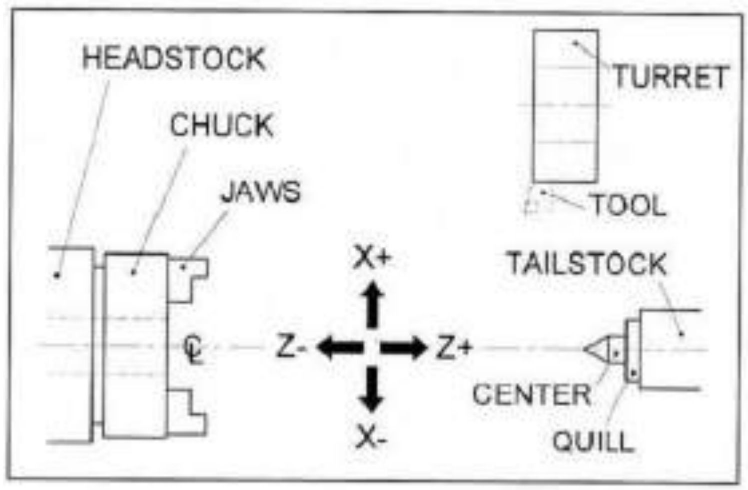
c. Three axis control system

d. Four axis control system

e. Five axis control system

***a.* Two axis control system:**

Two axis control system is a machine which give access to only two axis. The best example is the lathe machine. The machine driven with the servo motor allows you only 2 axis, i.e., X and the z Axis. Here in lathe the job will be rotating and the tool will be moving in 2 directions, indicating the depth Z and the cut in X direction. So the program is done with x and the Z direction.

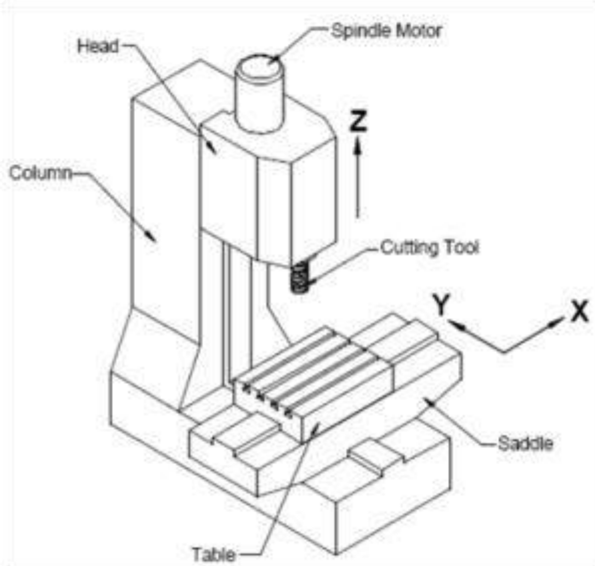


3.2 Two and half axis control system:

Two and half axis control system is also a three axis machine but the movement will not be 3 dimensional. These are the best example for drilling and tapping machines. First the X and the Y axis are moved to the position and then the third axis comes to effect. In some of the machines the first X movement is made and then the Y movement to reach the XY destination when it is in rapid. But while in feed or machining mode it does its work with 2 dimensional moving along x and y simultaneously, These type of machines may also called as two and half axis machines or 2.5 axis machine.

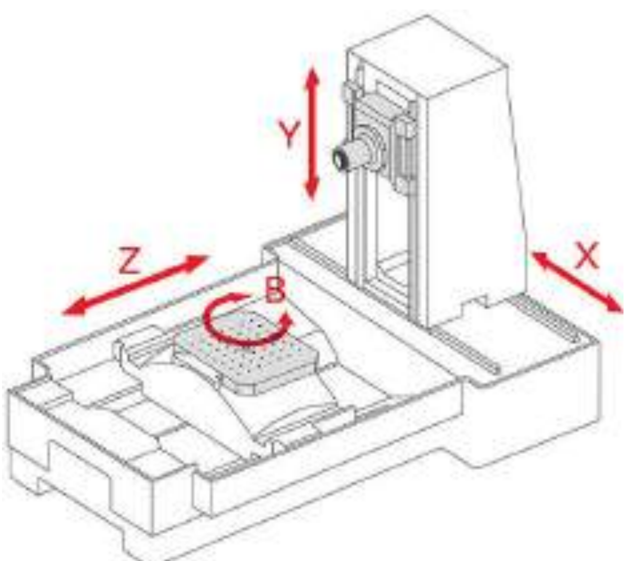
3.3 Three axis control system:

Three axis control systems are the machines which moves in three dimensional i.e., X, Y and Z axis move simultaneously. These are the most popular machines which can produce high accurate precision parts with three axis machines. The servo motor control the movements as per the instructions are given. The three axes have variety of machines based on their bed lengths. As the bed length increases the cost of the machine increases. The number of setups increases and the cost also increases, three axis machines are limited for simple jobs and straight forward 3 axis jobs.



3.4 Four axis control system:

In Four axis control system is a three axis machine with an extra rotation on B-axis. Four axis can be a vertical machine or an horizontal machine.

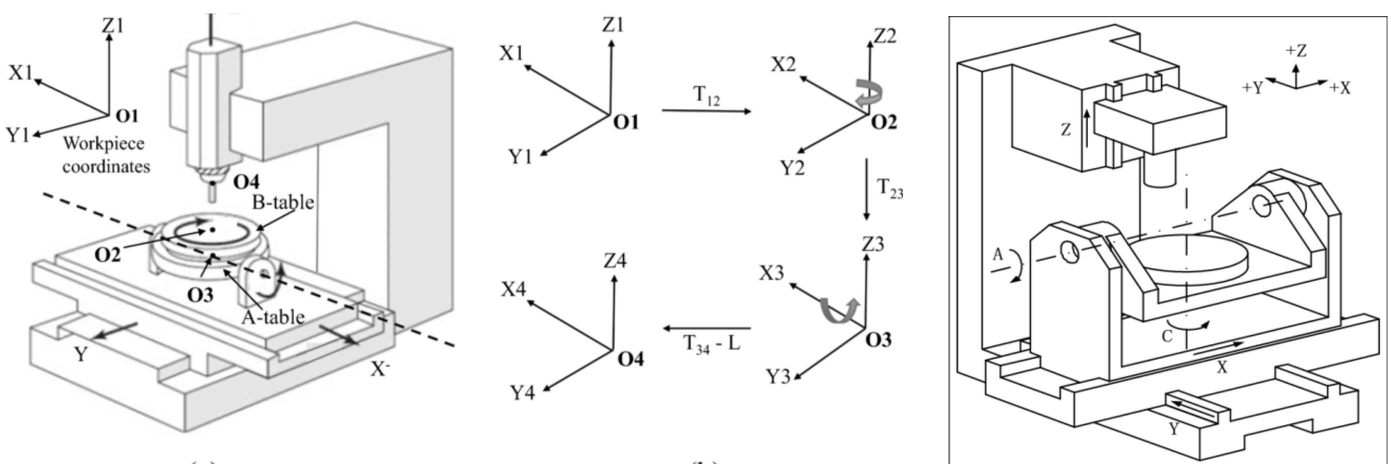


In vertical CNC machine the rotary head is added on the side of the machine bed. The machine works as the three axis machine but it has a rotary head for example if the holes have be machined on the tube, the tube can be mounted on the rotary B-axis and the drilling in the required angle. But vertical 4-axis machines are limited for the small jobs.

Horizontal 4-axis machines are used worldwide for successful machining of big parts in a single setup, where it takes 2-3 setups in a 3-axis machine. The part is mounted on the bed which has 360 rotations around Y direction i.e., B-Axis giving chance for the tool to cut on the angled faces in a single setup.

3.5 Five axis control system:

Five axis control system are the three axis machines with an extra rotation along Y and Z directions which are called B-axis and A-axis. As in Four axis machines B rotation is given by the bed and the A-axis rotation is given by the spindle movement called PIVOT point. Using a five axis machine reduces the cycle time by machining the complex parts in a single setup and improves the accuracy on the positional errors by eliminating the setups.



(a) (b)

3.6 Six axis control system:

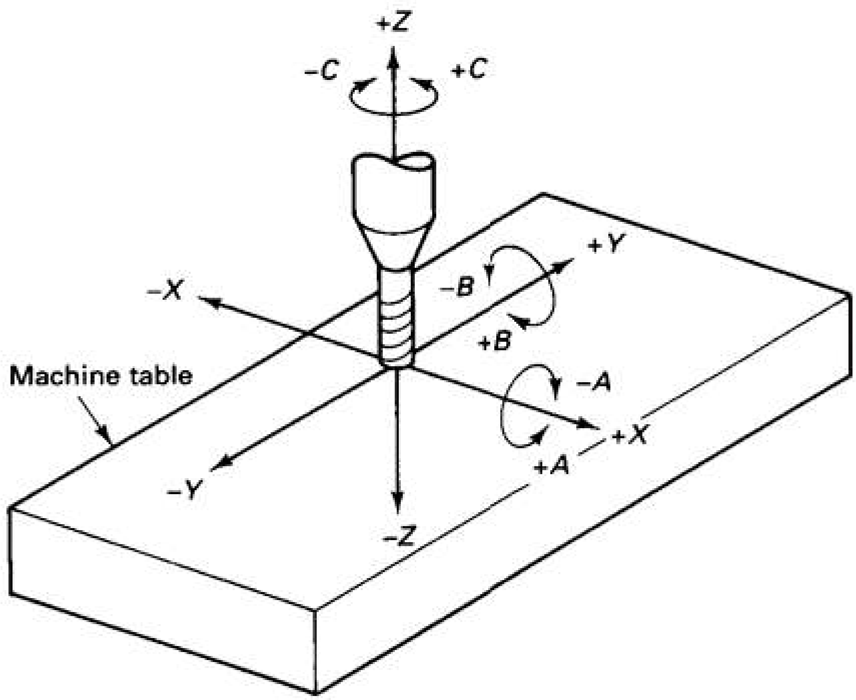
Six axis control system are the three axis machines with an extra rotation along X, Y and Z directions which are called A-axis, B-axis and C-axis respectively.

Importance of higher axes machining:

- Reduced cycle time by machining complex components using a single setup.

- In addition to time savings, improved accuracy can also be achieved as positioning errors between setups are eliminated.
- Improved surface finish and tool life by tilting the tool to maintain optimum tool to part contact all the times.

- Improved access to under cuts and deep pockets. By tilting the tool, the tool can be made normal to the work surface and the errors may be reduced as the major component of cutting force will be along the tool axis.
- Higher axes machining has been widely used for machining sculptures surfaces in aerospace and automobile industry.



1.5 CONSTRUCTIONAL FEATURES OF CNC

A CNC machine, which is now a days very popular consists of following features

1. Machine Structure
2. Drives
3. Actuation system
4. Slideways for machines
5. Automatic Tool Changer
6. Automatic Pallet changer
7. Transducers / Control system
8. Feed back devices

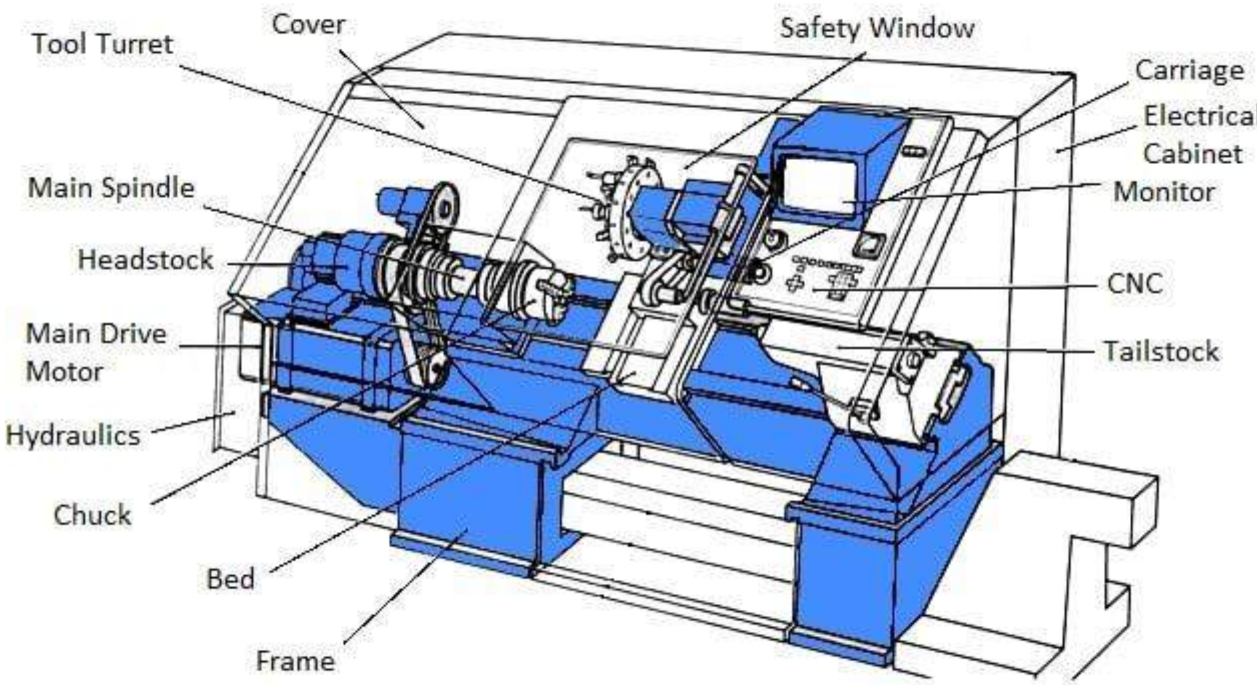
1. **Machine Structure**

The CNC machine tool structure consists of following main parts

a. Bed b. Table c. Column

a. Bed:

The bed of CNC machine is generally made of high quality cast iron with heavy ribbing to provide high stiffness and low weight. The cast rion structure provides the necessary damping, to reduce the vibrations produced due to high speed, large material removal rates and heavy duty machining. Another area of consideration is the design from chip disposal point of view. A slant bed structure is provided in turning centres which allows the chips to fall from the cutting zone. It also provides the operator easier and better access to the workpiece and tooling.



Drives

There are two basic applications where drives are used in CNC machines

1. Spindle drives 2. Feed drives

1. Spindle drives

Spindle drives are used to provide the main spindle for cutting. As large material removal rates are used in CNC, large power motors are used for spindle drives. Also, the speed required during operations is infinitely variable. Hence to provide such a speed, control for infinitely variable speed DC motors are used. The speed control for DC motors can be achieved by varying the voltage infinitely. The use of AC motors are preffered in the generation of currents in CNC machine tools. This is achived by development in the frequency converter.

2. Feed Drives

CNC machines are provided with independent axis drive to provide the feed movements for the slides. In order to obtain fast response and positional accuracy a special type of motor called servomotor is used to power the slides. Following are the feed drives that are used in the CNC machine tools

a. DC servomotors
b. Brushless DC servomotors
c. AC servomotors
d. Stepper motor
e. Linear motor

a. DC Servomotors

The force that rotates the motor armature is the result of the interaction between two magnetic fields (the stator field and the armature field). To produce a constant torque from the motor, these two fields must remain constant in magnitude and in relative orientation. This is achieved by constructing the armature as a series of small sections connected in sequence to the segments of a commutator.

Electrical connection is made to the commutator by means of two brushes. As successive commutator segments pass the brushes, the current in the coils connected to those segments changes direction. This commutation or switching effect results in a current flow in the armature that occupies a fixed position in space, independent of the armature rotation, and allows the armature to be regarded as a wound core. with an axis of magnetization fixed in space. This gives rise to the production of a constant torque output from the motor shaft. The axis of magnetization is determined by the position of the brushes. If the motor is to have similar characteristics in both directions of rotation, the brush axis must be positioned to produce an axis of magnetization that is at 90° to the stator field.

DC servomotors are high performance motors and are useful as prime movers in numerically controlled machine tools where starts and stops must be made quickly and accurately. The lightweight and low inertia armatures of DC servomotors respond quickly to the excitation voltage changes. Also low armature inductance in these motors results in a low electrical time constant (typically 0.05 to 1.5 ms) that further sharpens motor response to command signals.

b. Brushless DC servomotors

In the brush less motor, the construction of the iron-cored motor is turned inside out, so that the rotor becomes a permanent magnet and the stator becomes a wound iron core. The permanent magnet, located on the rotor, requires that the flux created by the current carrying conductors in the stator rotate around the inside of the stator in order to achieve motor action. The stator windings are interconnected so that introducing a three-phase excitation voltage to the three-stator windings produces a rotating magnetic field. This construction speeds heat dissipation and reduces rotor inertia. The permanent magnet poles on the rotor are attracted to the rotating poles of the opposite magnetic polarity in the stator creating torque. The magnetic field in the stator rotates at a speed proportional to the frequency of the applied voltage and the number of poles. In the brush less motor, the flux of the current carrying winding rotates with respect to the stator; but, like the dc motor, the current carrying flux stays in position with respect to the field flux that rotates with the rotor. The major difference is that the brush less motor maintains position by electrical commutation, rather than mechanical commutation.

C. AC servomotor

These are basically the AC synchronous motors with built-in brush less tacho and position encoders. The main advantage of this machine is the low rotor inertia and high power and low weight. This makes them very attractive since they are small in size compared to the equivalent DC servo motor.

In an AC servomotor, the rotor is a permanent magnet while the stator is equipped with 3-phase windings. The speed of the rotor is equal to the rotational frequency of the magnetic field of the stator, which is regulated by the frequency converter.

D. Stepper Motor

A stepper motor is a device that converts the electrical pulses into discrete mechanical rotational motions of the motor shaft. A stepper motor rotates (steps) in fixed angular increments. Step size, or step angle, is determined by the construction of the motor and the type of drive scheme used to control it. Typical step resolution is 1.8 degrees. However, micro-step motors are capable of 0.0144 degree steps. Micro-step motors are hybrid 200 steps per revolution motors that are electrically controlled to produce 25000 steps per revolution.

Stepper motors are usually used in open loop control systems, though an encoder may be used to confirm positional accuracy. There are many types of step-motor construction. However, **permanent magnet (PM) and variable reluctance (VR)** are the most common types.

Permanent magnet (PM) steppermotor

Rotor is a permanent magnet.

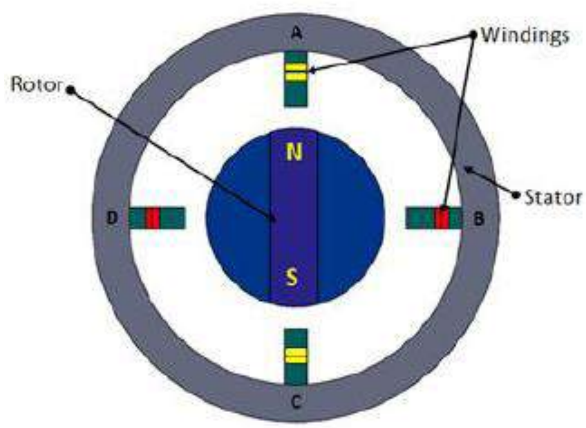
PM motor rotor has no teeth and is designed to be magnetized at a right angle to its axis.

Figure shows a simple, 90^0 PM motor with four phases (A-D).

Applying current to each phase in sequence will cause the rotor to rotate by adjusting to the changing magnetic fields.

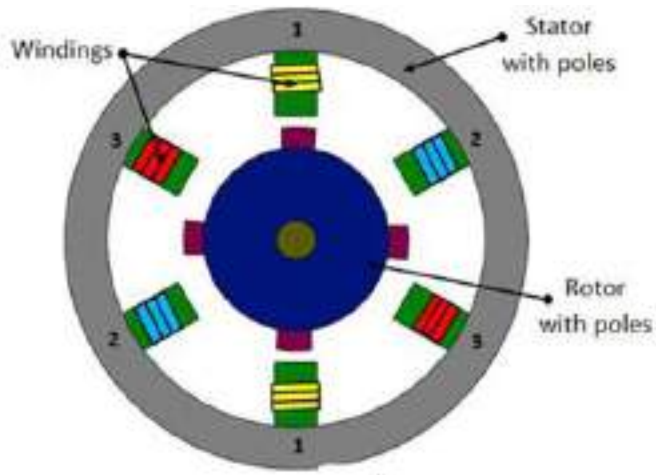
Although it operates at fairly low speed, the PM motor has a relatively high torque characteristic.

These are low cost motors with typical step angle ranging between 7.5^0to15^0



Permanent magnet stepper

Variable Reluctance Motor



The cylindrical rotor is made of soft steel and has four poles

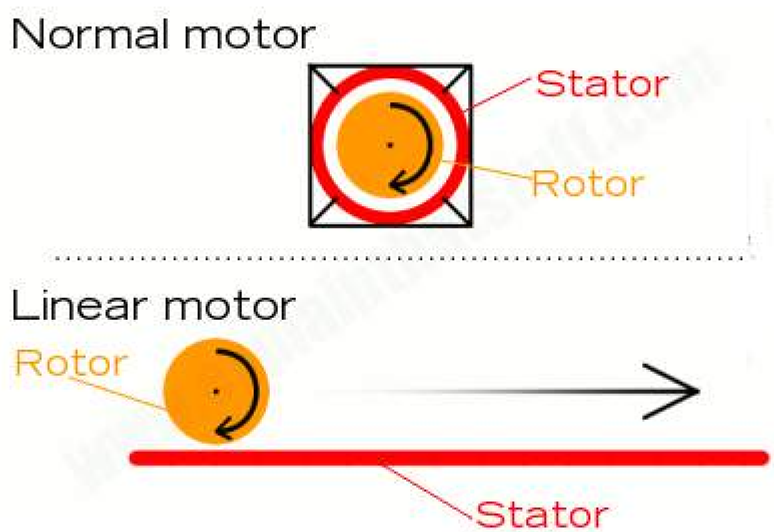
It has four rotor teeth, 90^0 apart and six stator poles, 60^0 apart.

Electromagnetic field is produced by activating the stator coils in sequence.

It attracts the metal rotor. When the windings are energized in a reoccurring sequence of 2,3,1, and so on, the motor will rotate in a 30^0 step angle.

In the non-energized condition, there is no magnetic flux in the air gap, as the stator is an electro magnet and the rotor is a piece of soft iron; hence, there is no detent torque.

Linear Motor



In a traditional DC electric motor, a central core of tightly wrapped magnetic material (known as the rotor) spins at high speed between the fixed poles of a magnet (known as the stator) when an electric current is applied. In an AC induction motor, electromagnets positioned around the edge of the motor are used to generate a rotating magnetic field in the central space between them. This "induces" (produces) electric currents in a rotor, causing it to spin. In an electric car, DC or AC motors like these are used to drive gears and wheels and convert rotational motion into motion in a straight line.

A linear motor is effectively an AC induction motor that has been cut open and unwrapped. The "stator" is laid out in the form of a track of flat coils made from aluminum or copper and is known as the "primary" of a linear motor. The "rotor" takes the form of a moving platform known as the "secondary." When the current is switched on, the secondary glides past the primary supported and propelled by a magnetic field.

Linear motors have a number of advantages over ordinary motors. Most obviously, there are no moving parts to go wrong. As the platform rides above the track on a cushion of air, there is no loss of energy to friction or vibration (but because the air-gap is greater in a linear motor, more power is required and the efficiency is lower). The lack of an intermediate gearbox to convert rotational motion into straight-line motion saves energy. Finally, as both acceleration and braking are achieved through electromagnetism, linear motors are much quieter than ordinary motors.

Linear motors are now used in all sorts of machines that require linear (as opposed to rotational) motion, including overhead traveling cranes and beltless conveyors for moving sheet metal. They are probably best known as the source of motive power in the latest generation of high-speed "maglev" (magnetic levitation) trains, which promise safe travel at very high speeds but are expensive and incompatible with existing railroads. Most research on maglev trains has been carried out in Japan and Germany.

3. ACTUATION SYSTEM

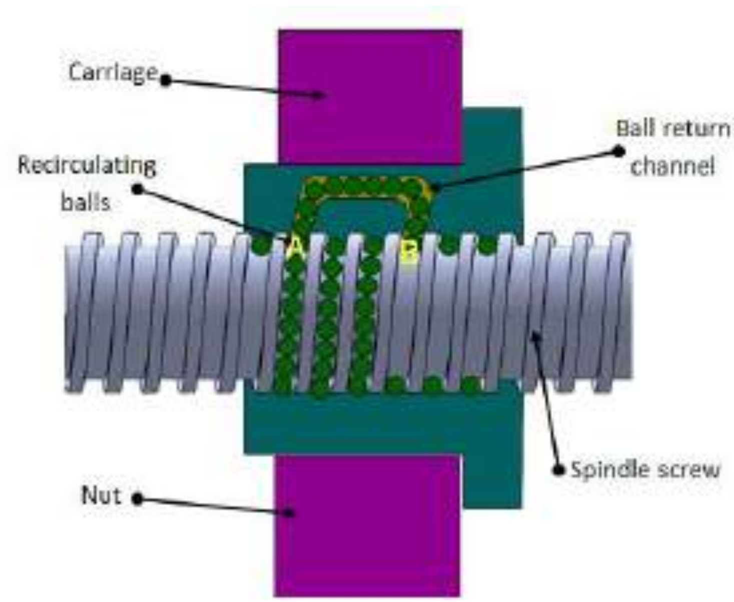
Recirculating ball screw

A recirculating ball screw consists a screw, a nut and a series of balls. The main difference between a recirculating ball screw and a power screw is that in former the sliding friction gets replaced by rolling friction. Since rolling friction is very less than sliding friction recirculating ball screw has very high efficiency of power transfer.

The threads of screw and nut in recirculating ball screw are semicircular so that they can accommodate rolling balls. Recirculating ball screw is also known as ball bearing *screw* or simply *ball screw*. There is almost no heat generation in recirculating ball screw due to negligible friction and they can be used at very high speeds up to 10 m/min.

It consists of a screw spindle, a nut, balls and integrated ball return mechanism as shown in Figure. The flanged nut is attached to the moving part of CNC machine tool. As the screw rotates, the nut translates the moving part along the guideways.

However, since the groove in the ball screw is helical, its steel balls roll along the helical groove, and, then, they may go out of the ball nut unless they are arrested at a certain spot.



Thus, it is necessary to change their path after they have reached a certain spot by guiding them, one after another, back to their “starting point” (formation of are circulation path). The recirculation parts play that role. When the screw shaft is rotating, as shown in Figure, a steel ball at point (A) travels 3 turns of screw groove, rolling along the grooves of the screw shaft and the ball nut, and eventually reaches point (B). Then, the ball is forced to change its path way at the tip of the tube, passing back through the tube, until it finally returns to point (A). Whenever the nut strokes on the screw shaft, the balls repeat the same recirculation inside the return tube.

When debris or foreign matter enter the inside of the nut, it could affect smoothness in operation or cause premature wearing, either of which could adversely affect the ballscrew's functions. To prevent such things from occurring, seals are provided to keep contaminants out.

There are various types of seals viz. plastic seal or brush type of seal used in ball-screw drives.

Advantages of recirculating ball screw

Compared to power screw they offer following advantages.

- They have very high efficiency (approx. 90%), compared to power screw (approx. 40%)
- No stick and slip phenomenon which results in durability
- It is virtually wear free
- They require Less starting torque
- No heat generation during operation
- They can be used for high speed operations
- They can be easily preloaded to eliminate backlash
- They have very smooth and noiseless operation
- They have high reliability and durability
- Their load carrying capacity is more than power screw of the same size

Disadvantages of recirculating ball screw

Compared to power screw they offer following disadvantages.

- They are much more costly
- Buckling of screw is serious problem at critical speed
- They require high degree of cleanliness
- They require thin film lubrication for satisfactory operation
- They can have vibrations
- They require periodic overhauling to maintain their efficiency
- They do not have self-locking properties

Applications of recirculating ball screw

- Machine tools controls
- Used in robotics where precise movement is vital
- Hospital beds adjusters

- Aircraft landing gear retractors
- X-Y recorders of CNC machines
- Power actuators
- Automobile steering gears

COMPARISON OF CONVENTIONAL SCREW AND RECIRCULATING BALL SCREW

s.No	Parameter	Conventional power screw	Recirculating ball screw
	Advantages of recirculating ball screw		
1	Efficiency	There is sliding friction, hence high input torque is required to overcome friction. Thus efficiency of the screw is as low as 40%	There is rolling friction, hence low input torque is required to overcome friction. Thus efficiency of screw is as high as 90%
2	Load carrying capacity	It has lower load carrying capacity as compared to recirculating ball screw	It has high load carrying capacity as compared to conventional power screw. For the same load carrying capacity, recirculating ball screw is more compact and light weight
3.	Stick slip phenomenon	In conventional power screw, stick-slip phenomenon is observed. This is due to difference between the value of coefficient static friction and coefficient of sliding friction	The operation of recirculating ball screw is smooth and free from any stick slip phenomenon.
4.	Compensation for wear and tear	It requires periodic adjustment to compensate for wear on the surfaces of the screw and nut	It is virtually wear free due to presence of lubricating film between the contacting surfaces
	Disadvantages of recirculating Ball screw		
5.	Cost	The initial cost is low	The initial cost of recirculating ball screw is very high
6.	Special operating	It can be operated in any	It requires high degree of

	environment	environment with satisfactory life	cleanliness and restricted entry of foreign particles
7.	Lubrication	It can be easily lubricated by grease	It requires a continuous thin film of lubricant between the balls and grooves in the nut and screw
8.	Self locking and over hauling	Due to high friction between thread surfaces, these screws are self locking	Due to negligible friction between balls and thread surfaces, these screws are over-hauling. Hence special brake is required to hold the load in its place.

4. Slideways for machine

Precise positioning and repeatability of machine tool slides are the major functional requirements of CNC machine. To eliminate stick slip, there are different slideway systems such as rolling friction slideway and slideways with low friction Poly Tetra Fluoro Ethylene (PTFE). These slides have low wear, good vibration damping, easy machinability, low coefficient of friction and low price. The plastic coated slideway have static coefficient of friction, which is less than dynamic coefficient of friction. With increase in speed, dynamic coefficient of friction increases upto a certain value and then remains constant.

Following are the techniques used to meet requirements in CNC machine tool slideways

1. Hydrostatic slideways 2. Linear bearings with balls 3. Rollers or needles 4. Surface coatings

1. Hydrostatic slideways

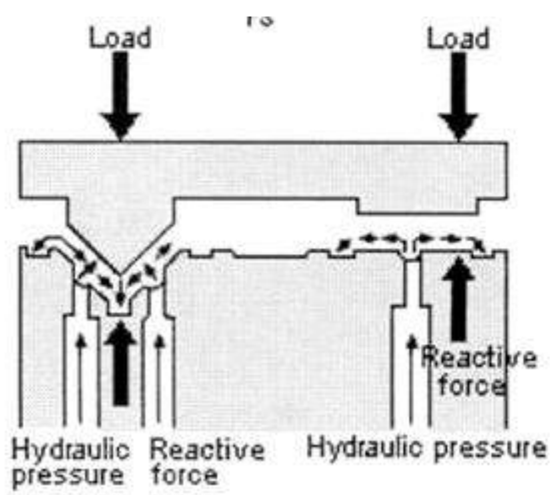


Figure 3.1 Hydrostatic bearing

Hydrostatic bearings utilize a pressurized fluid to keep the bearing "afloat" and not in contact with its lower channel. The main advantage of these types of bearings is their ability to handle very heavy loads without generating the friction (and heat) that a contacting slide would incur. **Figure 3-1** displays a typical set-up for a hydrostatic bearing:

It is interesting to note that one channel is in a "V" configuration and the other side of the bearing is a flat configuration. This is one of the most common forms of hydrostatic bearings. The "V" trough ensures pressurized fluid support in the vertical direction as well as in the lateral direction – keeping the bearing aligned without side contact.

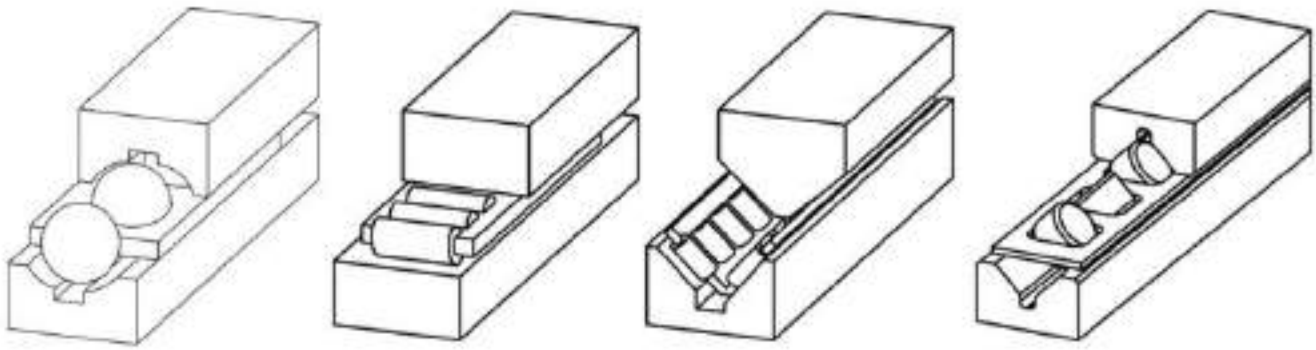
A rather new development in hydrostatic bearings is the use of water as the bearing fluid. This replaces certain oils used previously, and gives the user an advantage in the fact that they no long have to handle what is often called a hazardous chemical – hydraulic fluid. The water bearings eliminate the need for special disposal of the bearing fluid.

Linear bearings with balls

Linear ball and roller bearings can be broadly divided into two categories—recirculating and non-recirculating—depending on whether or not the rolling elements actually flow (or circulate) through the bearing housing. In order to understand what a recirculating linear bearing is, let’s first take a look at bearings that don’t recirculate.

Non-recirculating linear bearings

Non-recirculating bearings have balls or rollers that are contained in a housing and directly support a load. As the bearing moves, the rolling elements rotate about their own axes, but they do not travel within the housing. Although their basic construction principle is the same, there are several types of non-recirculating bearings, based on the type and arrangement of their rolling elements, as shown below.

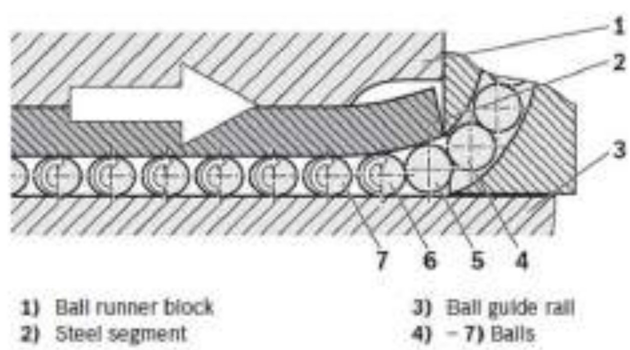


Left to right: non-recirculating ball bearings, flat-type roller bearings, v-type roller bearings, and crossed-roller bearings

Regardless of their different rolling mechanisms and designs, all non-recirculating bearings have a few things in common. First, the length of the bearing and the number of rolling elements limits the stroke that can be achieved. Second, because their rolling elements only rotate (no recirculation), they provide extremely smooth motion. And with machined top and bottom surfaces, they can have extremely high travel accuracy. Non-recirculating bearings are often the guide system of choice for high-precision stages and are commonly used in machine tool, precision scanning, and measuring applications.

Recirculating linear bearings

“Circulate” means “to move in a circle or circuit; to move or pass through a circuit back to the starting point.” And the term “recirculate” means to do this over and over again. Hence, recirculating bearings have rolling elements that move continuously through a circuit, or path, within the bearing. This design allows the bearing to travel any distance, regardless of the bearing length. In other words, where non-recirculating bearings have limited travel, in theory, recirculating bearings have unlimited travel, constrained only by the length of the rail or shaft guideway.



As the balls or rollers circulate through the bearing, they move from a non load-carrying zone (sometimes referred to as the return zone) to a load-carrying zone. This variation of the balls (or rollers) from a non-loaded to a loaded state causes pulsations, which affect the bearing's travel accuracy. Improving the smoothness of the circulation process has been a priority for manufacturers in recent years, with new designs for the recirculation zone yielding improved travel accuracies.

Recirculation also limits the maximum speed that the bearing can achieve, due to the forces created on the bearing end caps when the recirculating elements make the "turn" around the circuit. One way to reduce these forces is to reduce the mass of the balls. Some manufacturers have done just that by offering profiled rail bearings with ceramic balls. These versions can achieve speeds up to 10 m/s, but with a somewhat reduced load capacity.

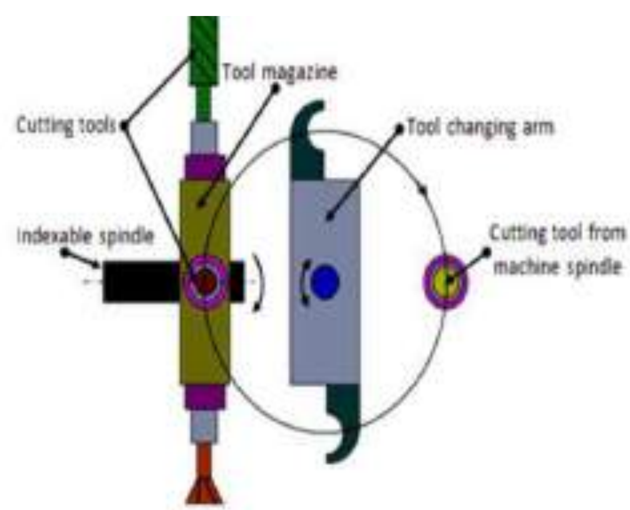
Although recirculating bearings generally have lower travel accuracies than non-recirculating types, they come in a wide range of sizes, preloads, and accuracy classes, making it easy to find a recirculating linear bearing that fits just about any application requirement.

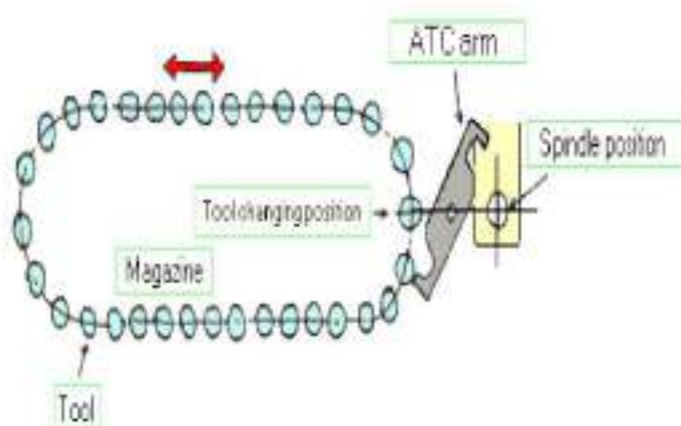
Surface Coatings

In this technique, the guidning surface is coated with low friction material such as poly tetra fluro ethylene (PTFE). Sometimes, replaceable strips of low friction material are used on guide ways

AUTOMATIC TOOL CHANGER

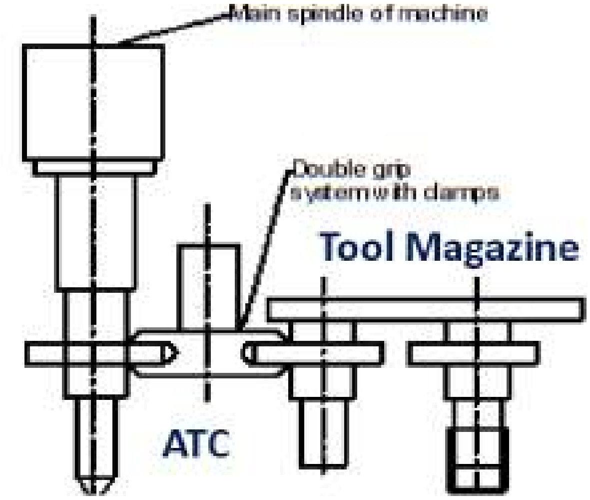
The device used to pick up a tool from Tool magazine and replace it with the other tool in the spindle within 3 to 7 seconds is called ATC so as to reduce the idle time during tool changing operations



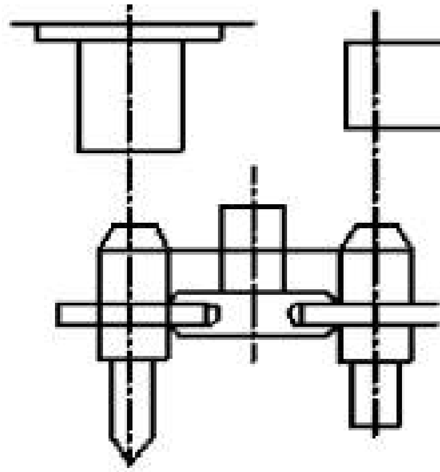


1. ATC arm Rotates through 90º from the Rest position

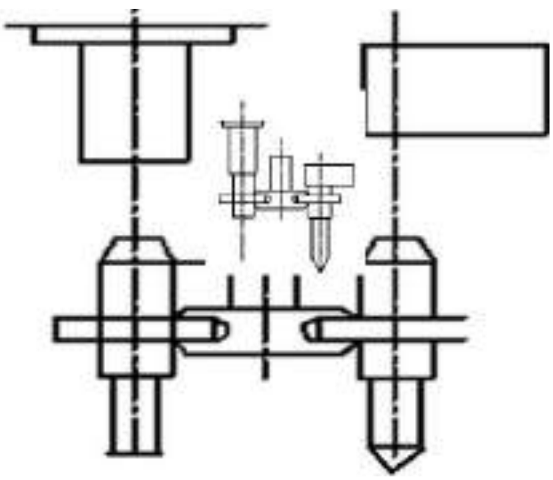
2. One end grips tool in the magazine & another end grips tool in the spindle
3. Both tools are pulled out
4. ATC arm rotates through 180º
5. It inserts new tool to the spindle & old tool to the magazine
6. ATC arm Rotates through 90º & goes to the Rest position



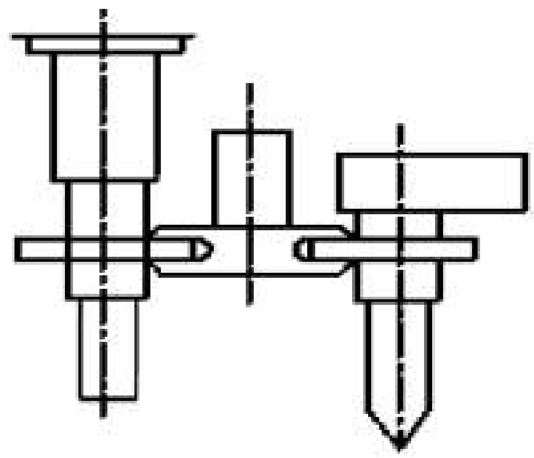
Arm comes from the rest position & Grips both Tools



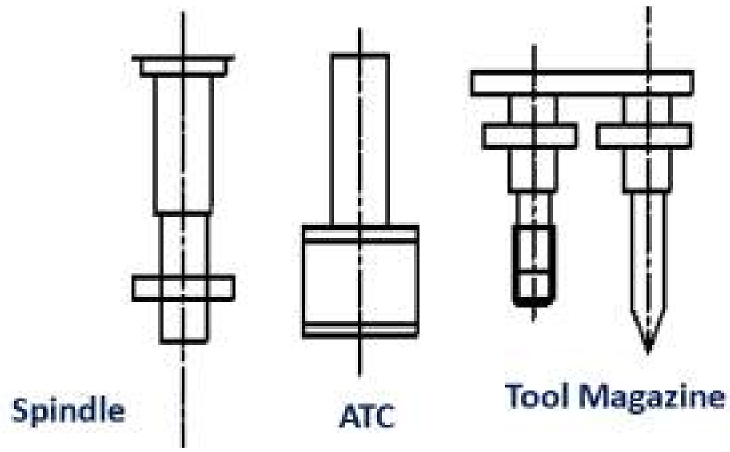
Arm pull the tools from the Mounting



Arm rotates through 180º



Carrier Inserts tools



Arm rotates & go to rest position

Tool Magazine

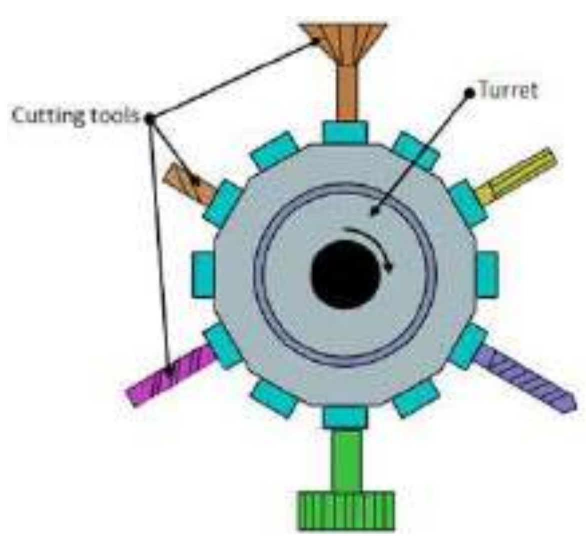
The system of arrangement which **holds large number of tools** is called Tool Magazine. It is specified by Storage Capacity & Shape. Storage capacity ranges from **12 – 200 tools.** Used in CNC turning centre & CNC machining centre

Tool Magazine Types

a. Turret type
b. Drum / Disc type
c. Chain type

Turret type tool Magazine

- It is the **Simplest form** of tool magazine
- It consists of tool storage **without tool changer**
- Turret is **indexed in required position** for desired operation
- Tool can be **easily identified**
- Tool **change time is more**
- The required turret station is specified on the program; hence the turret indexes from one tool station to the next desired station in accordance with the input data.



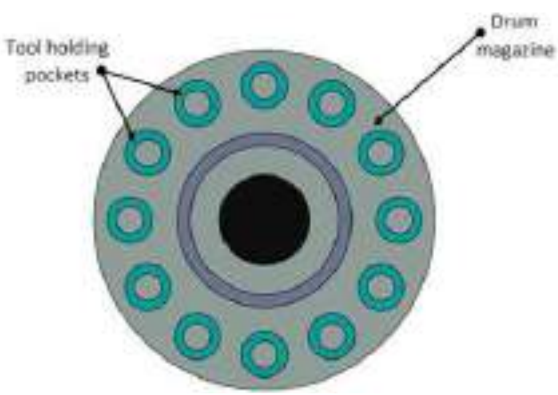
d. Drum / Disc type tool Magazine

Disc magazines are commonly used on small vertical machining machines. The “disc magazine” is commonly known as the “drum magazine” to distinguish it from the “bucket magazine” and the “chain magazine”. The disc-shaped magazine has a small capacity, with at most two or thirty knives. Tool exchange is required with ATC (Auto Tools Change). Two types of drum magazine are used in CNC machines

- The first is without a tool catcher and also called as **Carousel Type Tool Magazine**

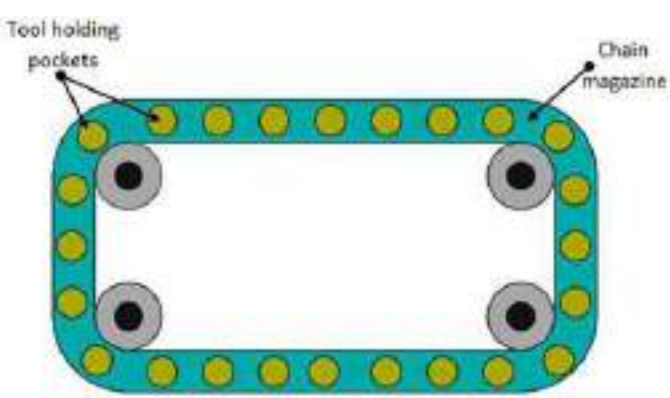
  Generally, only 16 to 24 tools can be stored, and the entire magazine moves to the spindle when the tool change is performed. When the tool on the spindle enters the slot of the magazine, the spindle moves up and out of the tool, at which point the magazine rotates. When the tool to be changed is directly below the positive spindle, the spindle moves down, so that the tool enters the spindle taper hole. After the tool is clamped, the magazine returns to its original position.

- **Second type with a special mechanism (tool catcher),** is employed to select the correct tool and transfer it to the machine spindle, where it is automatically clamped in position.



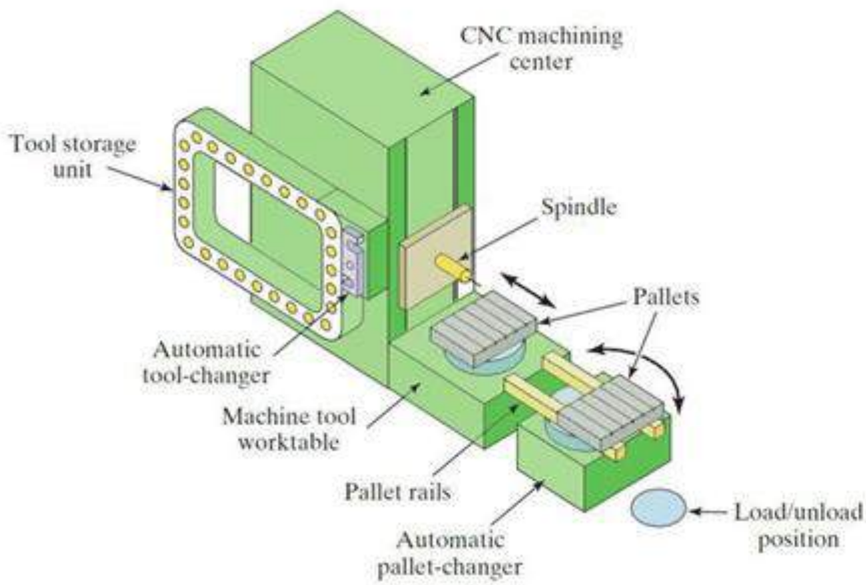
Chain type magazine

- It is the Capable of storing more number of tools
- Used in machining centre
- Tools are inserted into their pockets which are attached to the chain
- Chain moving on sprocket & sprockets are driven by motors



AUTOMATIC PALLET CHANGER (APC)

A pallet is a movable & interchangeable part of a machine tool which helps to transport raw or finished parts from the machine in order reduce downtime for part loading/unloading.



There are three main types of pallet changers in most CNC machine tools – manual, automated and robotic.

- **Manual pallet systems:**

The operator has to physically load and unload the pallet, with the raw / finished part, to / from a docking station located & mounted on top of the machine table. Of the three options, this is the most laborious and least automated version. However, you can still achieve a reduction of downtime with this system.

- **Automated pallet systems:**

These are normally built-into the machine and operated automatically by the machine itself. The human intervention comes from the loading and unloading of parts at the pallet station.

- **Robotic pallet systems:**

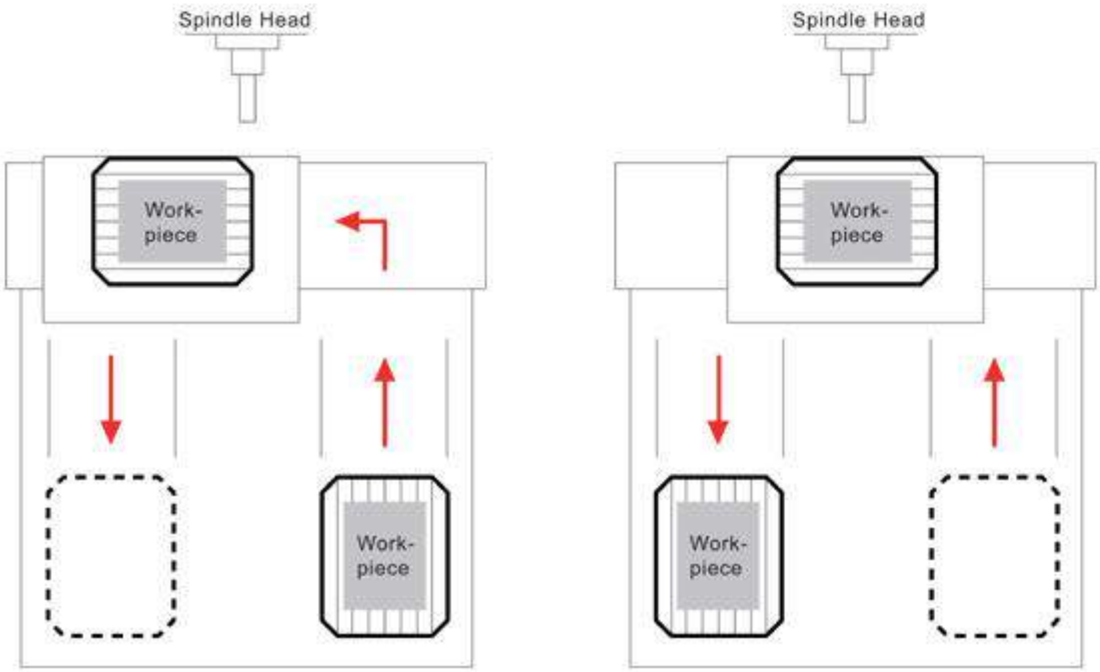
These usually utilize a robot to load and unload pallets. The robot is often able to serve one to multiple machines. However, these are fairly costly and may not be pragmatic in cost-sensitive markets. Over time, the state of constantly improving technology may make such systems increasingly cost-effective. Such systems are often sought in locations with high labour costs or insufficient skilled manpower. Difficulties for such systems may arise whenever customized and very specific manufacturing needs are required.

According to logical movement of pallet, the system can be **linear or rotary**

Linear Pallet Changer system

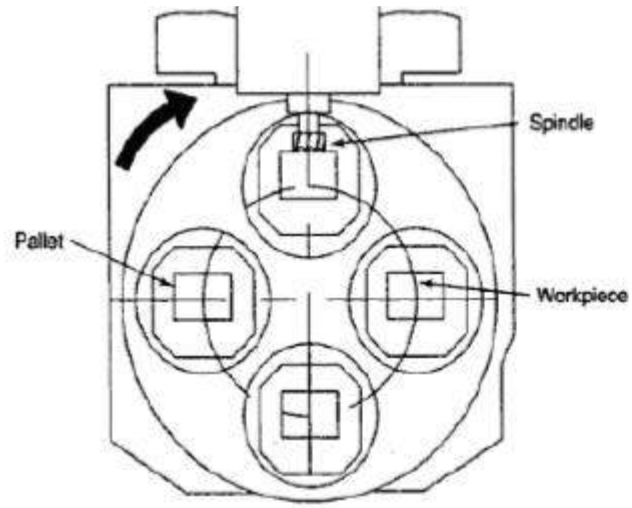
A typical linear pallet changer system is shown in figure.

In this system, the table moves in a linear motion, hence called as linear pallet changer system. In fig, the workpiece is on right side track is waiting for completion of machining operation of earlier workpiece. After completion of earlier workpiece, the finished workpiece moves onto the unloading table and the next component is ready to move into the machining table. In figure, the next component moves onto the machining table and the system continuous



Rotary pallet changer system

Rotary pallet changer system is shown in Figure, which is same as linear pallet changer system except that the table is rotated for the movement of the workpieces.



Transducers / Control Elements

In CNC applications, a transducer measures physical motion, then converts that measurement to an electrical input/output. There are different applications of this technology, but the prevalent one is using the transducer in a closed loop environment. This insures that the actual movement of the CNC machine as commanded by the control, then encoded to the motor for that particular axis is, in effect, actually occurring. If the system was not closed loop and had no means to verify movement, potential problems that could occur would be:

1.) CNC control sends signal to encoder to count the revolutions of the axial motor and sends signal to the axial motor telling it to revolve, resulting in linear axial movement across the machine.

2.) The machine momentarily strikes a rigid portion of the machine or fixturing on the machine, momentarily causing the machine to physically pause its linear movement.

3.) Because the motor is designed to "clutch slip" in this case to prevent damage to the motor itself, the motor shaft at the encoder end keeps revolving and the encoder keeps counting.

4.) At the end of its movement the control thinks the correct revolutions of the motor have been executed, therefore assuming the axis has moved linearly correctly also. However, the actual linear physical movement is inaccurate because of the physical pause which occurred.

If a transducer of some type were added to this scenario, now all of a sudden a signal is sent back to the control verifying that the actual physical movement the control thought had occurred, in fact has not. An error would be generated within the control to prevent further movement until the condition is corrected.

Feedback devices

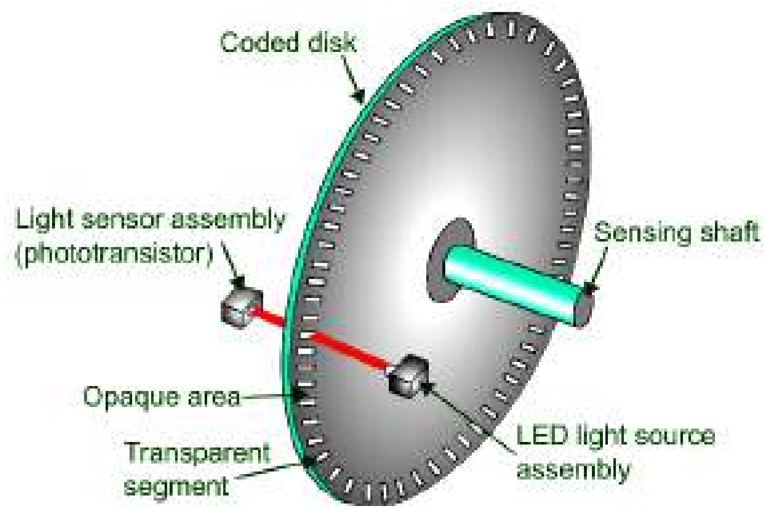
The NC machine tools generally are run with a closed loop control system. For this purpose it is necessary to provide appropriate feedback in order to achieve accurate control of the movement of the axes. The feedbacks that are normally used are the displacement and velocities of the individual axes in the machine tool. The typical positional sensors used in the NC machine tools are :

(a) Encoders, and

(b) Linear scales.

a. Encoders

The encoder is a transducer that is connected directly to the rotor or the lead screw and hence is the simplest arrangement requiring no additional gearing. An optical rotary encoder converts the rotary motion of the motor into a sequence of digital pulses. The pulses counted to convert to the position measurement. The optical encoder consists of a disc with a number of accurately etched equidistant lines or slots along the periphery. The encoder disc is attached to the shaft of the machine whose rotary position needs to be measured. The disc is placed between a light source and a light-measuring device. When the disc rotates the lines are interrupted and the light-measuring device counts the number of times the light is interrupted. By a careful counting and appropriate calculations it is possible to know the position traversed by the shaft.



b. Linear Encoder

Rotary encoder requires the pitch of the lead screw to be more accurate over its entire length for accurate sensing of the position. To obviate such a problem, it will be better if the exact position reached by the slide can be measured by means of a transducer rather than the indirect way with the encoder. This can be done with the help of a linear scale attached directly to the slide. In this case the positional measurement will be direct and hence any of the inaccuracies present will not be affecting the measurement. The linear scale consists of a finely graduated grating made of either glass or stainless steel, which is the measuring surface attached to one part of the slide. A scanning unit is fixed to the other part. The scanning unit consists of a light source, a glass grid with graduated windows and some photo diodes as receptors. The basic principle employed in such measurements is that when two gratings overlap each other, a fringe pattern is formed corresponding to the displacement. The actual distance moved can be calculated by measuring the shift in the fringe pattern.

1.6 Machining centers

Machining centers are very important type of CNC machine tools and are multi-function machines equipped with automatic tool changes and are capable of carrying out milling, drilling, reaming, tapping, boring, counter boring and allied operations without operator intervention for change of tools. Tool changing is carried out using an automatic tool changer and is accomplished in 4-6 seconds**.**

1.6.1 Step-wise Approaches to CNC Machine Center Development

- Development of CNC machine centers begins with vertical spindle configuration includes 3-axis. Two axes for table and third axis for the spindle head.
- Further turret machining center is developed for auto tool selection by indexing the required tool without operator during the machining and manual tool change is completely removed.
- It has developed by the introduction of a tool library or Tool magazine, where which we can store large number of tools in tool magazine with an auto tool changer for automatically retrieve tool from the tool magazine to the spindle and vice versa.
- Pertaining to a prism type job in a single setup, the concept of the horizontal machining centre developed and increased with an index or rotary table to orient the work piece for machining on various faces.
- Introduction of auto pallet changes, the idle time of machine like setting the job. Setting the tool, setting table has been fully eliminated.

1.6.2 Features of Machining Centers

- Higher spindle speed 10,000-30,000 rpm.
- Higher rapid traverse 40 to 60 meters / minute.
- Quick and faster tool changing system.
- It has digital servo control system of main spindle for accuracy.

- It has very efficient thermal control system on various machine elements like main spindle, balls and ball screws and ball bearings for accuracy

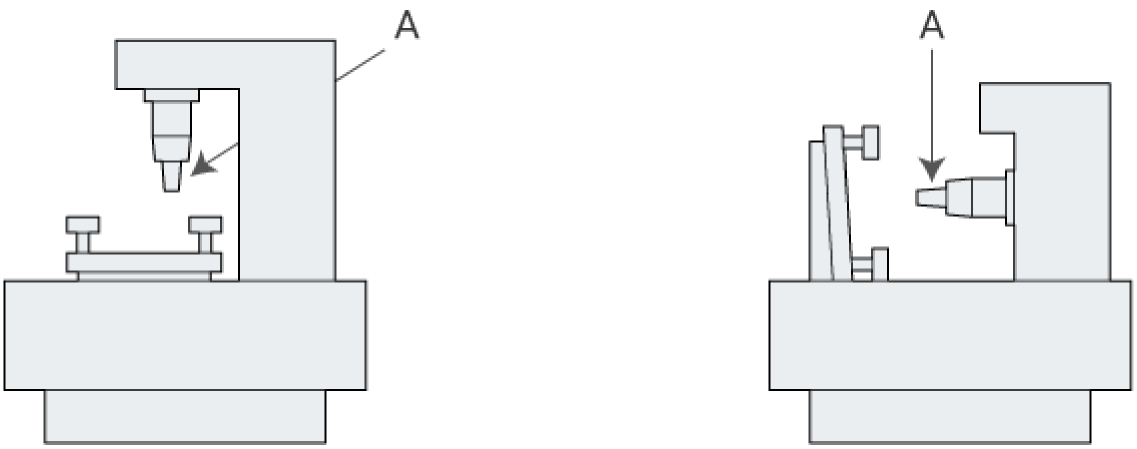
1.6.3 General Structure and Types of Machining Centers

Machining centers can be broadly classified into three types based on their structure: horizontal, vertical, and gantry types. The horizontal type—the first to be developed—can be defined simply as a machine where the spindle to which the cutting tool is attached is mounted horizontally (or parallel to the floor). In contrast, vertical types have the spindle set upright. Gantry types, on the other hand, have a gate-like structure with the spindle mounted on the ceiling of the gate, facing downward.

Using the horizontal type as an example, the general structure of a machining center consists of a base part called a bed at the bottom, a saddle that moves on the bed, a table attached on top of the saddle for placing the raw material, a column installed perpendicularly to the bed, and a spindle head where the cutting tools are attached.

1.6.4 Difference between Horizontal and Vertical Machining center

Horizontal machining centers have a blade-mounted spindle that comes out sideways, which machines workpieces in the horizontal direction. The column moves along the X axis, the saddle along the Y axis, and table along the Z axis, and this combination enables three-dimensional machining. Additionally, some models have a B axis that rotates the table horizontally, making it possible to machine materials using a total of four axes. One advantage of horizontal types is the capability to machine four surfaces of a workpiece—when using a four-axis machining center with B axis—all at once. This eliminates the need for operators to manually switch the four sides of the workpiece, and thus also



contributes to higher machining precision. Moreover, machining from the horizontal direction allows chips to fall down, which helps prevent the chips from accumulating on the workpiece and digging into the blade.

Conversely, vertical machining centers have the spindle in a vertical position, and workpieces are machined from above. Generally, the table travels horizontally on the X and Y axes, and the spindle moves vertically, enabling triaxial machining.

Compared with horizontal types where the spindle is situated to the side of the workpiece, vertical types take up less installation space, making them a popular choice. In addition, machining from above the workpiece allows operators to work while comparing the machining to the design drawings. However, machining on the top of the workpiece causes chips to accumulate on the workpiece, creating the need for a blower that uses compressed air, or rinsing with lubricant to remove the chips appropriately.

1.6.5 Components of machining Centers

VMCs all use a commonality of components, which are as follows:

1. **Rotating Spindle** — The Spindle, which is perpendicular to the working surface or table, can hold a variety of cutting tools (or mills as they are sometimes called). The spindle cartridge is mounted in a housing that moved up and down—this direction of motion is called the Z-Axis.
2. **Table** — The Table is a platform on which to mount workpieces—either directly or through a variety of fixtures like milled aluminum plates or hard clamping vises. The table has a motion of left and right, which we call the X-Axis, and front to back, which is called the Y-Axis. These two axes of motion, coupled with the Z-Axis, allow for virtually unlimited contouring across the planes of motion.
3. **Tool Changer** — A tool changer greatly increases a VMC's productivity by allowing for the automatic, computer-controlled selection of tools for a variety of tasks from rough cutting to fine hole boring.
4. **Coolant System** — To keep parts and cutters cool and lubricated, most VMCs employ some sort of recirculating coolant system; which is usually a mix of soluble oil and water but can be a variety of other liquids as well.
5. **Chip/Screw Conveyor—** To evacuate the chips from the work zone, a variety of chip conveyors and chip augers may be employed to increase productivity and reduce downtime from manually shoveling out waste chips.
6. **Full Covers/Enclosures—** These can be added to reduce the splash/splatter caused by milling operations and protect the operators and environment from the machining process.
7. **Rotating Tables** — Adding additional Axes to the machine can greatly increase its productivity by turning a simple three-axis machine into a four- or even five-axis system capable of machining complex components with varying surfaces (such as turbine blades).
8. **Quick Loading Loaders** — Another great addition to significantly increase productivity is the use of shuttle tables or other automatic part loading systems. These can decrease downtime and greatly increase the spindle “ON” time of most VMC systems.

1.7 CNC Turning centers

Lathe machines with additional options such as Y-axis, sub-spindles, or specific selected options for automation are commonly called Turning Centres. These sophisticated machine tools are capable of machining complex parts – these go beyond standard OD & ID turning operations and may incorporate milling, drilling and tapping operations to complete the part in one setting.

Taking a piece from raw part to finished product, such all-in-one machine tools significantly improves productivity.

In general, CNC Lathe machines comes in the following main configurations:

- Horizontal
- Vertical
- Slant Bed
- Flat Bed
- Standard
- Multi-axis
- Other types, e.g. Swiss Type, Multi-spindle, B-axis

Depending on their configuration, CNC Lathes may have the following components.

Machine Bed

The bed of a CNC Lathe or CNC Turning Centre forms the main base for the whole machine. It is where the different components of the machine are mounted on. For example, the spindle stock which houses the machine's main spindle; tailstock body; X and Z axis slide; optional Y-axis; and sub-spindle.

Main Spindle

Often known as the heart of a machine tool, the main spindle consists of a spindle assembly and the spindle drive system. These are some of the moving parts of the CNC machine tool, and they include motors, gears, and the chuck (more on it later). Tool holders are normally installed at the turret disk (a rotating disk allowing tools to be positioned and switched), they include both fixed and life tool holders. The life tool drive system is built-in.

Chuck

A chuck is a vice-like structure that grips the workpieces to be machined. It is attached to the main spindle that rotates both the chuck and the workpiece.

Guide Way

The guide way enables the tool to move horizontally and vertically to achieve a smooth cutting process. To ensure durability, rigidity and the highest accuracy are needed for this structure.

Headstock

Comprising the main motor, the headstock holds the main spindle where the chuck is mounted on. To ensure high performance, you need to consider if your machine tool's headstock can handle high torque at low speed. This is an important consideration for tough materials.

Tailstock

Tailstock are used to provide an extra support force on the other end of the component. This is necessary when machining long and extra-long work pieces (e.g. Shafts).

Tool Turret

The turret provides the possibility to change the cutting tools required for machining. Hence it is mounted on the tool turret. The size of the turret is determined by the number and size of tools that will be mounted on it.

1.8 CNC PART PROGRAMMING

A Part Program is a list of coded instructions which describes how the designed component, or part, will be manufactured.

Program consists of

Dimensional data - the size and shape of the component

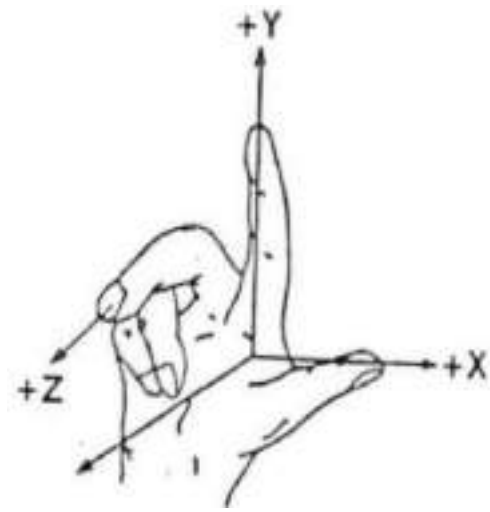
Technological data **-** Sequence of operations, Cutting speed, feed rate etc.

Miscellaneous data - Coolant ON/OFF, Spindle ON / OFF, Tool CLAMP /UNCLAMP etc.

1.8.1 Axis Designation in CNC

A program in CNC system, specify the various axis about which motion is required. For this purpose, a standard axis system is considered due to which relative tool position with respect to work must be obtained.

Machine axes are designated according to the "right hand rule“. When the thumb of right hand points in the direction of the positive X axis, the index finger points toward the positive Y axis, and the middle finger toward the positive Z axis. The main axes to be designated are the linear axes and the rotary axes.



The direction of each finger represents the positive direction of motion.

• *The axis of the main spindle is always Z*, and the positive direction is into the spindle.

• On a mill the longest travel slide is designated the X axis and is always perpendicular to the Z axis.

• *If you rotate your hand looking into your middle finger, the forefinger represents the Y axis.*

• The base of your fingers is the start point or (X0, Y0, Z0).

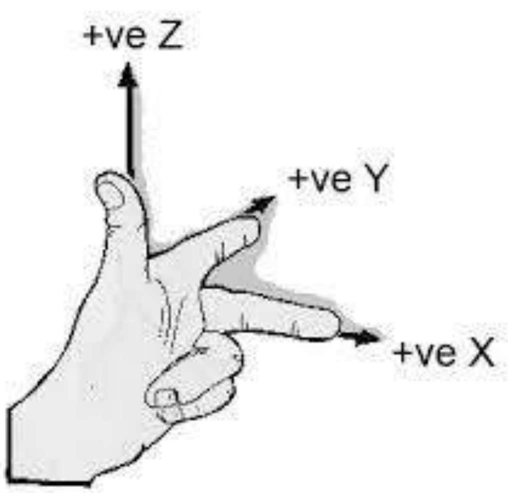
Rotary Motion

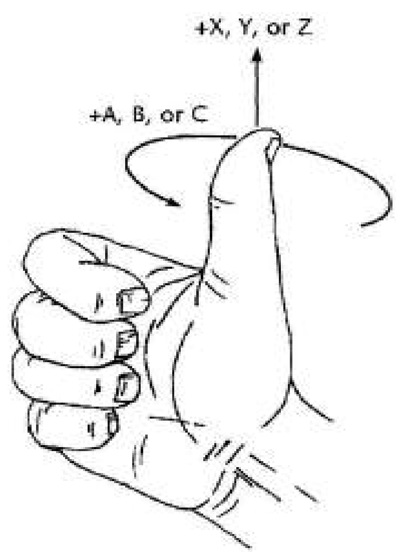
• The right-hand rule for determining the correct axis on a CNC machine may also be used to determine the clockwise rotary motion about X, Y,and Z.

• To determine the positive, or clockwise, direction about an axis, close your hand with the thumb pointing out.

- The thumb may represent the X, Y, or Z direction and the curl of the fingers may represent the clockwise, or positive, rotation about each axis.

- These are known as A, B, and C and represent the rotary motions about X, Y, and Z, respectively.





1. Z-axis

- Positive (+) Z direction increases the clearance between the cutting tool and the workpiece. For example, in a drilling machine the drill movement towards the workpiece is the negative Z direction. This helps in reducing the possible accidents because of wrong part program entry in the coordinate signs.
- The main spindle (axis of tool spindle or w/p rotates) is treated –Z axis.
- +ve direction away from the w/p and towards tool spindle holder.
- in case of multiple spindles –one spindle is selected as principal spindle & its axis as Z axis.

2. **X-axis**

- The X axis is the principle motion direction in the positioning plane of the cutting tool or the workpiece. It is perpendicular to the Z axis and should be horizontal and parallel to the work-holding surface wherever possible.
- Horizontal & parallel to the working surface.
- When Z -axis is horizontal: +ve X-axis to the right when looking from spindle towards the w/p
- When Z -axis is vertical: +ve X-axis towards right looking from the spindle towards supporting column.

3. Y-axis

- It is perpendicular to both X and Z axes and the direction is identified by the right hand Cartesian coordinate system.
- Perpendicular to both X-and Z -axis.
- For +ve Y direction rotate x axis advance right hand screw in +ve Z direction

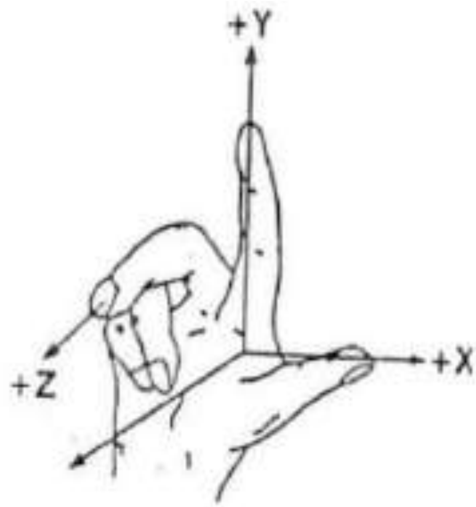


Figure — Axis Designation for Horizontal Z-axis

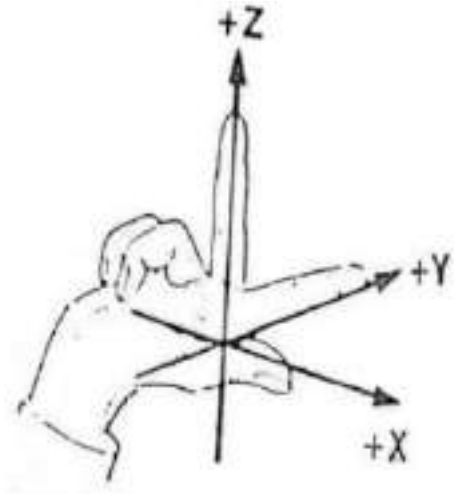
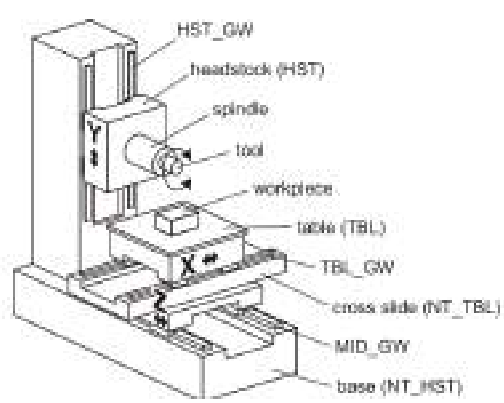
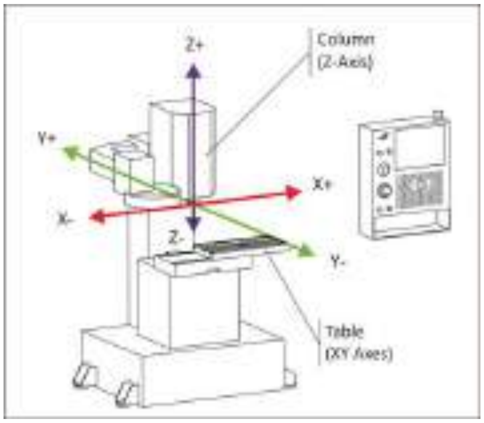


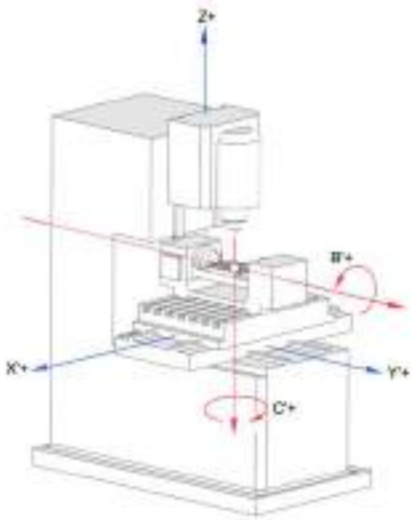
Figure — For Vertical M/c Z-axis



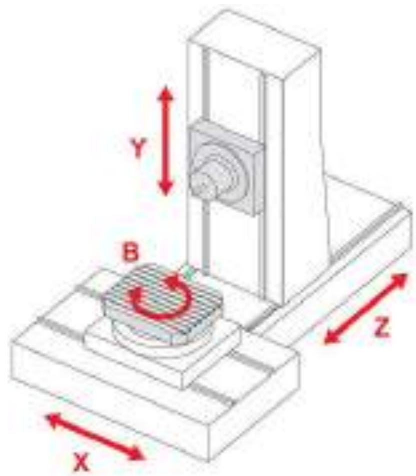


Axis designation in 3 axis VMC

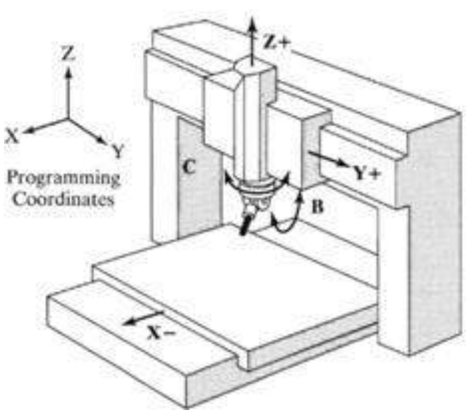
Axis designation in 3 axis HMC



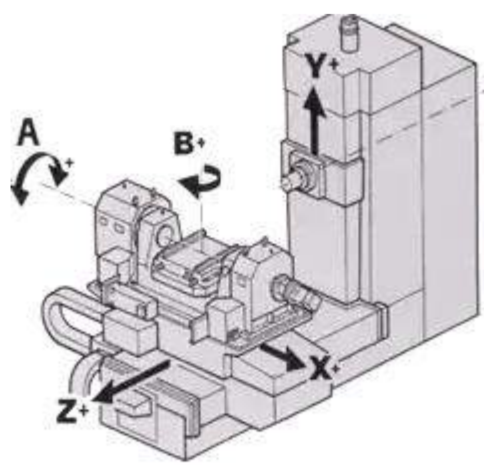
Axis designation in 4 axis VMC



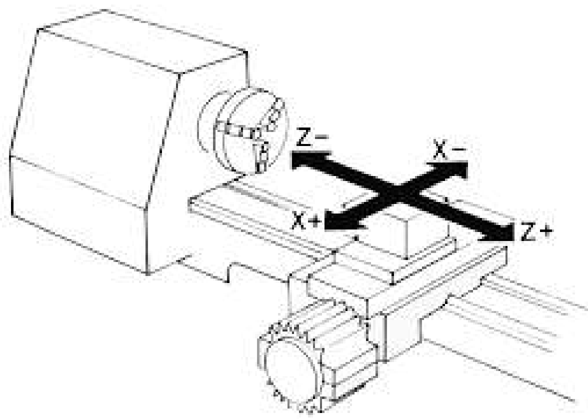
Axis designation in 4 axis HMC



Axis designation in 5 axis VMC



Axis designation in 5 axis HMC



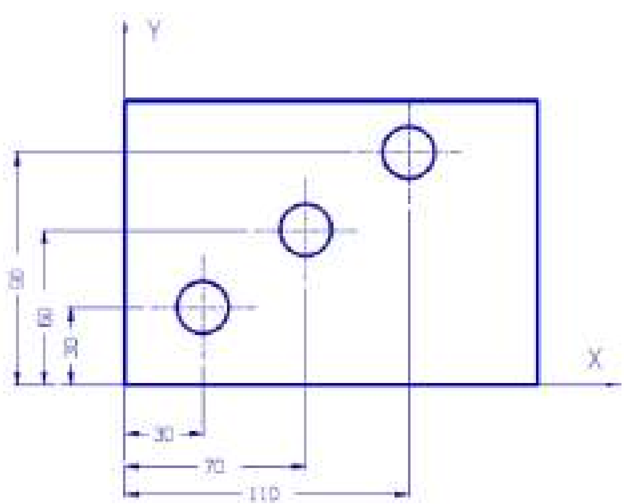
Axis designation in CNC lathe

1.8.2 Programming Systems

Two types of programming modes, the **incremental system and the absolute system**, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time. Most controls on machine tools today are capable of handling either incremental or absolute programming.

Absolute Coordinate System (G90)

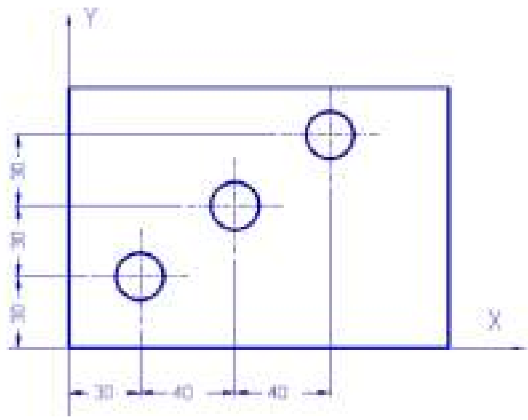
In an absolute system all references are made to the origin of the coordinate system. All measurements are made from the part origin established by the programmer and set up by the operator. Any programmed coordinate has the absolute value in respect to the absolute coordinate system zero point. The machine control uses the part origin as the reference point in order to position the tool during program execution



- A "X plus" (X+) command will cause the cutting tool to be located to the right of the zero or origin point.
- A "X minus" (X-) command will cause the cutting tool to be located to the left of the zero or origin point.
- A "Y plus" (Y+) command will cause the cutting tool to be located toward the column.
- A "Y minus" (Y-) command will cause the cutting tool to be located away from the column.

In absolute programming, the **G90** command indicates to the computer and MCU that the programming is in the absolute mode.

Relative / Incremental programming (G91)



In incremental programming, the tool movement is measured from the last tool position. The programmed movement is based on the change in position between two successive points. The coordinate value is always incremented according to the preceding tool location. The programmer enters the relative distance between current location and the next point

- A “X plus” (X+) command will cause the cutting tool to be located to the right of the last point.
- A “X minus” (X-) command will cause the cutting tool to be located to the left of the last point.
- A “Y plus” (Y+) command will cause the cutting tool to be located toward the column.
- A “Y minus” (Y-) will cause the cutting tool to be located away from the column.
- A “Z plus” (Z+) command will cause the cutting tool or spindle to move up or away from the workpiece.
- A “Z minus” (Z-) moves the cutting tool down or into the workpiece.

In incremental programming, the **G91 command** indicates to the computer and MCU (Machine Control Unit) that programming is in the incremental mode.

1.8.3 Reference Point

Part programming requires establishment of some reference points. Three reference points are either set by manufacturer or user.

a) Machine Origin

The machine origin is a fixed point set by the machine tool builder. Usually it cannot be changed. Any tool movement is measured from this point. The controller always remembers tool distance from the machine origin. This point is located at lower left corner / south west corner pf the table. This is also known as **Fixed zero / Machine datum**

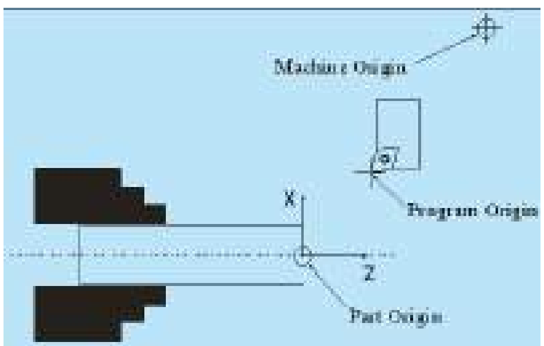
b) Program Origin

It is also called home position of the tool. Program origin is point from where the tool starts for its motion while executing a program and returns back at the end of the cycle.

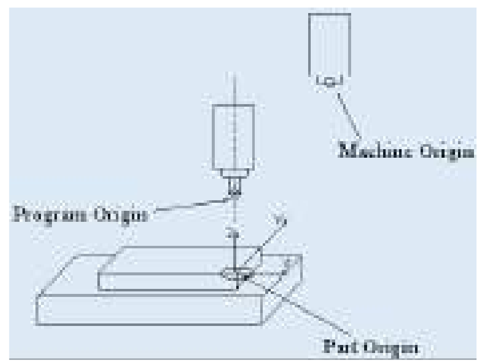
This can be any point within the workspace of the tool which is sufficiently away from the part. In case of CNC lathe it is a point where tool change is carried out.

c) Part Origin

The part origin can be set at any point inside the machine's electronic grid system. Establishing the part origin is also known as zero shift, work shift, **floating zero or datum**. Usually part origin needs to be defined for each new setup. Zero shifting allows the relocation of the part. Sometimes the part accuracy is affected by the location of the part origin. **Figure 1 and 2** shows the reference points on a lathe and milling machine.



Reference points and axis on a lathe



Reference points and axis on a Milling Machine

1.8.4 Modal and Non Modal Commands

Commands issued in the NC program may stay in effect indefinitely (until they explicitly cancelled or changed by some other command), or they may be effective for only the one time that they are issued. The former are referred as Modal commands. Examples include feed rate selection and coolant selection.

A modal G code will remain active until another G code from the same group is programmed into a block, or it is cancelled.

For example,
GØ1 and GØØ are modal G codes from group 1:

```
GØ1   X _ _ _ _ ;  ┐
      Y _ _ _ _ ;  ├ GØ1 is effective in this range.
      Z _ _ _ _ ;  ┘
GØØ   Z _ _ _ _ ;   -     GØØ replaces the GØ1 command.
```

Commands that are effective only when issued and whose effects are lost for subsequent commands are referred to as **non-modal commands**. A dwell command, which instructs the tool to remain in a given configuration for a given amount of time, is an example of a non-modal command.

A non-modal G code must be programmed into every block when it is required, ie, it is only effective in the block in which it is specified.

1.8.5 CNC program structure

There are four basic terms used in CNC programming **Character -> Word -> Block -> Program**

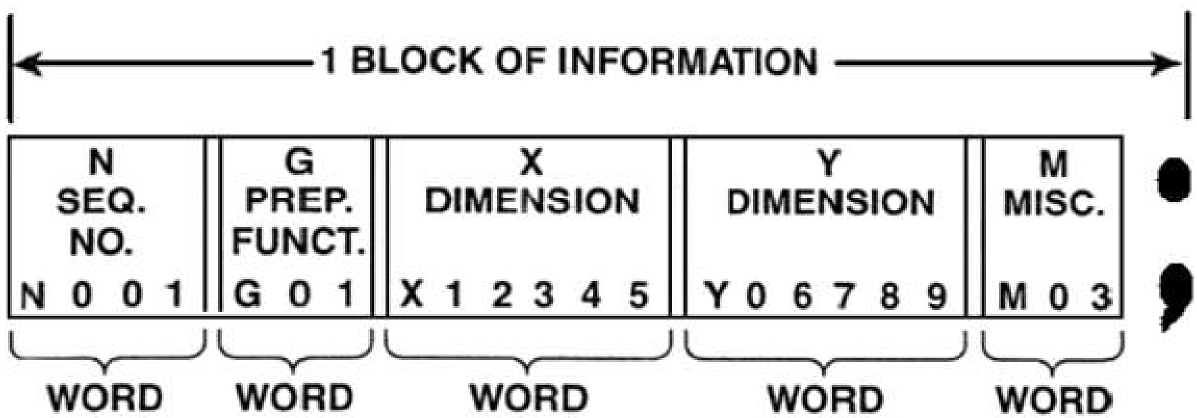
Character is the smallest unit of CNC program. It can have Digit / Letter / Symbol.

Word is a combination of alpha-numerical characters. This creates a single instruction to the CNC machine. Each word begins with a capital letter, followed by a numeral. These are used to represent axes positions, feed rate, speed, preparatory commands, and miscellaneous functions.

Several commands are grouped together to accomplish a specific machining operation, hence the use of a block of information for each operation.

•Each command gives a specific element of control data, such as dimension or a feed rate. Each command within a block is also called a word.

The way in which words are arranged within the block is called block format.



The words are as follows :

N001 – represents the sequence number of the operation.

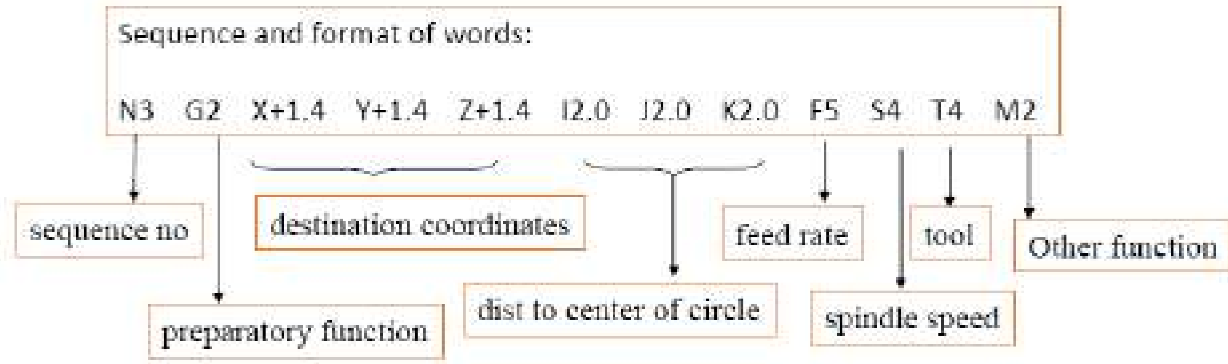
G01 – represents linear interpolation.

X12345 – will move the table in a positive direction along the *X*-axis.

Y06789 – will move the table along the *Y*-axis.

M03 – Spindle on CW and

; – End of block.



1.8.6 Types of CNC codes

1) Preparatory codes

Preparatory functions, called G codes, are used to determine the geometry of tool movements and operating state of the machine controller; functions such as linear cutting movements, drilling operations and specifying the units of measurement. They are normally programmed at the start of a block. A **preparatory function** is designated in a program by the word address G followed by two digits. Preparatory functions are also called **G-codes** and they specify the control mode of the operation.

Code	**Function**
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise (CW)
G03	Circular interpolation counterclockwise (CCW)
G20	Inch input (in.)
G21	Metric input (mm)
G24	Radius programming
G28	Return to reference point
G29	Return from reference point
G32	Thread cutting
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation positive (+) direction
G44	Tool length compensation minus (-) direction
G49	Tool length compensation cancels
G 53	Zero offset or M/c reference
G54	Settable zero offset

G84	canned turn cycle
G90	Absolute programming
G91	Incremental programming

Note : On some machines and controls, some may be differ.

2) Miscellaneous codes

Miscellaneous functions use the address letter M followed by two digits. They perform a group of instructions such as coolant on/off, spindle on/off, tool change, program stop, or program end. They are often referred to as machine functions or **M-functions**. Some of the M codes are given below.

Code	**Function**
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping
M11	Chuck - unclamping
M12	Tailstock spindle out
M13	Tailstock spindle in
M17	Tool post rotation normal
M18	Tool post rotation reverse
M30	End of tape and rewind or main program end
M98	Transfer to subprogram
M99	End of subprogram

Note : On some machines and controls, some may be differ.

1.8.7 Programming Format

Three different block formats are commonly used, **(Fixed sequential format, Tab sequential format and Word address format)**

Word Address Format

This type of tape format uses alphabets called address, identifying the function of numerical data followed. This format is used by most of the NC machines, also called variable block format. A typical instruction block will be as below :

N20 G00 X1.200 Y.100 F325 S1000 T03 M09 <EOB>

or

N20 G00 X1.200 Y.100 F325 S1000 T03 M09;

The MCU uses this alphabet for addressing a memory location in it. This is the widely used format

Tab Sequential Format

Here the alphabets are replaced by a Tab code, which is inserted between two words. The MCU reads the first Tab and stores the data in the first location then the second word is recognized by reading the record Tab. A typical Tab sequential instruction block will be as below :

>20 >00 >1.200 >.100 >325 >1000 >03 >09

Fixed Block Format

In fixed block format no letter address of Tab code are used and none of words can be omitted. The main advantage of this format is that the whole instruction block can be read at the same instant, instead of reading character by character. This format can only be used for positioning work only. A typical fixed block instruction block will be as below:

20 00 1.200 .100 325 1000 03 09 <EOB>

1.9 Preparatory Functions

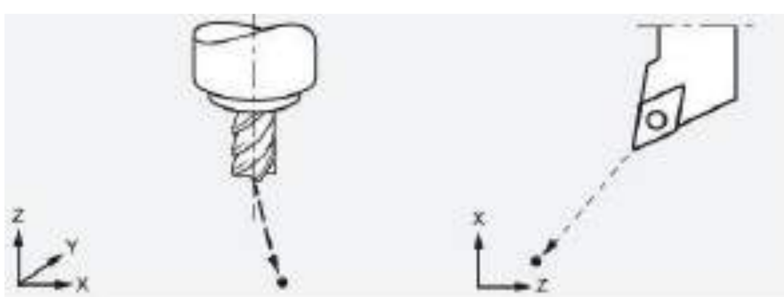
1.9.1 G00 Rapid traverse

When the tool being positioned at a point preparatory to a cutting motion, to save time it is moved along a straight line at Rapid traverse, at a fixed traverse rate which is pre-programmed into the machine's control system.

Typical rapid traverse rates are 10 to 25 m /min., but can be as high as 80 m/min.

Syntax: N010 [G90/G91] G00 X10 Y10 Z5

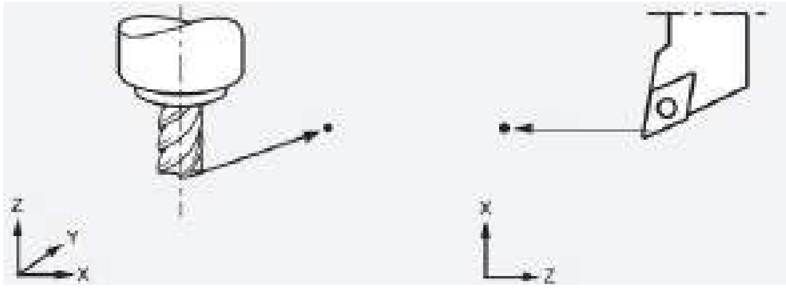
[For GOO, no need to mention the feed rate.]



1.9.2 G01 Linear interpolation (feed traverse)

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the feed rate.

•Syntax: N010 [G90/G91] G01 X10 Y10 Z5 F25



1.9.3 G02/G03 Circular interpolation

The format to program a circular interpolation in Cartesian co-ordinates is written as follows :

There are four ways to program **a clockwise circular path using the GØ2 code:**

G9Ø GØ2 X _ _ _ _ Y _ _ _ _ R _ _ _ F _ _ _ ;

G9Ø GØ2 X _ _ _ _ Y _ _ _ _ I _ _ _ J _ _ _ F _ _ _ ;

G91 GØ2 X _ _ _ _ Y _ _ _ _ R _ _ _ _ F _ _ _ _ ;

G91 GØ2 X _ _ _ _ Y _ _ _ _ I _ _ _ J _ _ _ F _ _ _ ;

There are four ways to program **an anticlockwise circular path using the GØ3 code:**

G9Ø GØ3 X _ _ _ _ Y _ _ _ _ R _ _ _ F _ _ _ ;

G9Ø GØ3 X _ _ _ _ Y _ _ _ _ I _ _ _ J _ _ _ F _ _ _ ;

G91 GØ3 X _ _ _ _ Y _ _ _ _ R _ _ _ _ F _ _ _ _ ;

G91 GØ3 X _ _ _ _ Y _ _ _ _ I _ _ _ J _ _ _ F _ _ _ ;

where,

GØ2 defines the clockwise direction circular interpolation.

GØ3 defines the counterclockwise direction circular interpolation.

G9Ø X _ _ _ _ Y _ _ _ _ defines the arc end point in the work co-ordinate system.

G91 X _ _ _ _ Y _ _ _ _ defines the signed distance of the arc end point from the arc start point.

I _ _ _ J _ _ _ defines the signed distance of the arc start point from the centre point of the arc.

R _ _ _ defines the length of the arc radius.

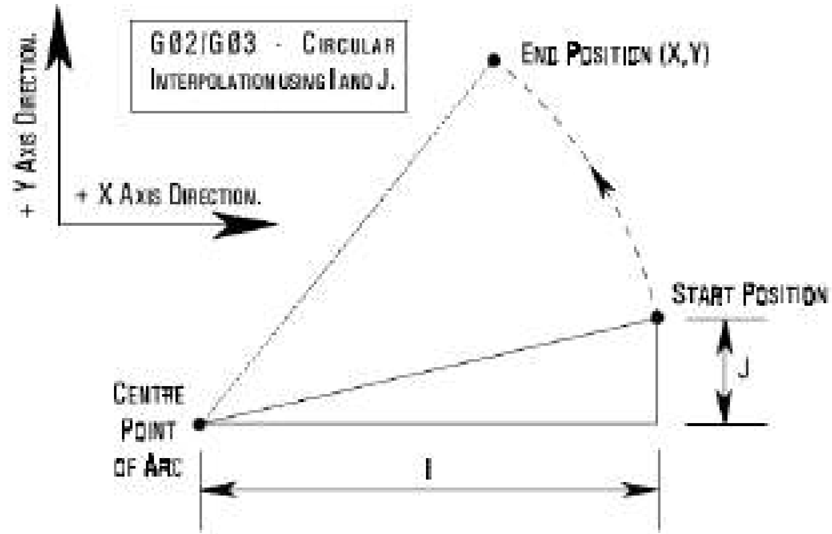
F _ _ _ defines the feedrate along the arc.

I AND J

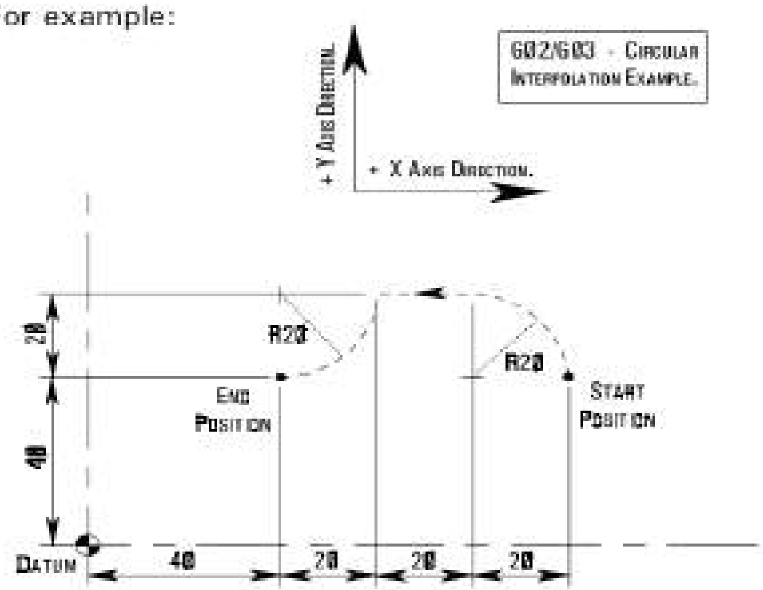
To program an arc when only the arc centre is given (the radius is unknown) use the address letters I and J.

I relates to the address X and is the incremental value and direction (+/-) from the start point of the arc in the X axis to the arc centre (see diagram below).

J relates to the address Y and is the incremental value and direction (+/-) from the start point of the arc in the Y axis to the arc centre (see diagram below).



For example:



The above tool path can be programmed as follows (In absolute mode, G9Ø):

GØ1 X1ØØ Y4Ø F125 ;

GØ3 X8Ø Y6Ø I-2Ø ;

GØ1 X6Ø ;

GØ2 X4Ø Y4Ø I-2Ø ;

or,

GØ1 X1ØØ Y4Ø F125 ;

GØ3 X8Ø Y6Ø R2Ø ;

GØ1 X6Ø ;

GØ2 X4Ø Y4Ø R2Ø ;

The above tool path can be programmed as follows (In incremental mode, G91):

GØ3 X-2Ø Y2Ø I-2Ø ;

GØ1 X-2Ø ;

GØ2 X-2Ø Y-2Ø I-2Ø ;

or,

GØ3 X-2Ø Y2Ø R2Ø ;

GØ1 X-2Ø ;

GØ2 X-2Ø Y-2Ø R2Ø ;

1.9.4 G04 Dwell

The GØ4 code is used to enter a set time delay into the program (called a "dwell").

A GØ4 command is written in the following format:

GØ4 X _ _ _ _ ;

or GØ4 P _ _ _ _ ;

where the dwell value is programmed using the address letters X (time in seconds) or P (time in 1/1ØØØ seconds), followed by a number indicating this dwell value.

For example :

GØ4 X1.5 ;

This command is read perform a dwell of 1.5 seconds duration.

GØ4 P25ØØ ;

This command is read perform a dwell of 2.5 seconds duration.

NOTE 1. A decimal point cannot be used with the address P.

NOTE 2. The dwell is performed at the start of the block in which it is programmed.

NOTE 3. The dwell begins when the commanded feed rate of the previous block reaches zero.

NOTE 4. The maximum value of a dwell time is 999 seconds.

NOTE 5. GØ4 is a non-modal G code. It is only active in the block in which it is programmed.

NOTE 6. A GØ4 code can be written into a program in two ways. GØ4 or G4.

1.9.5 G 20 & G21 – Imperial / Metric units

The machine controller can be programmed in either Imperial (inch) unit input (G2Ø) or Metric (millimetre) unit input (G21). The standard format for a CNC partprogram is to write the G2Ø or G21 code in the first block of the program.

G code.	Type.	Units.	Lowest input value.
G2Ø	Imperial	Inch	Ø.ØØØ1 inch
G21	Metric	Millimetre	Ø.ØØ1 mm

NOTE.

- The status of G2Ø or G21 in the machine controller is dependant on the option that is saved to the disc.
- A G2Ø code must not be changed for a G21 code (or vice versa) during the program.
- When switching between G2Ø and G21, the offsets must be set according to the units of measurement being used.
- G 2Ø and G21 are both modal G codes within the same modal group.

1.9.6 G28 Reference Point Return

The reference point is a fixed position on the machine, to which the tool can be moved. A G28 code instructs the tool to automatically move to this reference point.

A G28 command is written in the following format :

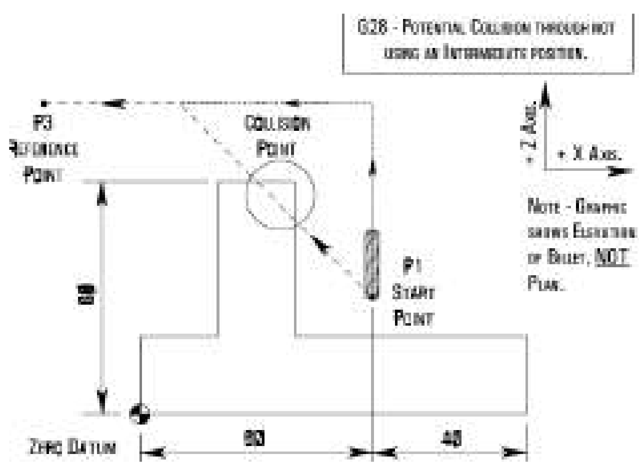
G9Ø G28 X _ _ _ _ Y _ _ _ _ Z _ _ _ _ ;

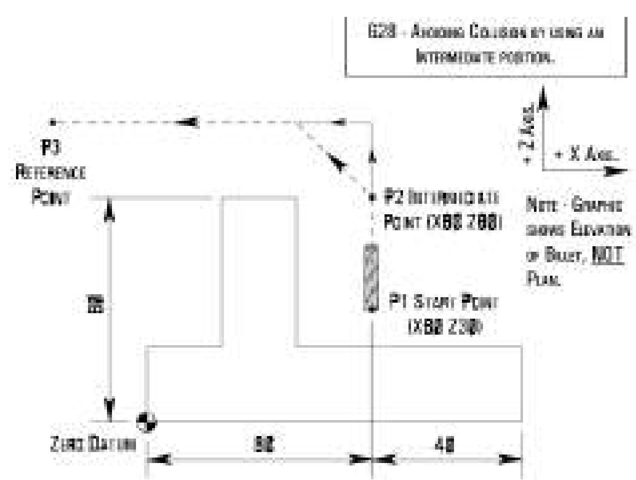
or G91 G28 X _ _ _ _ Y _ _ _ _ Z _ _ _ _ ;

where X, Y and Z can be used to indicate an intermediate point, through which the tool will pass, before continuing to the reference point.

NOTE 1

The diagram below shows how the tool could collide with the billet when directing towards the reference point. This is a result of the non-vectored movements forcing the tool to follow a path which "cuts" through the edge of the billet.





To avoid this collision, the tool is sent on a path which includes the additional, or intermediate, point P2. The intermediate point is used to allow the tool to move completely clear from the billet, before continuing onto the reference point, P3,

The above toolpath can be programmed as follows (In absolute mode, G9Ø):

G9Ø G28 X6Ø Z6Ø ;

The above toolpath can be programmed as follows (In incremental mode, G91):

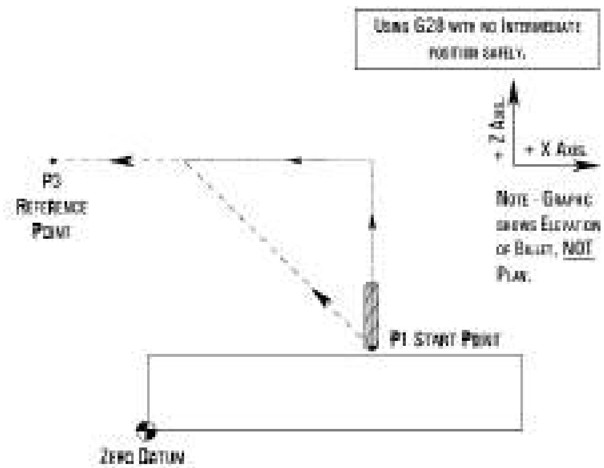
G91 G28 XØ Z4Ø ;

NOTE 2.

In the diagram below, the tool is in a position (P1) where no collision is possible. The intermediate point, in this case, is not required, so the block can be written as follows (In incremental mode, G91):

G91 G28 XØ YØ ZØ ;

The intermediate point co-ordinates are still stated, but all their values are set to zero, indicating no axis movement. Therefore, the tool will move from point P1 to the reference point, P3, along a non-vector type path.



(**G28 is a non-modal G code**. It is only active in the block in which it is programmed).

1.9.7 G40, G41 & G42

G40 - Cutter diameter / radius compensation cancel

G41 - Cutter diameter / radius compensation – left

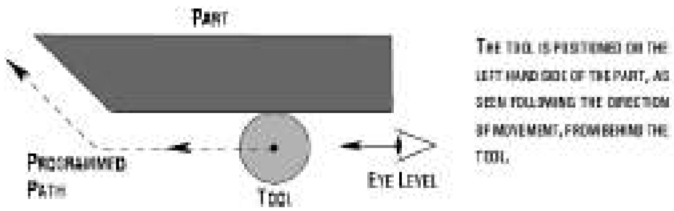
G42 - Cutter diameter / radius compensation – Right

The collection of G4Ø, G41 and G42 codes allow the machine controller to produce very accurate arcs and tapers on the billet, by compensating for the tool radius.

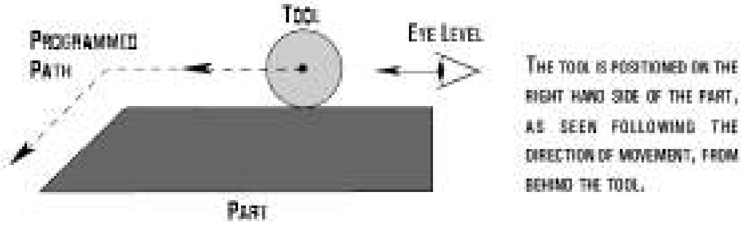
Complex workpiece shapes are therefore programmed with cutter compensation mode active. The radius of the tool (the offset amount) is measured, then entered into the offset file in the machine controller. Once set, the tool path can be offset by this value, regardless of the program.

The two diagrams below illustrate the direction of compensation codes G41 and G42, in relation to your eye level

G41 - LEFT HAND



G42 - RIGHT HAND

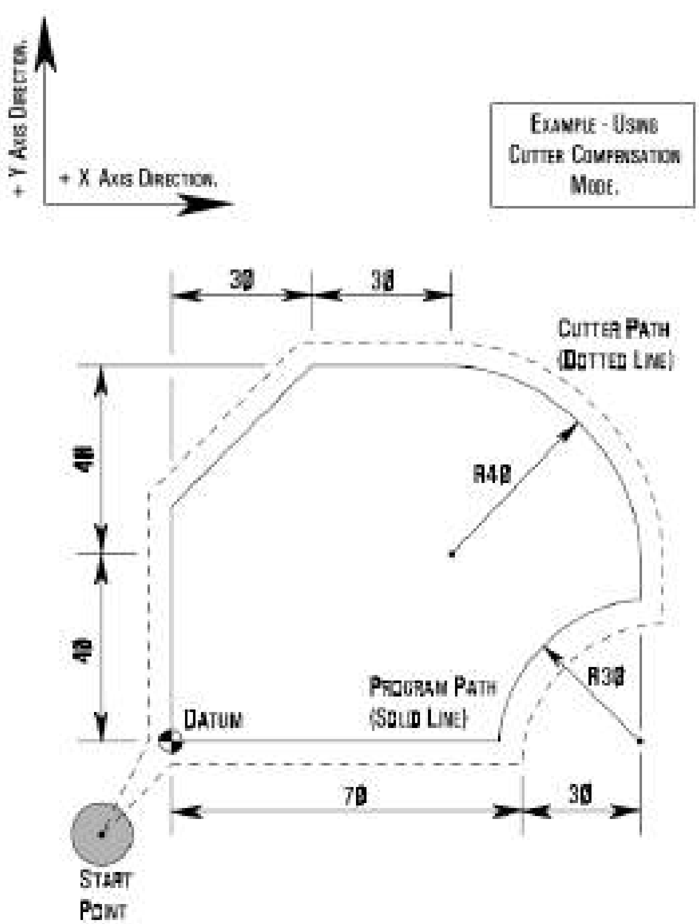


Note :-A GØ2 or GØ3 circular interpolation command cannot be specified in the start-up block.

Example

The following part program for a finishing pass shows the recommended method for start-up and cancellation of cutter compensation:

O ØØ1Ø



NØØ1Ø **GØØ X-15 Y-15** ;
NØØ2Ø **G41 XØ YØ F1ØØ** ; (Start-Up Move)
NØØ3Ø **Y4Ø ;**
NØØ4Ø **X3Ø Y8Ø** ;
NØØ5Ø **X6Ø ;**
NØØ6Ø **GØ2 X1ØØ Y4Ø R4Ø** ;
NØØ7Ø **GØ1 Y3Ø** ;
NØØ8Ø **GØ3 X7Ø YØ R3Ø ;**
NØØ9Ø **XØ ;**
NØ1ØØ **X-15 Y-15 ;** (Cancellation Move)

1.9.8 G73- G89 CANNED CYCLES

A **canned cycle** is a way of conveniently performing repetitive CNC machine operations. Canned cycles automate certain machining functions such as drilling, boring, threading, pocketing, etc. Canned cycles combine many programming operations and are designed to shorten the program length, minimize mathematical calculations, and use minimal tool motions. A canned cycle is also known as a **fixed cycle**. A canned cycle is usually permanently stored as a pre-program in the machine's controller and cannot be altered by the user.

G81 Drilling cycle

G82 Drilling cycle with dwell (Counter bore cycle)

G83 Peck drilling cycle / deep drill

G84 Right hand tapping cycle

G85 Boring / Reaming cycle

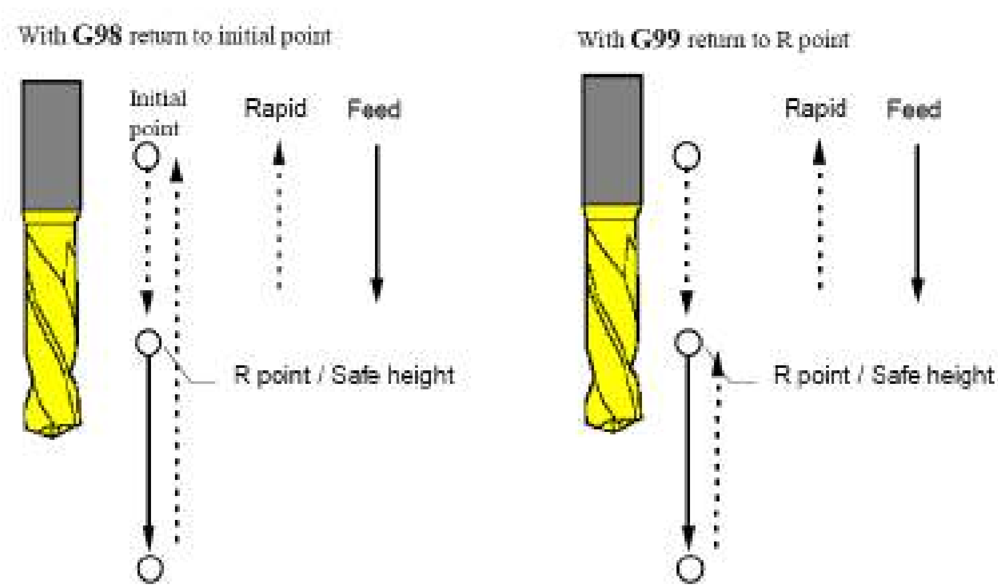
G86 Boring cycle

G87 Back boring cycle

G74 Left hand tapping cycle

G76 Fine boring cycle

1.9.9 G81 Drilling / Spot boring cycle



G81 (Drilling - Spot Boring) command is written in the following format:

(G9Ø) **(G98)**

or **or** **G81 X.... Y.... Z.... R.... F.... ;**

(G91) **(G99)**

G81is defined as the canned cycle.

X.... Y.... is defined as the hole position, in absolute or incremental value.

Z.... is defined as the distance from the R point to the bottom of the hole in incremental mode, or the position of the hole bottom in absolute mode.

R.... is defined as the distance from the initial level to the R point level in incremental mode, or the position of the Z datum in relation to the R point level in absolute mode.

Example for Canned Cycle

The following example shows how the canned cycle reduces the program length

4 holes are to be drilled in a MS plate of thickness 30 mm. The coordinates of the holes are as follows (10,10), (20,10), (30,10) and (40,10). The depth of 1st, 2nd and 4th hole are 15 mm respectively and the depth of 3rd hole is 25 mm (10 mm deeper than the other 3 holes). Write the CNC part program.

Without Canned Cycle

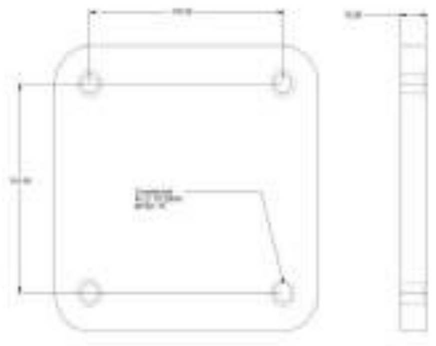
```
O 0010
N1    M06 T01;
N2    G21 G28;
N3    M03 S1200
N4    G00   X0    Y0    Z5 ;
F100
N5    G90;
N6    G01   X10   Y10         F100;
N7                      Z-15 ;
N8                      Z5;
N9          X20;
N10                     Z-15;
N11                     Z5;
N12         X30;
N13                     Z-25;
N14                     Z5;
N15         X40;
N16                     Z-15
N17                     Z5
```

With Canned Cycle

```
O 0010
N1    M06 T01;
N2    G21 G28;
N3    M03 S1200
N4    G90 G99 X10  Y10  Z-15  R2

N5              X20
N6              X30       Z-25
N7              X40       Z-15
N8    G80
N9    G01      X0  Y0
N10   G28   M05   M30
```

N18 X0 Y0

N19 G28 M05 M30



EXAMPLE -2

O0001 (Spot Drill Four Holes)

T01 M06 (**16 MM SPOT DRILL**)

G90 G00 G54 X-55 Y-55 S2500 M03

G43 H01 Z3 M08

G81 G98 Z-6 R1 F100

Y55

X55

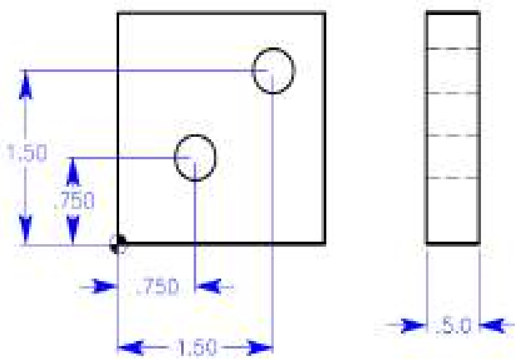
Y-55

G80

G00 G53

X0 Y0 Z0

M30



EXAMPLE 3

N1 T1 M06
N2 G90 G54 G00 X.75 Y.75
N3 S1200 M03
N4 G43 H01 Z1. M08
N5 G81 G99 Z-.625 R.1 F10.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G28 G91 Z0. M05
N9 M30

Repeat Drilling

With G81 drilling cycle drilling operation can be repeated multiple times. The drilling is repeated K times when that parameter is given with G81 drilling cycle.

(G9Ø) (G98)

or or G81 X.... Y.... Z.... R.... K….F.... ;

(G91) (G99)

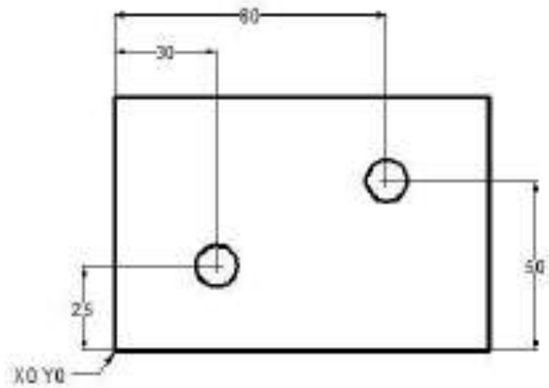
G81is defined as the canned cycle.

X.... Y.... is defined as the hole position, in absolute or incremental value.

Z.... is defined as the distance from the R point to the bottom of the hole in incremental mode, or the position of the hole bottom in absolute mode.

R.... is defined as the distance from the initial level to the R point level in incremental mode, or the position of the Z datum in relation to the R point level in absolute mode.

K…. Number of cycle repetitions



Repeat drilling is normally used with G91 Incremental mode, and a good example of repeated drilling is Grid-plate drilling. The example for repeat drilling is given below.

N10 T1 M06

N20 G90 G54 G00 X0 Y0

N30 S1200 M03

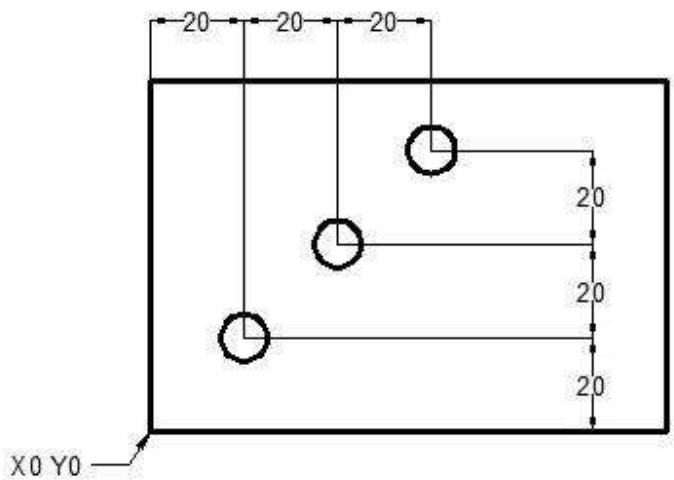
N40 G43 H01 Z5 M08

N50 G81 G99 G91 X30 Y 25 Z-10 R2 K2 F75

N60 G80

N70 G00 Z100 M09

N80 M30



Example

T1 M6

G00 G90 G21 G17 G94

G54 X0 Y0 S1000 M03

G43 H1 Z100

Z3

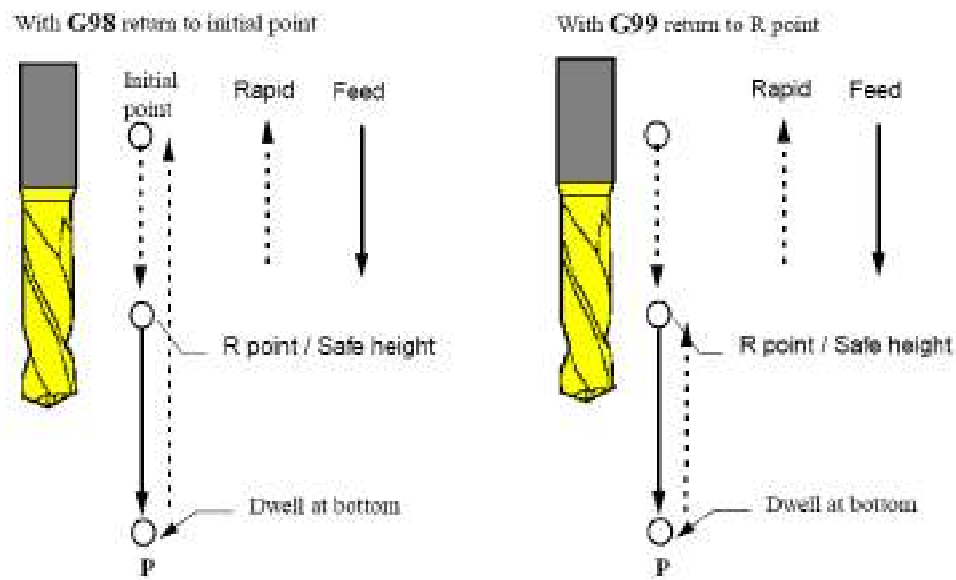
G81 G99 G91 X20 Y20 R3 Z-20 K3 F100 M08

G80

G00 G90 Z100

M30

1.9.10 G82 – Spot drilling / Counter bore cycle



FORMAT

(G9Ø)	**(G98)**	
or	**or**	**G82 X.... Y.... Z.... R...P…. F.... ;**
(G91)	**(G99)**	

G82 is defined as the canned cycle.

X.... Y.... is defined as the hole position, in absolute or incremental value.

Z.... is defined as the distance from the R point to the bottom of the hole in incremental mode, or the position of the hole bottom in absolute mode.

R.... is defined as the distance from the initial level to the R point level in incremental mode, or the position of the Z datum in relation to the R point level in absolute mode.

P…. Dwell at bottom

Sequence of moves in G82:

Op 1) Rapid position to X, Y and Z (the Initial level).

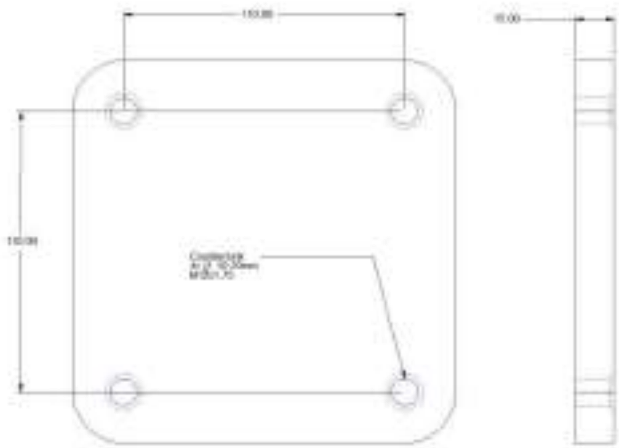
Op 2) Rapid traverse to R point level.

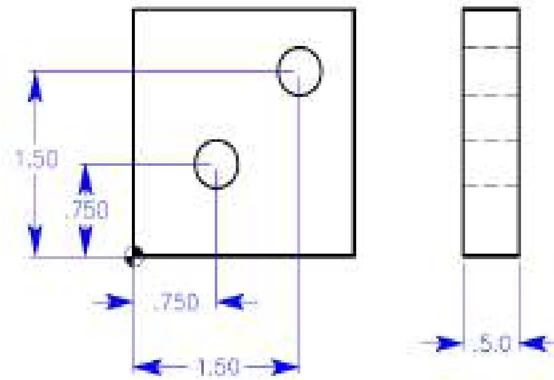
Op 3) Feed to Z depth.

Op 4) Dwell for value P.

Op 5) Rapid traverse to Initial level (G98) or R point level (G99).

Example 1





O0001 (Spot Drill Four Holes WITH DWELL)

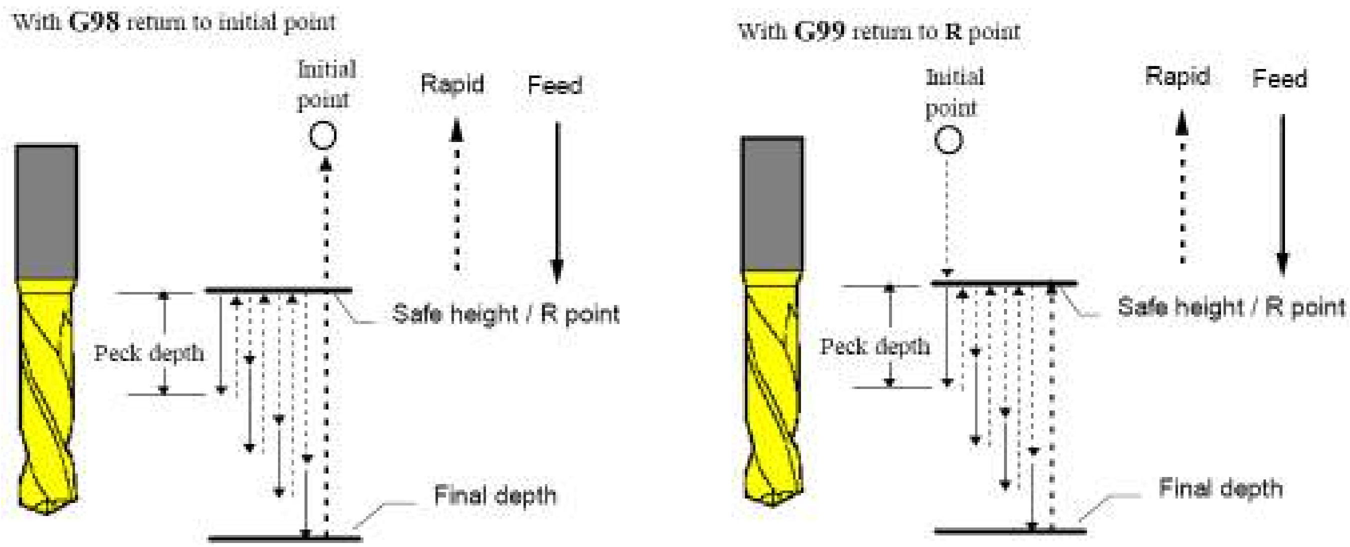
```
T01   M06 (16 MM SPOT DRILL)
G90   G00   G54   X-55   Y-55          S2500
      M03
G43   H01                       Z3
      M08
G82   G98                       Z-6    R1     P2000     F100
                         Y55
                         X55
                         Y-55
G80
GOO   G53         X0     Y0     Z0
M30
```

EXAMPLE 2

N1 T2 M06
N2 G90 G54 G00 X.75 Y.75
N3 S1200 M03
N4 G43 H02 Z1. M08
N5 G82 G99 Z-.625 P1.5 R.1 F8.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G28 G91 Z0. M05
N9 M30

1.9.11 G83 – PECK DRILLING / DEEP HOLE DRILLING CYCLE

The **G83** cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum).



FORMAT

(G9Ø) **(G98)**

or **or** **G83 X.... Y.... Z.... Q...P…. F.... ;**

(G91) **(G99)**

G83 is defined as the canned cycle.

X.... Y.... is defined as the hole position, in absolute or incremental value.

Z.... is defined as the distance from the R point to the bottom of the hole in incremental mode, or the position of the hole bottom in absolute mode.

R.... is defined as the distance from the initial level to the R point level in incremental mode, or the position of the Z datum in relation to the R point level in absolute mode.

P…. Dwell at bottom

Q…. Pecking depth amount value, always incremental

Sequence of moves:

Op 1) Rapid position to X, Y and Z (the initial level).

Op 2) Rapid traverse to R point level.

Op 3) Feed in to the value of Q.

Op 4) Rapid traverse out to R point. Rapid traverse back to within 1mm of depth of Q cut. Operation moves 2 and 4 are repeated until Z depth is reached.

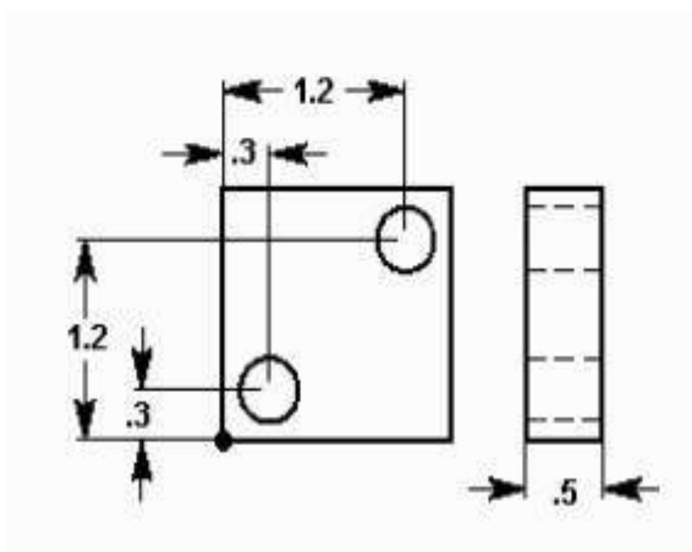
Op 5) Rapid traverse to Initial level (G98) or R point level (G99).

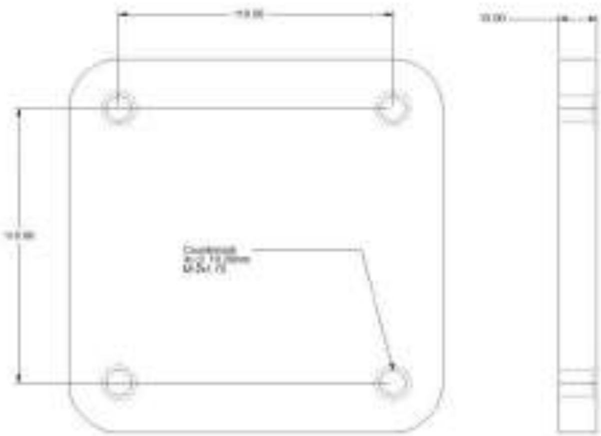
Example

```
N1  T1  M06
N2  G90 G54 GOO X.3  Y.3
N3  S1200  M03
N4  G43 H01 Z1 M08
N5  G83 Z-1.5 Q.5  R.1 F10
N6  X1.2  Y1.2
N7  G80
N8  G00  Z1
N9  G28  Z0 M05
N9  M30
```

EXAMPLE 2

```
O0001 (PECK DRILL CYCLE)
T01   M06 (10.2 MM DRILL)
G90   G00   G54   X-55  Y-55        S2500       M03
G43   H01                     Z3                M08
G83   G98                     Z-6   R1
      Q2    P2000       F100
                        Y55
                  X55
                        Y-55
G80
GOO   G53         X0    Y0    Z0
M30
```





EXAMPLE
(G83 using Q)

N1 T3 M06

N2 G90 G54 G00 X.75 Y.75

N3 S1200 M03

N4 G43 H03 Z1. M08

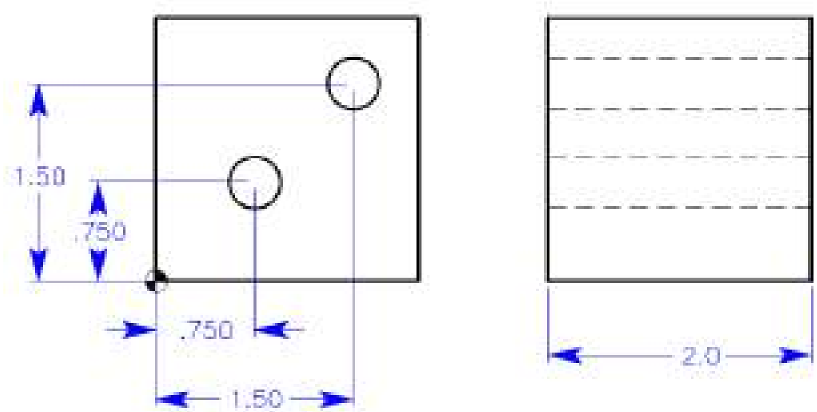
N5 G83 G99 Z-2.125 Q.5 R.1 F8.

N6 X1.5 Y1.5

N7 G80 G00 Z1. M09

N8 G28 G91 Z0. M05

N9 M30



1.9.11 G 83 Peck drilling with I, J, K options

(G9Ø) (G98)

or or G83 X.... Y.... Z.... I... J….R…F.... ;

(G91) (G99)

X Optional rapid X-axis command

Y Optional rapid Y-axis command

Z Bottom of hole (Feed point of Z depth starting from R plane)

Q The peck depth amount value, always incremental

I Optional size of first cutting depth (If Q is not used)

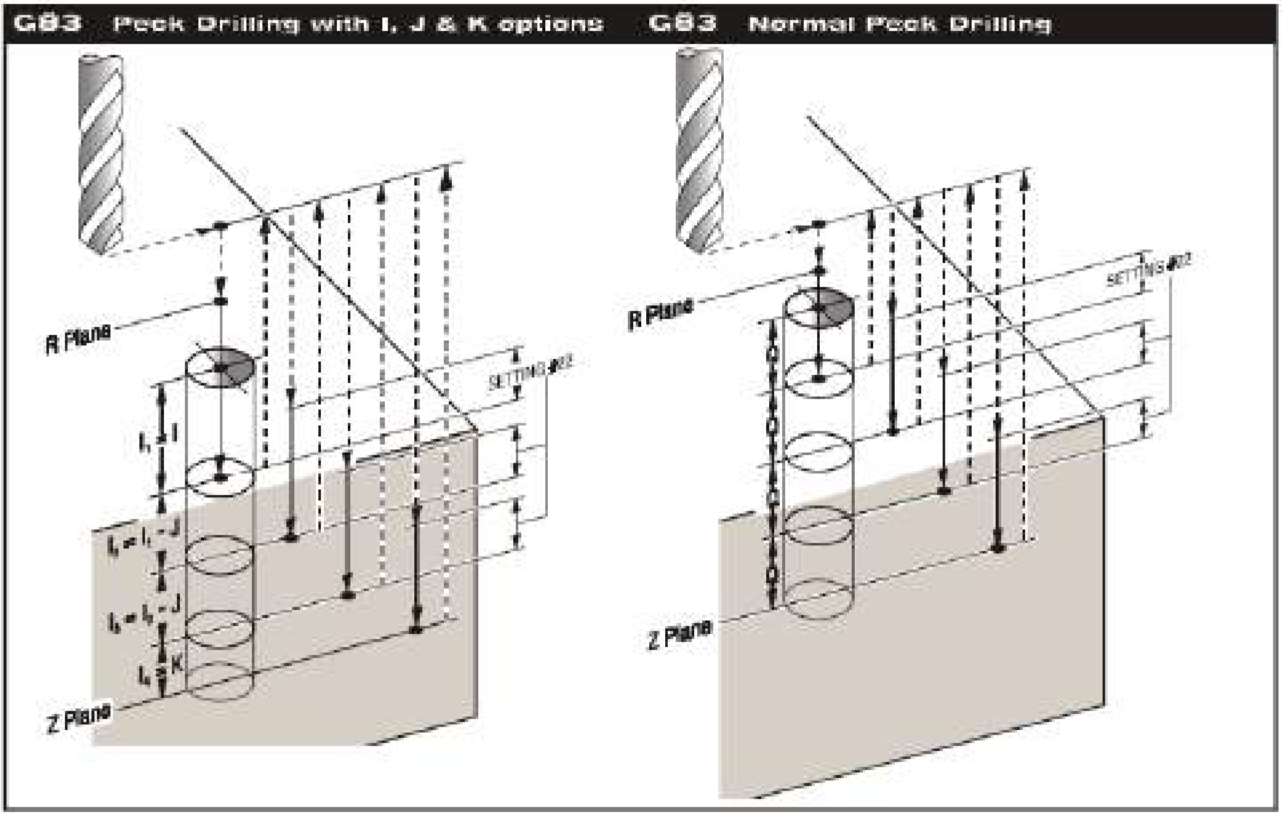
J Optional amount to reduce cutting depth each pass (If Q is not used)

K Optional minimum depth of cut (If Q is not used)

P The dwell time at the bottom of the hole

R Rapid position of the R plane (Where you rapid, to start feed)

F Feed Rate in inches (mm) per minute



EXAMPLE
(G83 using I, J & K)

N1 T3 M06

N2 G90 G54 G00 X.75 Y.75

N3 S1200 M03

N4 G43 H03 Z1. M08

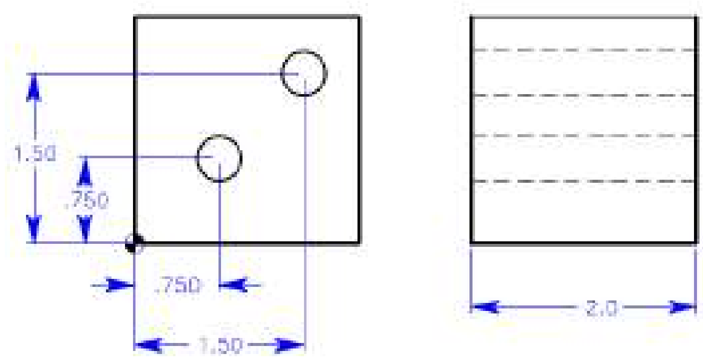
N5 G83 G99 Z-2.125 I.5 J.1 K.2 R.1 F8.

N6 X1.5 Y1.5

N7 G80 G00 Z1. M09

N8 G28 G91 Z0. M05

N9 M30



1.9.12 G84 - Tapping cycle – Right hand Tapping
FORMAT

(G9Ø) **(G98)**

or **or** **G84 X.... Y.... Z.... R...P…. F.... ;**

(G91) **(G99)**

G83 is defined as the canned cycle.

X.... Y.... is defined as the hole position, in absolute or incremental value.

Z.... is defined as the distance from the R point to the bottom of the hole in incremental mode, or the position of the hole bottom in absolute mode.

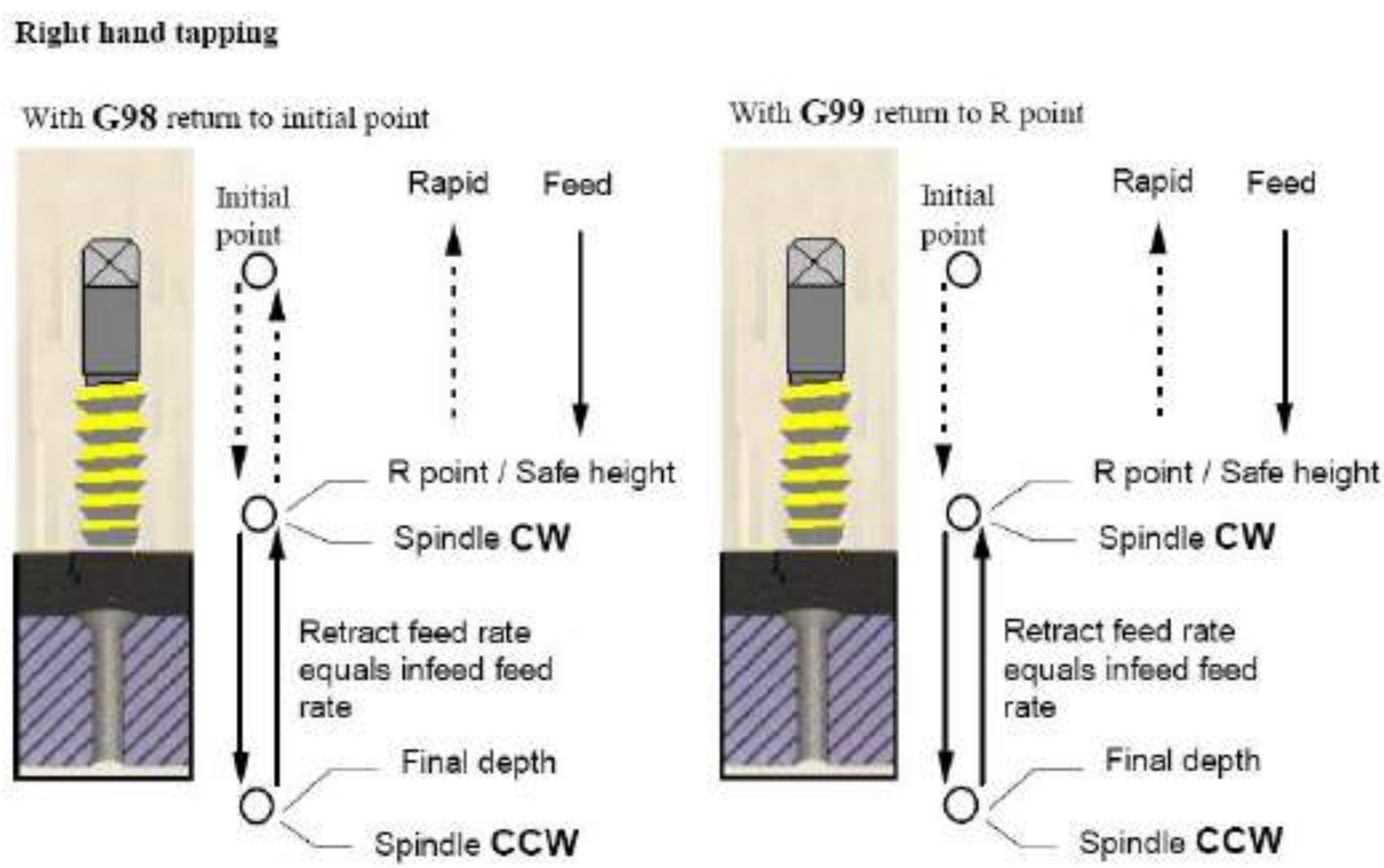
R.... is defined as the distance from the initial level to the R point level in incremental mode, or the position of the Z datum in relation to the R point level in absolute mode.

P…. Dwell Time

F…. Feed Rate = Pitch x RPM

Note :-

1. You don't need to start the spindle with a M03 for a tap that's using G84 because this cycle will turn on the spindle for you automatically and it will do it quicker.
2. Spindle direction in RH tapping ; CW during infeed , CCW during outfeed



Sequence of moves:

Op 1) Rapid position to X, Y and Z (the initial level).

Op 2) Rapid traverse to R point level.

Op 3) Feed to Z depth.

Op 4) Dwell P (time for spindle stop and start CCW direction).

Op 5) Feed to R point level.

Op 6) Dwell P (time for spindle stop and start CW direction).

If the G98 code is programmed within the cycle, the next move will be a rapid traverse to the Initial level. If the G99 code is programmed within the cycle, there will be no movement.

EXAMPLE

N1 T4 M06 (1/2-20 tap)

N2 G90 G54 G00 X.75 Y.75

N3 S800 M42 *(You don't need an M03. The G84 turns on spindle)*

N4 G43 H04 Z1. M08

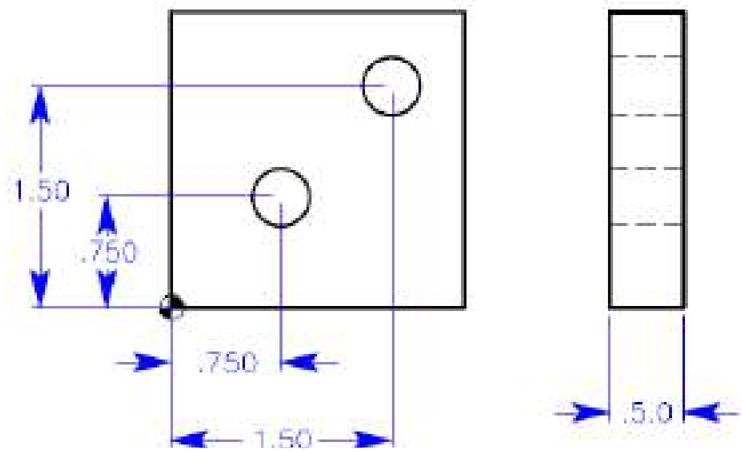
N5 G84 G99 Z-.65 R.1 F40.

N6 X1.5 Y1.5

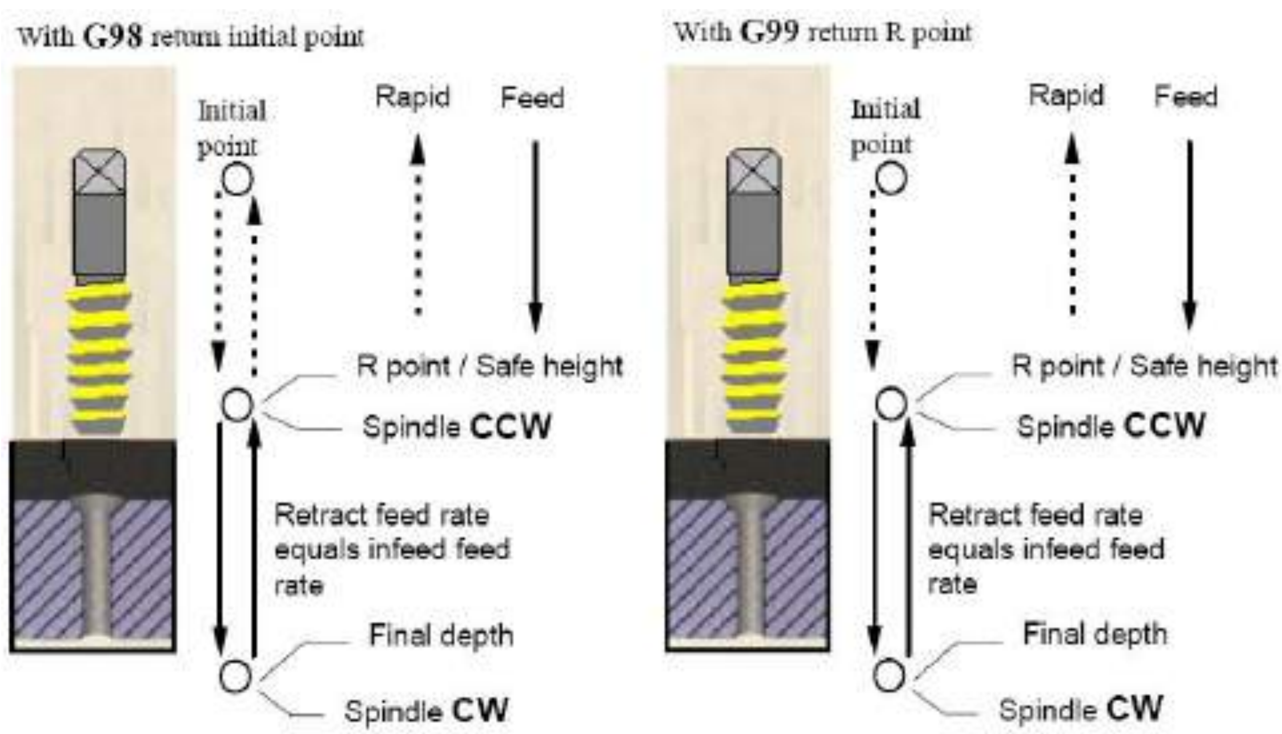
N7 G80 G00 Z1. M09

N8 G28 G91 Z0. M05

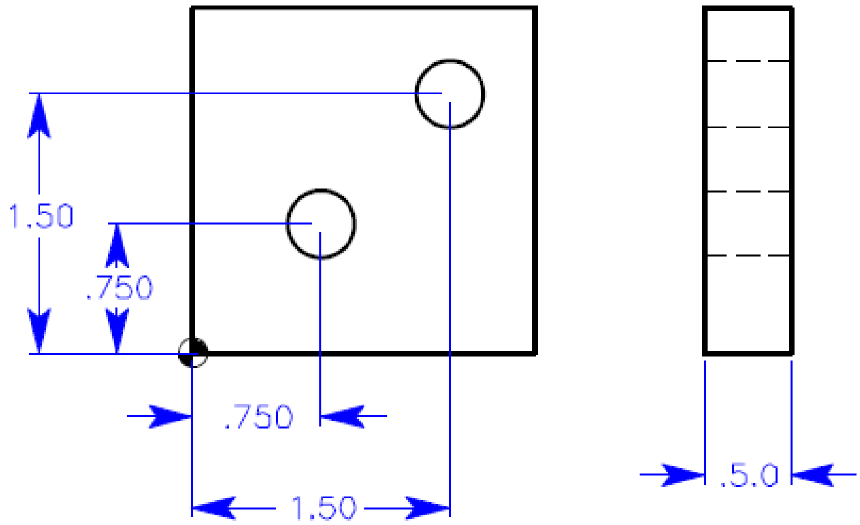
N9 M30



1.9.13 G74 - Tapping cycle – Left hand Tapping



Example



N1 T4 M06 (1/2-20 tap)

N2 G90 G54 G00 X.75 Y.75

N3 S800 MO4 (You don't need an M04. G84 turns on spindle)

N4 G43 H04 Z1. M08

N5 G74 G99 Z-.6 R.1 F40.

N6 X1.5 Y1.5

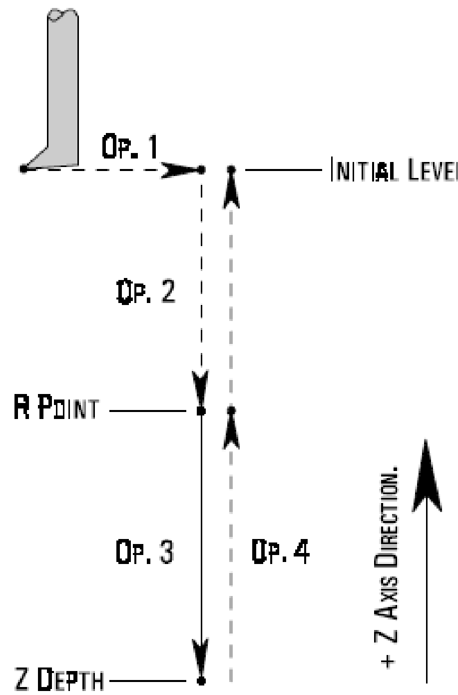
N7 G80 G00 Z1. M09

N8 G28 G91 Z0. M05

N9 M30

1.9.14 G85 - BORE IN AND BORE OUT CANNED CYCLE

(G9Ø)	**(G98)**	
or	**or**	**G85 X.... Y.... Z.... R...F.... ;**
(G91)	**(G99)**	



Sequence of moves:

Op 1) Rapid position to X, Y and Z (the initial level).

Op 2) Rapid traverse to R point level.

Op 3) Feed in to the Z depth.

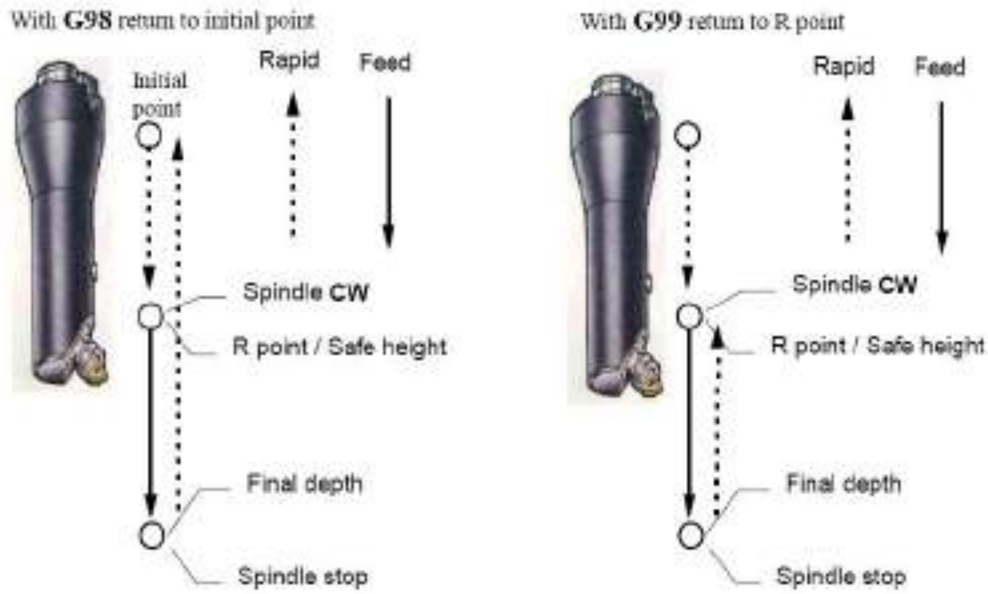
Op 4) Feed back to R point level.

If the G98 code is programmed within the cycle, the next move will be a rapid traverse to the Initial level. If the G99 code is programmed within the cycle, there will be no movement.

1.9.15 G86 - BORE IN, STOP, and RAPID OUT CANNED CYCLE

Format

(G9Ø)	**(G98)**	
or	**or**	**G85 X.... Y.... Z.... R...F.... ;**
(G91)	**(G99)**	



Sequence of moves:

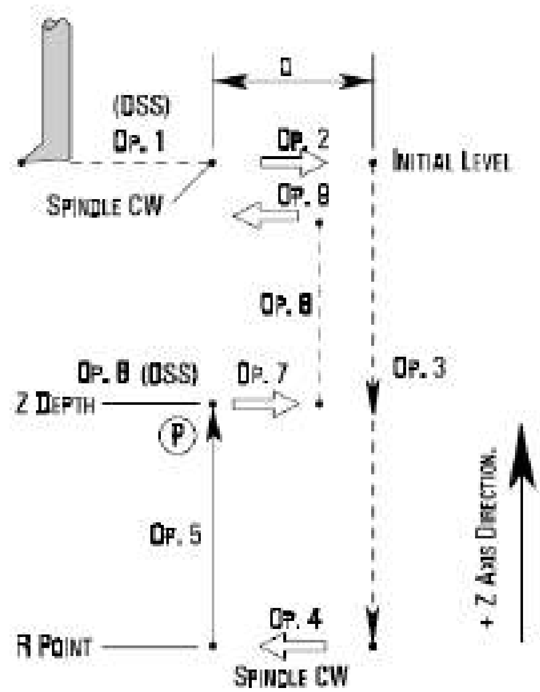
Op 1) Rapid position to X, Y and Z (the initial level).

Op 2) Rapid traverse to R point level.

Op 3) Feed to Z depth and spindle stop.

Op 4) Rapid traverse to the initial level and spindle CW for G98, or rapid traverse to R point level and spindle CW for G99.

1.9.16 G87 Back Boring Cycle



Format

(G9Ø) (G98)

or or G85 X.... Y.... Z.... R...Q…F.... ;

(G91) (G99)

Sequence of moves:

Op 1) Rapid position to X, Y and Z (the initial level).

Op 2) Spindle stop and orientation. Move the value of Q.

Op 3) Rapid traverse to R point level.

Op 4) Spindle CW and move back the value of Q.

Op 5) Feed in to Z depth (positive direction) and dwell P.

Op 6) Spindle stop and orientate.

Op 7) Move the value of Q.

Op 8) Rapid traverse to R point level.

Op 9) Move back the value of Q and spindle CW.

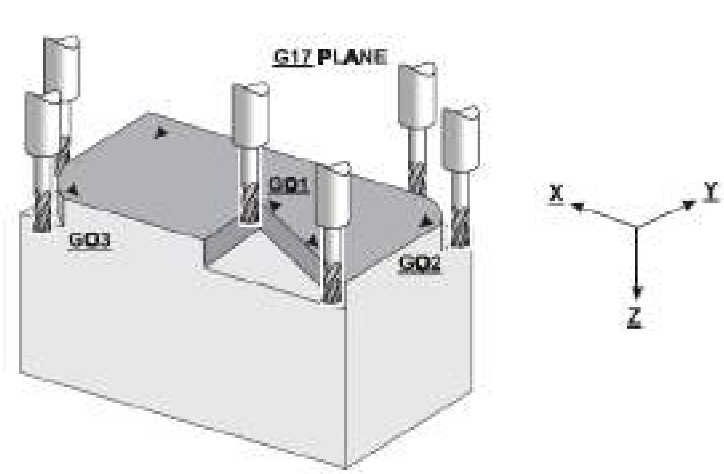
1.9.17 CIRCULAR PLANE SELECTION

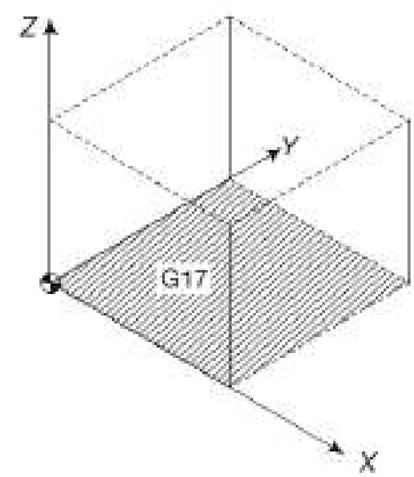
The plane used for circular motions must be comprised of two of the axes, X, Y, or Z. The plane selection is modal and stays in effect for all subsequent circular interpolation moves until you command another plane selection code.

There are three G codes used to select the circular plane**; G17 for the XY plane, G18 for the XZ plane, and G19 for the YZ plane.**

G17 XY CIRCULAR PLANE SELECTION

The G17 code is used to select the XY plane for circular motion. In this plane, circular motion is defined as clockwise for the operator looking down onto the X-Y table from above



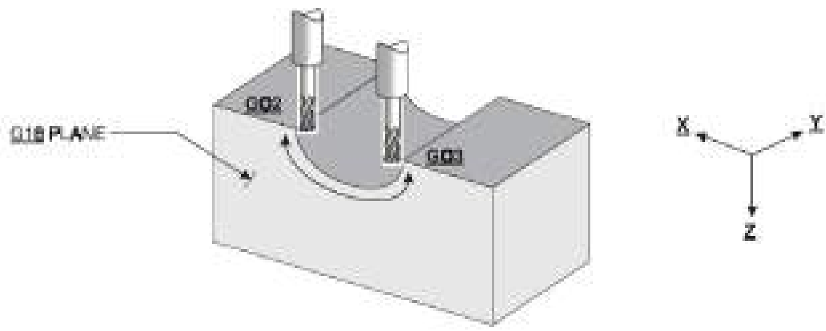


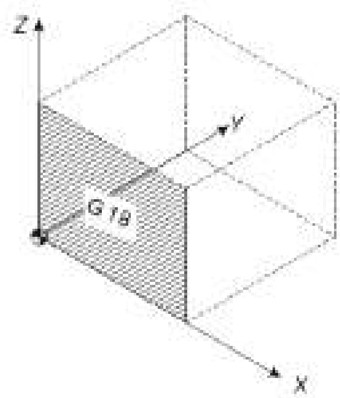
O01044 (G17 XY ARC PLANE EXAMPLE)
N1 T2 M06 (1/2 DIA. 2 FLT. END MILL)
N2 G90 G54 G00 X4. Y3.25 S2600 M03 (X Y start point of arc)
N3 G43 H02 Z0.1 M08
N4 G01 Z-.5 F50.
N5 G17 G02 X5.25 Y2. R1.25 F10. (G17 circular motion X Y plane)
N6 G00 Z1.
N7 X-.25 Y1. (G17 is default when you power up machine)
N8 G01 Z-.5 F50.
N9 G17 G03 X1. Y-.25 R1.25 F10. (G17 circular motion X Y plane)
N10 G00 Z1. M09
N11 G53 G49 Y0 Z0 M05

N12 M30

G18 Z - X CIRCULAR PLANE SELECTION

The G18 code is used to select the ZX plane for circular motion. In the X-Z plane (G18), circular motion is defined as clockwise for the operator looking from the rear of the machine out toward the control panel.

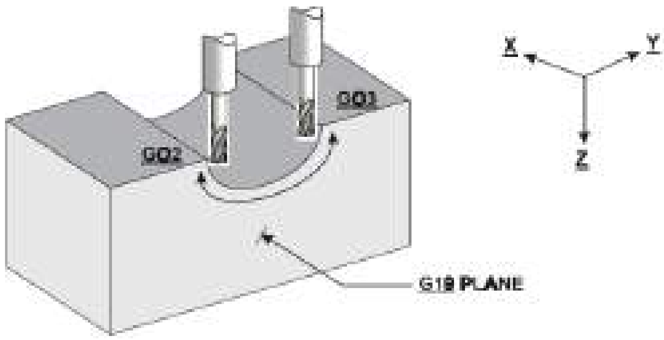


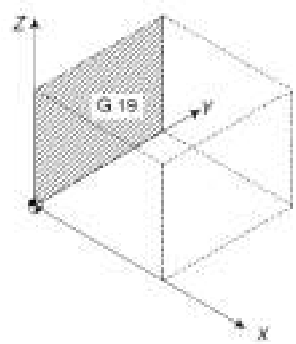


O01045 (MAIN PROGRAM G18 ZX ARC PLANE EXAMPLE)
N1 T3 M06 (1/2 DIA. 2 FLT. BALL END MILL)
N2 G90 G54 G00 X1.5 Y0. S2600 M03 (X Y start point of arc)
N3 G43 H03 Z0.1 M08
N4 G01 Z0. F20.
N5 M97 P100 L80 (Local sub-program call done 80 times with L80)
N6 G00 Z1. M09
N7 G53 G49 Y0. Z0. M05
N8 M30
(Local sub-program N100 called up by an M97 on line N5)
N100 G91 G01 Y-0.01
N101 G90
N102 G18 G03 X3. Z0. R.75 F12. (G18 circular motion Z X plane)
N103 G91 G01 Y-.01
N104 G90
N105 G18 G02 X1.5 Z0. R.75 F12. (G18 circular motion Z X plane)
N106 M99 (An M99 will cause the program to jump back to the next line after the M97 sub program call in the main program.)

G19 Y-Z CIRCULAR PLANE SELECTION

The G19 code is used to select the YZ plane for circular motion. In the Y-Z plane (G19), circular motion is defined as clockwise for the operator looking across the table from the side of





the machine the control panel is mounted.

O01045 (MAIN PROGRAM G19 YZ ARC PLANE EXAMPLE)
N1 T4 M06 (1/2 DIA. 2FLT BALL END MILL)
N2 G90 G54 G00 X.0 Y3. S2600 M03 (X Y start point of arc)
N3 43 H04 Z0.1 M08
N4 G01 Z0. F20.
N5 M97 P100 L80 (Local sub-program call done 80 times with L80)
N6 G00 Z1. M09
N7 G53 G49 Y0. Z0. M05
N8 M30
(Local sub-program N100 called up by an M97 on line N5)
N100 G91 G01 X-0.01
N101 G90
N102 G19 G03 Y1.5 Z0. R.75 F12. (G19 circular motion Y Z plane)
N103 G91 G01 X-0.01
N104 G90
N105 G19 G02 Y3. Z0. R.75 F12. (G19 circular motion Y Z plane)
N106 M99 (An M99 will cause the program to jump back to the next line after the M97 sub program call in the main program.)

Note related to Circular plane selection

The default plane selection when the machine is powered on is G17 for the X-Y plane. This means that circular motion in the plane of the X-Y table axis may be programmed without first selecting G17.

If cutter radius compensation is selected (G41 or G42), you can only use it for X-Y plane circular motions (G17). Cutter radius compensation is not available for circular motion in the G18 XZ or G19 YZ circular motion planes.

1.9.18 Tool Length Compensation (G43, G44 & G49)

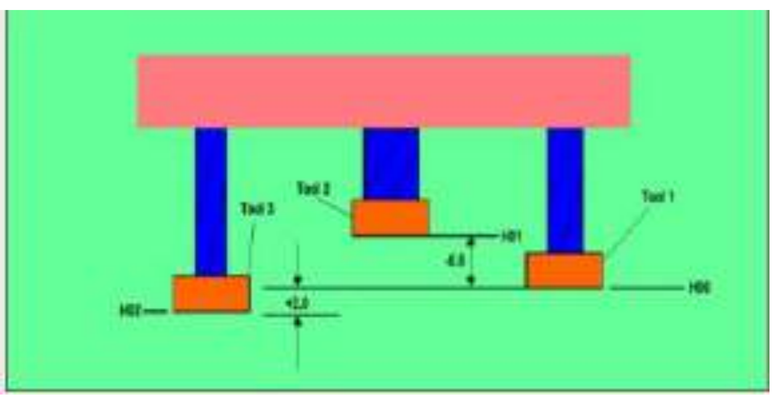
The procedure of mentioning the difference of length of tool assumed during programming and actual tool used for machining is called **"Tool length Compensation/offset."**

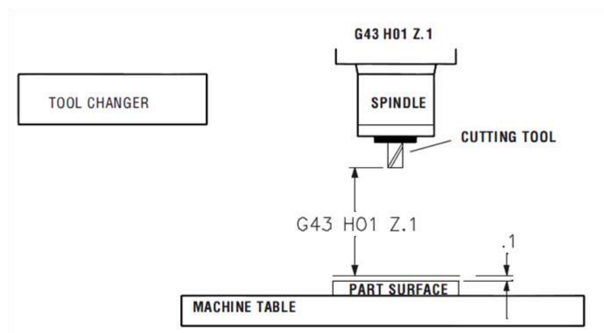
If more than one tool is being used, the first tool is set to zero on all axis (XYZ). To make life easier the first tool is given an offset(or length) of zero and subsequent tools are given a positive or negative offset based on the difference in length from tool No. 1.

Format:-

G43 Z_ H_;

G44 Z_ H_;





As shown in figure, tool 1 has been set to a tool length of H00. Now height of tool 2 is compared with first tool. The tool 2 is 5.5 mm lower than tool 1.when it is in contact with work, it is -5.5 mm lower than tool1. Now again the height of tool 3 is compared with frst tool. This time the tool is higher than tool no 1, so its offset value is positive, it means it is + 2.0 mm upper than tool 1.

The corresponding preparatory functions are

G43 - Positive Tool Length compensation

G44 - Negative Tool Length Compensation

G49 - Tool Length Compensation cancel.

1.10 OFTEN USED PREPARATORY (G) CODES:

G00 Rapid traverse motion; Used for positioning of non-cutting moves.

G01 Linear interpolation motion; Used for actual machining and metal removal. Program with a feedrate in inches (or mm) per minute. Maximum feedrate of machine is 300 inches per minute.

G02 Circular Interpolation - Clockwise

G03 Circular Interpolation - Counterclockwise

G28 Machine Home (Rapid traverse)

G40 Cutter Compensation m CANCEL

G41 Cutter Compensation LEFT

G42 Cutter Compensation RIGHT

G43 Tool Length Compensation – Positive

G44 Tool Length Compensation – Negative

G49 Tool Length Compensation - Cancel

G54 Work Coordinate #1 (Part zero)

G80 Canned Cycle Cancel

G81 Drill Canned Cycle

G82 Spot Drill Canned Cycle

G83 Peck Drill Canned Cycle

G84 Tapping Canned Cycle

G90 Absolute Programming Positioning

G91 Incremental Programming Positioning

G98 Initial Point Return in canned cycle

G99 Reference Plane Return in canned cycle

1.11 MISCELLANEOUS M CODES:

M00 The M00 code is used **to stop a program**. It also stops the spindle, turns off the coolant and stops interpretation lookahead processing. The CYCLE START button will continue program operation from the next block.

M01 The M01 command is identical to M00 except that it only stops if optional stop, OPT STOP key, is turned on from the control panel. A cycle start will continue program operation from the nest block.

M03 Starts the spindle Clockwise. Must have a spindle speed defined.

M04 Starts the spindle Counterclockwise. Must have a spindle speed defined.

M05 Stops the spindle.

M06 Tool change command along with a tool number will execute a tool change for that tool. This command will automatically stop the spindle, Z-axis will automatically move up to the machine zero position and the selected tool will be put in spindle. The coolant pump will turn off during a tool change.

M08 Coolant ON command.

M09 Coolant OFF command.

M30 Program end and rewind to beginning of program.

M97 Local subroutine call.

M98 Subprogram call.

M99 Subprogram return, or loop.

NOTE: Only one "M" code can be used per line. The "M" code will be the last command code to be performed in a line, regardless of where it's located in that line.

1.12 PROGRAM STRUCTURE

%	PROGRAM MUST BEGIN AND END WITH %
O01010	PROGRAM NUMBER, LETTER "O" WITH FOUR DIGIT NUMBER
(MILL PART PROGRAM EXAMPLE)	(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
N1 (DRILL 4 PLACES)	(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
T1 M06 (½ IN. DIA. STUB DRILL)	TOOL CHANGE TO TOOL #1, (NOTES TO OPERATOR)
G90 G54 G00 X-1.5 Y1.5 S1400 M03	ABS POSIT, WORK OFFSET#, RAPID X Y, SPINDLE ON CW
G43 H01 Z1. M08	TOOL LENGTH COMP #1, Z POSITION, COOLANT ON
G73 G99 Z-0.625 Q0.2R0.1 F5	HIGH SPEED PECK DRILLING, DRILL Z-.625 DEEP, .2 PECK
Y-1.5	DRILL ANOTHER HOLE RAPID PLANE IS AT R.1
/ X1.5	DRILL THIRD HOLE WITH AN OPTIONAL BLOCK DELETE

/ Y1.5	DRILL FOURTH HOLE WITH AN OPTIONAL BLOCK DELETE
G80 G00 Z1. M09	CANCEL CANNED CYCLE, RAPID Z1. COOLANT OFF
G28 G91 Z0 M05	RETURN Z TO MACHINE ZERO, SPINDLE OFF
N2 (COUNTERSINK 4 PLACES)	(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
T2 M06 (5/8 DIA. 90 DEG. C'SINK)	TOOL CHANGE TO TOOL #2, (NOTES TO OPERATOR)
G90 G54 G00 X-1.5 Y1.5 S900 M03	ABS POSIT, WORK OFFSET#, RAPID TO POSIT, SPINDLE ON CW
G43 H02 Z1. M08	TOOL LENGTH COMP #2, Z POSITION, COOLANT ON
G82 G99 Z-0.27 P0.5 R0.1 F12	SPOT DRILL CYCLE TO Z-.27 DEEP, DWELL .5 SECOND
Y-1.5	SECOND HOLE , RAPID PLANE IS AT R.1
: / X1.5	THIRD HOLE WITH AN OPTIONAL BLOCK DELETE
/ Y1.5	FOURTH HOLE WITH AN OPTIONAL BLOCK DELETE
G80 G00 Z1. M09	CANCEL CANNED CYCLE, RAPID Z1., COOLANT OFF
G28 G91 Z0 M05	RETURN Z TO MACHINE ZERO, SPINDLE OFF
N3 (SET D3 DIA. OFFSET VALUE TO .500)	(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
T3 M06 (½ DIA. END MILL)	TOOL CHANGE TO TOOL #3 (NOTES TO OPERATOR)
G90 G54 G00 X-2.3 Y2.3 S1100 M03	ABS POSIT, WORK OFFSET#, RAPID TO POSIT, SPINDLE ON CW
G43 H03 Z0.1 M08	TOOL LENGTH COMP #3, Z POSITION, COOLANT ON
G01 Z-0.625 F30	FEED TO DEPTH
G41 Y2. D03 F11	COMPENSATE CUTTER LEFT OF LINE
X2.0	CUT A 2.0 IN. SQUARE CW WITH TOOL DIA. COMP D03
Y-2.0	
X-2.0	
Y2.25	
G40 X-2.3 Y2.3	:G40 CANCELS COMP WHILE POSITIONING AWAY FROM PART
G00 Z1. M09	RAPID Z1., COOLANT OFF
G28 G91 Y0 Z0 M05	RETURN Y AND Z TO MACHINE ZERO
T1 M06	TOOL CHANGE BACK TO TOOL #1

M30 :	PROGRAM STOP AND REWIND TO BEGINNING
%	PROGRAM MUST BEGIN AND END WITH %

To change tools, all that is needed is an M06 even without a G28 in the previous line. A G28 can be specified to send all axes to machine home, or it can be defined to send a specific axis home with G28 G91 Z0 and/or Y0 and/or X0 to send just these axis specified to home position.

1.13 Some Word addresses

- A Angular dimension around X axis / Fourth axis rotary motion
- B Angular dimension around Y axis / Fifth axis rotary motion
- C Angular dimension around Z axis / Sixth axis rotary motion
- F Feed function
- G Preparatory function
- H Tool Length offset selection
- I Distance to arc centre or thread lead parallel to X
- J Distance to arc centre or thread lead parallel to Y
- K Distance to arc centre or thread lead parallel to Z
- Z Primary Z motion dimension
- M Miscellaneous function
- N Sequence number
- O Reference rewind stop
- S Spindle speed function
- T Tool function
- U Secondary motion dimension parallel to X*
- V Secondary motion dimension parallel to Y*
- W Secondary motion dimension parallel to Z*
- X Primary X motion dimension
- Y Primary Y motion dimension

1.14 SUB PROGRAM STRUCTURE.

A program which contains fixed sequences or frequently repeated patterns may be entered into memory as a Sub Program, in order to simplify the main program.

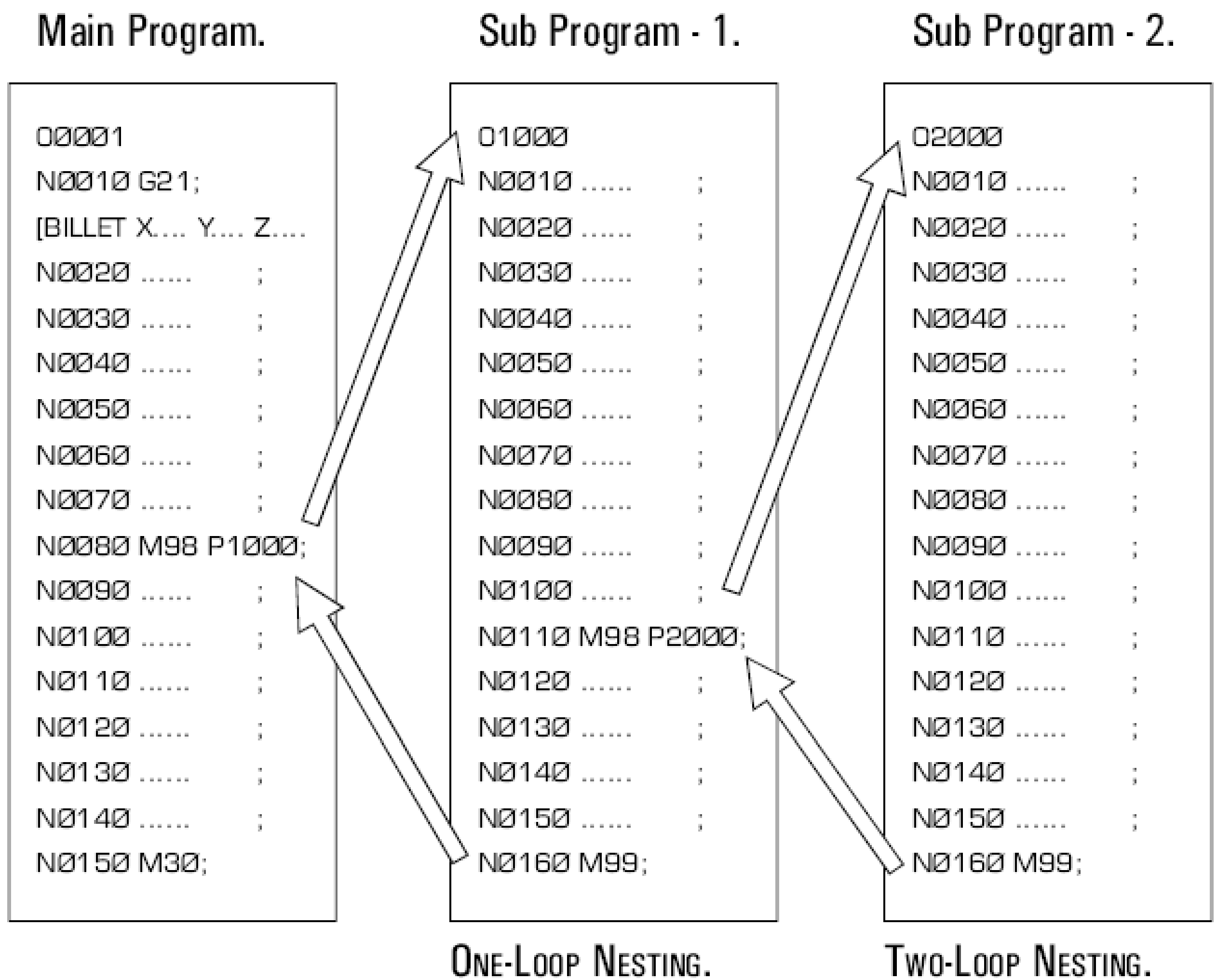
A sub program is entered into the machine controller memory in Edit Mode, in the same manner as the main program.

Differences between a sub and main program :

1) A sub program does not have a billet size definition at the top of the program listing.

2) A sub program is ended by the M99 code.

The sub program can be called into operation when the machine is set to run in Auto Mode. Sub programs can also call other sub programs into operation. When the main program calls one sub program into operation, the process is called a one-loop sub program call. It is possible to program a maximum four loop sub program call within the main program. Shown below is an illustration of a two-loop sub program call.



SUB PROGRAM COMMANDS - GENERAL NOTES.

NOTE 1.

A sub program must be saved to memory using a four digit number.

NOTE 2.

If cutter compensation is required on a tool and the co-ordinates for the tool are within the sub program, the cutter compensation must be applied and cancelled within the sub program.

NOTE 3.

To call a sub program the M98 code is used followed by PØØØØ (the number of the sub program required).

For example,

M98 P2ØØØ

This command is read call program number 2ØØØ.

NOTE 4.

A sub program call command (M98 PØØØØ) can be specified along with a move command in the same block.

For example,

GØ1 X42.5 M98 P1ØØØ;

NOTE 5.

At the end of a sub program, the M99 code is entered. This returns control to the main program. The M99 code will return control to the next block after the M98 sub program call block in the main program.

If the code M99 PØØØØ is entered, control will pass to the main program at a block with the N number equal to that of the P number stated after the M99 code.

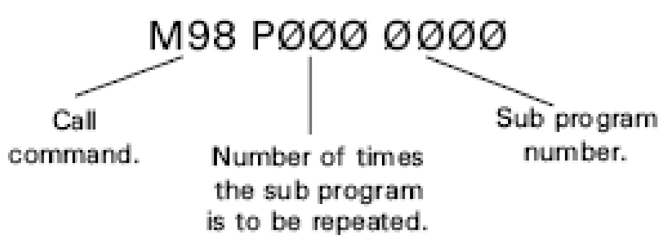
For example,

M99 PØ16Ø

This command is read return to the main program at block number NØ16Ø.

SUB PROGRAM REPEAT COMMAND:

A call command can be set to call a sub program repeatedly. This call can specify upto 999 repetitions of a sub program. A sub program repeat command has the following format :

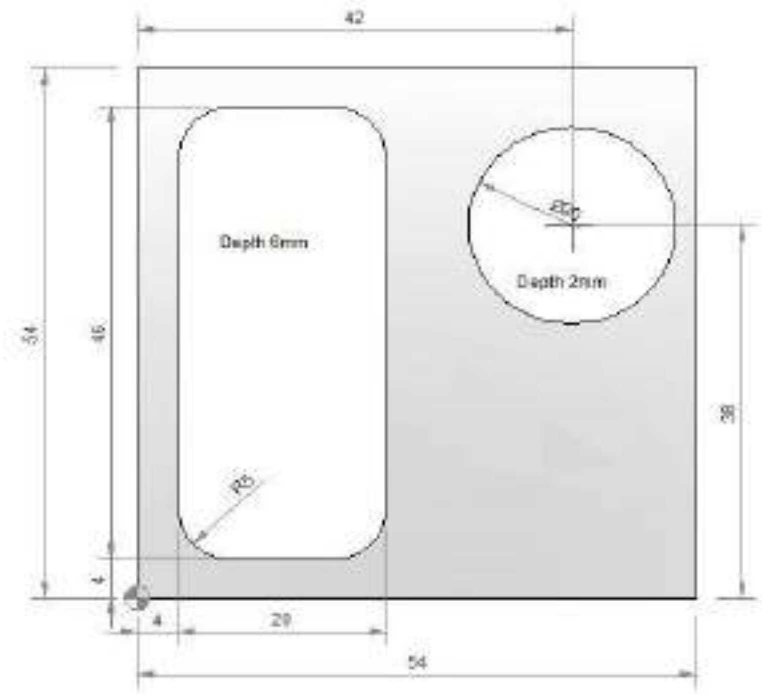


When the repetition is omitted, the sub program will be called once only.

For example,

M98 P1Ø ØØØ1

This command is read call the sub program number ØØØ1 ten times.



EXAMPLE - CNC Pocket Milling Program using Subprogram

Main Program

Milling cutter diameter: 10mm

```
N05   G55
N10   M6    T2    H3    G43   M3
N15   S1200       F60
N20   G0    X9    Y9    Z1
N25   G1    Z0
N30   M98   P03   0035
N35   G0    Z1    G90
N40   X42   Y38
N45   G1    Z-5   F30
N50   X47   F300
N55   G3    X47   Y38   I-5   J0
N60   G0    Z100
N65   G49
N70   M30
```

SUBPROGRAM

```
O 0035
N05   G1    Z-2   G91   F30
N10   X10   F100
N15   Y36
N20   X-10
N25   Y-36
N30   M99
```

ADDITIONAL PREPARATORY FUNCTIONS FOR TURNING CENTRES

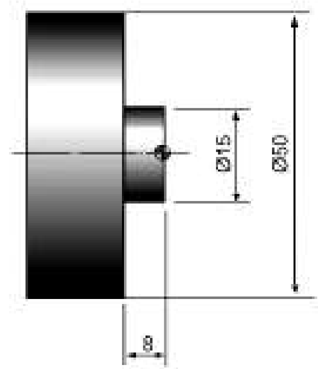
G76 THREADING CYCLE

G90 ABSOLUTE PROGRAMMING / TURNING CYCLE / TAPER TURNING CYCLE

G 72 /G94 FACING CYCLE

GROOVING CYCLE

KNURLING CYCLE



1.15 G94 FACING CYCLE

Fanuc G94 One Pass Facing Cycle

Fanuc G94 G code is used for rough facing.

Fanuc G94 facing cycle is used for simple facing (one-pass facing) however multiple passes are possible by specifying the Z-axis location of additional passes.

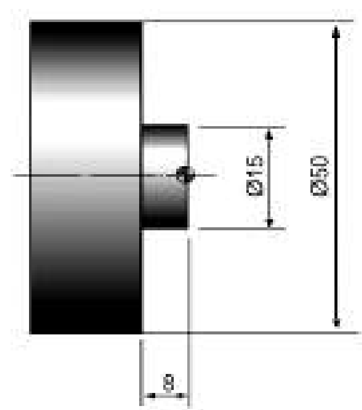
G94 X… Z…F….;

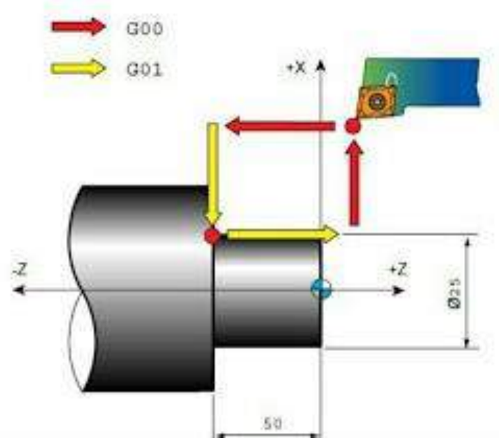
X: End point in X-axis.

Z: End point in Z-axis.

CNC program code using G94 Facing cycle

```
O0001
N10 G28;
N15 M06 T01;
N20 G90 M03 S1200;
N25 G00 X55 Z2;
N30 G94    X15   Z-2    F10;
N35              Z-4;
N40              Z-6;
N45              Z-8;
N50 G28 M05
N55 M30
```





CNC program code without using Facing cycle

O0001 (Note :- Without cycle, the number of lines are more)

```
N10 G28;
N15 M06 T01;
N20 G90 M03 S1200;
N25 G00    X55   Z2;
```

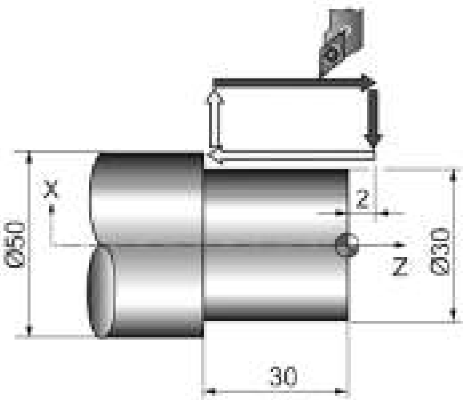
```
N30 G01           Z-2    F10;
N35         X15;
N40               Z2;
N45         X55;
N50               Z-4
N55         X15;
N60               Z2;
N65         X55;
N70               Z-6;
N75         X15;
N80               Z2;
N85         X55
N90               Z-8;
N95         X15;
N100              Z2;
N105        X55;
N110  G28   M05
N115  M30
```

1.16 G90 TURNING CYCLE

G90 turning cycle is called with many names like, G90 Fixed Cycle, G90 Straight Cutting Cycle, G90 Rough Turning Cycle, G90 Rough Turning Canned Cycle, G90 Box Cycle.

G90 turning cycle is used for simple turning however multiple passes are possible by specifying the X-axis location of additional passes.

G90 turning cycle is used for simple turning however multiple passes are possible by specifying the X-axis location of additional passes.



G90 turning cycle can be used for

- Straight turning.
- Boring operation.
- Taper cutting.

Format

G90 X….Z….F….;

OR

G90 U… W… F…;

X – Diameter to be cut.

Z – End point in z-axis.

U – x-axis incremental distance to target.

W – z-axis incremental distance to target.

Cancellation of G90 Turning Cycle

G90 turning cycle is a modal G code.

"Modal" G code meaning that they stay in effect until they are cancelled or replaced by a contradictory G code. It means G90 turning cycle remains active until another motion command is given like G00, G01 etc. As in the cnc program example above G90 G code is cancelled with G30 G code.

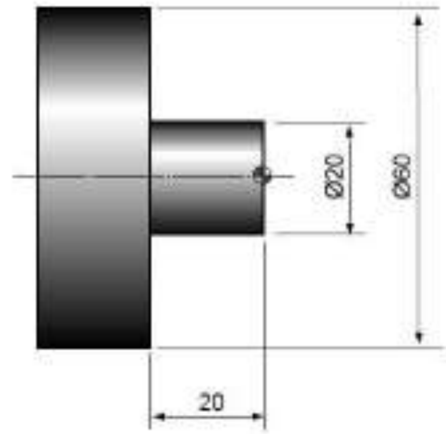
Put simply G90 turning cycle must be ended by making a motion command like G00, G01 etc.

Sample program for the component shown in figure

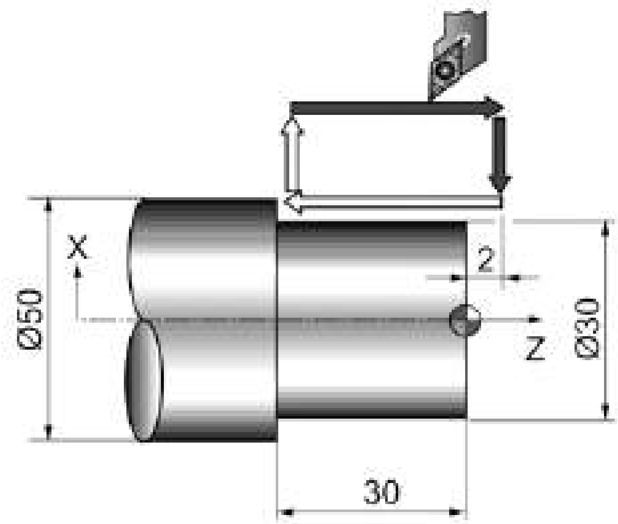
O0001

G28;
M06 T01;
G90 M03 S1200;

```
G00    X65    Z2;
G90    X55    Z-20    F20;
       X50;
       X45;
       X40;
       X35;
       X30;
       X25;
       X20;
G28 M05;
M30
```



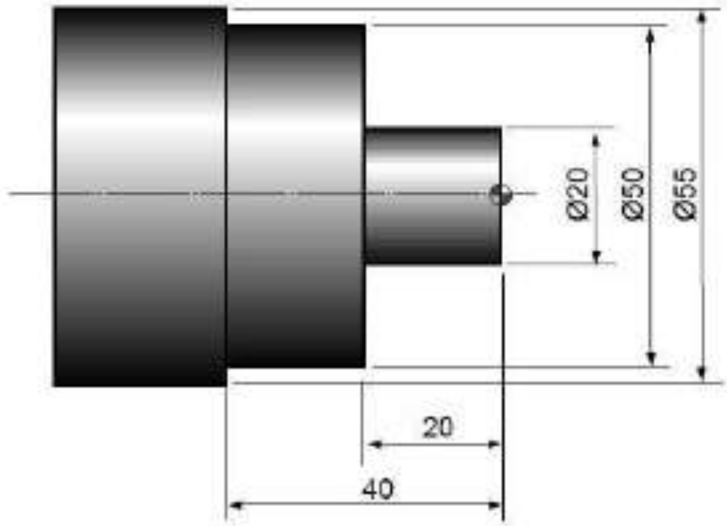
Sample program for the component shown in figure using U and W (Incremental)




```
O0001
G28;
M06 T01;
G90 M03 S1200;
G00    X55    Z2;
G90    X50    W-32    F20;
       X45;
       X40;
       X35;
       X30;
G28 M05
```

M30

Example – Step Turning



O0001

G28;

M06 T01;

G90 M03 S1200;

G00 X60 Z2;

G90 X55 Z-40 F10;

X50;

X45 Z-20;

X40;

X35;

X30;

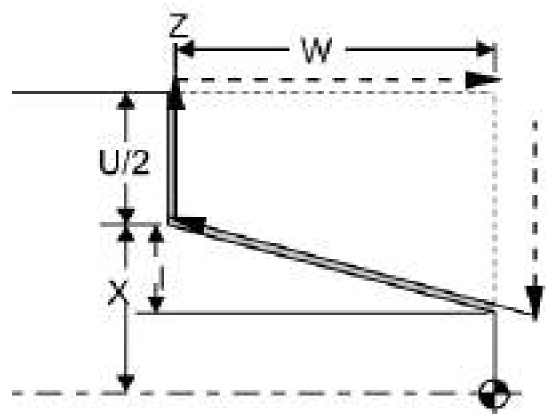
X25;

X20;

G28 M05

M30

1.17 G90 TAPER TURNING CYCLE



G90 rough turning canned cycle can be used for Straight turning and also for taper turning operations

Format

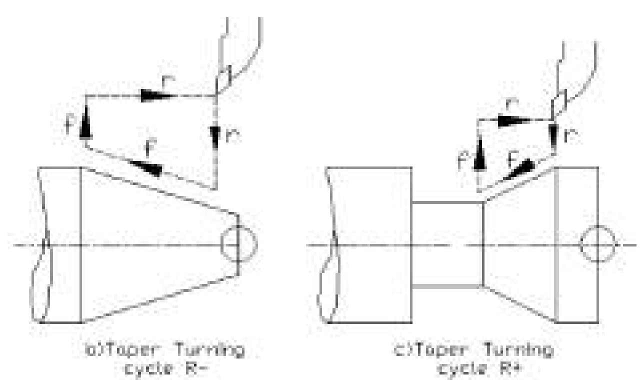
G90 X….Z….I….;

OR

G90 X… Z… R…;

OR

G90 U….W…..;



X – Diameter to be cut.

Z – End point in z-axis.

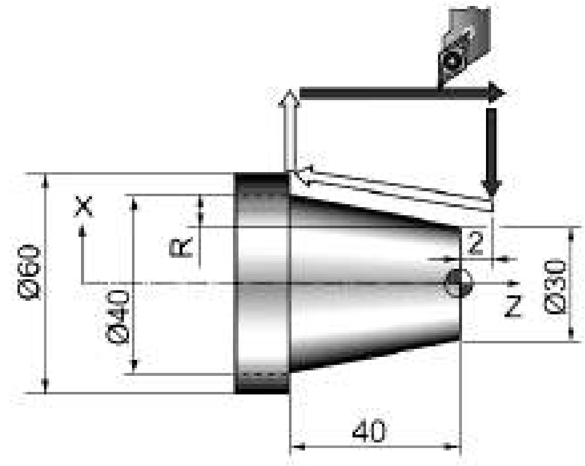
I & R – Distance and Direction of Taper (Radius

Value). It is the difference in incremental value of start cut radius and the end cut radius)

U – x-axis incremental distance to target.

W – z-axis incremental distance to target.

Example :- Taper Turning with G90 Turning Cycle



O0001

G28;

M06 T01;

G90 M03 S1200;

G00 X61 Z2;

G90 X55 Z-40 F10;

X50;

X45;

X40;

G90 X40 Z-40 R-2.5 F10;

R-2.5;

G00;

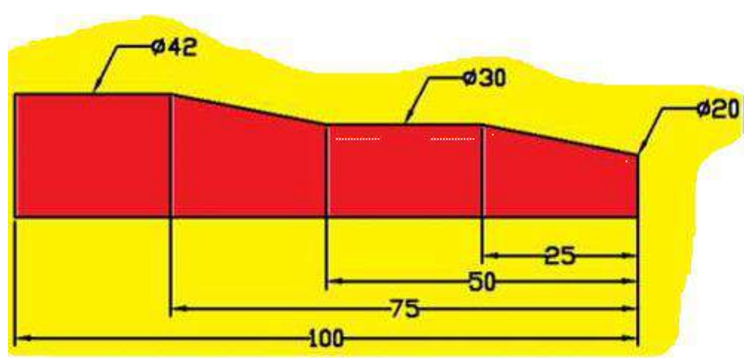
G28 M05;

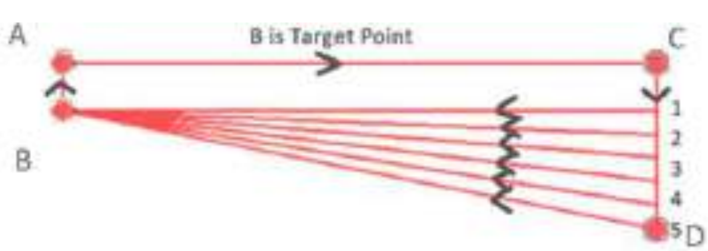
M30;

SAMPLE PROGRAM

Billet size⇒diameter 42,length 100

Tool used⇒Left hand tool(T1)





Depth of cut = 1 mm

G27 G97 G98

G28 U0 W0

M06 T1

M03 S3000

G00 X42 Z2 ***//simple turning***

```
G90  X40  Z-50  F60
     X38
     X36
     X34
     X32
     X30
```

G00 X44 Z-50 **//taper turning with R=6** $(\frac{30-42}{2}=6)$

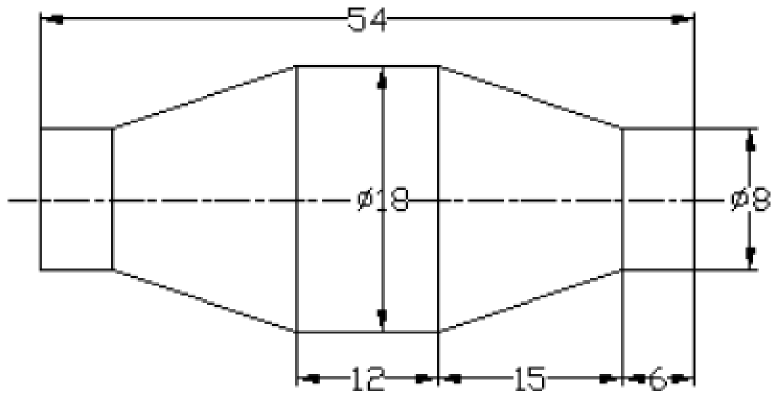
```
G90  X42  Z-70  R-1  F60
                R-2
                R-3
                R-4
                R-5
                R-6
```

G00 X22 Z0 **//taper turning with R=5** $(\frac{20-30}{2}=5)$

```
G90  X30  Z-25  R-1  F60
                R-2
                R-3
                R-4
                R-5
G28 U0 W0
M05
M30
```

EXAMPLE

Write a manual part program for Taper Turning Operation for the component shown in figure below.



BILLET SIZE = Φ22 X 54 MM

Depth of cut = 0.5 mm

Spindle speed = 1200 rpm Feed = 35mm/min

(CNC program for Taper Turning)

```
O1008                          --------- Program Number 1008
[BILLET X22 Z54]               --------- Defining Billet size dia : 22 length 54 mm
G21 G98                        --------- Initial settings
G28 U0 W0                      --------- Going to home position
M06 T0303                      ---------- Selecting Tool No. 3 with offset No 3
M03 S1200                      ---------- Setting spindle speed at 1200 rpm
G00  X22   Z1                  ---------- Tool moving to tool entry point X22 Z1
G90  X22   Z-54 F35 ----------- G90 Step Turning cycle
     X21
     X20
     X19
     X18
     X17   Z-6
     X16
     X15
     X14
     X13
     X12
     X11
     X10
     X9
```

X8

G00 X18 Z-6

G90 X18 Z-21 R0 F30 -------- Taper Turning – R= -5 $(\frac{8-18}{2}=5)$

R-1

R-2

R-3

R-4

R-5

G01 X20 Z-33

G90 X8 Z-48 R0 F50 -------- Taper Turning – R= +5 $(\frac{18-8}{2}=5)$

R1

R2

R3

R4

R5

G28 U0 W0 -------- Going to home position

M05 -------- Stop the spindle

M30 -------- Program stop and rewind.

1.18 G71 MULTIPLE TURNING CYCLE

This multiple turning cycle is used when the major direction of cut is along the 'Z' axis. This cycle causes the profile to be roughed out by turning. G71 turning cycle makes large diameter cutting easy. Cutting can be done in simple straight line or a complex contour can also be machined very easily. Two G71 blocks are needed to specify all the values. Through G71 turning cycle parameters cnc machinists can control

- Depth of cut.
- Retract height.
- Finishing allowance in x-axis and z-axis.
- Cycle cutting-feed, spindle speed.

FORMAT

G71 U(*u1)…. R…..

G71 P….. Q….. U(*u2)…… W….. F…… S……

Where

u1	–	Depth of cut (Radius Designation).
R	-	Relief amount / Retract height,
F	–	Feed rate,
S	-	Spindle Speed during G71 cycle
P	-	Line or block number of the start of the final profile.
Q	-	Line or block number of the end point of the final profile,
U2	–	Finishing allowance in the X axis.
W	-	Finishing allowance in the Z axis.

EXAMPLE

```
N60   G71   U10   R10
N70   G71   P80   Q90   U3    W0    F0.25
N80   G00   X60
N90   G01   Z-75
```

When G71 turning cycle is run the whole operation will be done in following sequence,

First-cut

1 – Tool will move in x-axis U (depth of cut) deep with programmed feed from starting-point.

2 – Tool will travel with feed in z-axis (destination point in z-axis is given in P Q blocks)

3 – Tool rapidly retracts R amount in both x-axis and z-axis (at 45 degrees).

4 – Tool rapidly travel in z-axis to start-point

Later Cuts

5 – Tool rapidly moves to last cut depth.

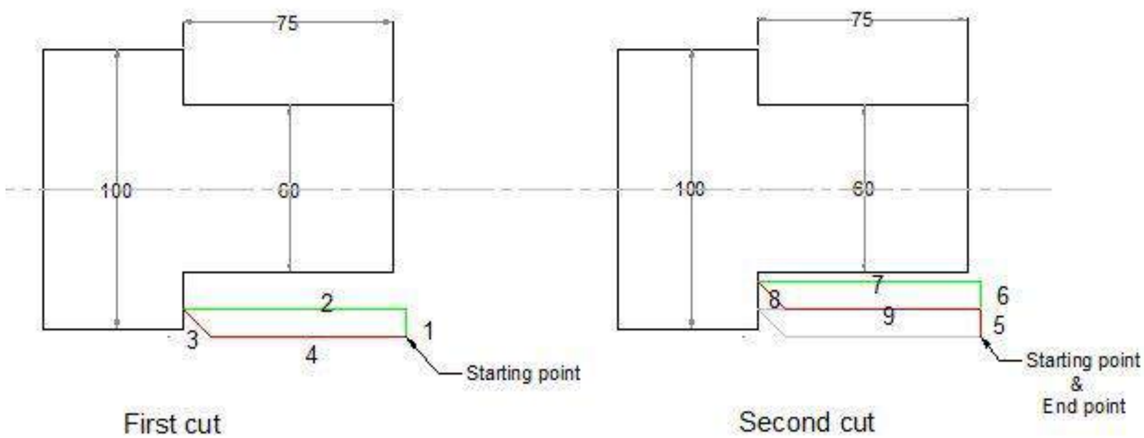
6 – Tool moves with feed in x-axis U deep (first-block U depth of cut).

7 – Tool with feed moves in z-axis (destination point given in P Q blocks).

8 – Tool rapidly retracts in x-axis and z-axis R amount (45 degrees).

9 – Tool rapidly moves to start-point only in z-axis.

This whole sequence of operation keep on going, until the destination point in x-axis is met. If finishing allowance is given tool will not make the exact diameter and length given in P Q blocks but will leave that much allowance, This finishing allowance can be later machined by calling G70 finishing cycle.



Fanuc G71 Turning Cycle

Fanuc G71 Example

Here is a cnc part-program which shows how G71 turning cycle can be used, this is the program for the drawing given above

N50 G00 X106 Z5 M03 S800

N60 G71 U10 R10

N70 G71 P80 Q90 U3 W0 F0.25

N80 G00 X60

N90 G01 Z-75

In this program G71 turning cycle will keep repeating the contour given inside P Q blocks shown below

```
N80  G00  X60

N90  G01  Z-75
```

These two cnc program blocks tell us that we want to remove material till X60 deep and in Z-75 in length. The depth of cut is given in first-block U10 retract amount is also given R10. Finishing allowance in x-axis is U3 but there is no finishing allowance given in z-axis W0.

1.19 G70 Finishing Cycle

If you programmed G71 turning cycle with finishing allowances then that finish allowances can be removed with G70 finishing cycle. G70 finishing cycle repeats the whole contour the G71 way, but in just one-cut removing the finishing allowances.

Why Use G70 Finishing Cycle

As material can be removed with G71 turning cycle, but if you want a different cutting-feed and spindle speed for the last cut, then it is recommended that you use G70 finishing cycle. G70 finishing cycle use F and S values which are given inside P Q programmed blocks. (G71 use F S values which are given inside G71 second block.)

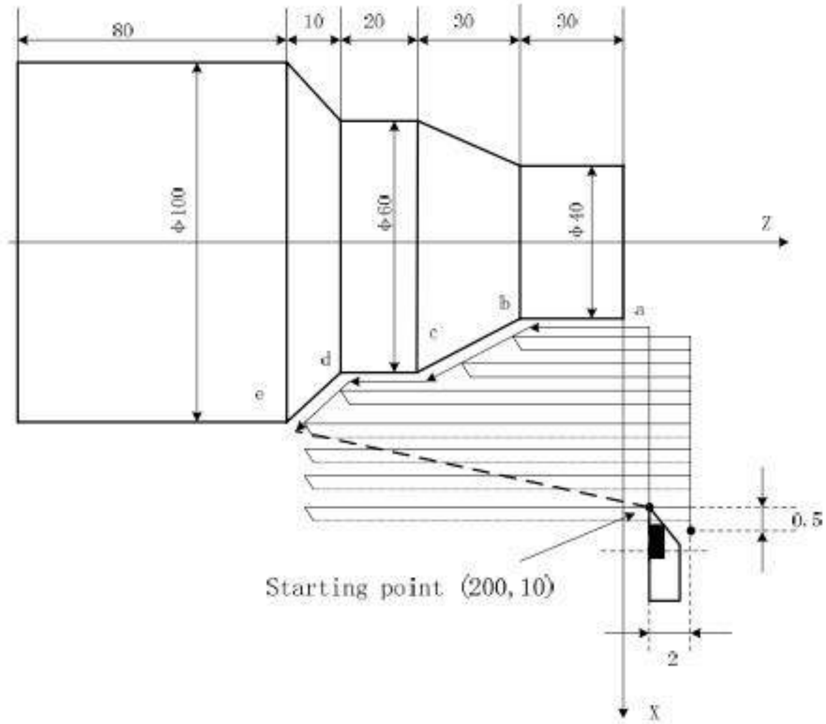
Fanuc G70 Example

```
N50  G00  X106  Z5    M03  S800

N60  G71  U10   R10

N70  G71  P80   Q90   U3   W0   F0.25

N80  G01  X60

N90  G01  Z-75  F0.15

N100 G0   0     X200  Z100

N110 G92  S1200

N120 T03  G96   S150  M03

N130 G00  X106  Z5

N140 G70  P80   Q90

N150 G00  X200  Z100
```

```
N160  M30
```

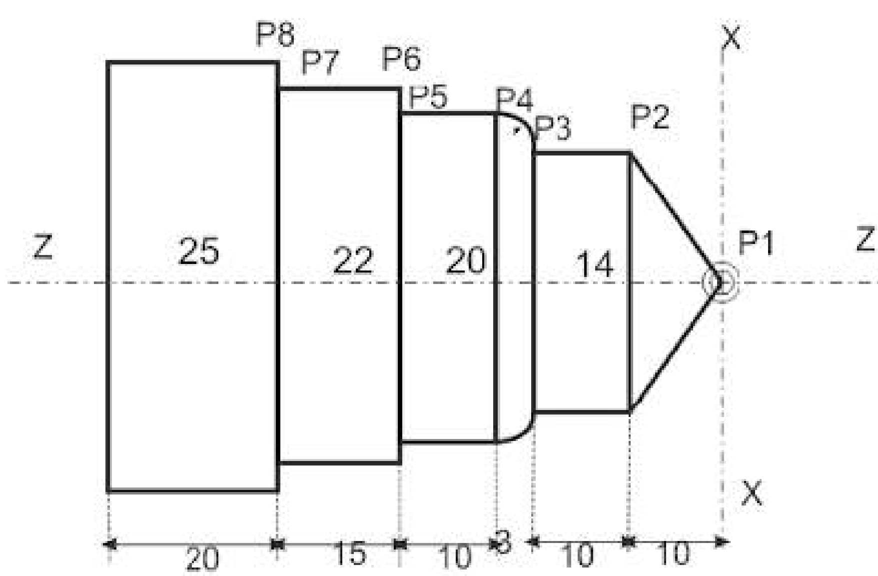
G70 G71 Example

```
O 0004
N50   G00   X200  Z10    M3    S800
N60   G71   U2    R1     F200
N70   G81   P80   Q120   U0.5  W0.2
N80   G01   X40   S1200
N90   G01         Z-30   F100
N100  X60   W-30
N110        W-30
N120  X100  W-10
N130  G70   P80   Q120
M30
```



EXAMPLE

FANUC STOCK REMOVAL ROUGH TURNING CYCLE G71 ..

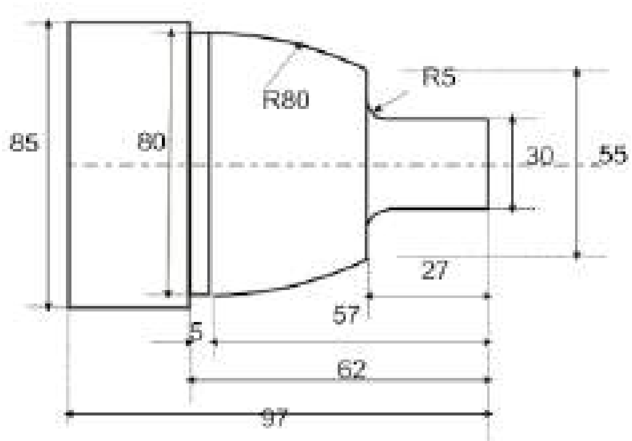


04253	04253- Name of main program.

N10 G90 G21 G99 F0.2 ;	N10- Absolute co-ordinate system , metric input in 'mm' , feed rate per revolution F0.2
N20 M06 T04 04 ;	N20- Tool changing command , select tool no 4
N30 M03 S1200 ;	N30- Spindle ON clockwise at speed 1200 rpm.
N40 M08 ;	N40- Coolant ON
N50 G28 U0 W0 ;	N50- Tool at reference point where U0 and W0
N60 G71 U0.5 R1 ;	N60- Turning cycle command where depth of cut for X axis is 0.5(total dia reduce each cut is 1) and tool relive distance 1 mm.
N70 G71 P80 Q150 U0.5 W0.1 ;	N70- Turning cycle command , actual operation start from block no. 80 and actual operation end at block no. 150 ,finishing along X axis 0.5 , and finishing along Z axis is 0.1
N80 G00 X0 Z0 ;	N80- Rapid command where tool at X 0 and Z 0 .[P1]
N90 G01 X14 Z-10 ;	N90- Linear interpolation command where X14 and Z-10 .[P2]
N100 G01 Z-20 ;	N100- Linear interpolation command where X14 and Z-20 .[P3]
N110 G03 X20 Z-23 R3 ;	N110- Circular interpolation counter clockwise (external radius), where X20 ,Z-23 and R3 .[P4]
N120 G01 Z-33 ;	N120- Linear interpolation command where X20 and Z-33 .[P5]
N130 G01 X22 ;	N130- Linear interpolation command where X22 and Z-33 .[P6]
N140 G01 Z-48 ;	N140- Linear interpolation command where X22 and Z-48 .[P7]
N150 G01 X25 ;	N150- Linear interpolation command where X25 and Z-48 .[P8]
N160 G28 U0 W0 ;	N160- Tool at reference point where U0 and

	W0.
N170 M05 M09 M30 ;	N170- Spindle stop , coolant off, main prog end .

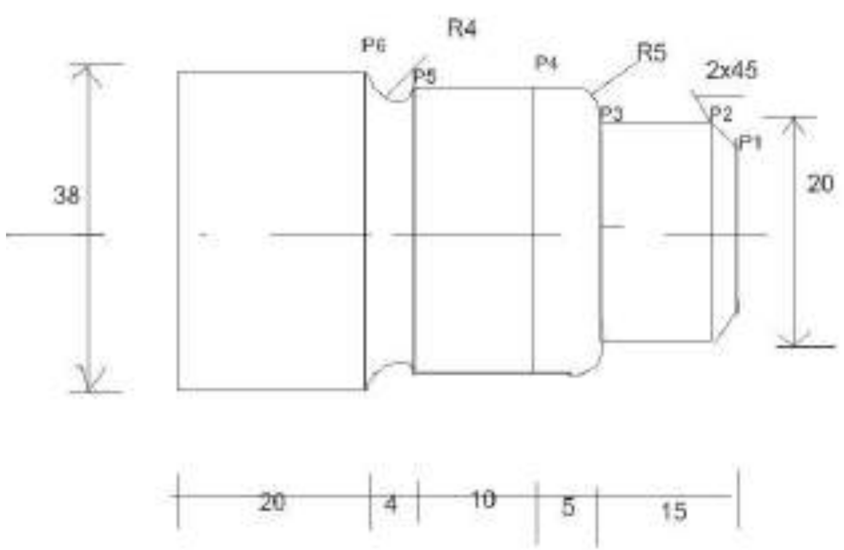
EXAMPLE - FANUC CNC STOCK REMOVAL ROUGH TURNING CYCLE G71



04253	04253- Name of main program.
N10 G90 G21 G99 F0.2 ;	N10- Absolute co-ordinate system , metric input in 'mm' , feed rate per revolution F0.2 .
N20 M06 T04 04 ;	N20- Tool changing command , select tool no 4.
N30 M03 S1200 ;	N30- Spindle ON clockwise at speed 1200 rpm.
N40 M08 ;	N40- Coolant ON
N50 G28 U0 W0 ;	N50- Tool at reference point where U0 and W0
N60 G71 U0.5 R1 ;	N60- Turning cycle command where depth of cut for X axis is 0.5(total dia reduce each cut is 1) and tool relive distance 1 mm.
N70 G71 P80 Q150 U0.5 W0.1 ;	N70- Turning cycle command , actual operation start from block no. 80 and actual operation end at block no. 150,finishing along X axis 0.5 , and finishing along Z axis is 0.1
N80 G00 X0 Z0 ;	N80- Rapid command , where X0 and Z0
N90 G01 X30 ;	N90- Linear interpolation command where X30 and Z0
N100 G01 Z-22 ;	N100- Linear interpolation command where X30 and Z-22
N110 G02 X40 Z-27 CR=5 ;	N110 - Circular interpolation clockwise command where X40 , Z-27 and radius 5 .

N120 G01 X55 ;	N120- Linear interpolation command where X55 and Z-27
N130 G03 X80 Z-57 CR=80 ;	N130- Circular interpolation counter clockwise command where X80 , Z-57 and radius 80 .
N140 G01 Z-62 ;	N140- Linear interpolation command where X80 and Z-62 ;
N150 G01 X85 ;	N150- Linear interpolation command where X85 and Z-62 ;
N160 G70 P80 Q150 F0.05 ;	N160- Finish machining cycle , actual operation start from block no. 80 and actual operation end at block no. 150 , feed rate for finishing cut 0.05
N170 G28 U0 W0 ;	N170- Tool at reference point where U0 and W0
N180 M05 M09 M30 ;	N180- Spindle stop , coolant off, main prog end .

EXAMPLE - FANUC CNC STOCK REMOVAL ROUGH TURNING CYCLE G71



04252	04252- Name of main program
N10 G90 G21 G99 F0.2 ;	N10- Absolute co-ordinate system , metric input in 'mm' , feed rate per revolution F0.2 .
N20 M06 T04 04 ;	N20- Tool changing command, select tool no 4.
N30 M03 S1200 ;	N30- Spindle ON clockwise at speed 1200 rpm.
N40 M08 ;	N40- Coolant ON
N50 G28 U0 W0 ;	N50- Tool at reference point where U0 and W0
N60 G71 U0.5 R1 ;	N60- Turning cycle command where depth of cut for X axis is 0.5(total dia reduce each cut is 1) and tool relive distance

	1 mm
N70 G71 P80 Q130 U0.5 W0.1 ;	N70- Turning cycle command , actual operation start from block no. 80 and actual operation end at block no. 130 , finishing along X axis 0.5 , and finishing along Z axis is 0.1
N80 G00 X16 ;	N80- Rapid command where tool at X 16 and Z 0 .[P1]
N90 G01 X20 Z-2 ;	N90- Linear interpolation its means tool start turning at X20 and Z-2.[P2](chamfering)
N100 G01 Z-15 ;	N100- Linear interpolation where X at same position and Z goes to -15. [P3]
N110 G03 X30 Z-20 R5 ;	N110- Circular interpolation counter clockwise (outer radius) X30 , Z-20 and R5. [P4]
N120 G01 Z-30 ;	N120- Linear interpolation where X at same position and Z goes to -30.[P5]
N130 G02 X38 Z-34 R4 ;	N130- Circular interpolation clockwise (internal radius) X38 , Z-34 and R4. [P6]
N140 G28 U0 W0 ;	N140- Tool at reference point where U0 and W0
N150 M05 M09 M30	.N150- Spindle stop , coolant off, main prog end .

Write a manual part program for Multiple Turning Operation for the component shown in figure below.

BILLET SIZE : Φ32 X 60

MATERIAL: ALUMINIUM

Multiple Rough turning; Tool Holder :SDJCR 1212H11; TOOL TIP: DCMT11T304; TOOL STATION:1, SPINDLE SPEED :1200 rpm; FEED: 35m/min

FINISHING : Tool Holder : SDJCR 1212H11; TOOL TIP: DCMT11T302; TOOL STATION : 2; SPINDLE SPEED : 1450rpm, FEED: 25m/min

(CNC program for Multiple Turning

O1009

[BILLET X32 Z60]

G21 G98

G28 U0 W0

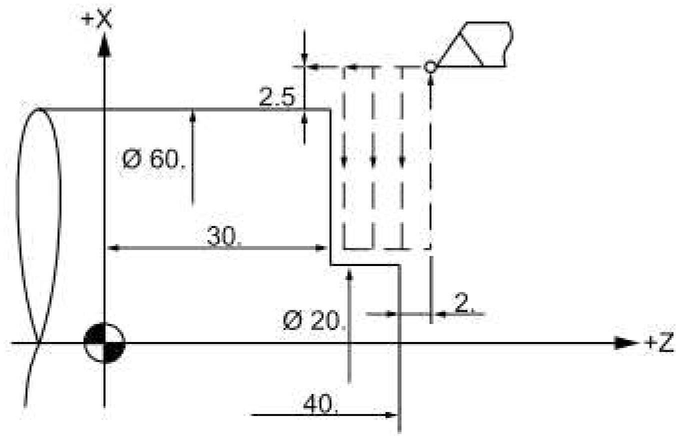
M06 T0303

M03 S1200

G00 X32 Z1

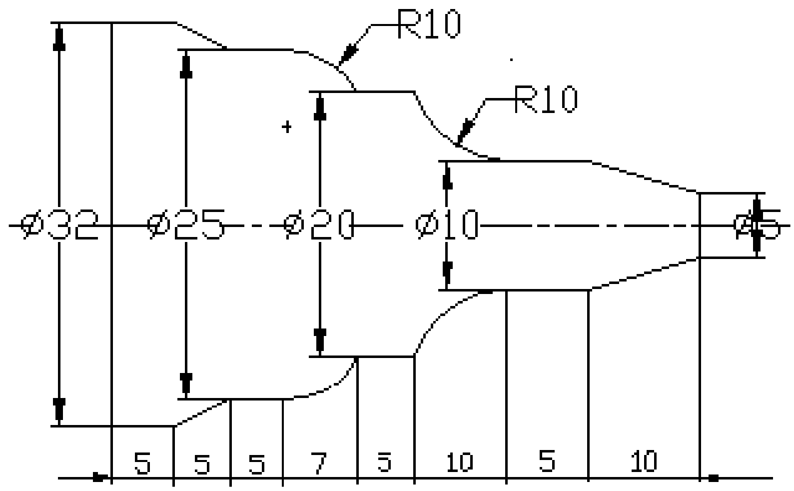
[G71 MULTIPLE TURNING
Depth of cut for each pass U=0.5 mm
Relief amount R= 1.0 mm
P and Q : Beginning and end of cycle sequence Nos. (Allowances on X(U) and Z(W) axis=0.1 mm respectively.
Feedrate= 35 mm/min.]

```
G71   U0.5  R1
G71   P10   Q20   U0.1   W0.1   F35
N10   G01         Z0
N20   G01   X5
N30   G01   X10   Z-10
N40   G01         Z-15
N50   G02   X20   Z-25   R10
N60   G01         Z-30
N70   G03   X25   Z-37   R10
N80   G01         Z-42
N90         X30   Z-47
N20               Z-52
G28 U0 W0
```



M06 T0202 ---------- ***Using RH Finishing tool.***

```
M03 S1450
      G00   X32   Z1
      G70   P10   Q20   F25
G28 U0 W0
M05
M30
```



1.20 G72 MULTIPLE FACING CYCLE

This multiple facing cycle is used when the major direction of cut is along the "X" axis. This cycle causes the profile to be roughed out by facing. Control passes on to after the last block of the profile. Two G72 blocks are needed to specify all the values.

FORMAT

G72 W….R…..

G72 P… Q… U… W… F…. S….

Where

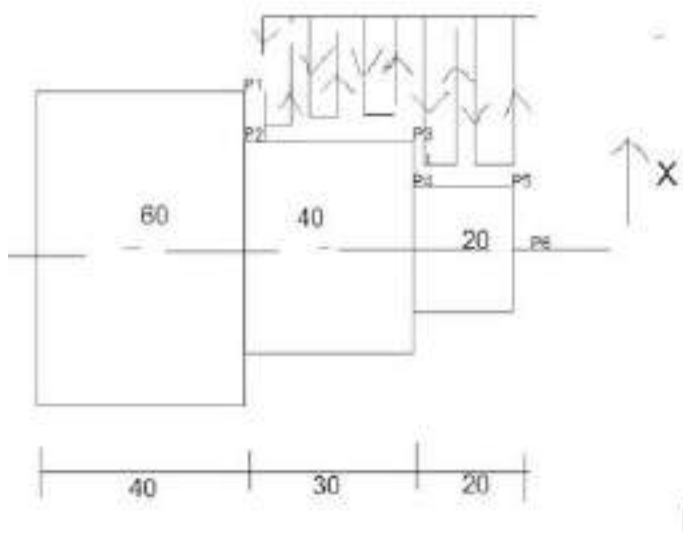
W – Depth of cut in Z axis

R – Escape of relief amount

P - The line number in the program marking the start of the finished form required.

Q - The line number in the program marking the end of the finished form required.

U – The amount and direction of the finishing allowance left in the X axis



W – The amount and direction of the finishing allowance left in the Z axis

F – Feed rate

S – Speed

The benefit of using F (feed-rate) in G72 second block is that during facing cycle machine will use this feed-rate, and will ignore any feed-rates given between P block and Q block program. The feed-rate given between P block and Q block program will only be used if you call G70 Finishing Cycle later in program with same P block and Q block numbers. This is very handy way gives cnc machinist opportunity to keep different feed-rates for “rough facing cuts” and “final finishing cut”.

EXAMPLE

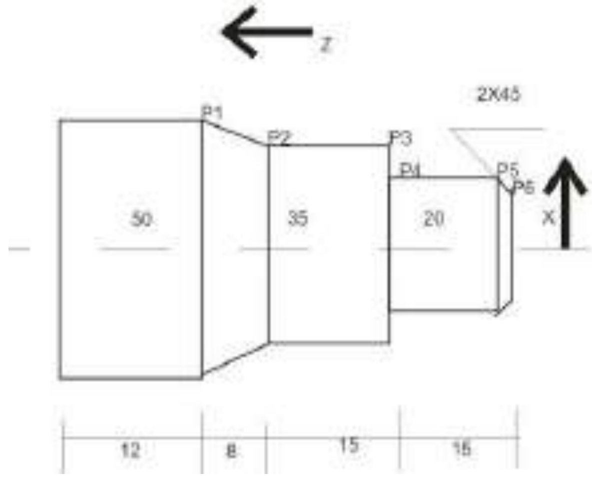
```
N5   G00  X65  Z2
N6   G72  W2   R2
N7   G72  P8   Q9   U0   W0   F0.3
N8   G01  Z30
N9   G00  Z30
```

EXAMPLE
CNC CYCLE G72 (STOCK REMOVAL ROUGH FACING CYCLE).

04252	04253- Name of main program.
N10 G90 G21 G99 F0.2 ;	N10- Absolute co-ordinate system , metric input in 'mm' , feed rate per revolution F0.2 .(Feed rate for rough cut F0.2)
N20 M06 T04 04 ;	N20- Tool changing command , select tool no 4.
N30 M03 S1200 ;	N30- Spindle ON clockwise at speed 1200 rpm.
N40 M08 ;	N40- Coolant ON
N50 G28 U0 W0 ;	N50- Tool at reference point where U0 and W0
N60 G00 X65 Z2 ;	N60- Rapid command, where X65 and Z2 ;
N70 G72 W2 R2 ;	N70- Facing cycle command where depth of cut for z axis is 2(w2) and tool relive distance 2 mm(R2).
N80 G72 P90 Q140 U0.5 W0.1 ;	N80- Facing cycle command , actual operation start from block no. 90 and actual operation end at block no. 140 , finishing allowance for X and Z axis is 0.5 and 0.1
N90 G00 Z-50 ;	N90- Rapid command where tool at X 65 and Z -52 .[P1]
N100 G01 X40 ;	N100- Linear interpolation command where X40 and Z-50 .[P2]
N110 G01 Z-20 ;	N110- Linear interpolation command where X40 and Z-20 .[P3
N120 G01 X20 ;	N120- Linear interpolation command where X20 and Z-20 .[P4]
N130 G01 Z0 ;	N130- Linear interpolation command where X20 and Z0 .[P5]
N140 G01 X0 ;	N140- Linear interpolation command where X0 and Z0 .[P6]
N150 G70 P90 Q140 F0.01 ;	N150- Finish machining cycle , actual operation start from block no. 90 and actual operation end at block no. 140 , feed rate for finishing cut 0.01
N160 G28 U0 W0 ;	N160- Tool at reference point where U0 and W0
N170 M05 M09 M30 ;	N170- Spindle stop , coolant off, main prog end .

EXAMPLE

CNC CYCLE G72 (STOCK REMOVAL ROUGH FACING CYCLE).



04254	04254- Name of main program
N10 G90 G21 G99 ;	N10- Absolute co-ordinate system , metric input in 'mm' , feed rate per revolution
N20 M06 T04 04 ;	N20- Tool changing command , select tool no 4.
N30 M03 S1200 ;	N30- Spindle ON clockwise at speed 1200 rpm.
N40 M08 ;	N40- Coolent ON
N50 G28 U0 W0 ;	N50- Tool at reference point where U0 and W0
N60 G00 X55 Z2 ;	N60- Rapid command, where X55 and Z2 ;
N70 G72 W2 R2 ;	N70- Facing cycle command where depth of cut for z axis is 2(w2) and tool relive distance 2 mm(R2).
N80 G72 P90 Q160 U0.5 W0.1 F0.2 ;	N80- Facing cycle command , actual operation start from block no. 90 and actual operation end at block no. 160 ,finishing allowance along Xand Z axis is 0.5 and 0.1 and 2 Feed rate for rough cut 0.2 mm/rev
N90 G00 Z-38 ;	N90- Rapid command where tool at X 65 and Z -38
N100 G01 X50 ;	N100- Linear interpolation command where X50 and Z-38 .[P1]
N110 G01 X35 Z-30 ;	N110- Linear interpolation command where X35 and Z-30 .[P2] (Taper)
N120 G01 Z-15 ;	N120- Linear interpolation command where X35 and Z-15 .[P3]
N130 G01 X20 ;	N130- Linear interpolation command where X20 and Z-15 .[P4]
N140 G01 Z-2 ;	N140- Linear interpolation command where X20 and Z-2 .[P5]
N150 G01 X16 Z0 ;	N150- Linear interpolation command where X16 and Z0 .[P6] (Chamfering)
N160 G01 X0;	N160- Linear interpolation command where X0 and Z0 .[P2]

N170 G70 P90 Q160 F 0.1;	N170- Finish machining cycle , actual operation start from block no. 90 and actual operation end at block no. 140 , feed rate for finishing cut 0.1
N180 G28 U0 W0 ;	N180- Tool at reference point where U0 and W0
N190 M05 M09 M30;	N190- Spindle stop , coolant off, main prog end .

EXAMPLE

Write a manual part program for Multiple Facing Operation for the component shown in figure

below.

BILLET SIZE : Φ32 X 60

MATERIAL: ALUMINIUM

Multiple Rough turning; Tool Holder :SDJCR 1212H11; TOOL TIP: DCMT11T304; TOOL STATION:1, SPINDLE SPEED :1200 rpm; FEED: 35m/min

FINISHING : Tool Holder : SDJCR 1212H11; TOOL TIP: DCMT11T302; TOOL STATION : 2; **SPINDLE SPEED** : 1450rpm, **FEED:** 25m/min

(CNC program for Multiple Facing cycle

O1006 --------- Program Number 1006

[BILLET X30 Z60 --------- Defining Billet size dia : 30 mm length 60 mm

G21 G98 --------- Initial settings

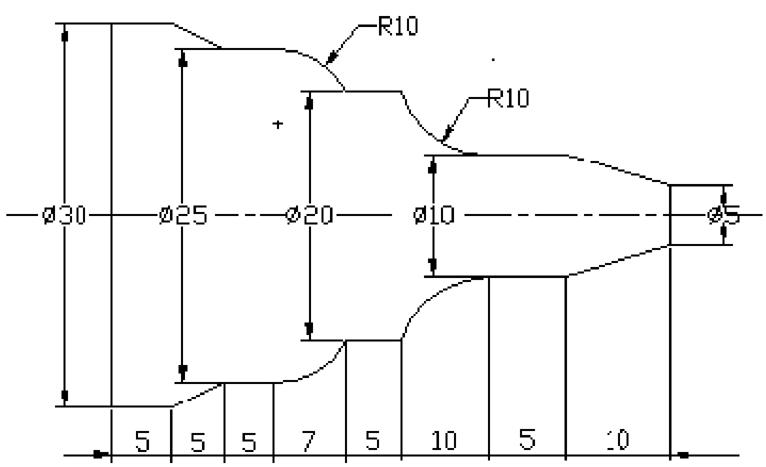
G28 U0 W0 --------- Going to home position

M06 T0303 ---------- Selecting Tool No. 3 with offset No 3

M03 S1200 ---------- Setting spindle speed at 1200 rpm

G00 X31 Z1 ---------- Tool moving to tool entry point X31 Z1

[MULTIPLE FACING CYCLE – G72, Depth of cut for each pass W=0.5 mm, (Relief amount R = 0.5 mm, P & Q – Beginning & end of cycles sequence Nos., Allowances on X and Z axis = 0.1 mm respectively,Feed rate F=35 mm/min.]



G72 W0.5 R0.5

G72 P10 Q19 U0.1 W0.1 F35

N10 G01 Z-52

N11 X30

N12 Z-47

N13 X25 Z-42

N14 Z-37

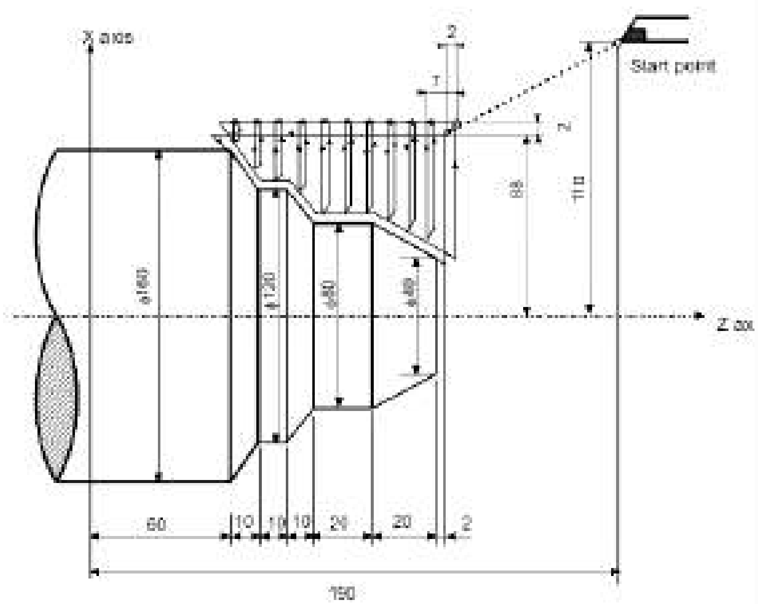
N15 G02 X20 Z-30 R10 F25

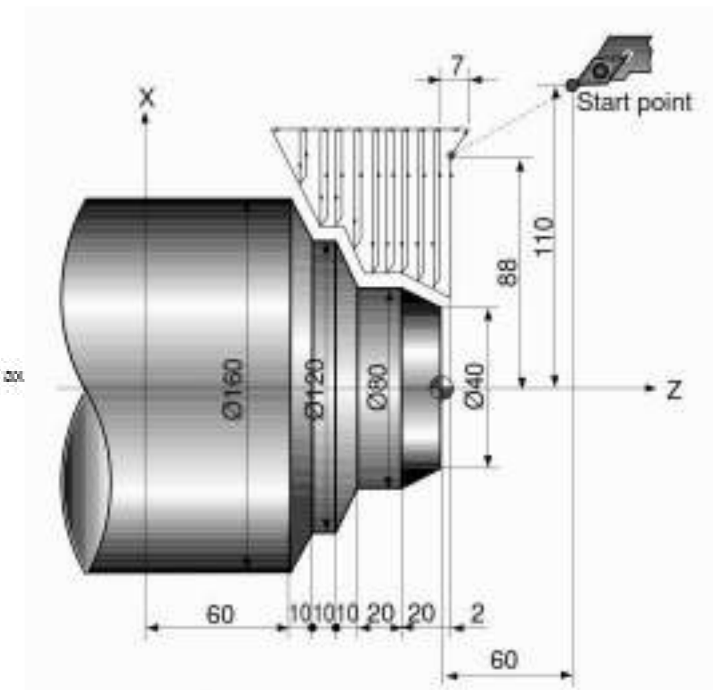
N16 G01 Z-25

N17 G03 X10 Z-15 R10

N18 G01 Z-10 F35

N19 X5 Z0





N20 G20 U0 W0

M06 T0202 -------- Using Left Hand Finishing tool

M03 S1450

G00 X31 Z1

G70 P10 Q19 F25 --------- Finishing Cycle

G28 U0 W0 --------- Going to Home position.

M05 -------- Stop the spindle

M30 -------- Program stop and rewind.

EXAMPLE

N10 G00 X220 Z60

N11 G00 X176 Z2.0

N12	G72	W7.0	R1.0			
N13	G72	P14	Q21	U4.0	W2.0 F0.3	S550
N14	G01		Z-70	S700		
N15		X160				
N16	G01	X120	Z-60			
N17			W10			
N18		X80	W10			
N19			W20			
N20		X40	W20			
N21	G28	U0	W0			
N22	G70	P14	Q21			
N23	G00	X220	Z60			
M04						
M30						

1.21 G75 – GROOVING CYCLE

Fanuc G75 grooving cycle can be used for outside (external) or inside (internal) grooving. The Fanuc G75 grooving cycle is very similar to G74 Peck drilling cycle, G74 is for drilling or grooving in z-axis and G75 is for grooving in x-axis. The cycle is commanded by two distinct lines of data.

FORMAT

G75 R

G75 X(u)….. Z(w)…. P… Q…. F;

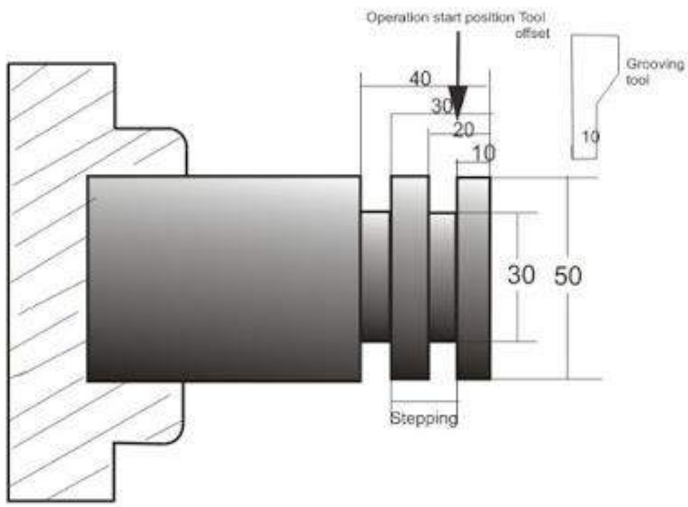
Where

R	–	Return amount
X	–	Groove depth along X axis (absolute)
U	–	Groove depth along X axis (Incremental)
Z	–	Total width along Z axis(absolute)
W	–	Total width along Z axis (Incremental)
P	–	Peck increment in X axis in microns (depth of cut in X axis)

Q – Stepping distance in Z axis in microns.

F - Feed rate in mm.

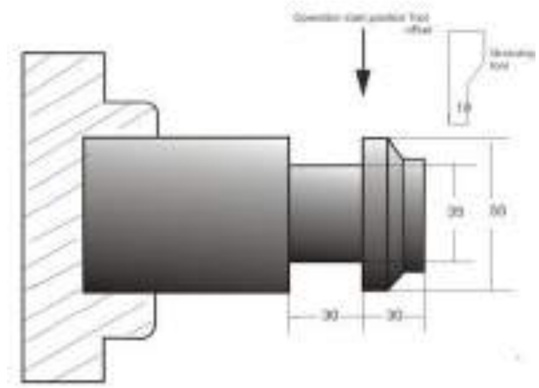
FANUC G75 CANNED CYCLE GROOVING CNC PROGRAM WITH description



01451	01451 - Name of main program
N10 G90 G21 G99 F0.15 ;	N10- Absolute co-ordinate system , metric input in mm , feed rate per revolution , feed is 0.15
N20 G50 S1500 ;	N20- Maximum spindle speed command , speed is 1500 rpm
N30 M06 T01 01 ;	N30- Tool change command , select tool no. 1
N40 M03 G96 S300 ;	N40- Spindle ON clockwise , constant surface speed command , speed is 300 ;
N50 G00 X52 Z-10 ;	N50- Rapid action command , where X52 and Z-10
N60 G75 R1 ;	N60- Grooving cycle command , distance of return 1mm.
N70 G75 X30 Z-30 P3000 Q20000 ;	N70- Grooving cycle command , grooving depth on x-axis is 30 , last groove position in z-axis is 30 , Peck increment in x-axis 3000 micron = 3 mm , stepping in z- axis is 20000 micron = 20 mm .
N80 G00 X60 ;	N80- Rapid action command , where X60 and Z-30
N90 G28 U0 W0 ;	N90- Reference point command , where X0 and Z0
N100 M05 M30 ;	N100- **Spindle** stop , main prog end .

EXAMPLE

G75 FANUC CANNED CYCLE GROOVING CNC PROGRAM ..



04521	01451 - Name of main program
N10 G90 G21 G99 F0.15 ;	N10- Absolute co-ordinate system , metric input in mm , feed rate per revolution , feed is 0.15
N20 G50 S1500 ;	N20- Maximum spindle speed command , speed is 1500 rpm
N30 M06 T01 01 ;	N30- Tool change command , select tool no. 1
N40 M03 G96 S300 ;	N40- Spindle ON clockwise , constant surface speed command , speed is 300 ;
N50 M08 ;	N50- Coolant ON
N60 G00 X57 Z-40 ;	N60- Rapid action command , where X57 and Z-40 (including 10 mm wideness of tool size at z axis)
N70 G75 R1 ;	N70- Grooving cycle command , distance of return 1mm
N80 G75 X35 Z-60 P3000 Q9000 ;	N80- Grooving cycle command , grooving depth on x-axis is 35 , last groove position in z-axis is 60 , Peck increment in x-axis 3000 micron = 3 mm , stepping in z- axis is 9000 micron = 9 mm (next groov by moving 9mm in z-axis)
N90 G00 X60 ;	N90- Rapid action command , where X60 and Z-60
N100 G28 U0 W0 ;	N100- Referance point command , where X0 and Z0
N110 M05 M30 ;	N110- **Spindle** stop ,coolant off, main prog end

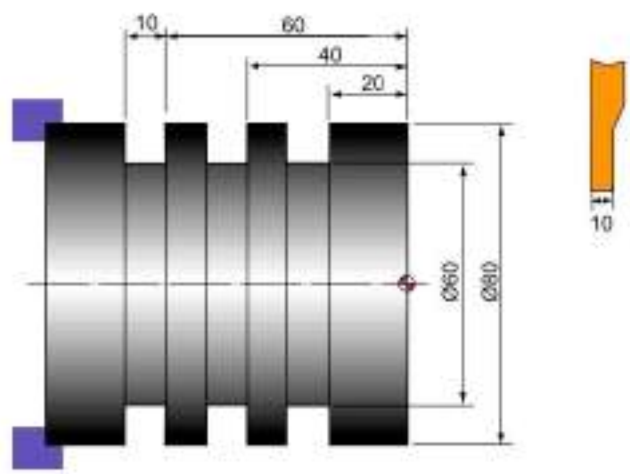
G75 Canned Cycle Grooving CNC Programming Example

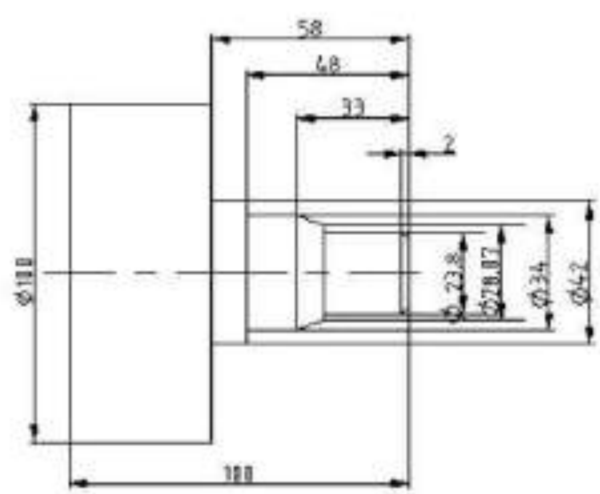
```
N10   G50   S500   T0100
N20   G97   S400   M03
N30   G00   X90    Z1     T0101
N40         X82    Z-60
N50   G75   R1.0
N60   G75   X60    Z-20   P3000  Q20000       F0.1
N70   G00   X90
N80   X200  Z200   T0100
N90   M30
```

EXAMPLE

G71 Turning Cycle

```
N10   T1    G97    S800   M03
N20   GOO   X45    Z2     G42
N30   G71   U2     R1
N40   G71   P50    Q120   U0.25  W0.1  F0.25
N50   G00   X19.8
N60   G01   X23.8  Z-2    F0.2
N70   G01          Z-25
N80   G01   28.07
N90   G01   X34    Z-33
N100  G01          Z-48
N110  G01   X42
N120  G01          Z-58
N130  G28   U0     W0
N140  M06   T0202
      M03   S1400
N150  G70   P50    Q120
N160  G00   X100   Z100
N170  M30
```





1.22 G76 – THREADING CYCLE

This is a "Box type" cycle that is repeated a given number of times. After the first pass subsequent passes cut with one edge of the threading tool only to reduce the load at the tool tip. This cycle requires two distinct blocks of data. When the cutting depth of one cycle becomes smaller than the limit, the actual amount of cut is clamped at the minimum cut depth.

FORMAT

G76 Pxxyyzz Q1 R1

G76 Xn Zn Pn Q2 F

P : P actually consists of multiple values which control the thread behavior,

XX : Number of spring passes or spring cuts.

YY : Thread run out at 45 degree

ZZ : Angle of Thread

Q : Minimum cutting depth times

R : Depth of Last or Finish cut (Finishing cut)

Xn : The end value in x-axis.

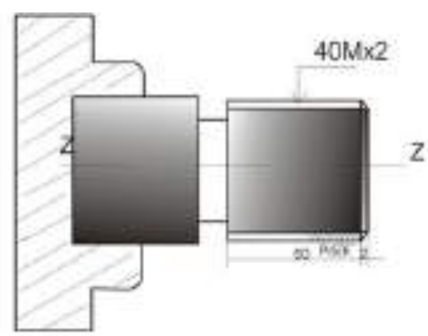
Zn : The end value in z-axis.

P : Thread depth (as radius value) / Height of thread times 1000

Q : Depth of first cut times 1000

F : Thread Pitch

G76 FANUC THREADING CYCLE CNC PROGRAM WITH DESCRIPTION (Metric thread)



N10 M06 T01 01 ;	N10- Tool change command , select tool no. 1
N20 M04 G97 S1000 ;	N20- Spindle ON anti clockwise , constant spindle speed command , speed is 1000 rpm
N30 G00 X45 Z5 ;	N30- Rapid action command where X45 and Z5
N40 G76 P020060 Q100 R20 ;	N40- Threading cycle command , **P020060** (P02 = No. of finished path 00 = Chamfer amount at end 60 = Angle of tool tip) , **Q100** = Each cut is 0.1 mm , R20 = finishing allowance 0.02mm
N50 G76 X35.54 Z-50 P1227 Q100	N50- Threading cycle command , Minor dia X axis ,

F2 ;	threading along Z- axis up to -50 , Threading depth , Depth of finish cut 0.1 mm , pitch is 2 . M40X2 Major diameter is 40, Pitch is 2 **Thread depth calculation** = Pitch x 0.61363 = 2 x 0.61363 = 1.227 mm in micron is 1227 **Minor diameter** = 40-2 x1.23 = 37.54 mm
N60 G00 X45 Z5 ;	N60- Rapid action command where X45 and Z5
N70 M05 M09 M30 ;	N70- Spindle off , coolant off , main program end

Tapered Threading with Fanuc G76 Threading Cycle CNC Program

FORMAT

G76 Pxxyyzz Q1 R1

G76 Xn Zn Pn Q2 R F

P : P actually consists of multiple values which control the thread behavior,

XX : Number of spring passes or spring cuts.

YY : Thread run out at 45 degree

ZZ : Angle of Thread

Q : Minimum cutting depth times

R : Depth of Last or Finish cut (Finishing cut)

Xn : The end value in x-axis.

Zn : The end value in z-axis.

P : Thread depth (as radius value) / Height of thread times 1000

Q : Depth of first cut times 1000

R : Taper thread parameter = [(end dia. - start dia)] / 2

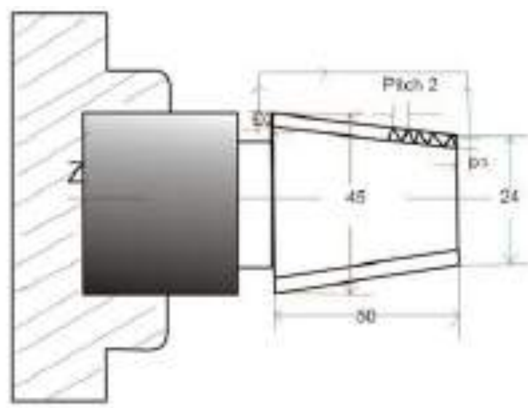
F : Thread Pitch

G76 FANUC TAPER THREADING CYCLE EXAMPLE

01542	
N10 M06 T03 03 ;	N10- Tool change command , select tool no. 3
N20 M04 G97 S1000	N20- Spindle ON anti clockwise , constant spindle speed command , speed is 1000 rpm
N30 M08 ;	Coolant on
N40 G00 X50 Z2 ;	N40- Rapid action command where X45 and Z5
N50 G76 P010060 Q100 R50 ;	N50- Threading cycle command , **P020060** (P02 = No. of finished path 00 = Chamfer amount at end

	60 = Angle of tool tip) , **Q100** = Each cut is 0.1 mm , R20 = finishing allowance 0.02mm
N60 G76 X45 Z-55 P1227 Q200 R10.5 F2	N60- Threading cycle command , X45 is end diameter , tool threading upto -55 in Z axis , P thread depth 1.227mm , depth of finish cut is 0.2 mm , R taper thread parameter is 10.5 , F is pitch is 2. P Depth of thread = pitch x 0.6136 = 2 X 0.6136 = 1.227 mm = 1227 in micron R taper thread parameter = (end dia. - start dia) / 2 = (45-24) / 2 = 10.5
N70 G00 X50 Z2 ;	N60- Rapid action command where X45 and Z5
N80 M05 M09 M30	N70- Spindle off , coolant off , main program end

;



;

;

.

.

EXAMPLE – G76 THREADING CYCLE

N10 M06 T01 01 ;

N20 M04 G97 S1000 ;

N30 G00 X45 Z5 ;

N40 G76 P020060 Q100 R50 ;

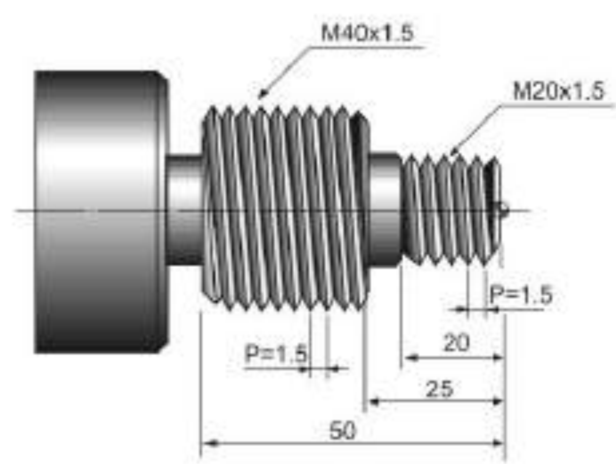
N50 G76 X18.16 Z-20 P920 Q100 F1.5 ;

N60 G00 X50 Z-20 ;

N70 G76 P020060 Q100 R50

N80 G76 X38.16 Z-52 P920 Q100 F1.5

N90 G00 X200 Z200



N100 M30

1.23 G73 PATTERN REPEAT CYCLE

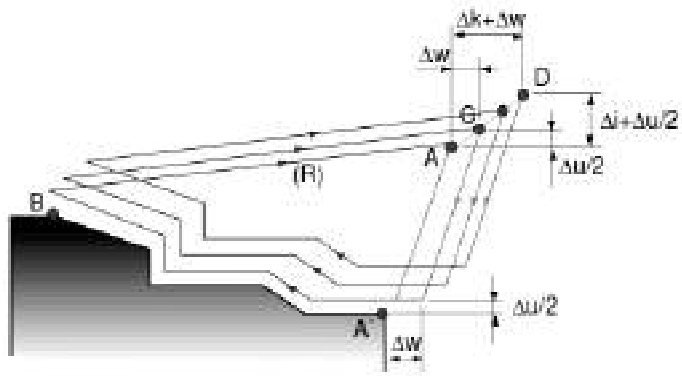
The standard G71 cycle roughs the profile using linear moves along the Z-Axis. The G72 cycle is used for facing and the G73 pattern repeating cycle is used when we are machining a profile that is already cut, for example, a casting or a pre-machined part.

The tool will cut in the shape of the profile that we defined with a subroutine when using the G73 G-Code. If used on a billet, some of the tool paths will be cutting in fresh air. This is why it is normally used when we already have the profile of the part pre-cut or cast. The tool will cut the shape of the profile of the part on each pass, indexing in both X and Z by the amount that we add to the first G73 line after each pass until the finished size is achieved.

The G73 cycle block should look like this example.
Each part is broken down and explained below.

G73 U(1) W(1) R;

G73 P Q U(2) W(2) F;



G73 - PATTERN REPEATING CYCLE
U(1) - DEPTH OF CUT IN X-AXIS
W(1) - DEPTH OF CUT IN Z-AXIS
R - AMOUNT OF ROUGHING PASSES / REPEAT OF CUT
P - FIRST LINE OF SUBROUTINE
Q - LAST LINE OF SUBROUTINE
U(2) - AMOUNT LEFT ON FOR FINISHING IN X
W - AMOUNT LEFT ON FOR FINISHING IN Z
F - FEED RATE

G73 tells the machine that we wish to use the pattern cycle
The first 'U' word defines the depth of cut of each roughing pass in the X-axis. 'W' is the amount that we wish to cut in Z-axis. The R is the number of passes we require.
The 'P' and 'Q' words let the control know the location of the subroutine of the profile that we are using. These values can be any value as long as it matches the 'N' numbers of the subroutine. This will look like the code below.

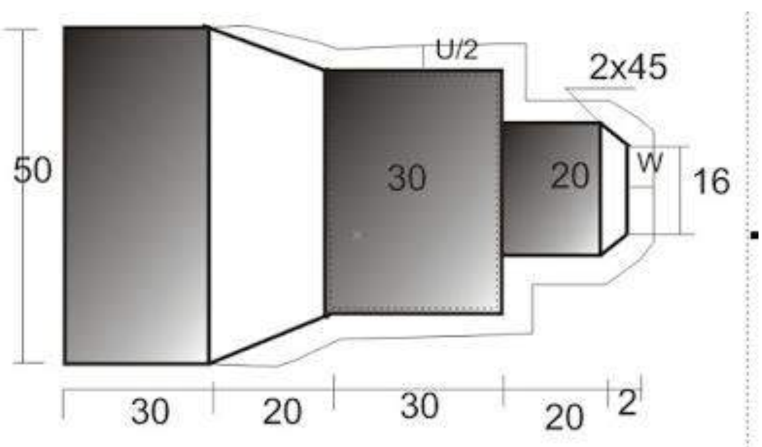
N150;

SUBROUTINE OF PROFILE;

N250;

In this example, 'P' would be P150 and 'Q' would be Q250 so they match the 'N' numbers. The 'U' on the second G73 line is the amount of material that we wish to leave on for a finishing pass along the X-Axis and 'W' is the finishing allowance along the Z-Axis.'F' is the command we use to specify a feed rate

Fanuc G73 Pattern Repeating Cycle Program Example

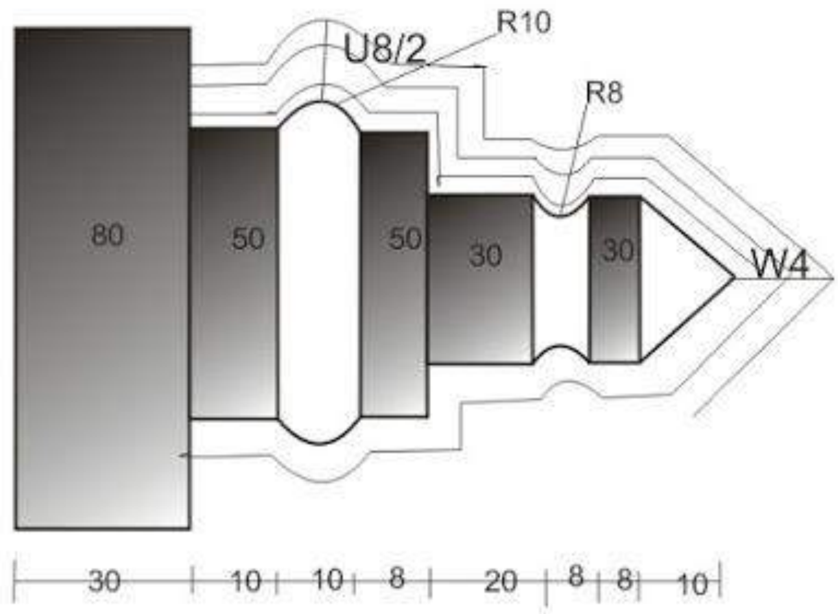


04534	04534- Name of main program
N10 G90 G50 S1200 ;	N10 - Absolute coordinate system, maximum spindle speed command , speed 1200
N20 G99 F0.25 ;	N20- Feed rate per revolution is 0.25 (F0.25)
N30 M03 G96 S200 ;	N30- Spindle on clockwise , constant surface speed command, speed is 200.
N40 M06 T01 01 ;	N40- Tool change command , select tool no. 1
N50 M08 ;	N50- Coolant
N60 G28 U0 W0 ;	N60- Reference point command where X0 and Z0
N70 G00 X25 Z2 ;	N70- Rapid command , where X25 and Z2
N80 G73 U4 W2 R2 ;	N80- Pattern repeating cycle command ,U4 is amount of material cut in X axis and W2 is amount of material cut in Z axis AND R2 is repeat of cut

N90 G73 P100 Q150 U0.5 W0.1 ;	N90- Pattern repeating cycle command , actual operation start from block no. 100 and actual operation end at block no. 150, finishing allowance in X axis is 0.5 and finishing allowance in Z axis is 0.1
N100 G00 X16 ; ;	N100- Rapid action command , where X16 and Z2
N110 G01 X20 Z-2 ;	N110- Linear interpolation command where X20 and Z-2. (CHAMFERING)
N120 G01 Z-22 ;	N120- Linear interpolation command where X20 and Z-22
N130 G01 X30 ;	N130- Linear interpolation command where X30 and Z-22
N140 G01 Z-52 ;	N140- Linear interpolation command where X30 and Z-52
N150 G01 X50 Z-72 ;	N150- Linear interpolation command where X50 and Z-72 (TAPER)
N160 G70 P100 Q150 F0.1 ;	N160- Finish machining cycle , actual operation start from block no. 100 and actual operation end at block no. 150 , feed rate for finishing cut 0.1
N170 G28 U0 W0 ;	N170- Reference point command where X0 and Z0
N180 M05 M09 M30 ;	N180- Spindle stop , coolant off, main prog end .

This cycle is used in predefined shape like forging , casting

CNC FANUC G73 PATTERN REPEATING CYCLE CNC.....EXAMPLE 2

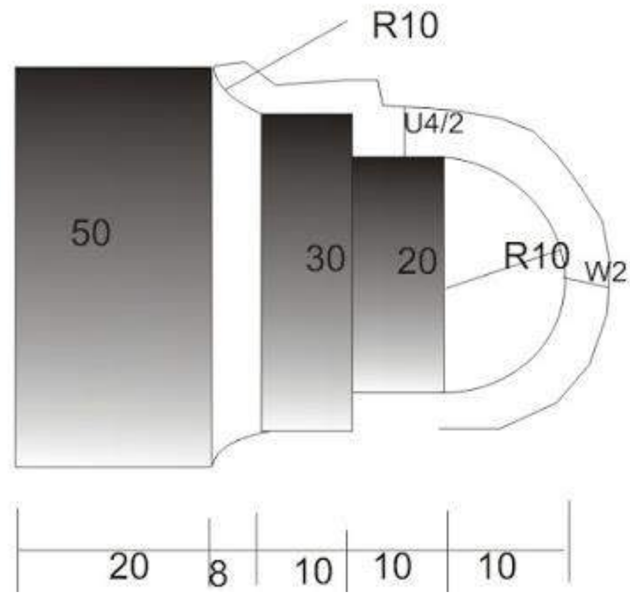


04534	04534- Name of main program
N10 G90 G50 S1200 ;	N10- Absolute cordinate system , maximum spindle speed command , speed 1200
N20 G99 F0.25 ;	N20- Feed rate per revolution is 0.25 (F0.25
N30 M03 G96 S200 ;	N30- Spindle on clockwise , Constant surface speed command, speed is 200
N40 M06 T01 01 ;	N40- Tool change command , select tool no. 1
N50 M08 ;	N50- Coolant
N60 G28 U0 W0 ;	N60- Reference point command where X0 and Z0
N70 G00 X20 Z2 ;	N70- Rapid command , where X20 and Z2
N80 G73 U4 W2 R3 ;	N80- Pattern repeating cycle command ,U4 is amount of material cut in X axis and W2 is amount of material cut in Z axis AND R3 is repeat of cut
N90 G73 P100 Q150 U0.5 W0.1 ;	N90- Pattern repeating cycle command , actual operation start from block no. 100 and actual operation end at block no. 180, finishing allowance in X axis is 0.5 and finishing allowance in Z axis is 0.1
N100 G00 X0 ;	N100- Rapid action command , where X0 and Z2
N110 G01 X30 Z-10 ;	N110- Linear interpolation command where X30 and Z-10
N120 G01 Z-18 ;	N120- Linear interpolation command where X30 and Z-18
N130 G02 X30 Z-26 CR=8 ;	N130- Circular interpolation clockwise , where X 30 , Z-26

	and CR=8
N140 G01 Z-46 ;	N140- Linear interpolation command where X30 and Z-46
N150 G01 X50 ;	N150- Linear interpolation command where X50 and Z-46
N160 G01 Z-54 ;	N160- Linear interpolation command where X50 and Z-54
N170 G03 X50 Z-64 CR=10 ;	N170- Circular interpolation counter clockwise , where X 50 , Z-64 and CR=10
N180 G01 Z-74 ;	N180- Linear interpolation command where X50 and Z-74
N190 G70 P100 Q150 F0.1 ;	N190- Finish machining cycle , actual operation start from block no. 100 and actual operation end atblock no. 180 , feed rate for finishing cut 0.1
N200 G28 U0 W0 ;	N200- Reference point command where X0 and Z0
N180 M05 M09 M30 ;	N210- Spindle stop , coolant off, main prog end

This cycle is used in predefined shape like forging , casting

CNC G73 PATTERN REPEATION CYCLE FOR LATHE ..EXAMPLE 3

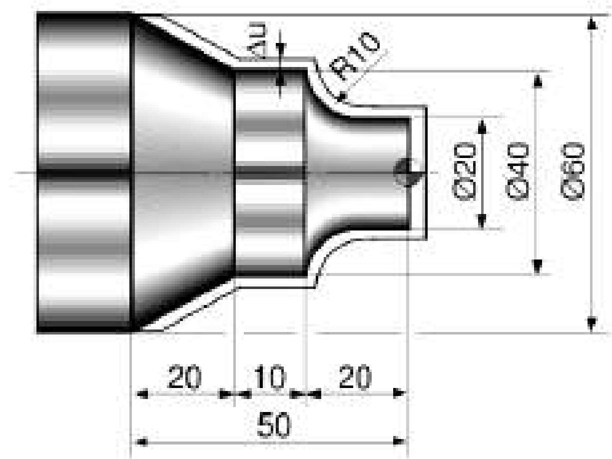


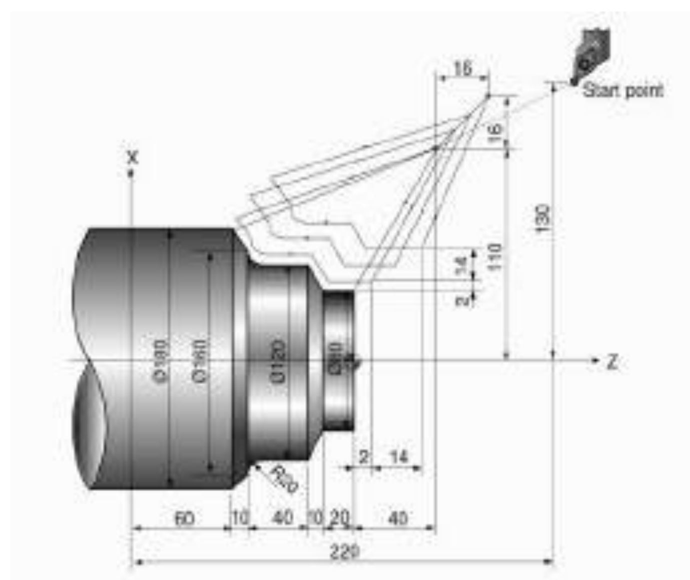
04532	04532- Name of main program
N10 G90 G50 S1200 ;	N10- <u>Absolute cordinate system</u> , maximum spindle speed command , speed 1200
N20 G99 F0.25 ;	N20- <u>Feed rate per revolution</u> is 0.25 (F0.25)
N30 M03 G96 S200 ;	N30- Spindle on clockwise , constant surface speed command, speed is 200.
N40 M06 T01 01 ;	N40- Tool change command , select tool no. 1
N50 M08 ;	N50- <u>Coolant ON</u>
N60 G28 U0 W0 ;	N60- Reference point command where X0 and Z0
N70 G00 X25 Z2 ;	N70- Rapid command , where X25 and Z2
N80 G73 U4 W2 R3 ;	N80- Pattern repeating cycle command ,U4 is amount of material cut in X axis and W2 is amount of material cut in Z axis AND R3 is repeat of cut
N90 G73 P100 Q150 U0.5 W0.1 ;	N90- Pattern repeating cycle command , actual operation start from block no. 100 and actual operation end at block no. 150, finishing allowance in X axis is 0.5 and finishing allowance in Z axis is 0.1
N100 G00 X00 Z00 ;	N100- <u>Rapid action command</u> , where X00, z00

N110 G03 X20 Z-10 CR= 10 ;	N110- Circular interpolation counter clockwise , where X20 , Z-10 and external radius is 10
N120 G01 Z-20 ;	N120- Linear interpolation command where X20 and Z20
N130 G01 X30 ;	N130- Linear interpolation command where X30 and Z20
N140 G01 Z-30 ;	N140- Linear interpolation command where X30 and Z30
N150 G02 X50 Z-38 CR=10 ;	N150- Circular interpolation clockwise , where X50 , Z-38 and internal radius is 10
N160 G70 P100 Q150 F0.1 ;	N160- Finish machining cycle , actual operation start from block no. 100 and actual operation end at block no. 150 , feed rate for finishing cut 0.1
N170 G28 U0 W0 ;	N170- Reference point command where X0 and Z0
N180 M05 M09 M30 ;	N180- Spindle stop , coolant off, main prog end .

Fanuc G73 Pattern Repeating Cycle Programming Example -4

```
04532
N10  G90 G50 S1200 ;
N20    G99 F0.25 ;
N30    M03 G96 S200 ;
N40    M06 T01 01 ;
N50    M08 ;
N60    G28 U0 W0 ;
N70    G00    X220  Z40;
N80    G73    U14    W14    R3
N90    G73    P100  Q150  U4.0    W2.0   F0.3
       S0180
N100  G00    X80     Z2
N110  G01             W-20  F0.15  S0600
N120  G01    X120  W-10
N130  G01             W-40
N140  G02    X160  W-10  R20
N150  G01    X180  W-10
N160  G70 P100 Q150 F0.1 ;
N170  G28 U0         W0
N180  M05    M30
```





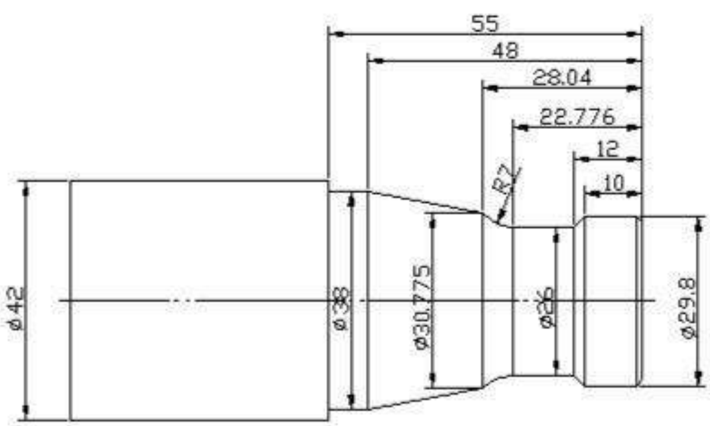
Fanuc G73 Pattern Repeating Cycle Programming Example -5

```
04532
N10  G90 G50 S1200 ;
N20    G99 F0.25 ;
N30    M03 G96 S200 ;
N40    M06 T01 01 ;
N50    M08 ;
N60    G28 U0 W0 ;
N70    G00    X70     Z10;
N80    G73    U3.0    W2.0  R2
N90    G73    P100  Q140  W0.1   F0.25
```

```
N100  G00   X20   Z2
N110  G01         Z-10   F0.15
N120  G02   X40   Z-20   R10
N130  G01         Z-30
N140  G01   X60   Z-50
N150  G73   P100  Q140   F0.1
N160  G28   U0    W0
N170  M05   M30
```

CNC Programming Example Fanuc G73 Pattern Repeating Cycle – EXAMPLE-6

```
04532
N10  G90 G50 S1200 ;
N20   G99 F0.25 ;
N30   M03 G96 S200 ;
N40   M06 T01 01 ;
N50   M08 ;
N60   G28 U0 W0 ;
N70   G00   X46    Z10;
N80   G73   U2.0   W2.0  R7
N90   G73   P100   Q190  U0.6   W0.3   F0.2
N100  G00   X27.8  Z1    S1200
N110  G01          Z0    F0.15
N120  G01   X29.8  Z-1;
N130               Z-10;
N140  G01   X26    Z-12;
N150  G01          Z-22.76;
N160  G02   X30.775      Z-28.04      R7;
N170  G01   X38    Z-48;
N180  G01          Z-55
N190  G01   X42;
N200  M05
N210  M06   T0303
N220  G97   S1200        M03
N230  G00   X42    Z1
```



N240 G70 P100 Q190

N250 G00 X200 Z200

N260 M05 M30

EXAMPLE

Write the CNC lathe program for a FANUC controlled machine using subroutine codes. Take the diameter of the work piece = 40mm, depth of cut = 0.5mm, speed = 1200rpm. Assume feed and other data suitably.

MAIN PROGRAM

N0 G90 F0.2 S1200 T0101 M04

N10 G00 X42 Z2

N20 M98 P012000 (CALL SUBPROGRAM 2000 ONCE)

N150 G28 U0 W0

N160 M05 M30

SUBPROGRAM

O2000

N30 G73 U2 W2 R7

N40 G73 P50 Q130 U0.05 W0.05

N50 G01 X16 Z0

N60 G01 Z-20

N70 G03 X30 Z-28 R8

N80 G01 Z-45

N90 G01 X40 Z-60

N100 G01 Z-68

N110 G02 X40 Z-78 R8

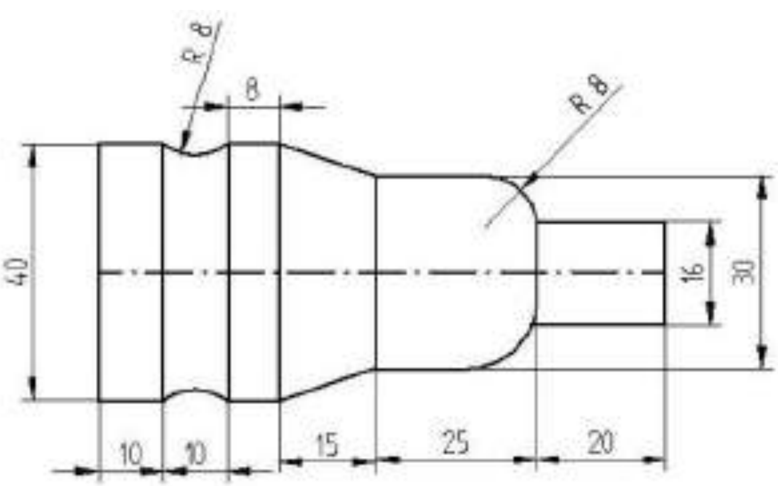
N120 G01 Z-88

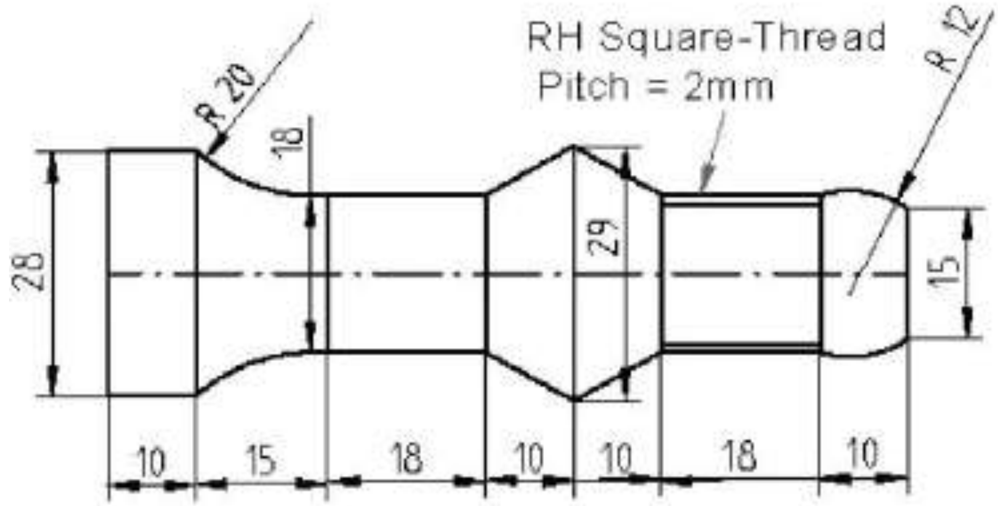
N130 G01 X50 Z2

N140 M99

EXAMPLE

Write the CNC lathe programming for a FANUC controlled machine. Take the diameter of the work piece = 30mm, speed = 1200rpm. Assume feed and other data suitably.





This program uses

G73 Pattern Repeating Cycle &

G76 Threading Cycle

```
N5 G90 F0.2 S1200 T0101 M04
N10 G00   X30   Z2
N15 G00         Z0
N20 G01   X0 (Facing operation)
N21 G00   X30   Z2
N30 G73   U7    R14 (Pattern Repeating Canned Cycle)
N40 G73   P50   Q130  U0.05 W0.05
N50 G01   X15   Z0
N60 G03   X18   Z-10  R12
N70 G01         Z-28
N80 G01   X29   Z-38
N90 G01   X18   Z-48
N100 G01        Z-66
N110 G02  X28   Z-81  R20
N120 G01        Z-91
N130 G01  X32   Z2
N140 G28  U0    W0
N150  S400  T0202     M06 (Threading Tool)
N160  G00   X22   Z0
N161  G01   X18   Z-10
N170  G76   P010100   Q10
```

N180 G76 X16 Z-28 P1000 Q100 F2

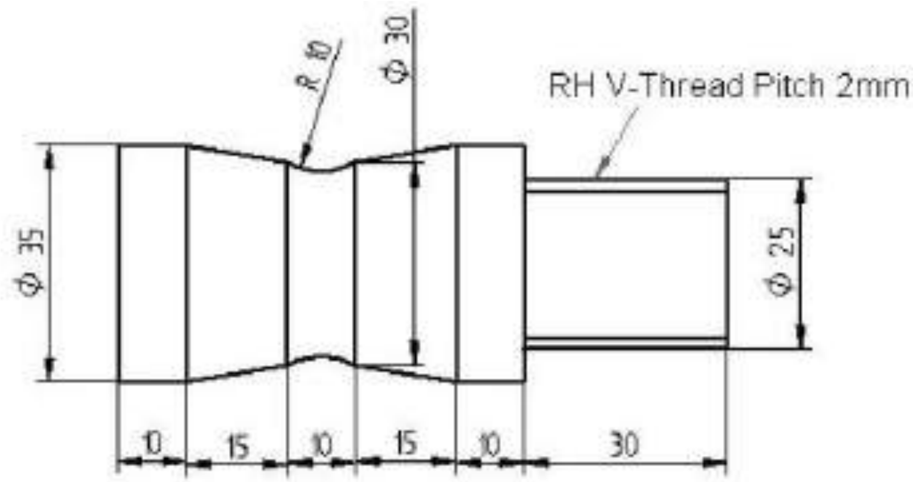
N190 G28 U0 W0

N200 M05

M30

EXAMPLE

Write a part program for a FANUC controlled CNC Lathe for the given component using canned cycle. Take the depth of cut 0.5mm & speed 1200rpm. Assume suitable cutting conditions and cutting tools.



This program uses

G73 Pattern Repeating Cycle &

G76 Threading Cycle

To determine U= (Max. Dia of wp- min dia of profile)/2 = (35-25)/2 = 5

N0 G90 F0.2 S1200 T0101 M04

N10 G00 X38 Z2

N30 G73 U5 R10

N40 G73 P50 Q120 U0.05 W0.05

N50 G01 X25 Z0

N60 G01 X25 Z-30

N70 G01 X35 Z-30

N80 G01 X35 Z-40

N90 G01 X30 Z-55

N100 G02 X30 Z-65 R10

N110 G01 X35 Z-80

N120 G01 X35 Z-90

N130 G01 X38 Z2

N140 G28 U0 W0

N150 S400 T0202 M06

N160 G00 X26 Z0

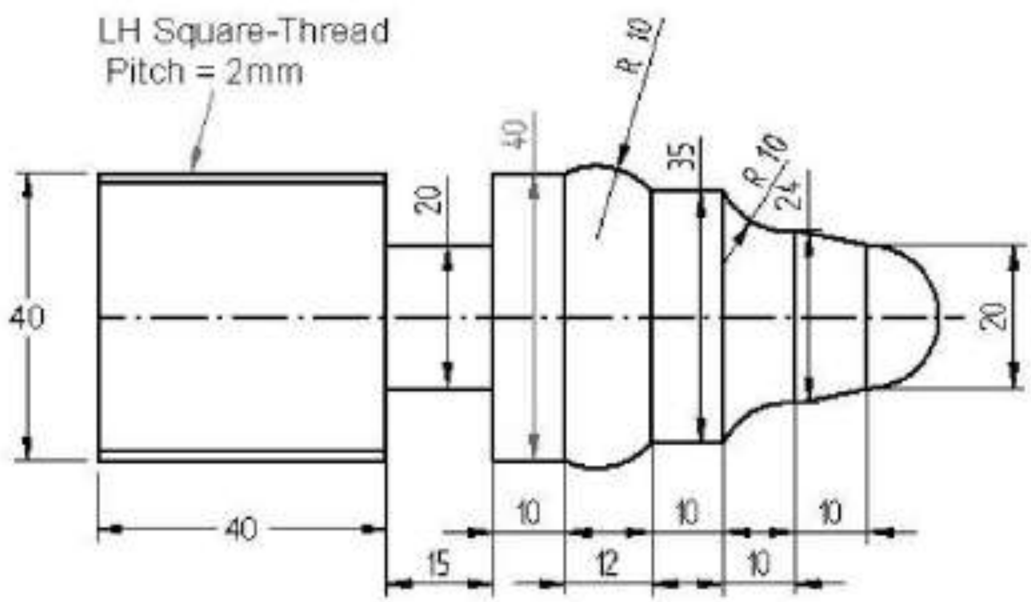
N170 G76 P010160 Q5

N180 G76 X22.6 Z-30 P1200 Q200 F2

N190 G28 U0 W0

N200 M05 M30

CNC lathe programming example using G76 to cut Left hand threads on component and Grooving using G75 Grooving Cycle.



N0 G90 F0.2 S1200 T0101 M04

N10 G00 X42 Z2

N21 G71 U1 R0.5

N22 G71 P50 Q90 U0.05 W0.05

N50 G03 X20 Z-10 R10

N60 G01 X24 Z-20

N70 G02 X35 Z-30 R10

N71 G01 X35 Z-40

N80 G03 X40 Z-52 R10

N90 G01 X40 Z-117

N111 G00 X45 Z2

N112 G28 U0 W0

N180 T0202 M06

N190 G00 X45 Z-64

N191 G00 X40 Z-64

N200 G75 R1

N210 G75 X20 Z-77 P1000 Q1999 (If width of parting tool is 2 mm)

N220 G00 X45 Z-65

N230 G28 U0 W0

N300 T0303 M06

N310 G01 X45 Z-115

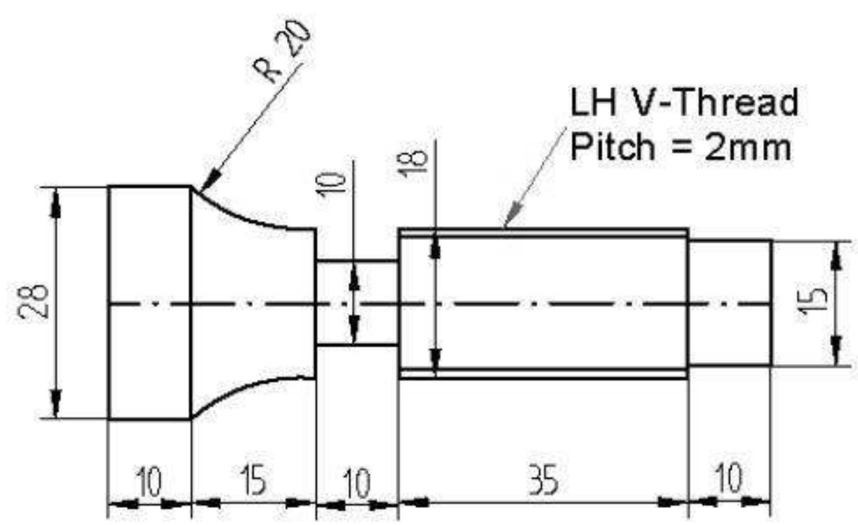
N320 G01 X40 Z-115

N330 G76 P010100 Q10

N340 G76 X38 Z-77 P1000 Q100 F2

N350 M05 M30

CNC lathe program to cut LH (left hand) thread on a component and also to cut the component profile using G71 turning cycle, Grooving using G75 grooving cycle. Write an ISO part programming for the FANUC controlled CNC Lathe using canned cycle. Work piece diameter = 30mm, Work piece material = Mild Steel, Feed = 0.2mm/rev, Speed for turning = 1200rpm, Depth of cut = 0.5mm.



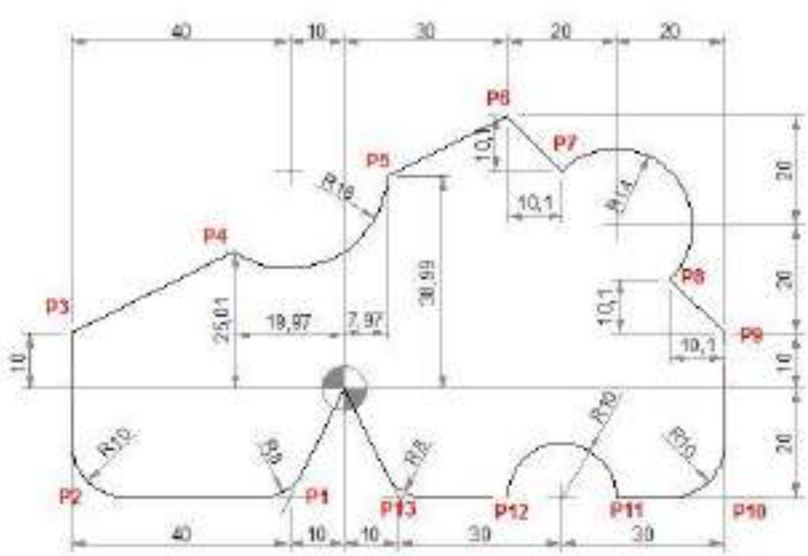
N0 G90 F0.5 S1200 T0101 M04

N20 G00 X30 Z2

```
N40 G71 U0.5 R0.5 (Roughing Cycle)
N50 G71 P60 Q110 U0.05 W0.05
N60 G01     X15    Z0
N70 G01     X15    Z-10
N80 G01     X18
N90 G01            Z-55
N100 G02    X28    Z-70   R20
N110 G01    Z-80
N120 G01    X30    Z2
N130 G70    P60    Q110 (Finishing cycle)
N135 G00    X30    Z2
N140 G28 U0 W0
N150 S400 T0202 M06
N160 G01    X20    Z-45
N170 G01    X18    Z-45
N180 G76 P010160 Q20
N190 G76    X15.6 Z-10 P1200 Q200 F2
N200 G01    X30    Z2
N210 G28 U0 W0
N220 S1200 T0303 M06
N230 G01    X30    Z-45
N240 G01    X18    Z-46
N250 G75    R1
N255 G75    X10    U1 Z-55      P1000 Q1000
N260 G28 U0 W0
N300 M05 M30
```

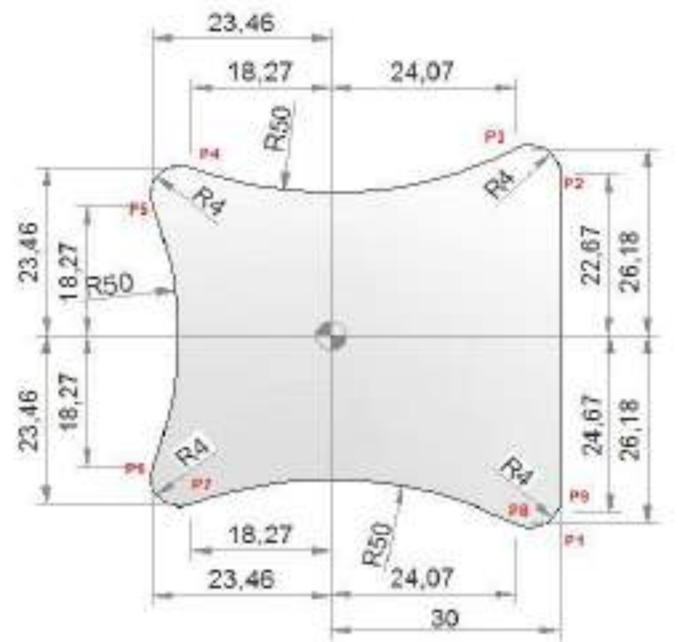
CNC Mill Example Program

```
N40 G90 G00 X0 Y0
N50 G01 X-10 Y-20 R8              (P1)
N60 G01 X-50 R10                  (P2)
N70 Y10                           (P3)
N80 X-19.97 Y25.01                (P4)
N90 G03 X7.97 Y38.99 R18          (P5)
```



```
N100 G01 X30 Y50               (P6)
N110 G91 X10.1 Y-10.1          (P7)
N120 G90 G02 X59.9 Y20.1 R14 (P8)
N130 G01 X70 Y10               (P9)
N140 Y-20 R10                  (P10)
N150 X50                       (P11)
N160 G03 X30 R10               (P12)
N170 G01 X10 R8                (P13)
N180 X0 Y0
```

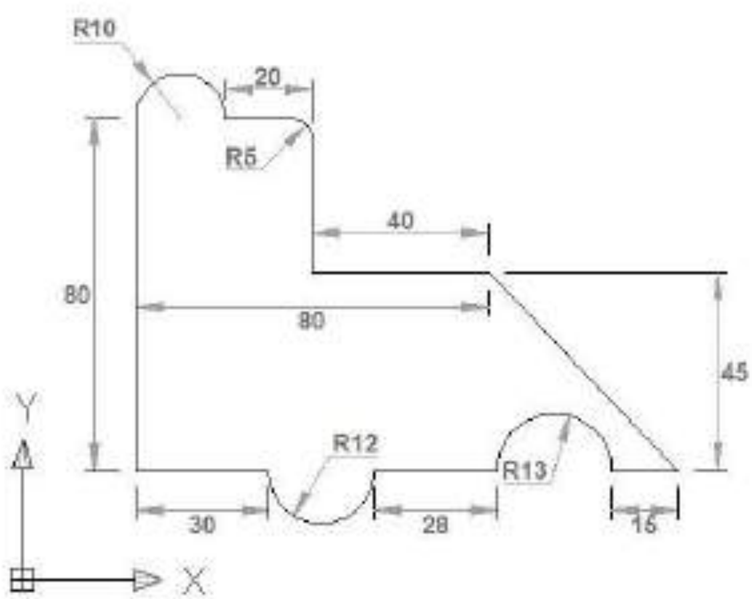
G02 G03 Example CNC Mill




```
G0 X30 Y-30             (P1)
G1 Y22.67               (P2)
G3 X24.07 Y26.18 R4     (P3)
G2 X-18.27 Y23.46 R50   (P4)
G3 X-23.46 Y18.27 R4    (P5)
G2 X-23.46 Y-18.27 R50  (P6)
G3 X-18.27 Y-23.46 R4   (P7)
G2 X24.07 Y-26.18 R50   (P8)
G3 X30 Y-24.67 R4       (P9)
G1 X33
```

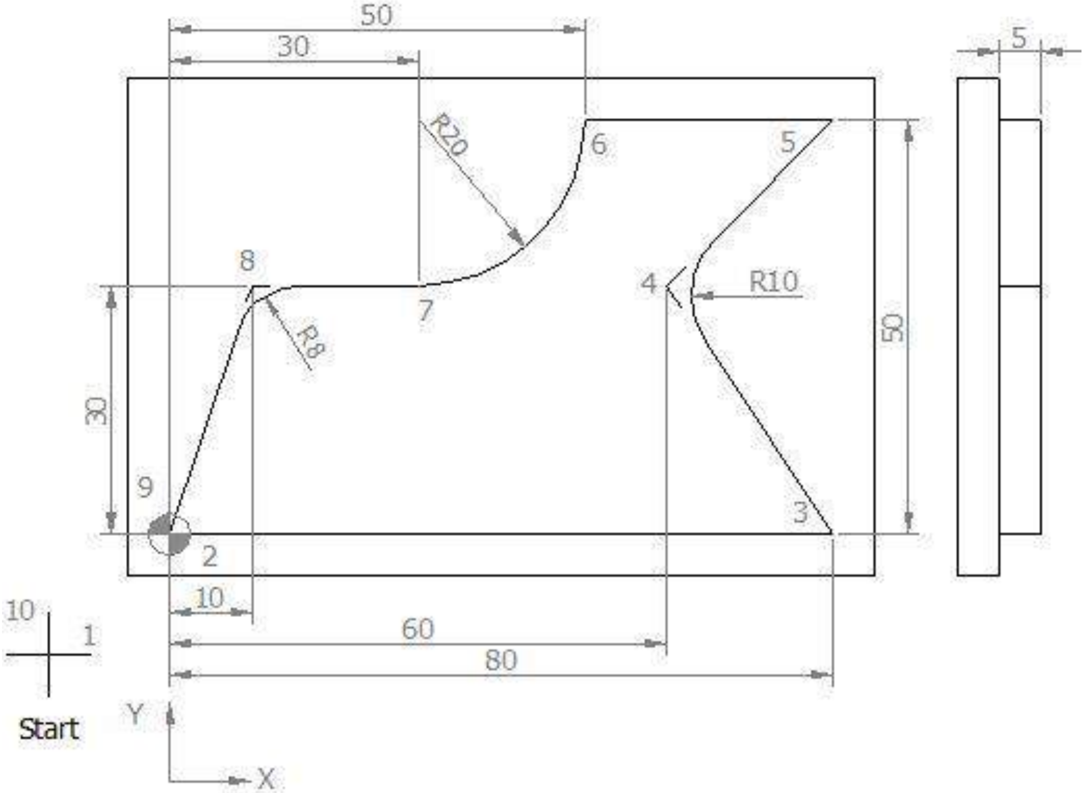
CNC mill program to show the use of

- **G02 Circular Interpolation CW**
- **G03 Circular Interpolation CCW**




```
G90 G01 X0 Y0
X30
G03 X54 R12
G01 X82
G02 X108 R13
G01 X123
X80 Y45
X40
```

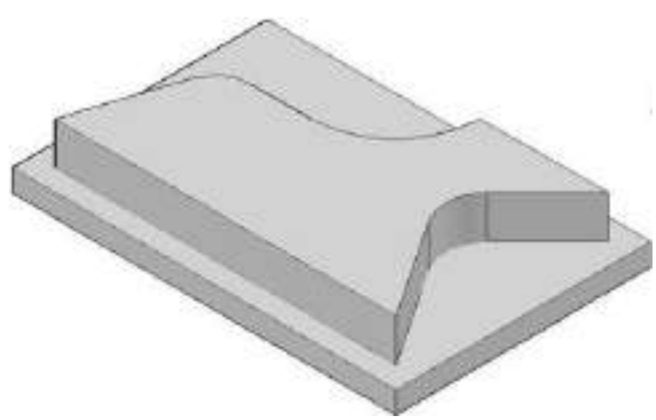
```
Y75
G03 X35 Y80 R5
G01 X20
G03 X0 Y80 R10
G01 Y0
M30
```



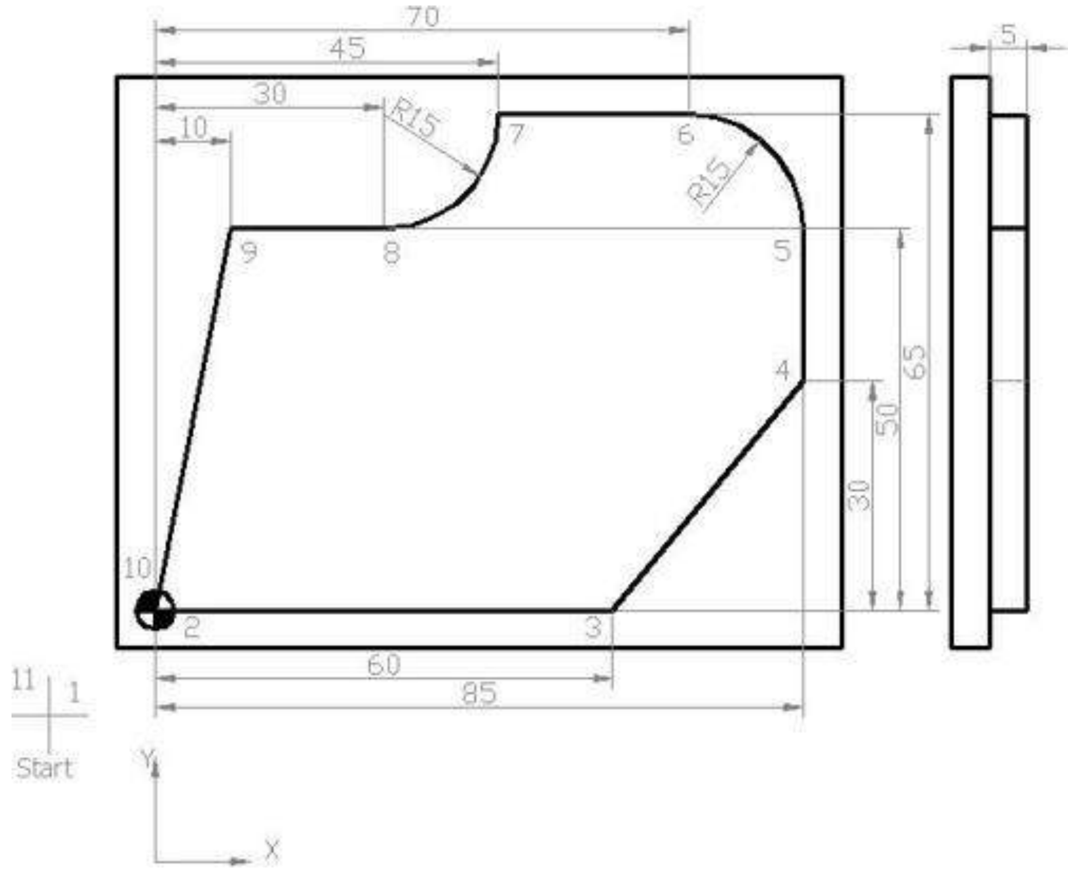
Circular

Interpolation Programming Example

```
N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50
N10 S450 M03 F250 D01 (12.5 MM DIA)
N15 G00      X00    Y00
N20                         Z5
N25 G01                     Z0
N30                         Z-5
N35 G42      X0     Y0               M08
N40          X80    Y0
N45          X60    Y30     U10
N50          X80    Y50
N55          X50    Y50
N60 G02      X30    Y30     U20
N65 G01      X10    Y30     U8
N70          X0     Y0
N75 G40      X-20   Y-20
N80 G00                            Z50     M09
N85                 Y100
N90 M30
```



Circular Interpolation Programming Example 1 (Use of R)



N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50

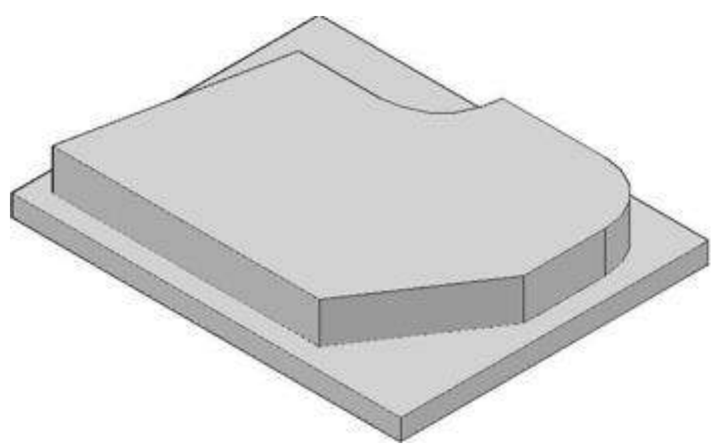
N10 S450 M03 F250 D01 (12.5 MM DIA)

N15 C0

N20 Z5

N25 G01 Z0

N30 Z-5



N35 G42 X0 Y0 M08 (Linear motion)

N40 X60 Y0 (Linear motion)

N45 X85 Y30 (Linear motion)

N50 X85 Y50 (Linear motion)

N55 G03 X70 Y65 U15 (Circular motion G03 for Counter Clockwise motion and U for arc radius)

N60 G01 X45 Y65 (Linear motion)

N65 G02 X30 Y50 U15 (Circular motion G02 for Clockwise motion and U for arc radius)

N70 G01 X10 Y50 (Linear motion)

N75 X0 Y0 (Linear motion)

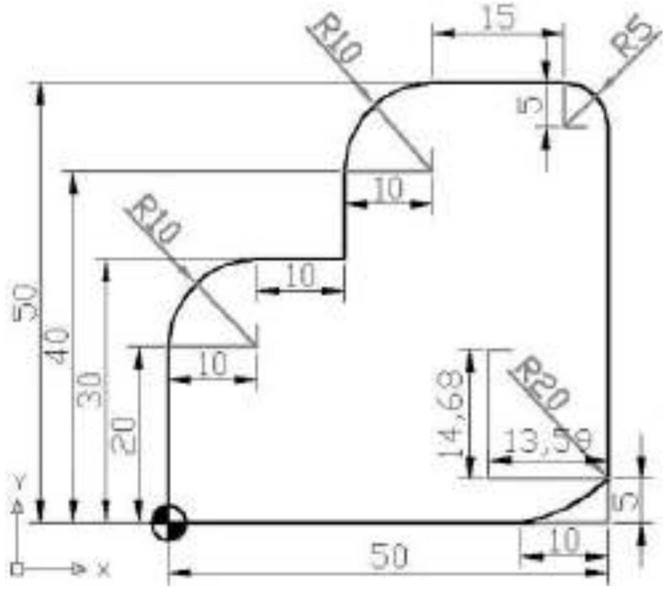
N80 G40 X-20 Y-20

N85 G00 Z50 M09

N90 Y100

N95 M30

Circular Interpolation using G90 and G91



CNC PROGRAM WITH G90	CNC PROGRAM WITH G91
G41 G90 G01 X0 Y20	G41 G91 G01 X0 Y20
G02 X10 Y30 I10 J0	G02 X10 Y10 I10 J0
G01 X20 Y30	G01 X10 Y0
G01 X20 Y40	G01 X0 Y10
G02 X30 Y50 I10 J0	G02 X10 Y10 I10 J0
G01 X45 Y50	G01 X15 Y0
G02 X50 Y45 I0 J-5	G02 X5 Y-5 I0 J-5
G01 X50 Y5	G01 X0 Y-40
G02 X40 Y0 I-13.59 J14.68	G02 X-10 Y-5 I-13.59 J14.68
G01 X0 Y0	G01 X-40 Y0