UNIT-I

THEORY OF METAL CUTTING

Mechanics of Chip formation:

A Chip is a combination of reshaping and refracturing. If a material reshaped, it is said to have exceeded its plastic limit. The deformed chip is separated from the parent material by fracture.

The following points are worth to note the:

* Shear plane is actually a narrow zone, of the order of about 0.025 mm.

* Cutting edge of the tool is formed by two intersecting surfaces.

* Surface along with the chip moves upwards is called rake surface.

* Surface which is relieved to avoid rubbing with the machined surface is called flank.

During cutting process, the following properties of the workpiece material are quite important.

i) Hardness
ii) Abrasive qualities
iii) Toughness
iv) Tendency to weld
v) Inherent hard Spots and Surface inclusions.

The desirable properties of tool material are hardness, Strength, toughness and wear resistance.

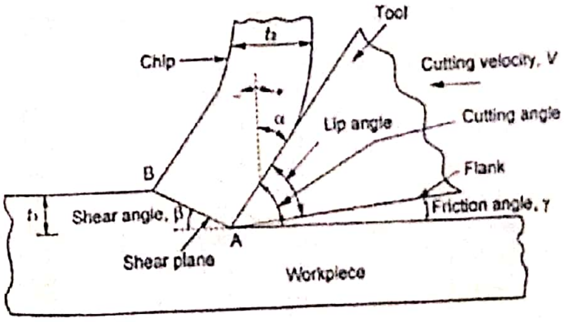


Figure 1.9 Mechanism of metal cutting

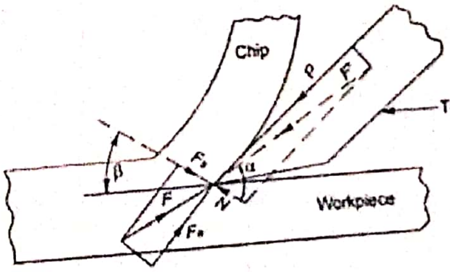


Figure 1.16 Cutting forces on chip

Types of Chips

Generally there are following three types of Chips.

1. Continuous Chip
2. Discontinuous chip
3. Continuous Chip with buildup edge

Continuous Chip:

During cutting of ductile material, a continuous ribbon-like chip is produced due to the pressure of the tool cutting edge in compression and shear. These types of chips are in the form of a long coil and have the same thickness throughout the length.

The following conditions favour the formation of continuous chips.

- Ductile material such as low carbon steel, aluminium, copper etc...
- Smaller depth of cut
- High cutting speed
- Large rake angle
- Sharp cutting edge
- Proper cutting fluid
- Low friction between tool face and chip interface.

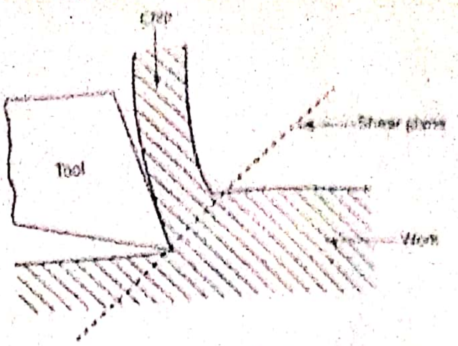


Figure 1.10 Continuous chip

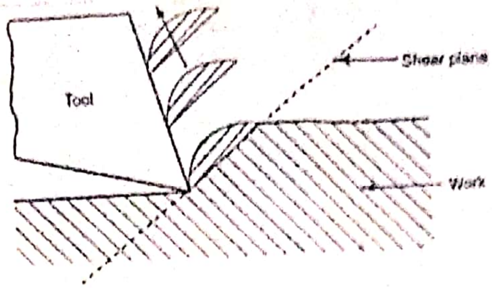


Figure 1.11 Discontinuous chip

Discontinuous or Segmental Chip:

Discontinuous chips as shown in fig. are produced while machining brittle materials such as grey cast iron, bronze, high carbon steel at low cutting speeds without fluid when the friction exists between tool and chip interface.

The following conditions are favour the formation of discontinuous chips

- * Machining of brittle material
- * Small rake angle
- * Higher depth of cut

- * low cutting speeds
- * Excess cutting fluid
- * Cutting ductile material at very low feeds with small rake angle of the tool.

Continuous Chips with Built-up-Edge:

During cutting process, the interface temperature and pressure are quite high and also high friction between tool-chip interface. It causes the chip material to weld itself to the tool face near the nose as shown in fig is called 'built-up edge'.

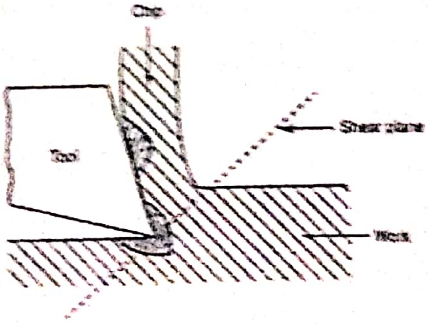


Figure 1.12 Continuous chip with built-up edge

The following conditions favour the formation of continuous chips with built-up edge

- Low cutting Speed
- Small rake angle
- Coarse feed
- Strong adhesion between chip and tool interface
- Insufficient cutting fluid
- Large uncut thickness

Single Point Cutting Tool:

The single point cutting tool has only one cutting point or edge. These tools used for turning, boring, Shaping or planning operations. These tools used on lathe, boring and shaper machines.

Nomenclature of a single point cutting Tool:

Naming the various angles and parts is known as nomenclature. The parts of a single point cutting tool are shown in fig.

i) shank:

The body of the tool which is not ground is called shank.

ii) Face:

The Surface over which the chip of metal slides is known as face.

iii) Flank:

The surface of the tool which is facing the workpiece is known as flank.

iv) Base:

It is the bottom surface of the shank. Generally it is flat in nature.

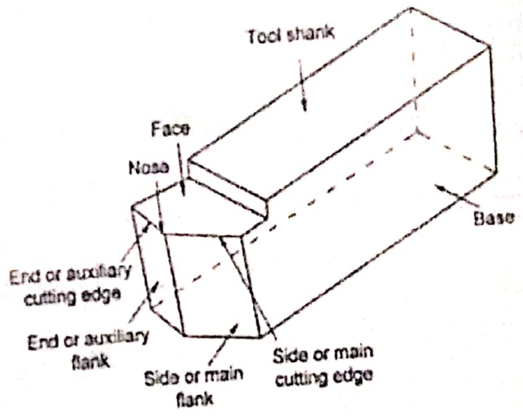


Figure 1.6 Parts of a single point cutting tool

v) Nose:

The junction of sides and end cutting edges are called nose.

vi) Cutting edge:

It is the junction of face and flank. It is generally denoted by two types of cutting edges.

1) End or auxiliary cutting edge
2) Side or main Cutting edge

Angles of single point cutting tool:

The various angles of a single point cutting tool are shown in fig.

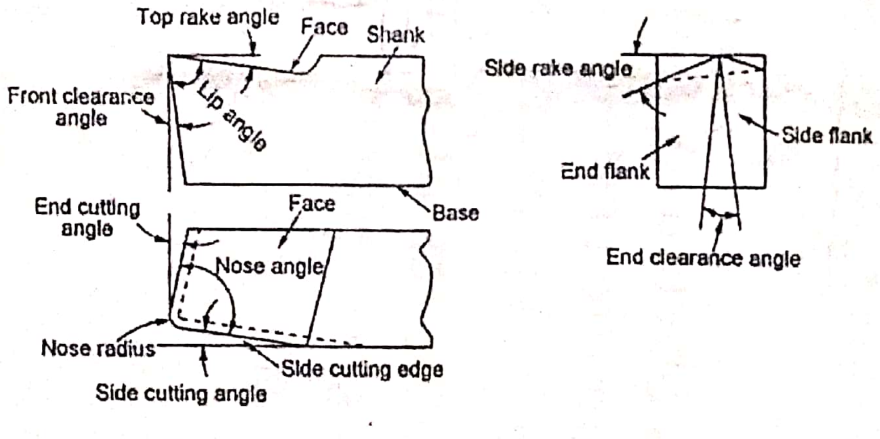


Figure 1.7 Nomenclature of a single point cutting tool

i) Back rake angle or Top rake angle:

It is the angle between the face of the tool and the line ||l to the base of the tool. It is the slope given to the face or surface of the tool.

ii) Side rake angle (or) Top rake angle:

It is the slope given to the face or the top of the tool. It is the angle between tool face and line ||l to the base of the tool as shown in fig.

iii) True rake:

A combined front and side rake are provided on the tool suitably. The resultant slope is known as true rake.

iv) Relief angle or Clearance angle:

It is the slope given downwards from cutting edges.

a) Front clearance or Side clearance angle (or) Side relief angle

The angle between side flank and a line ⊥r to the base of tool is known as 'side relief angle'.

b) End clearance angle or End relief angle:

The angle between end flank and a line ⊥r to the base of the tool measured at right angle to the end flank is known as end relief angle.

Its ~~is~~ value varies from 6° to 10°.

v) Cutting edge angles:

There are two cutting edge angles namely

a) Side cutting edge angle:

Side cutting edge angle is the angle between side cutting edge and side of the tool shank. Otherwise it is the angle between side cutting edge and longitudinal axis of the tool.

b) End cutting edge angle:

It is the angle between end cutting edge and a line ⊥r to the tool shank. Otherwise it is the angle between face of the tool and a plane ⊥r to the side of the shank.

vi) Nose radius:

Nose is the junction of side cutting edge and end cutting edges. A slight curved profile is provided at this junction called nose radius.

vii) Lip angle:

It is also called cutting angle. It is the angle between face and end surface of the tool.

Forces in Machining:

During cutting (turning) process on a Solid bar, the following three components of cutting forces are mutually acting right angle.

i) Longitudinal force or Feed force (F_x)

The longitudinal force acts in the direction parallel to the axis of the work but in the direction opposite to feed.

ii) Radial force or Thrust force (F_y).

The radial force acts in a radial direction from the centre line of the workpiece i.e) ⊥r from the centre line of the workpiece

iii) Tangential force (or) Cutting force (F_z).

It acts in a direction tangential to the revolving workpiece and it represents the resistance to the rotation of the workpiece.

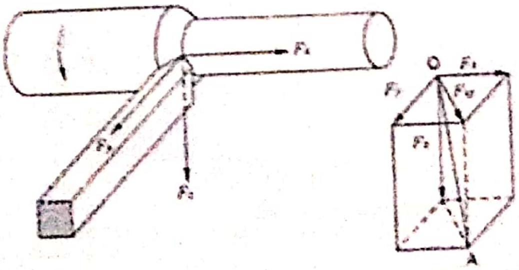


Figure 1.15 Cutting forces during turning

Resultant force $R = \sqrt{F_x^2 + F_y^2 + F_z^2}$

Merchant Circle Diagram of Forces:

Fig. shows a Merchant circle diagram which is convenient to determine the relation between various forces and angles.

Let P → Frictional resistance on tool

N → Normal force on tool

F_z → Cutting force

F_x → Feed force

α → Rake angle

β → Shear angle

γ → Friction angle

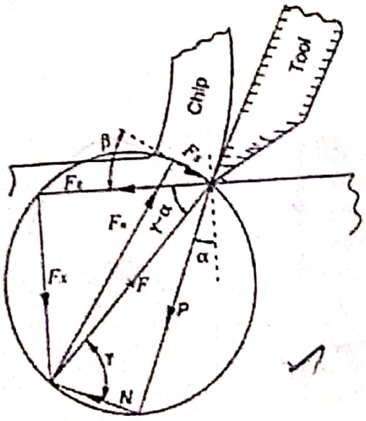


Figure 1.17 Merchant's circle diagram

The cutting force F_z and feed force F_x can be determined by using a dynamometer. After measuring F_x & F_z, they are drawn in to a suitable scale. The resultant of F_x & F_z is the diameter of Circle (F). The rake angle (α) is measured from the tool and forces P & N are determined.

The shear angle (β) are obtained from the relation. Then all force components on the chip are determined from the geometry.

When the chip slides over the tool face under the pressure, there is some friction (μ) between these two. Therefore,

Coefficient of friction $\mu = \dfrac{P}{N} = \tan\gamma$

As mentioned earlier, the shear angle (β) is obtained from the eqn (from fig)

$$\tan\beta = \frac{r\cos\alpha}{1-\cancel{\cos}\,r\sin\alpha} \qquad \text{(formula)}$$

Frictional resistance, $P = F_x\cos\alpha + F_z\sin\alpha$

Normal force $N = F_z\cos\alpha - F_x\sin\alpha$

Resultant force $F = \sqrt{F_z^2 + F_x^2}$

Cutting force $F_z = F\cos(\gamma - \alpha)$

Shear force $F_s = F\cos\theta$

Where, $\theta = \beta + \nu - \alpha$

$$F = \frac{F_s}{\cos\theta}$$

Substituting F value in F_z equation,

$$F_z = \frac{F_s \cos(\nu - \alpha)}{\cos\theta}$$

$$F_z = \frac{F_s \cos(\nu - \alpha)}{\cos(\beta + \nu - \alpha)} \qquad [\theta = \beta + \nu - \alpha]$$

W.k.T $\mu = \tan\nu$

$$= \frac{P}{N} = \frac{F_x \cos\alpha + F_z \sin\alpha}{F_z \cos\alpha - F_x \sin\alpha}$$

$$\mu = \frac{F_x + F_z \tan\alpha}{F_z - F_x \tan\alpha}$$

The relationship for F_s & F_n are given by

$$F_s = F_z \cos\beta - F_x \sin\beta$$

$$F_n = F_z \sin\beta + F_x \cos\beta$$

Types of Metal cutting process:

The metal cutting processes are mainly classified into two types.

i) Orthogonal cutting process (2D cutting)

ii) Oblique cutting process (3D cutting)

i) Orthogonal Cutting Process:

In orthogonal cutting, the cutting edge is Straight, ||l to the original plane surface on the workpiece and ⊥r to the direction of cutting. For example: Lathe cut-off operation. Straight milling are. It involves only two forces and it makes the analysis simpler.

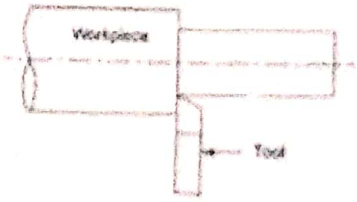


Figure 1.1 Orthogonal cutting

ii) Oblique Cutting process:

In oblique cutting, the cutting edge is inclined at an acute angle with the normal to the cutting direction. The analysis is more complex. In actual mk ng such as turning, milling etc are examples

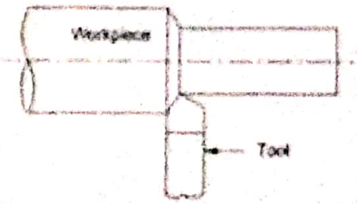


Figure 1.2 Oblique cutting

Thermal Aspects

In the metal cutting process, the energy dissipated at the cutting edge is converted into heat. This heat influences a tool wear on cutting tools and it develops friction between cutting edge of the tool and chip interface.

So plastic deformation takes place and it leads to convert the whole energy into heat. Finally, the energy used for m/c ing is stored in the material or workpiece as strain energy. Therefore the heat is generated in three regions such as shear zone, chip tool ~~work~~ interface region and ~~tool~~ work interface region.

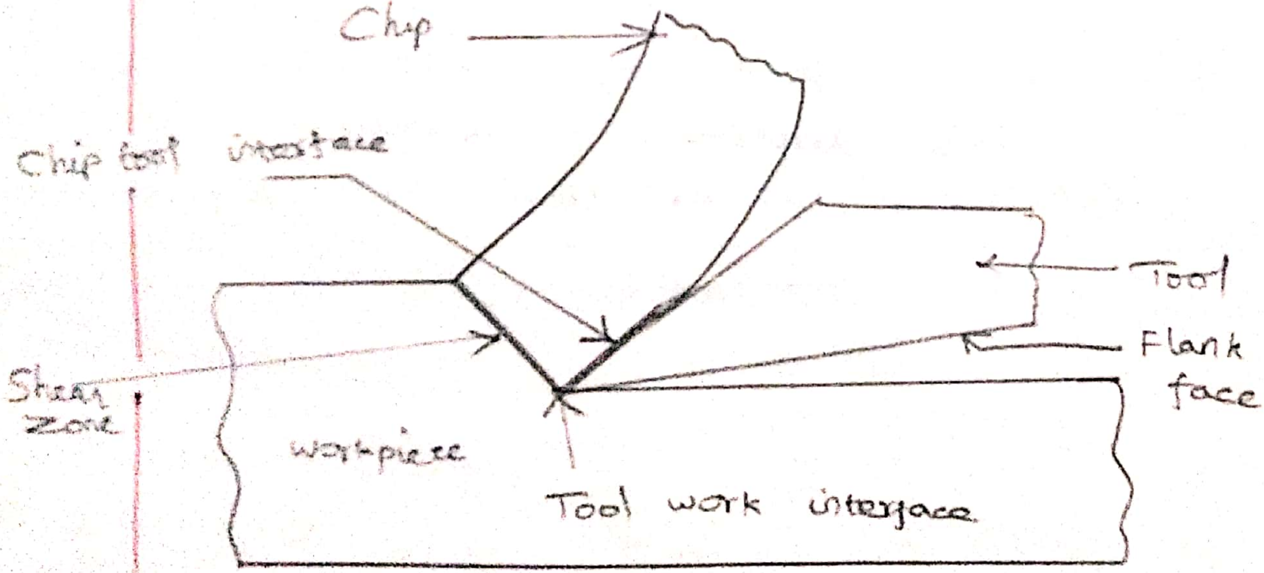


Fig Regions of heat generation in metal cutting process

(i) Shear Zone:

The Zone which is affected by the energy required to shear the chip or to Separate the chip and work is called Shear Zone. So the energy required to shear the chip is the source of heat. Nearly 80-85% of heat is generated in this region.

ii) Chip - tool interface region:

In this region the energy required to overcome the friction completely is the source of heat. Here Some plastic deformation also takes place. The heat generation is 15-20%.

iii) Tool - work interface region:

In this region, the energy required to overcome the rubbing friction between flank face of the tool and workpiece is the source of heat. In this region, the heat generation is in the range of 1-3%.

The tool temperature increases due to the following factors such as

i) cutting Speed
ii) Feed
iii) Properties of tool materials etc...

Cutting tool materials:

The various materials are used to remove the metal from workpiece.

Properties / characteristics of Cutting Tool Material:

1. Hot hardness:

The tool must maintain its hardness at high temperature.

2. Wear resistance:

It is the ability to resist wear. It leads to the poor surface finish.

3. Toughness:

It is the combined property of strength & ductility. It should have sufficient toughness to withstand shock and vibrations.

4. Low friction:

The coefficient of friction between tool and workpiece must be low. It reduces friction, heat developed and tool wear.

5. Cost of tool:-

Tool material should be economical in production.

Other Properties:

* High thermal conductivity
* Resistance to thermal shock
* Easy to grind and sharp
* Low mechanical and chemical affinity for the work material.

Classification of Tool Materials:

(a) Carbon tool steel
(b) High speed steel
(c) Cemented carbides
(d) Ceramics
(e) Diamonds

The selection of cutting tool material depends on the following factors

- * Volume of production
- * Tool design
- * Type of machining process
- * Physical and chemical properties of work material
- * Rigidity and condition of machine.

Tool Wear:

During maching process, the tool is subjected to three important factors such as forces, temperature and sliding action due to relative motion between tool & workpiece.

Tool Materials	Composition	Types	Characteristics
Carbon tool Steel	Carbon - 0.8 to 0.13% Manganese - 0.1 to 0.4% Silicon - 0.1 to 0.4%	* Carbon tool Steel * Carbon Vanadium Steel * Carbon Chromium Steel	* Low hot hardness and poor harden ability * It is used for cutting soft material like wood, plastic aluminium & copper * It is used for machining soft materials
Medium alloy Steel	Carbon - 1.2 to 1.3%. Manganese - 0.1 to 0.4%. Silicon - 0.1 to 0.4%.	-	* It is used for making drills, taps and reamers
High Speed Steels	Tungsten - 18%. Chromium - 5.5%. Carbon - 0.7%.	* High tungsten HSS * High molybdenum HSS * W-Mo HSS	* High hot hardness and good harden ability * It has high wear resistance * It is used in broaches, reamers and milling cutters.
Cast alloys	Non ferrous alloy which contains W, Cr, Co & C	-	* It is high hardness more than HSS * It is very brittle * It has less toughness
Cemented carbide tools	Produced by pressing & bonding	* Straight cemented carbides * Ti-W cemented carb * Ti-Ta-W carbides	* High hardness, heat resistance, wear resistance * It can be used up to 1000°C * Low impact resistance
Diamond tool	It is very hardest material	* Carbons * Ballas * Boarts * Ornamental Stones	* Very hard and brittle * It has abrasion resistant with low coefficient of friction * It has burn to CO_2 at 800°C

Classification of Tool wear:

The tool wear is generally classified as follows.

1. Flank wear
2. Face wear or C
3. Nose wear

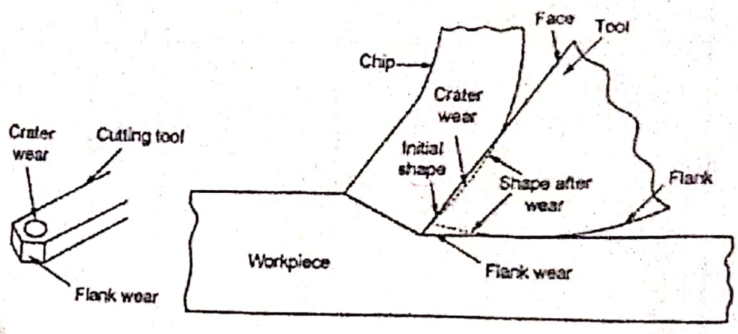


Figure 1.26 Tool wear

Flank wear:

This is also called 'edge wear'. Flank wear is a flat worn out portion behind the cutting edge. The wornout region of the flank is known as wear land.

Crater Wear: [Face wear]

The face of the tool is always contacted with the chip. The chip slides over the face of the tool. Due to the pressure of the sliding chip, the tool face gradually wears out. A cavity is formed on the tool face. The cavity is called crater. This type of wear is known as Crater wear.

Nose wear:

The wear occurs on the nose radius of the tool. When the nose of the tool is rough, more heat will be generated. It is more prominent than flank wear.

Tool Life:

Tool life is defined as the cutting time required for reaching a tool life criterion or time elapsed between two consecutive tool re sharpening.

The following are some ways of expressing tool life.

i) Volume of metal removed per grind
ii) Number of workpieces machined per grind
iii) Time unit

Factors Affecting Tool Life:

1. Cutting Speed:

It has greater influence on the tool life. There is a definite relationship between cutting speed and tool life. This relation is given by Taylor's formula as follows:

$$VT^n = C$$

Where, V - Cutting Speed in m/min
T - Tool life in minutes
n - Exponent or Index which depends on tool & work
= 0.1 to 0.5 for high speed steel tools
= 0.2 to 0.4 for tungsten carbide tools
= 0.4 to 0.6 for Ceramic tools
C = Constant, which gives a tool life of One minute.

2. Feed and depth of cut:

The life of cutting tool is influenced by the amount of metal removed by the tool per minute.

The effect of feed and depth of cut on tool life is given by formula

$$V = \frac{257}{T^{0.19} \times f^{0.36} \times t^{0.08}} \text{ in m/min}$$

Where, V - Cutting Speed

T - Tool life

f - feed in mm/min

t - Depth of cut in mm

3. Tool Geometry:

The rake angle, relief angle, side cutting edge angle influenced the weakens of the tool. The increase in nose radius improves the tool life.

The relationship between Cutting speed (V) tool life (T) and nose radius (r) as follows

$$VT^{0.0927} = 331\, r^{0.244}$$

4. Tool material

Both physical and chemical properties of tool material will influence on tool life. For a given cutting speed, HSS tool is more durable than Carbon Steel tool.

5. Cutting fluid:

The cutting fluid which directly controls the amount of heat at the Chip - tool interface is given by

$$T\theta^{n} = C$$

where T = Tool life
θ = Temperature of chip tool interface in °C
n = An index which depends on shape and material of the cutting tool.

6. Work piece material:

Tool life does also depend on the microstructure of the workpiece material. Tool life will be more when machining soft metals than hard metals such as cast iron and alloy steel.

7. Rigidity of work, tool and machine:

A strongly supported tool on a rigid machine will have more life than tool machining under the vibrating machine. Loose workpiece will decrease the tool life.

Surface Finish:

Generally the Surface finish of any product depends on the following factors:

i) Cutting Speed
ii) Feed
iii) Depth of cut

1. Cutting Speed:

The better surface finish can be obtained at high cutting speed. Rough cutting takes place at low cutting speed.

2. Feed:

Surface finish will not be good when Coarse feed is applied. But a better finish can be obtained in fine feeds.

3. Depth of cut:

Larger cuts provide a good surface finish to the workpiece. If the depth of cut increases during machining, the quality of surface finish reduces.

Therefore the above three parameters ensure a good surface finish.

Cutting Fluids:

During metal cutting, heat is generated due to plastic deformation of metal and friction at the tool - workpiece interface.

Functions of cutting Fluids:

i) Cutting fluid cools the Cutting tool & workpiece. The heat produced during machining is carried away by the fluid.

ii) When the friction is decreased at the tool - Chip interface, the tool life increases and the surface finish also increases.

iii) It improves the Surface finish as stated ~~earlier~~ earlier.

iv) It causes the Chips to break up into Small parts.

v) It washes away the Chips from the tool. It prevents the tool from fouling.

vi) It prevents the corrosion of work & m/c.

Properties of cutting fluids:

- It Should have good lubricating properties to reduce frictional forces and to decrease the power consumption.
- It Should have a high heat absorbing Capacity.

- * It should have a high flash point
- * It should be odourless
- * It should be non-corrosive to work & tool
- * It should be harmless to operators and the bearings.

Types of cutting fluids:

There are basically two main types of cutting fluids.

a) Water based cutting fluid

b) Straight or heat oil based cutting fluid

i) Mineral oil

ii) Straight fatty oil

iii) Mixed oils or Compounded oil

iv) Sulphurised oil

v) Chlorinated oil.

Methods of applying cutting fluids:

The cutting fluids are used in many ways such as,

i) drop by drop under gravity

ii) flood under gravity

iii) form of liquid jet

iv) atomised form with compressed air

v) through centrifugal action

Machinability:

Machining may be easier in some materials where as it may be difficult in other. This difference may be attributed to the machinability of various materials.

It is defined as the ease with which a material canbe Satisfactory machined. It can also be measured by the following factors.

Variables Affecting Machinability:

1. Work Variables:
2. Tool Variables
3. Machine Variables
4. Cutting conditions

Evaluation of Machinability:

- * tool life per grind
- * rate of metal removal per tool grind
- * magnitude of cutting forces and power Consumption
- * Surface finish
- * Dimensional Stability of the finished work
- * heat generated during cutting
- * ease of chip disposal
- * Chip hardness
- * Shape & Size of Chip

Problem on Tool life:

1 The following cutting speed & cutting time observations has been noted in a machining process.

Calculate
1) 'n' and C of Taylor's equation
2) Recommend the cutting speed for a desired tool life of 60 minutes.

Cutting speed, V	25 m/min	35 m/min
Cutting time, T	90 min	20 min

Given data: $T_1 = 90$ min $V_1 = 25$ m/min
$T_2 = 20$ min $V_2 = 35$ m/min

Solution:

By Taylor's equation

$$VT^n = C$$

$$V_1 T_1^n = V_2 T_2^n$$

$$25 \times 90^n = 35 \times 20^n$$

$$\left(\frac{90}{20}\right)^n = \frac{35}{25}$$

$$n \log\left(\frac{90}{20}\right) = \log\left(\frac{7}{5}\right)$$

$$n = 0.224$$

$$V_1 T_1^n = C$$

$$C = 25 \times 90^{0.224}$$

$$C = 68.4$$

ii) $V \times 60^{0.224} = 68.4$

$$V = \frac{68.4}{60^{0.224}} = 27.34 \text{ m/min}$$

Problem on Merchant Theory :

1. In an orthogonal cutting test with a tool of rake angle 10°, the following Observations were made :

Chip thickness ratio = 0.3

Horizontal Component of Cutting force = 1290 N

Vertical component of cutting force = 1650 N

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

Given data:

$\alpha = 10°$

$F_z = 1290 \text{ N}$

$F_x = 1650 \text{ N}$

$r = 0.3$

Soln:-

Shear angle, $\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1 - r\sin\alpha}\right]$

$$= \tan^{-1}\left[\frac{0.3\cos 10^\circ}{1 - 0.3\sin 10^\circ}\right]$$

$$\beta = 17.31^\circ$$

The frictional force, $P = F_x\cos\alpha + F_z\sin\alpha$

$$P = 1650\cos 10^\circ + 1290\sin 10^\circ$$

$$P = 1848.94\ N$$

The normal force $N = F_z\cos\alpha - F_x\sin\alpha$

$$N = 1290\cos 10^\circ - 1650\sin 10^\circ = 983.88\ N$$

The co-eff of friction at chip - tool interface

$$\mu = \frac{P}{N} = \frac{1848.94}{983.88} = 1.88$$

$$\mu = \tan\gamma$$

$$\gamma = \tan^{-1}\mu = \tan^{-1}(1.88) = 61.99^\circ$$

Shear force, $F_s = F_z\cos\beta - F_x\sin\beta$

$$= 1290 \times \cos 17.31 - 1650 \times \sin 17.31$$

$$F_s = 740.63\ N$$

Force acting ⊥r to shear plane, $F_n = F_x\cos\beta + F_z\sin\beta$

$$= 1650 \times \cos 17.31^\circ + 1290 \times \sin 17.31^\circ$$

$$F_n = 1959.1\ N$$

Resultant force, $F = \sqrt{F_s^2 + F_n^2} = \sqrt{740.63^2 + 1959.1^2}$

$$\boxed{F = 2094.42\ N}$$

2. In an orthogonal cutting test with a tool of rake angle 8°, the following observations were made

Chip thickness ratio = 0.2

Horizontal component of cutting force = 1190 N

Vertical component of cutting force = 1450 N

From Merchant's theory, calculate the various components of the cutting forces and the co-eff of friction at the chip tool interface. (Similar Problem)

Ans:- $P = 1601.51 N$, $N = 976.62 N$, $F_s = 876.45 N$

$F_n = 1658.44 N$, & $F = 1875.79 N$

---

3. During an orthogonal cutting a chip length of 160 mm was obtained from an uncut chip length of 350 mm. The cutting tool has 22° rake angle and a depth of cut is 0.8 mm. Determine shear plane angle & chip thickness.

Given $l_2 = 160 mm$; $l_1 = 350 mm$, $\alpha = 22°$; $\overset{\text{depth of cutt}}{t_1} = 0.8 mm$

To find Shear plane angle (β) & Chip thickness (t_2)

Soln

Chip thickness ratio $r = \frac{l_2}{l_1} = \frac{160}{350} = 0.46$

Shear plane angle, $\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1 - r\sin\alpha}\right]$

$$= \tan^{-1}\left[\frac{0.46 \times \cos 22°}{1 - 0.46 \times \sin 22°}\right]$$

$$\beta = 27° 15'$$

W.K.T, Chip thickness $r = \frac{t_1}{t_2}$

$$0.46 = \frac{0.8}{t_2}$$

$$t_2 = 1.74 mm$$

UNIT - II

TURNING MACHINES

Centre Lathe:

A lathe is a father of all machine tools. It is the most important machine used in any workshop.

The following operations can be done by using Lathe: turning, taper turning, eccentric turning, Chamfering, facing, drilling, boring, reaming, tapping, knurling, forming, grooving, Polishing, Spinning & thread cutting.

Constructional Features of a Lathe:

The principal parts of an engine lathe are labelled and shown in fig. The following are the principal parts of the lathe.

i) Bed
ii) Headstock
iii) Tailstock
iv) Carriage
v) Feed mechanism

A brief description of these parts is as follows.

A brief descri

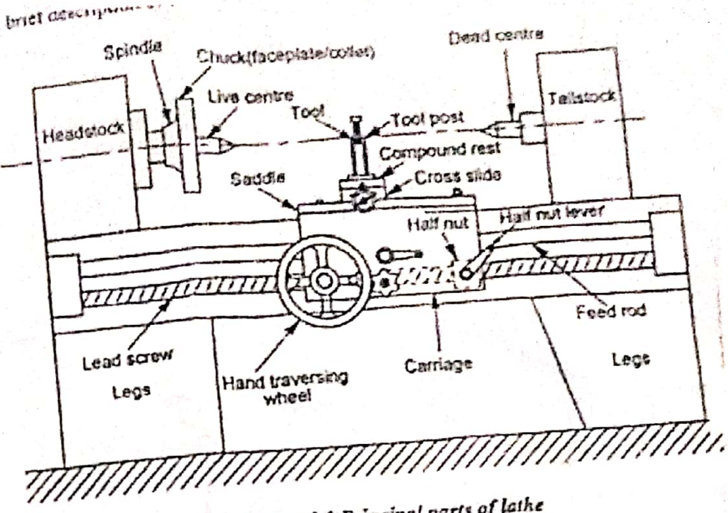
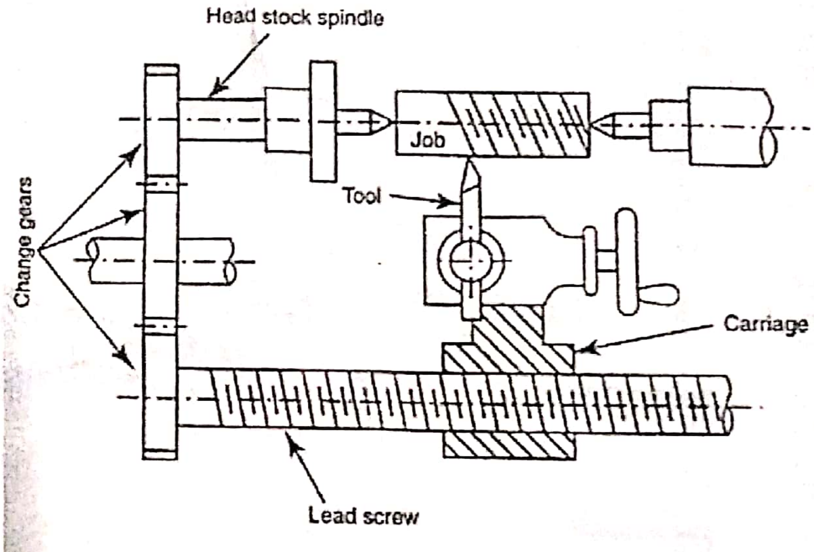


Figure 2.1 Principal parts of lathe



(b) Thread cutting operation arrangement (Top view)

Figure 2.43 Thread cutting

1. Bed:

Bed is the base of machines. It carries a headstock on its left end and tailstock on its right end. The carriage is mounted at the middle of bed. The bed has V and dovetail guideways as shown in fig.

Headstock:

The headstock assembly is permanently fastened to the left end of the bed. It carries a hollow spindle so that bass can be passed through it when it is required. The nose of the spindle is threaded to hold the chuck or faceplate.

A live centre can be attached to the spindle. This centre is called live centre because it turns with the work.

Tailstock:

Tailstock is situated at the right end of the bed. It is used for supporting the right end of work. It consists of a taper hole adjusting screw and hand wheel.

Tailstock consists of two main parts. The lower part directly rests on bed ways and the upper part rests on the lower part (base). A dead centre is fixed into the taper hole of the spindle for supporting the right end of the work.

Carriage:

The carriage is a moving part that slides over the guideways between handstock and tailstock. It ~~carriage~~ carries the following parts as shown in fig.

i) Saddle
ii) cross slide
iii) Compound rest
iv) Tool post
v) Apron

Specification of Lathe:

The size of the lathe is generally specified as follows

1. The length of bed
2. Maximum distance between dead and live centres
3. Type of bed
4. The height of centres from the bed
5. Swing over the bed
6. Swing over the cross slide
7. Width of the bed
8. Spindle bore.
9. Spidle speed
10. HP Of main motor and rpm.

Types of Lathe

1. Speed Lathe
    - (a) Wood working lathe
    - (b) Metal Spinning lathe
    - (c) Metal turning lathe
    - (d) Polishing lathe
2. Engine Lathe
    - (a) Step cone pulley drive lathe
    - (b) Geared lathe
    - (c) Variable speed lathe
3. Bench lathe
4. Tool room lathe
5. Semi automatic lathe
    - (a) Capston lathe
    - (b) Turret lathe
6. Automatic lathe
7. Special purpose lathe
    - (a) Crankshaft lathe
    - (b) Wheel lathe
    - (c) Duplicating lathe
8. Copying lathe

Taper Turning Methods:

(a) Form tool method
(b) Tailstock setover method
(c) Compound rest method
(d) Taper turning attachment method.

Form tool method:

It is one of the simplest methods to produce short taper. The form tool is ground to the required angle. When the workpiece rotates, the tool is fed ⊥r to the lathe axis as shown in fig.

Taper length < tool cutting edge length.

It is done at slow speed.

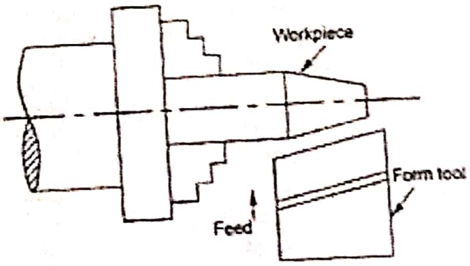


Figure 2.50 Form tool method

2. Tailstock Set Over Method:

This method is employed when the angle of taper is very small (less than 8°). The workpiece is held between live centre and dead centre. Now the tailstock is moved crosswise (or) $\perp^r$ to the lathe axis by turning the set over screw. This process is called 'tailstock set over'.

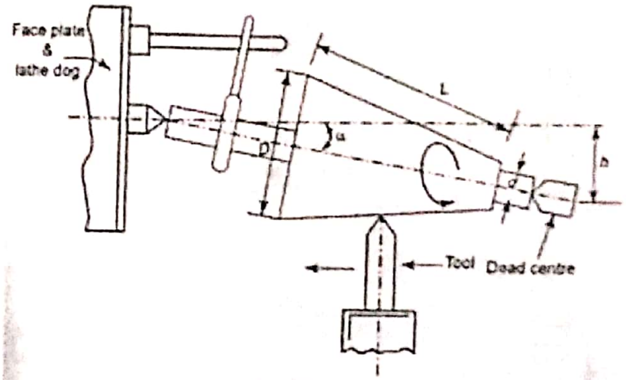


Figure 2.51 Tailstock set over method

Tailstock set over, $h = \frac{D - d}{2l} \times L$

Where, D - Major diameter of the workpiece
d - Minor diameter of the workpiece
l - Required length on which taper being made
L - Full length of the workpiece.

3. Compound rest method:

This method is used to produce a short and steep taper. In this method, the work is held in a chuck and it is rotated about the lathe axis. The compound rest is swiveled to the required angle and clamped in position. The angle is determined by using the formula, $\tan\alpha = \frac{D-d}{2l}$.

This method is used for both internal & external tapers. The compound rest can be swivelled up to 45° on both sides. Tool should be moved by hand.

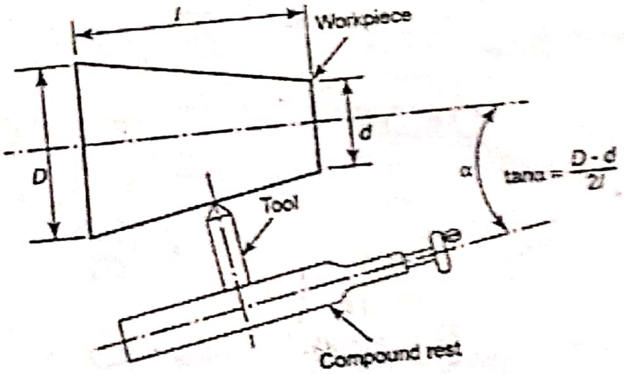


Figure 2.52 Compound rest method

4. Taper Turning Attachment Method:

A taper turning attachment is attached to the rear end of the bed by using a bottom plate or bracket. It has guide bar which is pivoted as its centre. This guide bar can swing and set at any required angle up to 10

It has a guide block which connects to the rear end of the cross slide and it move on the guide bar.

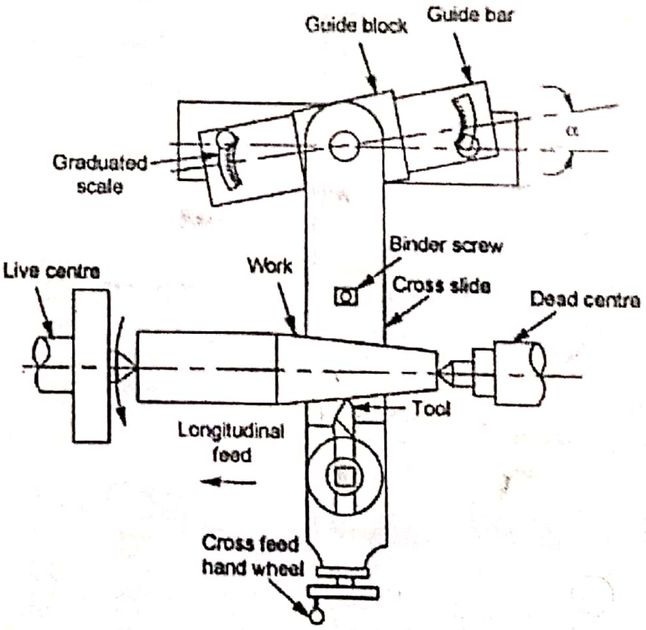


Figure 2.53 Taper turning attachment method

Special Attachments:

1. Milling attachments
   (a) for cutting grooves or keyways
   (b) for cutting multiple grooves and gear wheel

2. Grinding attachment

3. Copy Turning Attachment
   a) Mechanical copy turning attachment
   (b) Hydraulic copy turning attachment

4. Spherical turning attachment

Machining Time and Power Estimation:

Machining time depends on the various process parameters such as workpiece material, tool material, cutting speed, feed, depth of cut etc... and hence, it is necessary to select the proper process parameters in right combination for optimum process conditions.

a) Cutting speed (V):

The length of the chip removed per minute is its measure.

$$\text{Cutting speed, } V = \frac{\pi D N}{1000} \text{ in m/min}$$

Where, D - Diameter of the workpiece in mm

N - Speed of rotation of the workpiece in rpm.

b) Feed (f)

Feed is the amount of advancement of tool parallel to the surface being machined per revolution of the job

$$f = \frac{L}{NT} \text{ mm/rev}$$

L - Length of travel of the tool per pass, mm

N - Speed of rotation of workpiece in rpm

T_m - Cutting / machining time, min

c) Depth of cut (d):

The depth of cut is the advancement of the tool in the job in a $\perp r$ to the surface being machined.

$$\text{Depth of cut, } d = \frac{D_1 - D_2}{2}$$

Metal Removal Rate (MRR):

The metal removal rate is the volume of material removed per unit time. The volume of metal removed is a function of speed, feed and depth of cut.

Metal removed/rev = Volume of chip having length $\pi \times D_1$
× Uncut chip area (A_c)

$$MRR = \pi D_1 A_c \text{ mm}^3$$

$MMR = \pi D_1 A_c N$ mm³/min

$MMR = 1000\ A_c V$ mm³/min

$V = \frac{\pi D_1 N}{1000}$ in m/min

The uncut chip area canbe calculated as follows

Uncut chip area A_c = Width of chip (b) × thickness of uncut chip (t)

= feed (f) × depth of cut (d)

~~MMR~~ $MMR = 1000\ f d V$ in mm³/min

Estimation of Machining Time (T_m):

Machining time is the time required for turning one pass on the metal.

$T_m = \frac{L}{fN}$ minutes per pass

$L = l + x + y$

x - Length of tool approach

y - Length of tool over travel

$L = \frac{D}{2} + x + y$

$n = \frac{\text{Total machining allowance}}{\text{Material removed per cut}}$

$\text{Total m/c ing allowance} = \frac{D_1 - D_2}{2}$

Total m/c ing time, $T_{total} = T_m \times n$

Estimation of Power

Power is the product of Cutting force and Velocity.

Power required, P = Cutting force (F_c) × Velocity (V)

$$P = F_c \times V$$

Cutting force, $F_c = k \times d \times f$

so, $P = k \times d \times f \times V$

$$P = P_s + P_f$$

P_s – Power due to shear

P_f – Power due to friction

$$F_c \times V = F_s \times V_s + F_f \times V_f$$

where, F_s - Shear force

V_s - Velocity of shear

F_f - Friction force

V_f - Velocity due to friction

In oblique cutting,

$$F_c = \sqrt{F_x^2 + F_y^2 + F_z^2}$$

where, F_x - Force acting on X-X direction

F_y - Force acting on Y-Y direction

F_z - Force acting on Z-Z direction

Capstan and Turret Lathes:

Capstan and turret lathes are the natural developments of a centre lathe. These lathes are built to machine workpiece that are large in numbers and on repetitive basis for batch production jobs.

Principal Parts of Capstan and Turret Lathes:

The principal parts of the capstan & turret lathes are as follows.

1. Bed
2. Headstock
3. Turret head and Saddle
4. Cross slide

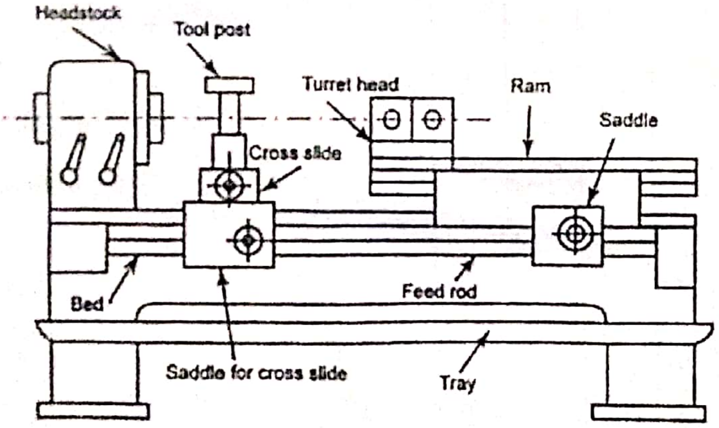


Figure 2.62 Capstan and Turret lathe

1 Bed:-

Bed is the base of machines. It carries a headstock on its left end and tailstock on its right end. The carriage is mounted at the middle of the bed.

A pinion gear is meshed with rack for moving the carriage when the hand wheel is turned. The bed is made of cast iron alloyed with nickel and chromium.

2. Head stock:

The headstock assembly is permanently fastened to the left end of the bed. It carries a hollow spindle so that bars can be passed through it when it is required.

3. Tailstock:

Tailstock is situated at the right end of the bed. It is used for supporting the right end of work. It consists of taper hole adjusting screw and hand wheel.

4. Carriage:

The carriage is a moving part that slides over the guideways between headstock and tailstock. It carries the following parts as shown in fig.

i) Saddle
ii) Cross slide
iii) Compound rest
IV) Tool post
V) Apron

Specification of a Lathe:

The size of the lathe is generally specified as follows.

1. The length of bed
2. Maximum distance between dead and live centres
3. Type of bed
4. The height of centres from the bed
5. Swing over the bed
6. Swing over the Cross slide
7. Width of the bed
8. Spindle bore
9. Spindle speed
10. HP of main motor and RPM
11. Number of Spindle speeds
12. Spindle nose diameter
13. Feeds
14. Floor space required

Tool Layout:

Turret and Capstan lathes are mainly used for machining workpieces at a rapid rate. Before starting the production, the following works are carried out.

1. Selection of tools
2. Designing of special tools
3. Selection of speed
4. Selection of feed
5. Setting the required length of workpiece and tool travel length.

These planning of operation sequence and preparation of turret or capstan are termed as tool layout.

The tool layout mainly consists of three stages.

1. Planning and Scheduling stage
2. Detailed sketching of various stages
3. Sketching the plan showing the various tools fitted unto the hexagonal turret faces.

Rules for Preparing Tool layout of Turret & Capstan Lathe :

The following rules should be kept in mind and followed in laying out the sequence of operation on the lathe.

1. For small batch production, simple tool layout should be used with standard tools.
2. Different machining operations should be done simultaneously as far as possible.
3. Similarly, the handling operations can also be combined with the machining operations.
4. During simultaneous multiple cutting operations the cutting tools should be arranged in such a way that the cutting forces by various cutting tools get balanced.
5. The finishing cut should be done full length of the workpiece involving multiple rough cuts with different tools.
6. It is important to drill a centre hole before final drilling in the case of small diameter holes.
7. In the case of stepped holes, large diameter hole should be drilled first and then the smaller hole should be drilled.

Step by Step Procedure for preparing Tool layout of Turret and Capstan Lathe:

1. The component to be machined is thoroughly studied and the required total length of the work is calculated

2. The number of operations involved in the component starting from the right end is roughly listed.

3. From the rough list of operations involved in the proper sequence is decided.

4. Various tools according to the sequence of operations are selected.

5. The selected tools are fitted either on a hexagonal turret or on cross slide according to the operation sequence.

6. The proper cutting speed, feed and depth of cut for each and every operation are selected

7. The ~~tool~~ time required per piece is determined. The total time includes the time.

8. The detailed drawing of the workpiece is drawn along with the turret tools and cross slide tools in a position.

Automatic Lathes:

Automatic lathe machines are machine tools with fully automatic work cycles. In this lathe, both the workpiece handling and machining operations are performed automatically.

Semi-automatics Lathes:

Semi automatics are machine tools ~~are~~ performing only machining operations automatically. Other operations such as loading the bar stock, starting the machine, checking the work size and unloading the finished component are done manually.

Single Spindle Automatic Lathes:

A single spindle automatic lathe is a modified form of turret lathe. These machines have an addition to a 6 station or 8 station turret, a maximum of 4 cross slides. These cross slides are operated by disc cams. The cams are mounted on a shaft which draws power from the main spindle through a set of gears called cyclic time change gears.

The following types of single spindle automatic lathe are mostly used.

1. Automatic cutting off machine
2. Automatic Screw cutting machine
3. Swiss type automatic screw machine.

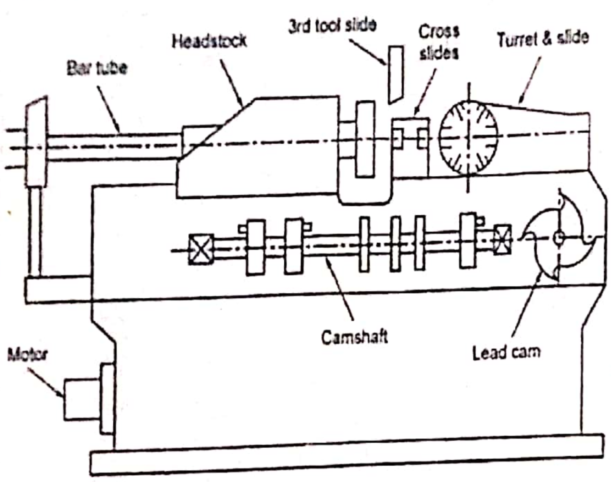


Figure 2.99 Automatic screw cutting machine

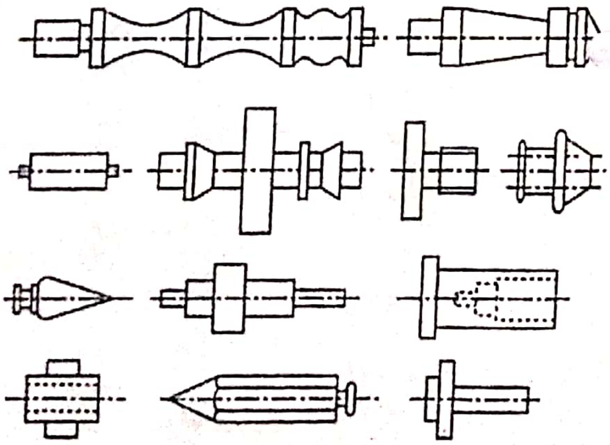


Figure 2.100 Parts produced by automatic screw-cutting machine

1. Automatic Cutting Off machine:

These machines are simple in design and they are used for producing large quantities of parts of smaller diameter and shorter length. The components of simple shapes are produced in this m/c. The principle of an automatic cutting off machine is shown in fig.

Two cross slides are located on the bed. one at the front end of the spindle and other at the rear end of the spindle.

Front cross slide tools are used to perform the main operations such as forming. Rear cross slide tools can perform the operations such as facing, chamfering, cutting off etc...

Cams on a camshaft are actuating the working movements of the cross slides through a system of levers.

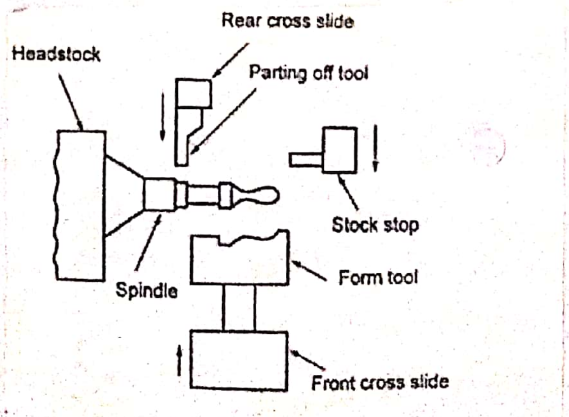


Figure 2.94 Automatic cutting off machine

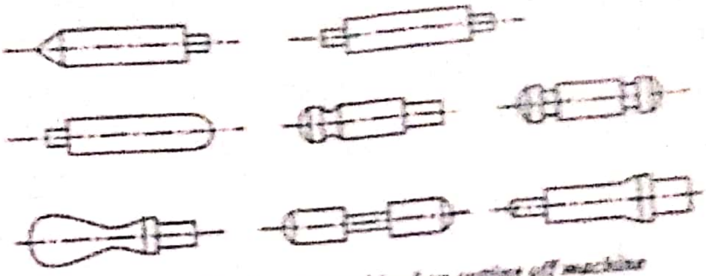


Figure 2.55 Simple parts machined on cutting off machine

Salient features of automatic cutting off machine.

(a) It is more compact in size. So it allows an operator to operate more than on units simultaneously.

(b) It has good adaptability which improves the productivity.

(c) There is no need to align the screws

(d) It is widely applicable in screws of various types, length and head shape

(e) It has unique testing loop which allows for the minimal Tuning time.

Swiss Type Automatic Lathes

(Sliding Head Automatic Lathes)

This type of automatic lathe is suitable for small, long and slender parts such as parts of wristwatches. There is a distinct difference between conventional automatic lathes and Swiss type automatic lathes. In the later, the work is fed against the tool.

The headstock carrying the bar stock moves back and forth for providing the feed movement in the longitudinal direction. Hence this type of automatic lathe is also called a sliding head automatic lathe. The m/c is used for producing long accurate parts of small diameter (2 to 25 mm). In this, parts can be machined to an accuracy of 0.005 mm to 0.0125 mm.

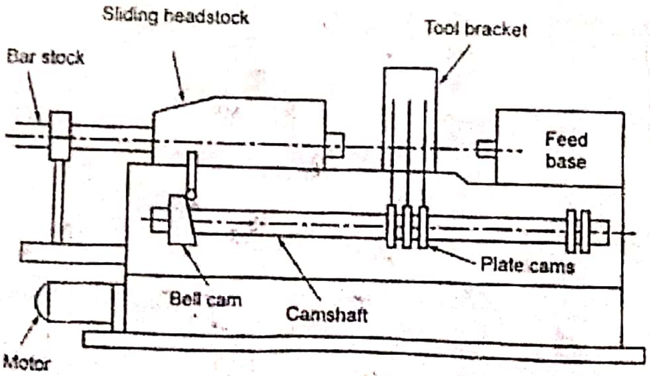


Figure 2.96 Swiss type screw cutting machine

Working Principle :

The bar Stock is held in the rotating spindle by a collet chuck. Headstock slides along the bed ways with the rotating bar stock. Thus headstock movement gives a longitudinal feed to the work. All tools in the tool slides remove material from the workpiece at the same time.

After the workpiece is machined, the headstock slides back to the original position. One revolution of camshaft produces one component.

Most of the turning and forming operations are done by the tools held on the front and rear tool slides. The vertical tool slides are mainly used for undercutting, ~~Ea~~ Chamfering, knurling and cutting off.

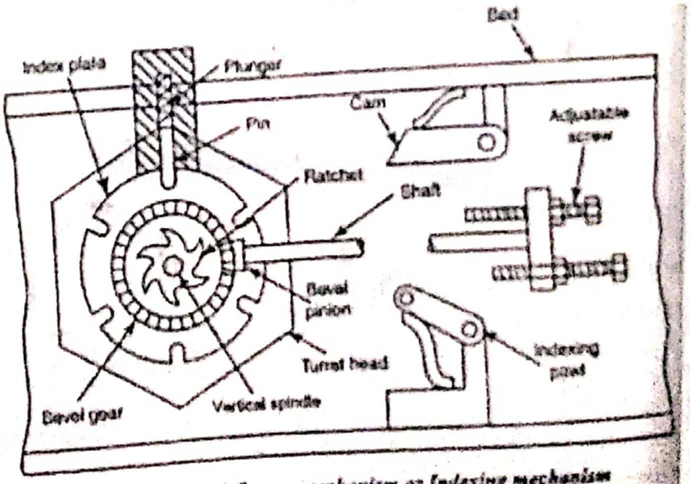


Figure 2.66 Geneva mechanism or Indexing mechanism

on the same

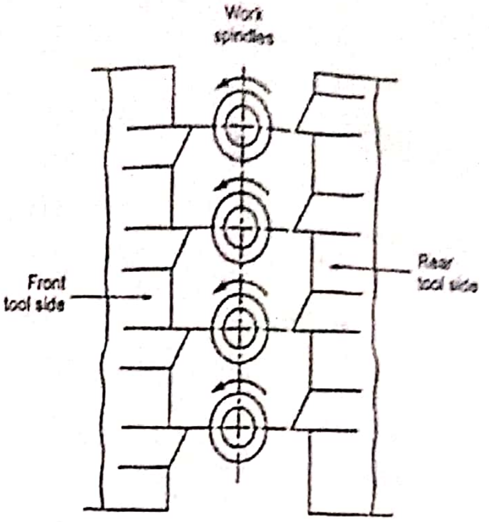


Figure 2.102 Parallel action multi spindle automatic machine

Advantages of Swiss type screw machine:

1. It is used to manufacture precision turning of small parts
2. It has five tool slides
3. A wide range of speeds is available
4. It is rigid is construction
5. Micrometer tool setting is possible
6. Interchangeability of cams is possible
7. imple design of cams is enough
8. Tolerance of 0.005 to 0.0125 mm is obtained
9. Numerous working stations are available.

Single Spindle Automatic Screw Cutting Machine:

These machines are essentially automatic bar type turret lathes. They are widely used for the production of all sorts of small turned parts. It mainly consists of a cross slide & turret.

Two cross slides, one front cross slide and another rear cross slide are provided for cross feeding tools. An additional vertical slide is also employed in this machine.

This third slide is installed above the work spindle. The line diagram of this machine is shown in fig.

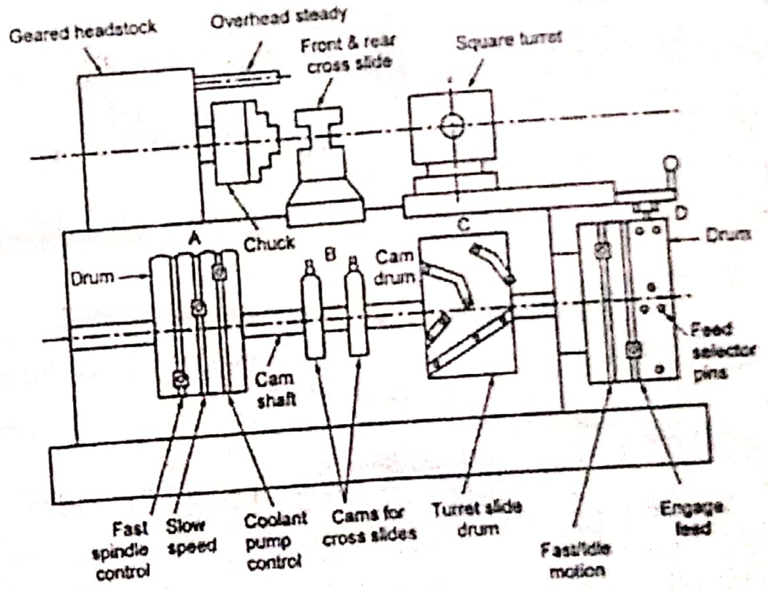


Figure 2.93 Single spindle automatic lathe

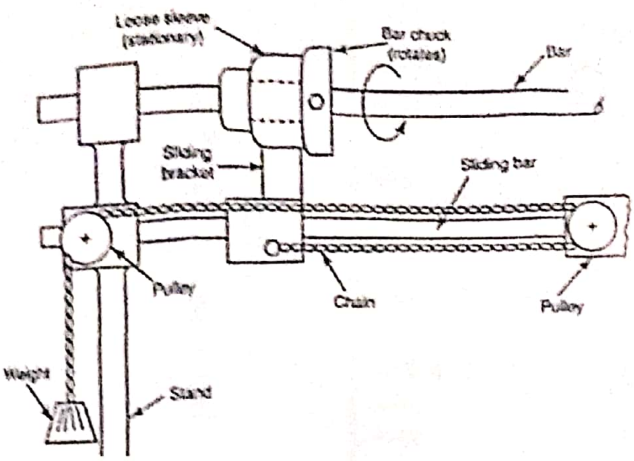


Figure 2.67 Bar feeding mechanism

The dics cams are used to control the cross slide. All operations such as turning, drilling, boring, threading, reaming, spot facing, knurling can be performed on the machine.

Special attachments are also available to perform slotting work, milling flats, cross drilling etc...

Applications:

It is used for producing smal jobs, screws, stopped pins, taper pins, bolts etc...

Multiple Spindle Automatic Lathes

Multiple spindle automatic lathes are machines which can produce larger workpieces than single spindle automats. The principle advantage of the multi-spindle automat is that it has a tool slide working simultaneously on the jobs on all spindles and hence, the time for producing a piece is the time for the longest cut.

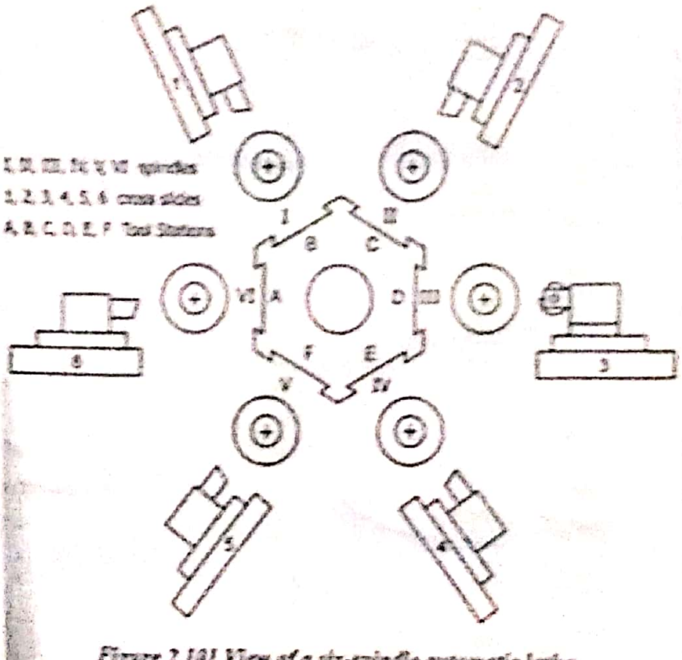


Figure 2.101 View of a six-spindle automatic lathe

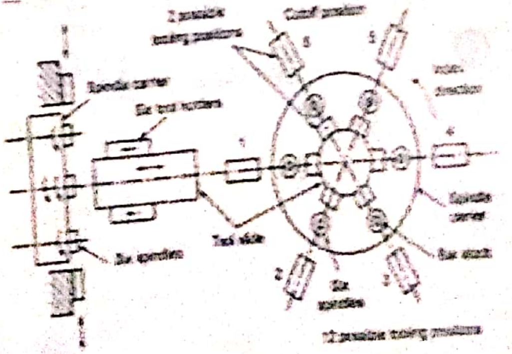


Figure 2.103 Progressive action multi-spindle machine

Each Spindle position has a separate cross slide which is operated by independant cams Cross slides are directly mounted on the headstock. One of the Spindle positions is used for stock loading in the case of magazine feeding or feeding of bar Stock.

Classification of Multi-Spindle Automatic Lathes:

Multi Spindle automatic lathes are classified as follows.

1. According to the type of workpiece (Stock) used
   a. Bar type machine
   b. Chucking type machine
2. According to the arrangement of Spindle
   a. Horizontal Spindle type
   b. Vertical Spindle type
3. According to the principal of Operation
   a. Parallel action type
   b. Progressive action type.

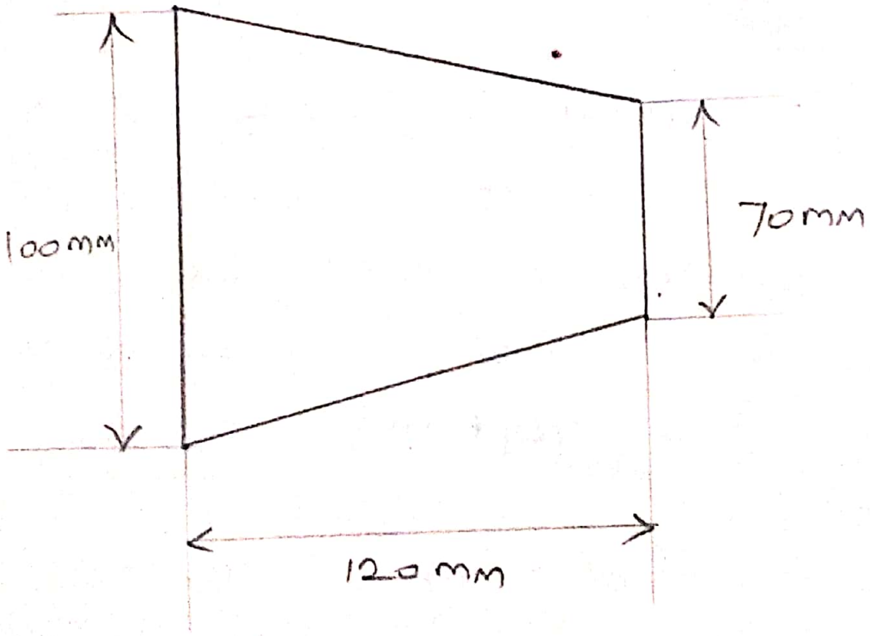
Problems on Taper Turning Methods:

1. Determine the conicity of the workpiece when the major and minor diameters of the work are 100 mm and 70 mm. Length of the workpiece is 120 mm

Given:

$D = 100 mm$; $d = 70 mm$; $l = 120 mm$

Solution:



$$\text{Conicity}, \; K = \frac{D-d}{l} = \frac{100-70}{120} = \frac{30}{120} = \frac{1}{4}$$

Conicity, $\frac{1}{4}$ means the amount of taper is 1:4 or in a length of 4 mm, the diameter of workpiece is reduced by 1 mm.

2. Find the angle of taper for the workpiece having a major diameter 80 mm, minor diameter 70 mm and length 150 mm.

Given data:

$D = 80\,mm$; $d = 70\,mm$; $l = 150\,mm$

Soln: W.K.T $\tan\alpha = \frac{D-d}{2l} = \frac{80-70}{2\times150} = 0.033$

$\alpha = 1^\circ 54'$

The full taper angle, $2\alpha = 3^\circ 48'$

---

3. Calculate the amount of Set over on the tailstock of a lathe for turning a workpiece 120 mm long if the taper is 90 mm long. The major and minor diameters of the workpiece ends are 40 mm and 10 mm respectively.

Given data:

$L = 120\,mm$; length of taper, $l = 90\,mm$

$D = 40\,mm$; $d = 10\,mm$

Soln:-

Tailstock set over, $h = L \times \frac{(D-d)}{2l}$

$= 120 \times \frac{(40-10)}{2\times90}$

$h = 20\,mm$

④ A shaft 250 mm long has a taper of 1:50 for a distance of 150 mm from one end. The maximum diameter of the shaft is 75 mm. What length of the dead centre should be set out of the lathe? Also calculate the minimum diameter of the shaft.

Given $L = 250$ mm; Taper $= 1:50$; $l = 150$ mm

$D = 75$ mm

Soln:

$$\text{Taper} = \frac{D-d}{l} = \frac{1}{50}$$

$$\frac{1}{50} = \frac{75-d}{150} \Rightarrow d = 72 \text{ mm}$$

Set over of the dead centre, $h = L \times \frac{(D-d)}{2l}$

$$h = 250 \times \frac{75-72}{2 \times 150} = 2.5 \text{ mm}$$

---

⑤ Calculate the amount of set over required for the tailstock for turning a workpiece of length 300 mm if the taper is 1:100.

Given $L = 300$ mm, Taper $= 1:100$

Soln

$$\text{Set over } h = \frac{L \times \text{Taper}}{2} = \frac{300 \times \frac{1}{100}}{2} = 1.5 \text{ mm}$$

Problem on MRR:

1. A solid metal shaft of 200 mm long and 60 mm diameter is to be reduced to 58 mm in one pass of turning. Calculate the material removal rate and machining time if the spindle speed is 350 rpm and feed is 210 mm/min.

Given:

$L = 200mm$; $D_1 = 60mm$; $D_2 = 58mm$; $n = 1$ Pass

$N = 350$ rpm; $f =$ ~~0.5~~ 210 mm/min

Soln:-

feed rate, $f = \frac{L}{N} = \frac{210}{350} = 0.60$ mm/rev.

Depth of cut, $d = \frac{D_1 - D_2}{2} = 1$ mm

$$MRR = \pi D_1 A_c N \quad mm^3/min$$

Uncut chip area, A_c = width of chip (b) × thickness of uncut chip (t)

(OR)

= Feed (f) × depth of cut (d)

$$A_c = 0.6 \times 1 = 0.6\ mm^2$$

$$MRR = \pi D_1 (f \cdot d) N$$

$$= \pi \times 60 \times 0.6 \times 1 \times 350$$

$$MRR = 39584.07\ mm^3/min$$

Machining time, $T_m = \frac{L}{fN} = \frac{200}{0.6 \times 350}$

$$T_m = 0.952\ min$$

Problem on Tool Layout:

1. Draw the tool layout for manufacturing the given component as shown in fig. on Capstan lathe.

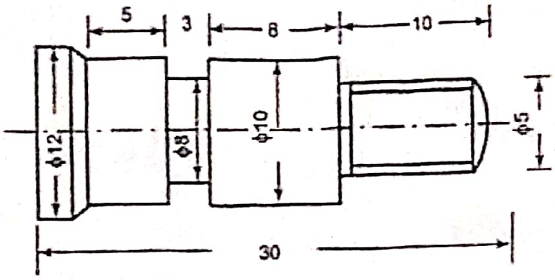


Figure 2.85 Component for Problem 2.23

Soln:-

Stage 1:

1. The component drawing is drawn
2. The total length of the work is calculated and 10 mm is added to provide clearance.
3. The number of operations is listed.
4. The Sequence of operations is listed.
5. The proper machine of 75 mm Capstan lathe is selected
6. The proper material of mild Steel Square bar is selected.
7. All tools and equipment as per the Operation sequence are collected and fitted on turret faces or on cross slides as per our convenience.

Stage II

The tool layout is drawn providing uniform balancing.

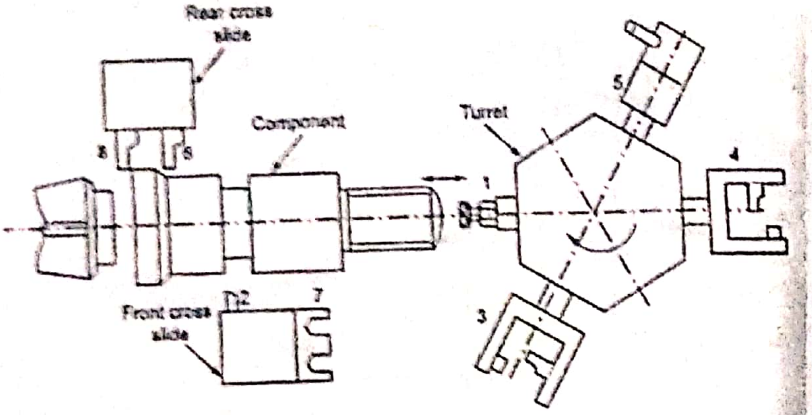


Figure 2.36 Tool layout for Problem 2.23

Stage III

Tooling schedule Chart (to mk given component)

max size of workpiece diameter

MACHINE: 75mm Capstan lathe

MATERIAL: Square mild Steel bar.

Operation Sequence	Description of Operation	Tool Position	Tools used
1	Holding the square bar in collet and setting the required length of 35mm (30+5)	Turret Position-1	Bar stop
2.	Turn to 12 mm diameter to a length 35 mm (from right end)	Front cross slide on the first face -2	Single point cutting tool
3	Turn to 10mm diameter and from the right end of the bolt for a length of 26mm	Turret Position - 3	Roller steady bar ending tool
4	Turn to 5mm diameter and from the right end of the bolt for a length of 10mm and from the end	Turret Position - 4	Roller steady bar ending tool
5.	Make the external thread cutting of 5mm diameter to a length of 5 mm (from right) 2 mm is provided for clearance	Turret Position -5	Self opening die head with chases of 5mm
6.	Undercut to 8mm diameter but the cutting should start 21 mm from right & turn it for 3mm towards ~~right~~ left side	Rear cross slide - 6	Parting off tool
7.	Chamfering the component head on 12 mm diameter	Front cross slide on the second face -7	Chamfering tool
8.	Parting off the component at a distance of 30mm from right end.	Rear cross slide - 8	Parting off tool

Problem 2

(From 2.90 2.95)

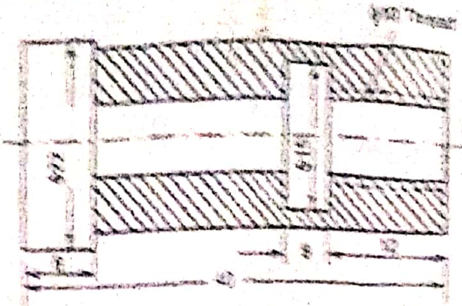


Figure 2.87 Component for Problem 2.24

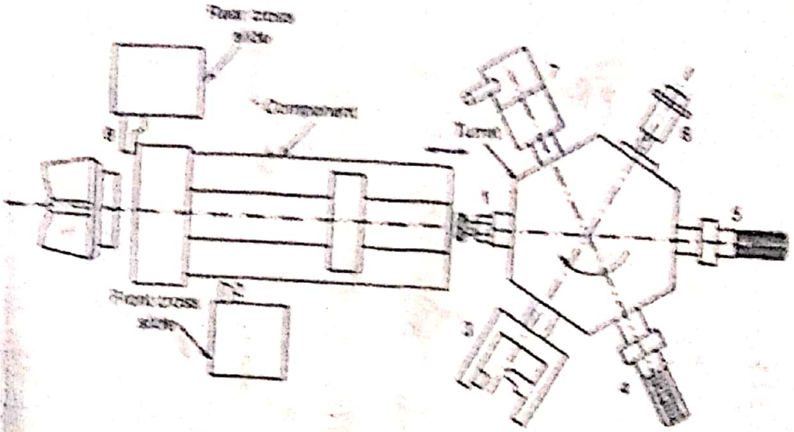


Figure 2.88 Tool layout for Problem 2.24

Problem 3

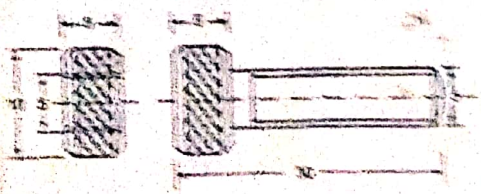


Figure 2.91 Component for Problem 2.26

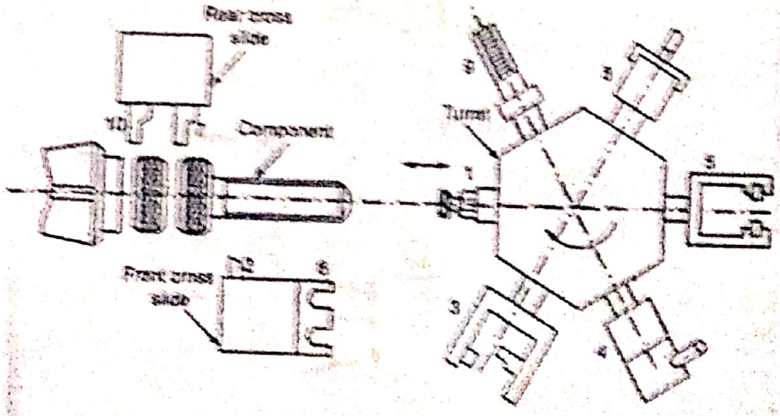


Figure 2.92 Tool layout for knurled screw and nut for Problem 2.26

⑥

UNIT - III

SHAPER, MILLING AND GEAR CUTTING MACHINES

Shaper:

The Shaper having a reciprocating type of machine tool with single point cutting tool is used to produce flat surfaces.

Various types of flat surfaces can be machined by the shaper as follows:

1. The table is moved ~~by~~ in a cross-wise direction to machine the horizontal surfaces.
2. The tool head is moved ⊥r to the table in downward direction to machine the vertical surfaces
3. The tool head is fed at an angle to produce inclined surfaces.

Principal Parts of Shaper:

The different parts of a Shaper are listed and described below.

1. Base
2. Column
3. Cross rail
4. Saddle
5. Table
6. Ram
7. Tool head

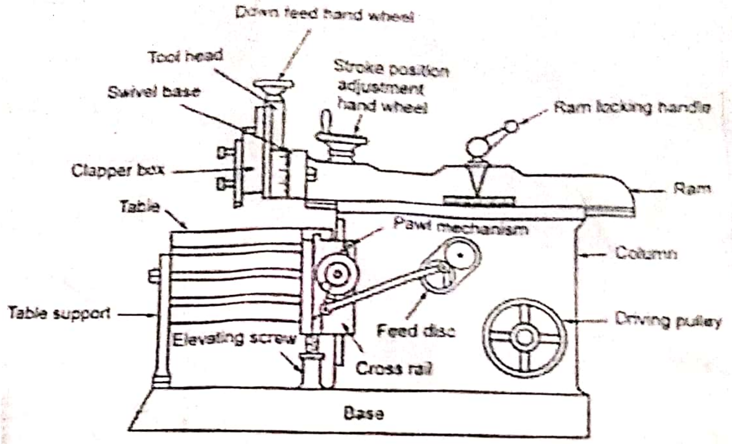


Figure 3.1 A standard shaper

1. Base:

The base is heavy and robust in construction which is made of cast iron by a casing process. It is the only part to support all other parts because all parts are mounted at the top of this base. So it should be made to absorb vibrations due to load and cutting forces while machining.

2. Column:

The column has box type structure which is made of cast iron. The inside surface is made as hollow to reduce the total weight of the shaper. It is mounted on the base. The ram driving (Quick return) mechanism is housed. The two guide ways are provided at the top. The ram reciprocates on this guide way. Similarly, there are two guide way at the front vertical face of the column to move the cross rail along these guide ways.

3. Cross rail:

It is also heavy in construction made of cast iron. It slides on the front vertical ways of the column with two mechanisms.

A saddle slides over two guide ways already provided on the front face of the cross slide. The crosswise movement of the table is obtained by a cross feed screw and the vertical movement of the cross rail is obtained by the elevating scre

4. Saddle:

It is mounted on the cross rail which holds the table in position without any shake.

Scanned with CamScanner

It is also a box type rectangular hollow cast iron block. This table slides along the horizontal guide ways of the cross rail. The work is held on the table. The table has machined surface at the top and it has T- slots for clamping work. It can Vertically be moved by the elevating Screw.

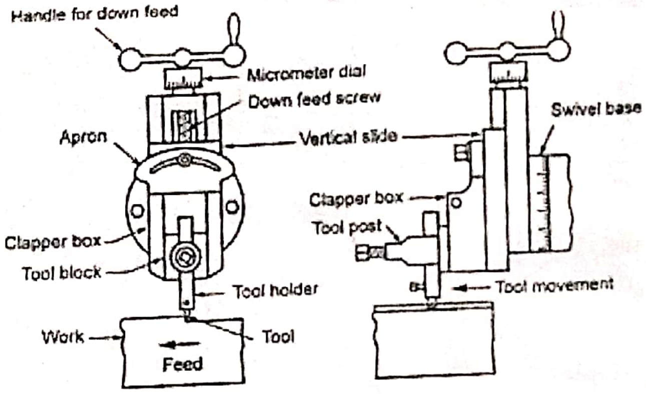


Figure 3.2 Tool head

6. Ram:

Ram is made of cast iron and has cross ribs for rigidity. Generaly it is a reciprocating type which slides over the guide ways at the top of the column. It is connected to driving mechanism of any one type and it also carries the tool at the front end.

7. Tool head:

Tool head is used to hold the tool rigidly. It is having Swivel base with degree graduation. So, the tool head canbe swivelled to any angle as required. The tool head has a vertical slide and apron to provide vertical and angular feeds to the tool. A feed screw with graduated dial moves the vertical slide vertically to set the accurate movement.

Quick Return Mechanism:

The following three types of quick return mechanisms are used in the shaper.

1. Hydraulic drive
2. Crank and Slotted link mechanism
3. Whitworth quick return mechanism

Hydraulic Drive:

A piston reciprocates within the hydraulic cylinder. Oil is sucked by a gear pump from the reservoir at a particular pressure. This high pressure oil goes to the cylinder through a four-way valve. The oil is allowed from the pump to the left side of the piston which forces the piston to move the ram towards right (R)

It is called forward or cutting stroke. In this stroke, oil flows out to the right side entry to the reservoir through the four way valve and drain pipe. The lever hits the trip dog (P_1) at the end of this stroke. Now the lever position is changed. Due to this, the supply pipe supplies the oil to the right side of the piston which moves the ram towards left (L), called return stroke or non-cutting stroke.

During this stroke, the high pressure oil of the same quantity covers less area on the cylinder due to the piston rod which increases the pressure.

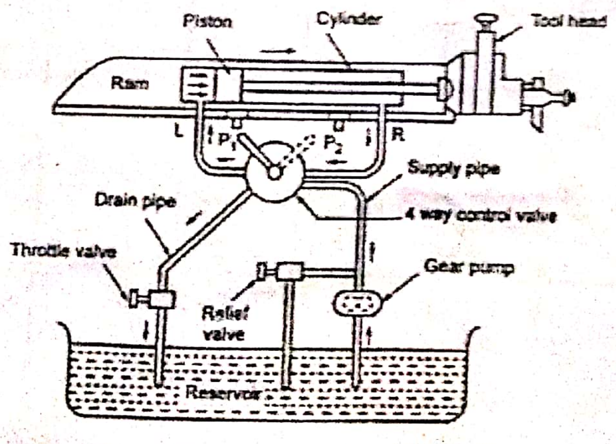


Figure 3.3 Hydraulic drive

Advantages of hydraulic drive

1. Smooth cutting operation can be obtained by uniform speed.
2. Changing of cutting speed is easier.
3. Higher cutting to return ratio can be obtained
4. Infinite range of cutting speeds is available
5. The operation is safe due to the relief Valve
6. Stroke length can easily be adjusted without stopping the machine.

Crank and Slotted Link Mechanism:

When the pinion gear rotates along with the bull gear, the crank will also rotate. Due to this, the rocker arm sliding block also rotates in the same circle. Simultaneously, the sliding block slides up and down in the slot. This movement is transmitted to the ram which reciprocates. Hence, the rotary motion is converted into reciprocating motion.

Quick return Principle:

From fig A_1 & A_2 are rear and forward extreme positions of the link. S_1 & S_2 are two extreme positions of a crank pin.

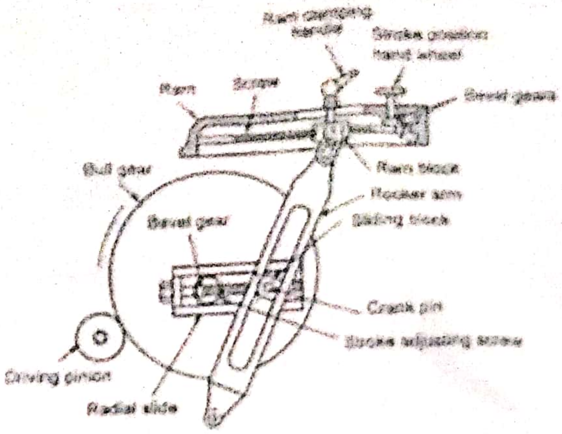


Figure 3.4 Crank and slotted lever mechanism

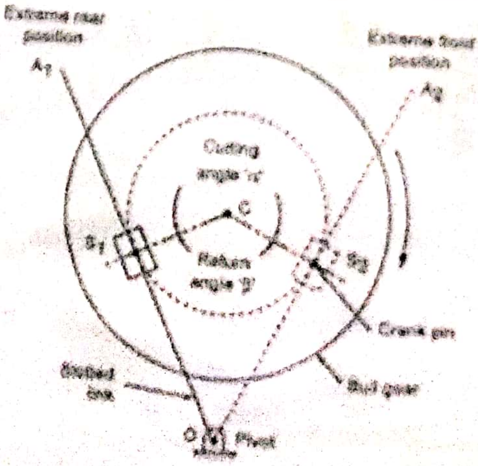


Figure 3.5 Quick return motion

During forward stroke, the link moves from A_1 to A_2 as the sliding block moves from S_1 to S_2 in the clockwise direction at an angle 'α'.

During return stroke, the sliding block goes from S_2 to S_1 in the CW direction through an angle of β. But the speed of bull gear is constant throughout. Therefore, the time taken during these two strokes is directly proportional to these angles α and β. But the angle β is smaller than α. So the time taken by the return stroke will be reduced.

$$m = \frac{\text{Cutting time}}{\text{Return time}} = \frac{\alpha}{\beta} = \frac{\text{Cutting angle}}{\text{Return angle}}$$

The value of m varies from 2:1 to 3:2

Whitworth quick return mechanism:

Figure illustrates the arrangement of various elements in Whitworth quick return mechanism. The shaft of an electric motor drives the pinion which rotates the bull gear The bull gear has a crank pin. A sliding block slides over this crankpin and it slides inside the slot of a crank plate.

This crank plate is eccentrically pivoted at point S. A connecting rod connects the crank plate by a pin at P at one end and ram at the other end M. When the pinion rotates, the bull is also rotated along with the crank pin.

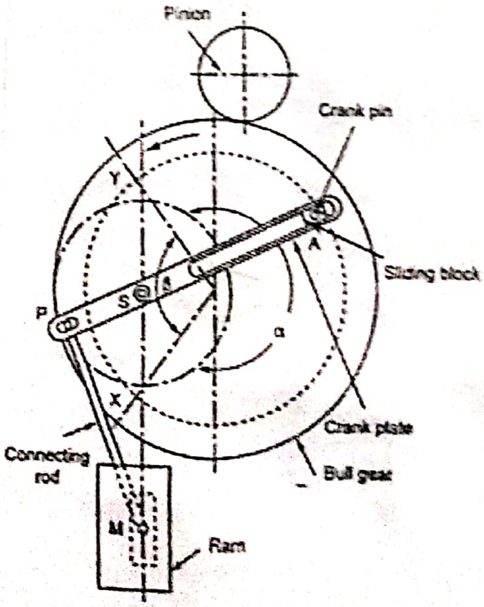


Figure 3.6 Whitworth quick return mechanism

The two important cases are discussed below

1 When the Pin A is at X, the ram will be in forward Stroke. At that time, the bull gear rotates in the anticlockwise direction at angle of α.

2. When the bull gear rotates further in the Same direction from Y to X at an angle of β. the Return Stroke will take place. Here the angle β is less than α. So the time taken for the return Stroke is reduced.

$$m = \frac{\text{Cutting time}}{\text{Return time}} = \frac{\alpha}{\beta}$$

Shaping Operations:

The following operations can be performed on a Shaper.

1. Machining horizontal Surface
2. Machining Vertical Surface
3. Machining angular Surface
4. Machining Slots, grooves and keyways
5. Machining irregular Surfaces.

a) Machining horizontal Surface:

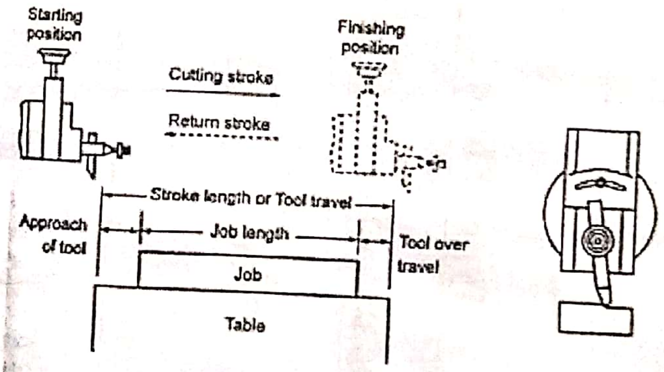


Figure 3.15 Machining horizontal surface

The work is held on a table and the tool is fitted on the tool post with minimum overhang. It should prevent the rubbing of the tool on the work while returning.

The tool is vertically adjusted by some clearance and the stroke length is set longer than the workpiece (e) 12 mm tool approach and 8 mm tool overtun are added to the length of the work as shown in fig. Then the proper cutting speed and feed are chosen.

In any machines, the roughing cut is performed by giving more depth of cut with slow cutting speed and faster feed. Similarly, the finishing cut is performed by giving less depth of cut with faster cutting speed and slow feed.

(b) Machining Vertical Surface:

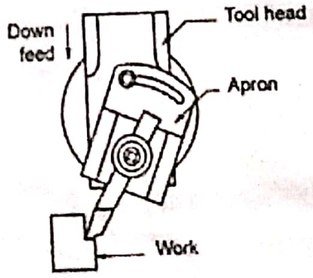


Figure 3.16 Machining vertical surface

The job is held on the table and the tool is set on the tool holder. The tool position and the stroke length are adjusted to required dimensions. Then the value on the vertical slide dial is set at zero. The apron is swivelled as shown in fig. to avoid the rubbing of the tool

(c) Machining angular Surface

The job is mounted on the table and the tool is set at required angle on the tool head as shown in fig. Position and stroke length are adjusted. Also the proper cutting speed and feed are chosen. The apron is set away from the machining surface. The method of giving depth of cut and feed are similar to making the vertical surface

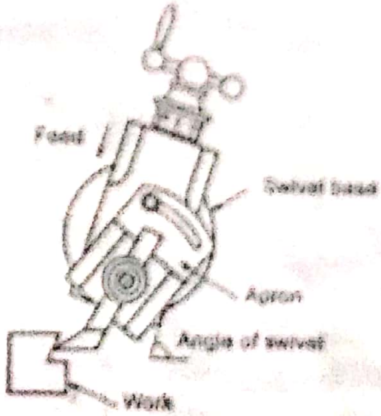


Figure 3.17 Machining angular surface

For example to make the dovetail shape on the right hand side of the workpiece, the vertical slide with the right hand tool is set at the required angle on the right side of the work. By giving the feed and depth of cut, the right side dovetail is finished.

d) Machining slots, grooves and keyways:

The work is held in a vice using V blocks and parallels. First a hole is drilled to a required keyway depth at the end of the workpiece. The diameter of the hole should be greater than the width of the keyway. Then, the position and stroke length are adjusted. The keyway cutting tool is set on the tool head. Finally the external keyway is machined with reduced speed.

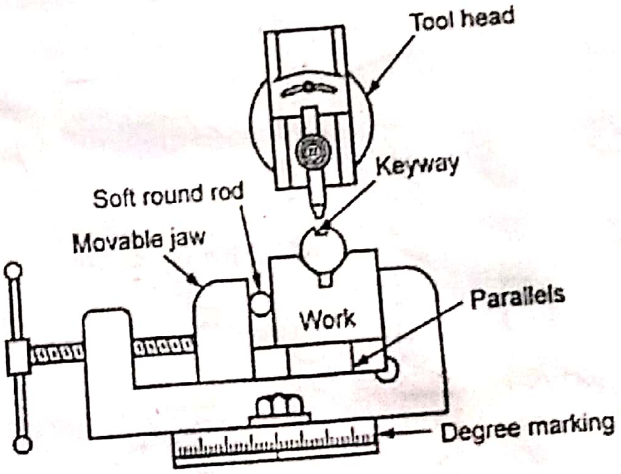


Figure 3.18 Machining grooves and slots

e) Machining irregular surface

For machining the irregular surface, a round nose tool is set on the tool head. By giving both the cross feed and vertical feed simultaneously, the irregular surface is obtained. The cross feed is given through the table and the vertical feed is given by the tool head. The apron is fitted to some angle away from the machined surface to avoid the rubbing of the tool on the work during return stroke.

Drilling Machine :

Drilling is the process of producing the hole on the workpiece by using a rotating cutter called drill. The m/c on which the drilling is carried out is called drilling machine. The drilling m/c ~~so~~ sometimes is called drill press as the machine exerts the vertical pressure to originate a hole.

Portable Drilling Machine :

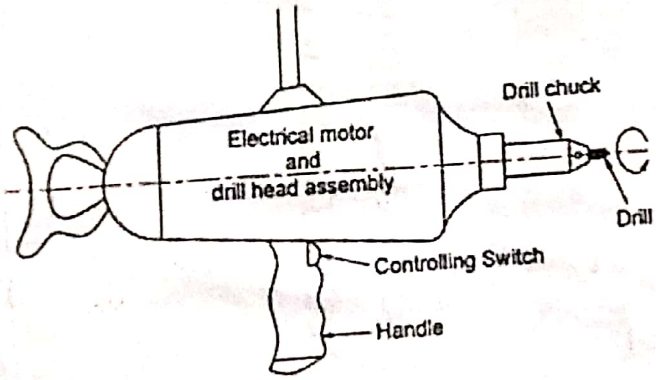


Figure 3.20 Portable drilling machine

This type of m/c is light in weight, compact is a smaller unit and easily handled w.r.to the workpiece. It is used for making small holes (upto 15 m) in large workpiece. It is operated by hand power, Pneumatic power or electric power. Fig shows the electrically operated portable drilling machine.

Sensitive Drilling Machine

Sensitive drilling m/c are light weight, high-speed machines which are generally bench type drilling m/c but Pillar type machines are also available. It is used for light duty work and drill holes up to 15 mm diameter. There is no power feeding arrangement but feeding is purely on hand Control of the operator so that the operator can sense the feeling or can control the feeding. Therefore the machine is called 'Sensitive drilling machine'.

1.

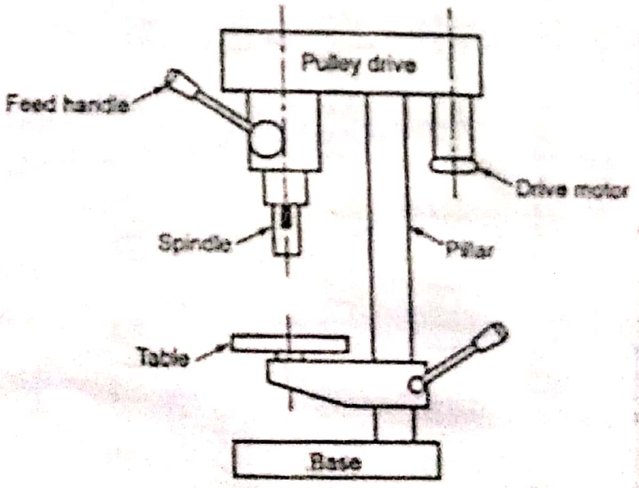


Figure 3.21 Sensitive drilling machine

1. Column:

The column vertically stands on the base. It is a cylindrical post. It supports the table, the spindle head, motor and driving mechanism.

2. Table:

The job on which the hole to be produced is mounted on the table. It can vertically be moved along the column and clamped at any position. It can also radially be adjusted around the column. It has T-slots for clamping workpiece or work holding device.

3. Spindle and driving mechanism:

It is mounted at the top of the column. It has an electrical motor on one side whereas it has the spindle assembly on the other side. The motor drives the spindle through a cone pulley and V-belt arrangement. The belt can be shifted to different sets of pulleys to get different spindle speeds.

The spindle manually feed into the workpiece using a hand lever. The spindle has a Morse taper bore at its bottom end to hold the drill chuck. Drill chuck holds the drill bit.

Upright or Pillar Drilling Machine

Upright drilling m/c is a higher capacity version of the sensitive drilling machine. It is a stationary floo mounted drilling m/c. It is used for medium sized workpieces and having medium speed.

The spindle head and the drive arrangement in this m/c are |||ly to a sensitive drilling m/c. But in this case, Power-feeding arrangements are available. The main parts of the machine are base, column, worktable and spindle head.

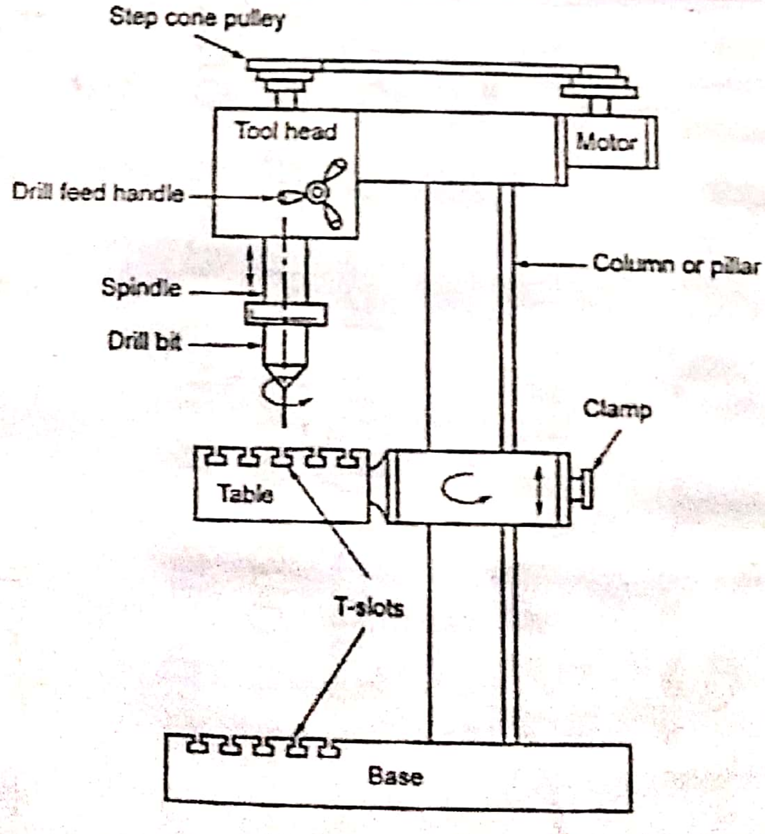


Figure 3.22 Upright drilling machine

Radial Drilling Machine:

The most significant feature of this m/c is a radial arm which can swing about a column. The arm can also be moved up and down with respect to the column which can be locked at any desired position as per Job size.

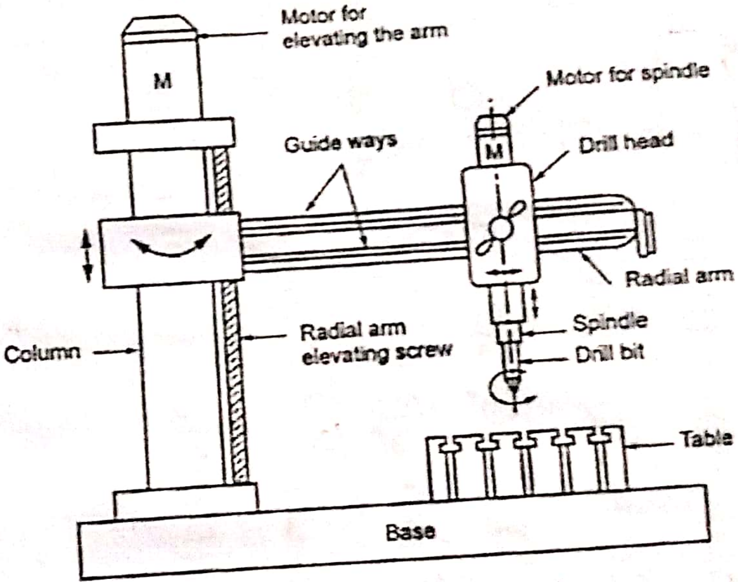


Figure 3.23 Radial drilling machine

Spindle head and feed mechanism:

The various type of radial drilling m/c are :

i) Plain type
ii) Semi-Universal type
iii) Universal type

i) Plain type:

The following adjustments are available in this type.

- * Vertical movement of the radial arm with r to column
- * Circular movement of the radial arm about the colum
- * Horizontal movement of the tool along the arm ways.

ii) Semi-Universal type:

In addition to the above three movements as in the case of a plain type, the fourth movement of the tool post can be swung about a horizontal axis ⊥r to the arm. This arrangement permits for drilling a hole inclined at any angle to the horizontal plane.

iii) Universal type:

In addition to above four movements as in the case of Semi-universal type, the fifth movement permits to rotate (tilt) the radial arm about a horizontal axis. All these movements enable the universal drilling machine to drill on a job at any angle in either horizontal plane or vertical plane or in both planes.

Multi - Spindle Drilling Machine:

This m/c is more suitable for mass production. In this m/c several holes of different sizes can simultaneously be drilled. It has several spindles. They are driven by a single motor by using a set of gears. The centre distance of spindles may be adjusted to any desired length. All spindles holding the drills are fed into the work at the same time. The feed is given either by raising the table or by lowering the spindle head. Drill jigs are sometimes used to guide drills accurately into the work.

Gang Drilling Machine:

When a number of single spindles with essential speed and feed are mounted side by side on one base and they have a common work table known as gang-drilling machine.

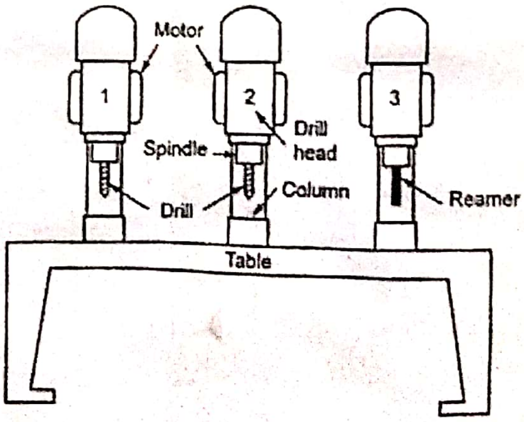


Figure 3.24 Gang drilling machine

The number of Spindles Varies from four to six numbers. The drilling heads of each spindles have individual driving motors as shown in fig. Hence the speed & feeds of the spindles are interdependently controlled.

A series of operation can be done one by one. Each spindle of this m/c is fixed with respective speed, feed, tool and position of the spindle as per operations required for a production job with the arrangement of job movement by using suitable jig and fixture.

Automatic Drilling Machine :

A series of drilling m/c are arranged to perform more than one operation at a time in the sequence of successive workstations is called 'automatic drilling machines'. During mking the workpiece moves automatically to the next stat by using transfer line. The different types of operations such as drilling, reaming, boring, milling etc.. can be carried out one after the other in this type drilling m/c.

Deep Hole Drilling Machine:

Deep hole drilling M/c are used when the drill hole depth exceeds the normal drill hole depth. For very deep holes L/D ratio 6 to even 30, the use of riffle barrels, long spindles, oil holes, bearings and connecting rods are difficult to apply cutting fluids and chip removal.

A drill has a straight flute with single point cutting edge for the whole length of the drill. A hydraulic system is used for forcing oil under high pressure for the whole drill.

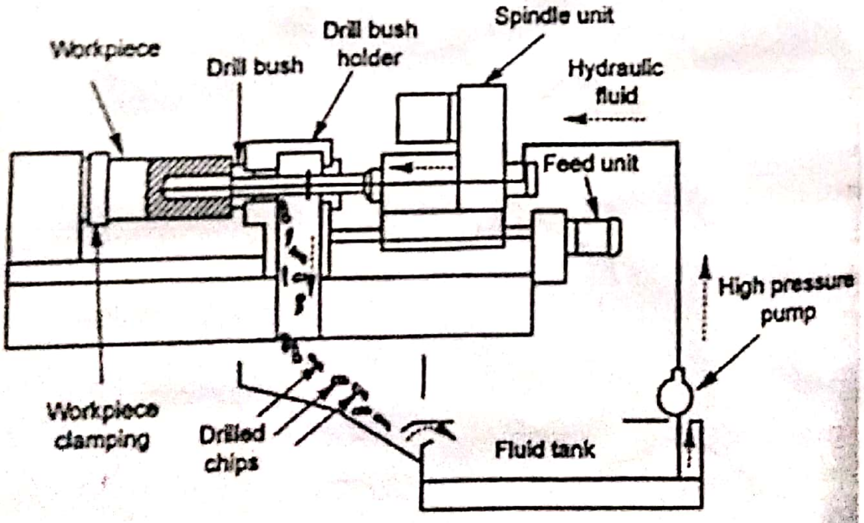


Figure 3.25 Deep hole drilling machine

Drilling Operations:

(i) Drilling:

Drilling is the operation of cutting a round hole ~~&~~ by a rotating tool called drill. Before drilling the centre of hole is located on the workpiece.

Drilling does not produce an accurate hole. The internal surface produced by drilling will be rough. The hole is slightly larger than the size of the drill used due to the vibration of drill.

ii) Reaming:

Reaming is the process of sizing and finishing the already drilled hole. The tool used for reaming is known as reamer. Reamer is a cylindrical tool having many cutting edges as shown in fig. It can't produce a hole. It simply follows the path of an already drilled hole.

iii) Boring:

Boring is an operation of enlarging a hole by a single point cutting tool as shown in fig. Boring is done where the suitable size drill is not available. If the hole size is very large, it can't be drilled. Then boring is done to enlarge the hole to get required size.

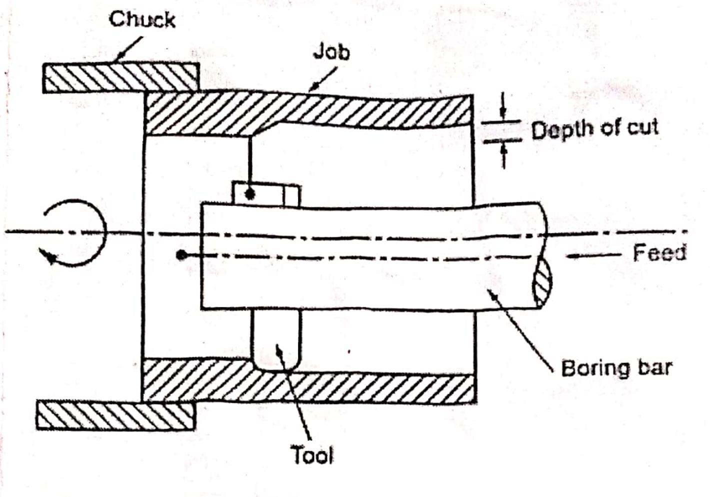


Figure 3.45 Boring operation

(V) Counter Sinking:

The operation of making a cone shaped enlargement of the end of a hole is known as Counter Sinking.

vi) Spot facing:

The operation of squaring and smoothing the surface around a hole is known as Spot facing. Fig. illustrates the process of spot facing.

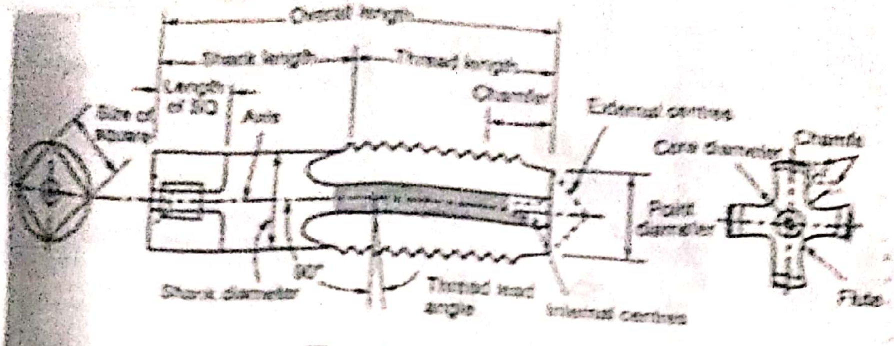


Figure 3.44 Tapping tool

vii) Tapping:

It is an operation of cutting internal threads in a hole by using a cutting tool called tap. When the tap is screwed into the hole, it will remove metal and cut internal threads.

Tap drill size = 0.8 × Outside diameter of thread.

viii) Trepanning:

The operation of producing a large hole (>50 mm) by removing the metal along the circumference of a hollow cutting tool is known as trepanning.

ix) Undercutting:

The operation of enlarging the hole somewhere between its ends as shown in Fig is known as undercutting.

Drill tool nomenclature.

1) A drill or twist drill is a fluted end cutting tool used for making holes in solid material. It basically consists of two parts.

1. The body consists of the cutting edges
2. The shank used for holding purposes

1. Body:

The body of the twist drill has spiral flutes cut on it. These flutes serve to provide clearance to chips produced at the cutting edge. They also allow the cutting fluid to reach cutting edges.

2. Shank:

It is a part that fits into the drill chuck or sleeve. It may be a ∥ shank or taper shank. Smaller diameter drills have a straight shank. Morse taper is commonly provided for large diameter tapered drills.

4. Point:

It is the cone shaped end of the drill. The point is shaped to produce lip, face, flank and chisel edge or dead centre.

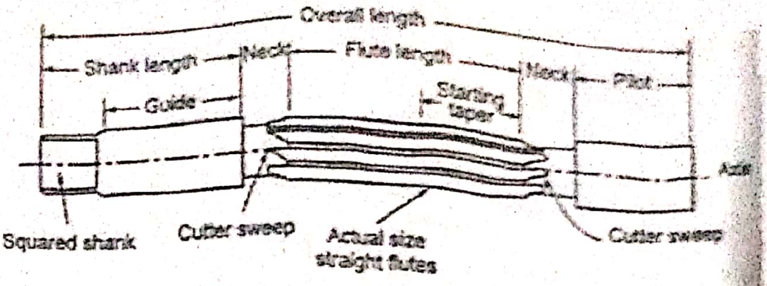


Figure 3.43 Reamer with straight flutes

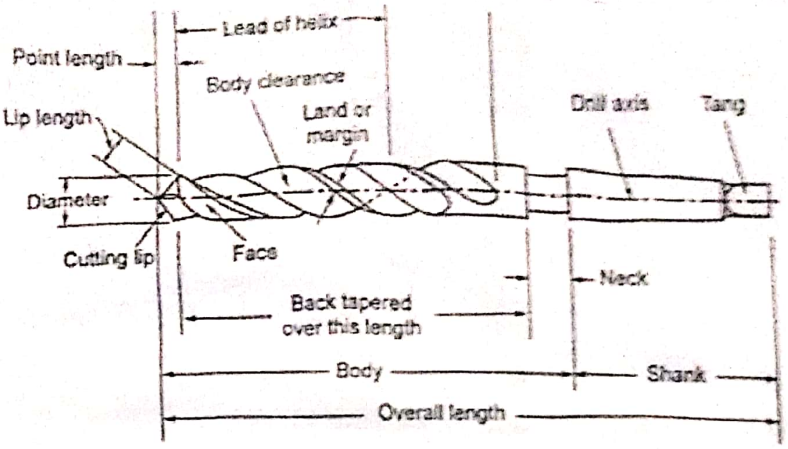


Figure 3.36 Drilling tool nomenclature

3. Neck:

It is the undercut portion between body and shank. Generally, the size and other details are marked at the neck.

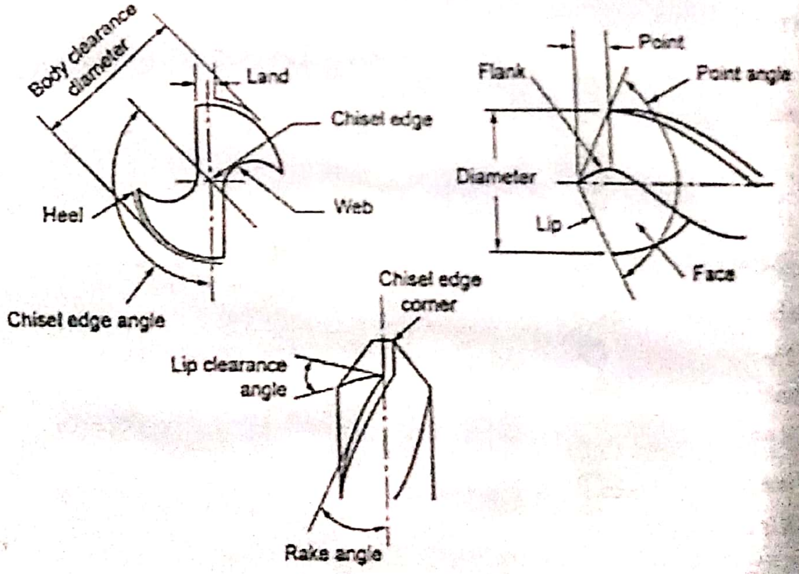


Figure 3.37 Nomenclature of twist drill

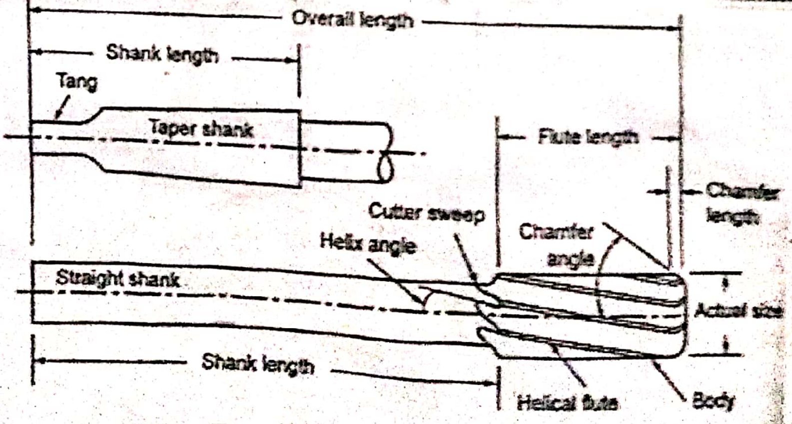


Figure 3.42 Reamer with helical flutes

5. **Land or Margin:**

It is a narrow strip. It extends back to the edges of the drill flutes. The size of the drill is measured across the lands at the point end. The land keeps the drill aligned.

6. **Web:**

It is the central portion of a drill situated between roots of flutes and extending from the point ~~to~~ towards the shank.

7. **Chisel edge:**

The intersection of flanks forms the chisel edge. It acts a flat drill. It cuts a small hole in the workpiece at the begining. Then the cutting edges remove further material to complete the hole

8. **Lip or Cutting edge:**

The cutting edges of a drill are known as lips. Both lips should have equal length, same angle of inclination and correct clearance.

9. **Flank:**

The surface behind the lip following the flute is called flank.

10. **Face:**

This is the portion of flute surface adjacent to the lip. The chip impinges on it.

11. Heel:

The edge which is formed by the intersection of the flute surface and the body clearance is known as ~~the~~ heel.

12. Point angle:

It is the angle between cutting edges. It is generally 118°. Its value depends on the hardness of the workpiece to be drilled. For harder material, larger angles are used.

13. Rake angle:

It is the angle between the face and line ll to the drill axis. At the periphery of the drill, it is equal to the helix angle. The usual values of rake angle are 30° and 45°.

14. Helix angle:

It is the angle between leading edge of the land and axis of the drill. It is also called spiral angle.

15. Lip clearance angle:

It is the angle formed by the portion of the flank adjacent to the land and plane at right angle to the drill axis measured at the ~~Pa~~ periphery of the drill.

16. Chisel edge angle:

It is the obtuse angle between chisel ~~angle~~ edge and lip. Generally this angle is 120° & 135°.

Boring Machine:

Boring is the process of enlarging previously drilled holes with a single point cutting tool as shown in fig. The boring machine is one of the most versatile machine tools.

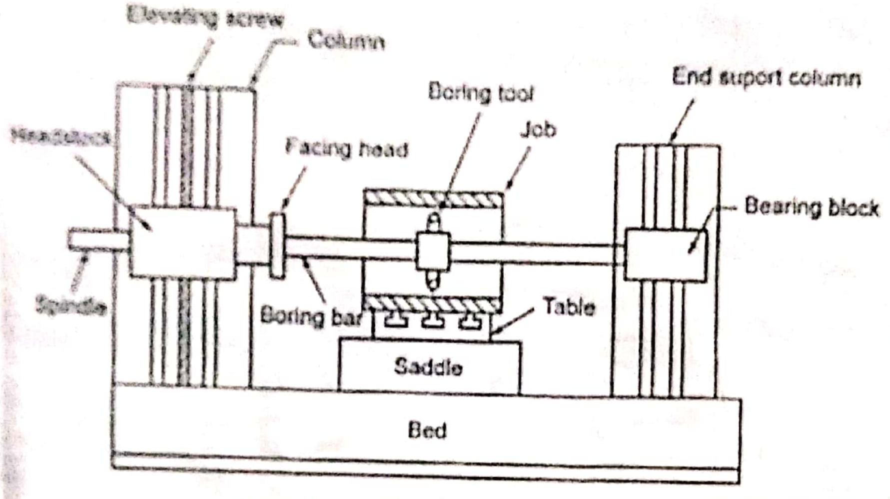


Figure 3.46 Horizontal boring machine

Boring Operations:

(a) Drilling
(b) Boring
(c) Counterboring
(d) Spot facing
(e) Internal and external thread cutting
(f) face milling
(g) facing and turning Cylindrical Surfaces
(h) Gear cutting etc.

Milling Machine:

Milling is the process of removing metal by feeding the workpiece against a rotating multipoint cutter. The metal is removed in the form of Small Chips.

Plain (or) Horizontal Milling machine

It is a horizontal column and knee type milling machine, Otherwise, Simply a horizontal milling machine.

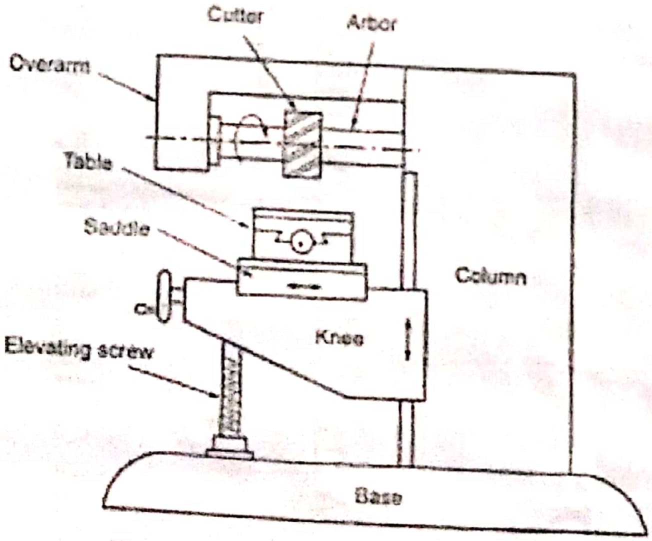


Figure 3.65 Horizontal milling machine

Base:

It is the foundation of the machine made of grey cast iron. All other parts are mounted on it. It also serves as a reservoir for cutting fluid.

Column:

It is the main support of the m/c. The motor and other driving mechanisms are housed in it. It supports and guides the knee in its vertical travel.

Knee:

The knee projects from the column and slides up and down through dovetail guides. It supports saddle and the table. An elevating screw provides its vertical movement (up and down)

Saddle:

It supports and it carries the table. It provides the traversed movement.

Over arm:

It is mounted and guided by the top of the column. It is used to hold the outer end of the arbor to prevent it from bending.

Arbor:

Arbor is an accurately machined shaft. Cutters are mounted on the arbor which is rigidly supported by the over arm, spindle and end braces.

Vertical Milling Machine

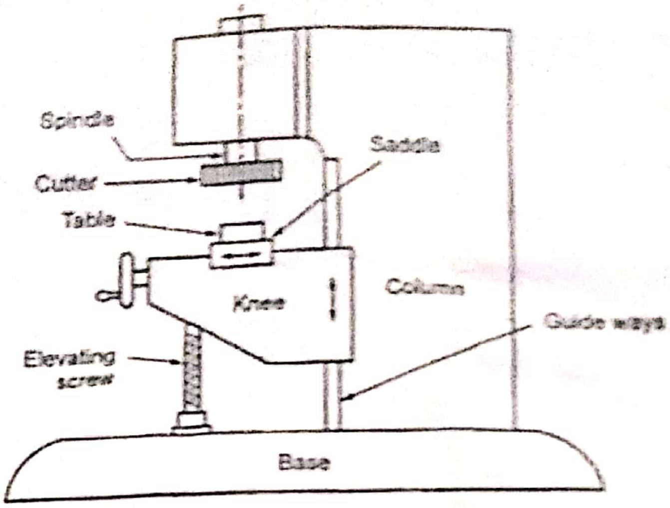


Figure 3.66 Vertical milling machine

Omniversal milling machine :

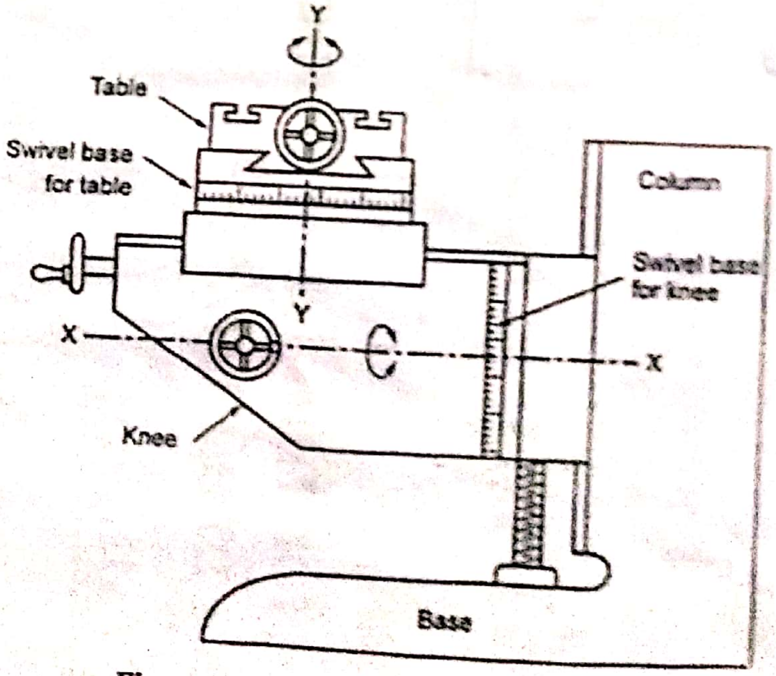
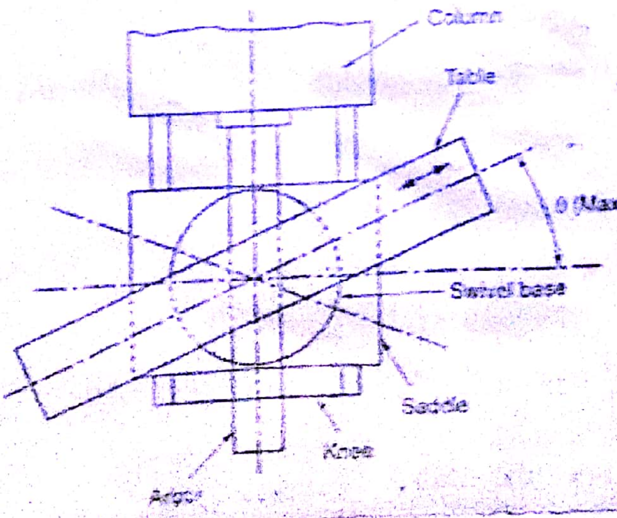


Figure 3.67 Omniversal milling machine

Universal millig machine:

In appearance, a universal milling m/c is similar to horizontal millig machine. The worktable of this machine is provided with another extra swivel movement with an index or dividing head located at the end of the table. Thus Universal milling m/c table has the following movements.

1. Vertical movement - through the knee
2. Crosswise movement - through the saddle
3. Longitudinal movement of the table.
4. Angular movement of the table by swivelling the table on the swivel base.



Nomenclature of Plain Milling Cutter

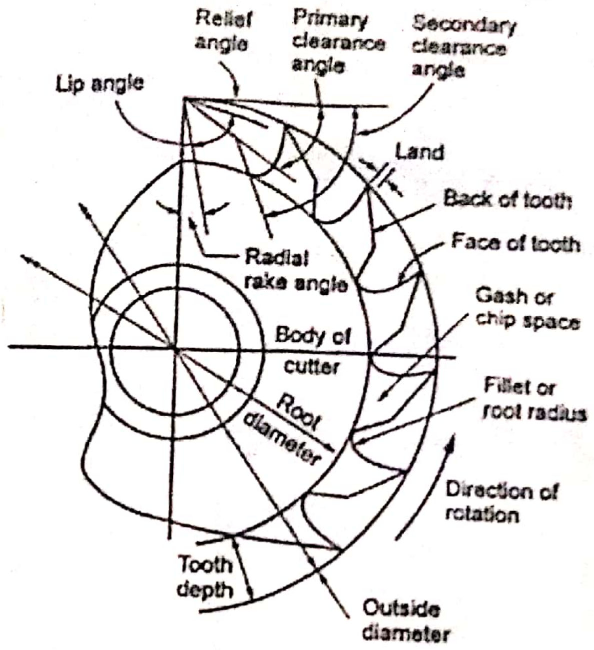


Figure 3.93 Nomenclature of a plain milling cutter

1. Body of cutter:

The main frame of the cutter on which the teeth rest to form an integral part is known as body of the cutter.

2. Cutting edge:

The edge formed by the intersection of teeth and the circular land of the surface left by the provision of primary clearance is known as cutting edge.

3. Face:

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

4. Fillet:

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

5. Gash or Chip space:

The Chip space between the back of one tooth and face of the next tooth is called gash.

6. Lead:

The cutter advances the distance in one complete revolution of turn.

7. Land:

The part of the back of the tooth adjacent to the cutting edge which is relieved to avoid interference between the surface being machined and the cutter is called Land.

8. Outside diameter:

The diameter of the circle passing through the peripheral cutting edge is called outside diameter.

9. Root diameter:

The diameter of circle passing through the bottom of the fillet is called root diameter.

10. Cutter angles:

A milling cutter is provided with a rake, clearance and other cutting angles for the efficient removal of chips. The different angles provided on cutters are now discussed.

Relief angle:

The angle between the land of the tooth and the tangent to the outside diameter of the cutting edge is known as relief angle.

Rake:

The inclination is in such a way that the keenness of the cutting edge increases. The rake angle is classified as zero, positive or negativ

Helix angle:

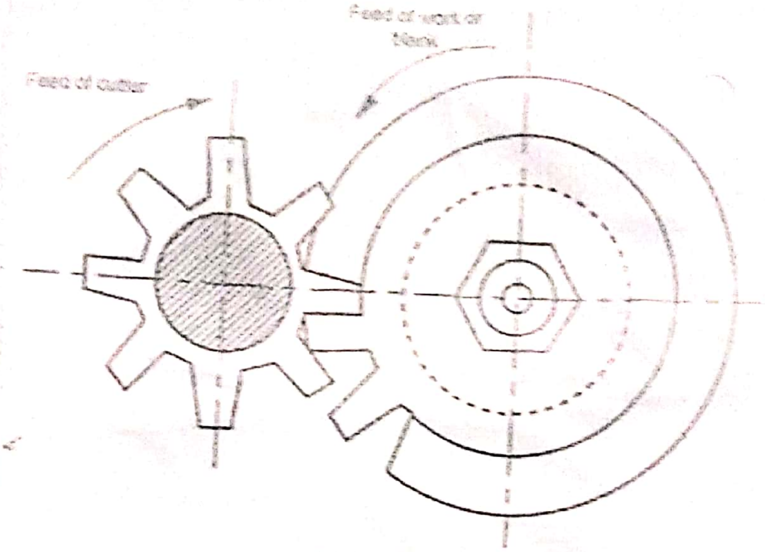
It is the inclination of a helical curve relative to its axis.

Lip angle:

It is the included angle between land and face of the tooth.

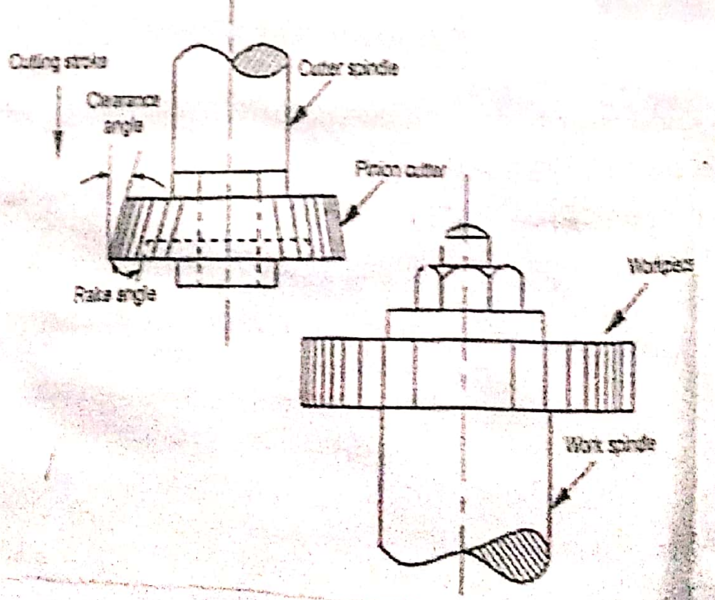
Gear Shaping:

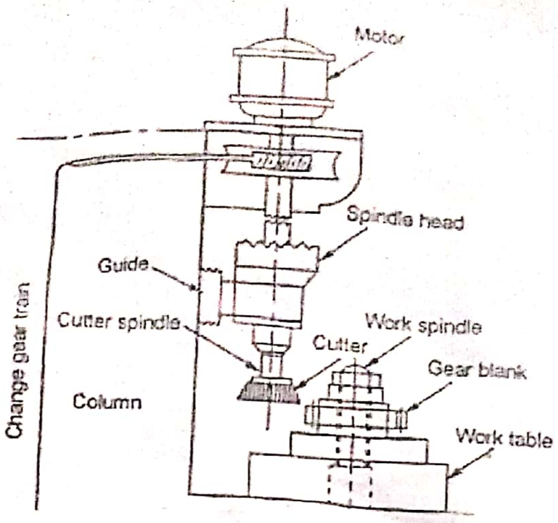
Gear Shaping is one of the generation methods used for cutting Cylindrical gears. It is done by Gear Shaper.



(b) Top view

Figure 3.115 Gear shaping process





Gear Shaping machine

Applicat

1. Gear Shaping is used for generating both internal and external Spur gears.
2. Helical gears can also be generated using special attachments.

Advantages:

1. Both internal and external gears can be generated
2. Various sizes of gears can be generated using a single cutter.
3. The mechanism is simple.

Limitations:

1. Worm gears & cluster gears cannot be produced
2. There is no cutting in the return stroke of gear cutter. So there is a need to make return stroke faster than the cutting stroke.

Gear Hobbing:

The process of generating a gear by means of a multipoint rotating cutter called hob is known as hobbing. The hob has helical threads. It looks similar to a worm gear having a number of straight flutes all around its periphery ||l to its axis as shown in fig.

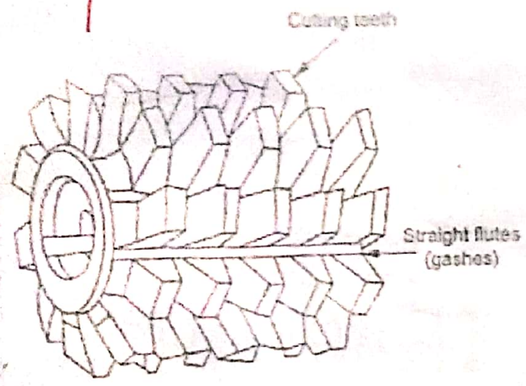


Figure 3.118 Three dimensional view of a hob

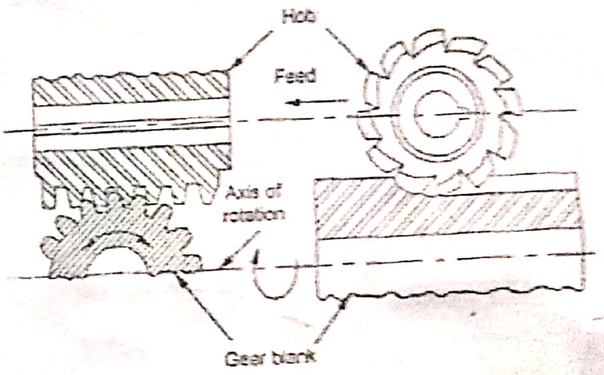


Fig. Gear Hobing process

Applications:

Hobbing is used for generating spur, helical and worm gears.

Advantages:

1. Using a single hob any number of teeth of same module can be produced.
2. Spur and helical gears can be produced using hob.

Limitations:

1. Internal gears can't be generated
2. Hobbing process can't be applied very near to shoulders

Finishing of Gears

Generally, the gear teeth are produced by anyone of the gear forming or generating process.

1. Gear Shaving

Gear shaving is a process of finishing of gear teeth by rotating at very high rpm through meshing a gear shaving tool as shown in fig. A gear shaving tool is of a type rack or pinion having hardened teeth provided with serrations.

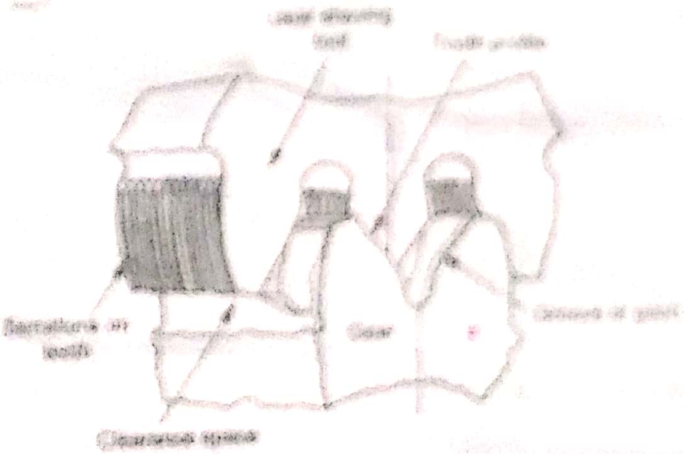


Figure 1.111 Gear shaving

Gear shaving can be used when:

* for finishing high volume of small gears
* for heat treated gears having poor accuracy in profile and lead

2. Roll finishing of Gear tooth:

Two hardened rolling dies are used to remove the rough surface in the gear to be finished. The dies have very accurate tooth profile of the gear to be finished. The gear to be finished is held between these dies.

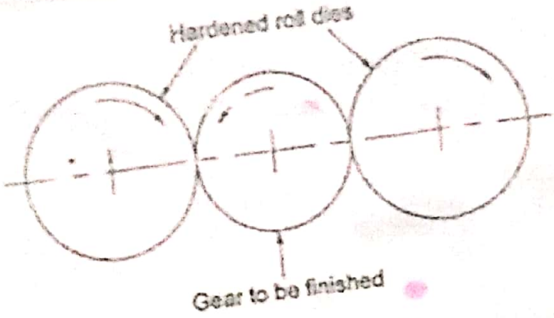


Figure 3.122 Roll finishing

3. Gear Burnishing:

Gear burnishing is another method of surface finishing for gear teeth of a gear which is done before heat treatment. It consists of rolling the work gear with the hardened rotating gears called burnishing gears whose teeth are very hard, smooth and accurate.

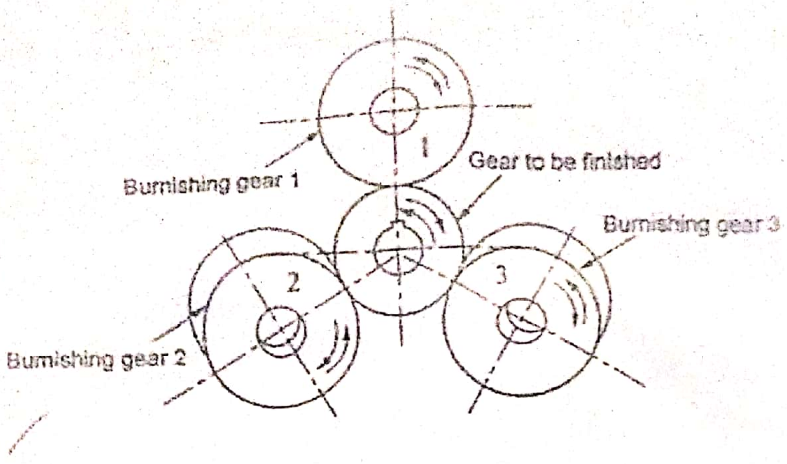
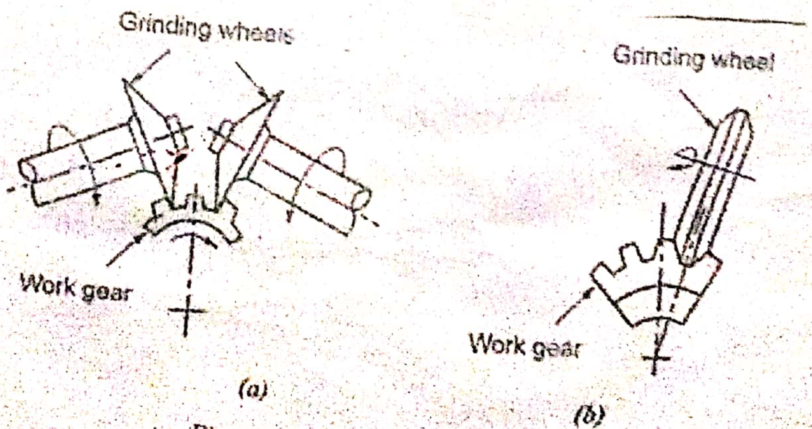


Figure 3.123Gear burnishing

4. Gear Grinding:

The abrasive grinding wheel of a required shape and geometry is used to finish the gear teeth. In addition, the abrasive grinding wheel should highly be heat treated to increase hardness.



(a) (b)

Figure 3.125 Gear teeth grinding

Machining Calculations for Shaper:

1. Cutting Speed (V):

$$V = \frac{LN(1+m)}{1000}$$

Where, L → Length of cutting stroke in mm

N → Speed in rpm

m → Ratio between cutting time & return time

2. Machining Time (T):

$$T = \frac{L}{N \times f} = \frac{L}{V} + m\frac{L}{V} = \frac{L}{V}(1+m)$$

Where, L → Length of stroke = l + approach length + over run

N → Speed in rpm

f → Feed per stroke

l → workpiece length

L/V → ~~Time~~ Time for cutting stroke

mL/V → Time for return stroke

3. Number of Strokes required (S_N):

$$S_N = \frac{W}{f}$$

Where W → width of work

f → feed per stroke

4. Total machining time (T_t):

It is the time required for machining the entire surface of the work as per requirements.

$$T_t = T \times S_N$$

Where $T \rightarrow$ Machining time

$S_N \rightarrow$ Number of Strokes required

5. Metal Removal Rate (MRR):

$$MRR = fdLS \text{ or } fdv$$

where
f - Feed
d - Depth of Cut
L - Length of work
S - Strokes per minute
v - cutting speed

6. Power required:

$$P = k \times MRR \text{ in } HP \qquad kW = HP \times 0.736$$

Where, k - Machining Constant

7. Number of Pass:

$$n = \frac{S_r}{d}$$

Where,

$S_r \rightarrow$ Stock to be removed

$d \rightarrow$ Depth of Cut

8. Percentage of time when the tool is not contacting the workpiece

$$= \frac{360 - \theta_f}{360}$$

($\theta_f \rightarrow$ forward stroke angle)

Angle of return stroke $\theta_r = 360 - \theta_f$

Scanned with CamScanner

1. A Shaper is operated at 125 cutting strokes per minute and is used to machine a workpiece of 300 mm in length and 125 mm in width. Use a feed of 0.6 mm per stroke and a depth of cut of 6 mm. Calculate the total machining time for machining the component. The forward stroke is completed in 230°. Calculate the percentage of the time when the tool is not contacting the workpiece and the ratio between the cutting time and return time.

Given

$S = 125$ strokes/min

Workpiece length, $l = 300$ mm

$w = 125$ mm

$f = 0.6$ mm/stroke

$d = 6$ mm

Forward stroke angle, $\theta_f = 230°$.

Soln:-

Let us assume the approach & over run = 25 mm

Stroke length, $L = l + 25 = 300 + 25 = 325$ mm

Number of strokes, $S_N = \frac{w}{f} = \frac{125}{0.6} = 208.33 \simeq 209$

∴ The time for completing one stroke, $T = \frac{L}{S} = \frac{325}{125} = 2.6$ min

Total machining time, $T_t = T \times S_N$

$= 2.6 \times 209$

$T_t = 543.4$ min

The % of time when the tool is not contacting the workpiece

$$= \frac{360 - \theta_f}{360}$$

$$= \frac{360 - 230}{360} \times 100$$

$$= 36.11\%$$

✓ Then, $m = \frac{\text{Angle of Cutting Stroke}}{\text{Angle of return Stroke}}$

But, the angle of return Stroke $= 360° - \theta_f$

$$= 360° - 230°$$

$$= 130°$$

∴ Ratio $m = \frac{230}{130}$

$$= 1.769$$

2

Calculate the power required for shaping the Steel with a depth of cut of 2.8 mm, Cutting speed 65 m/min and the work length 50 mm. The feed rate is 0.5 mm/rev. Take machining cost $k = 79 \times 10^{-6}$.

Given

$d = 2.8\,mm$; $V = 65\,m/min = 65000\,mm/min$

$l = 50\,mm$; $f = 0.5\,mm/rev$; $k = 79 \times 10^{-6}$

Soln: $MRR = fdV = 0.5 \times 2.8 \times 65000 = 91000\,mm^3/min$

Power required, $P = k \times MRR$

$$= 79 \times 10^{-6} \times 91000$$

$$= 7.189\,H.P$$

$$= 7.189 \times 0.736$$

$$P = 5.29\,KW$$

Drilling Calculations :

1. Cutting Speed

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

D → Diameter of the drill in mm

N → rpm of drill spindle

V → Cutting speed in m/min

2. Feed

Feed per minute $= N \times f$

N – rpm of the drill spindle

f – Feed in mm/rev

3. Depth of cut :

Depth of cut $= \frac{D}{2}$; D – Diameter of the drill

4. Machining time : (t)

$$t = \frac{\text{Length of the tool travel in mm}}{\text{Feed in mm/rev} \times \text{rpm of the spindle}}$$

where length of tool travel = Thickness of metal in mm

$L = t_p + (0.3D + \text{Over travel})$

'0' if not given

5. Power of drilling :

$$P = \frac{2\pi N T}{60} \text{ in watts}$$

N = speed of drill in rpm

$T = C \times f^{0.75} \times D^{1.8}$ in Nm

f – feed in mm/rev

D – Diameter of drill in mm

Material to be drilled	Value of C
Alluminium	0.11
Mild steel	0.56
Cast iron	0.07
Soft brass	0.084
Carbon tool Steel	0.4

6. Volume of metal removal/minute =

Area of hole × feed × Speed

$= \frac{\pi}{4} D^2 \times f \times N$

7. Energy consumption $= \frac{\text{Volume of metal removal/minute}}{\text{Power}}$

1. Calculate the feed in mm/rev to drill a hole of 30 mm in one minute to a plate thickness of 40 mm and using a spindle speed of 500 rpm

Given:

Diameter of hole $D = 30$ mm

Machining time $t = 1$ min

Plate thickness $t_p = 40$ mm

Spindle speed $N = 500$ rpm

Soln

Machining time, $t = \frac{\text{Length of the tool travel in mm}}{\text{Feed in mm/rev} \times \text{rpm of the spindle}}$

$$1 = \frac{t_p + (0.3D)}{f \times N}$$

$$1 = \frac{40 + (0.3 \times 30)}{f \times 500}$$

$$f = 0.098 \text{ mm/rev}$$

2. Calculate the spindle speed to drill a hole of 50 mm using the cutting speed as 25 m/min

Given

Diameter of hole $D = 50$ mm

Cutting speed $V = 25$ m/min

Soln Cutting speed $V = \frac{\pi D N}{1000}$

$$25 = \frac{\pi \times 50 \times N}{1000}$$

$$N = 160 \text{ rpm}$$

5. Calculate the machining time required for making 15 holes on a MS plate of 30 mm thickness with the following data

Drill diameter = 25 mm

Cutting speed = 20 m/min

Feed = 0.13 mm/rev

Given

Number of holes to be drilled = 15

Thickness of plate $t_p = 30$ mm

Drill diameter $D = 25$ mm

Cutting speed $V = 20$ m/min

feed $f = 0.13$ mm/rev

Soln. Cutting speed $V = \frac{\pi D N}{1000}$

$$20 = \frac{\pi \times 25 \times N}{1000}$$

$$N = 260 \text{ rpm}$$

m/cing time $t = \frac{\text{Length of tool travel in mm}}{\text{feed in mm/rev} \times \text{rpm of spindle}}$

$$= \frac{t_p + 0.3D}{f \times N}$$

$$= \frac{30 + (0.3 \times 25)}{0.13 \times 260}$$

$$t = 1.109 \text{ min/hole}$$

Total m/cing time $= 1.109 \times 15$

$= 16.642$ min

4. A 40 mm HSS drill is used to drill a hole in a cast iron block of 80 mm thick. Determine the time required to drill the hole if the feed is 0.2 mm/rev. Assume an over travel of drill as 5 mm. The cutting speed is 22 m/min.

Given:

Drill Dia = 40 mm; Thickness of CI block = 80 mm

Feed $f = 0.2$ mm/rev ; Over travel $S = 5$ mm

Cutting speed $V = 22$ m/min.

Soln:

Cutting speed, $V = \frac{\pi D N}{1000}$

$$22 = \frac{\pi \times 40 \times N}{1000}$$

$$N = 175 \text{ rpm}$$

Length of travel of drill $= t_p + 0.3D +$ over travel

$$= 80 + (0.3 \times 40) + 5$$

$$= 97 \text{ mm}$$

Machining time $t = \frac{L}{fN} = \frac{97}{0.2 \times 175} = 2.77$ min

5. Calculate the power required to drill 25 mm diameter hole in aluminium plate at a feed of 0.2 mm/rev and at a drill speed 400 rpm. Also determine the volume of metal removed / unit minute.

Given $D = 25$ mm; Material = Aluminium;
$f = 0.2$ mm/rev, $N = 400$ rpm

Soln

Torque, $T = C \times f^{0.75} \times D^{1.8}$

from table for aluminium $C = 0.11$

$$T = 0.11 \times (0.2)^{0.75} \times 25^{1.8} = 10.8 \text{ Nm}$$

$$P = \frac{2\pi N T}{60} \text{ W} ; \quad T \text{ in Nm}$$

$$P = \frac{2\pi \times 400 \times 10.8}{60} = 452.4 \text{ W}$$

Volume of metal removal /minute = Area of hole × feed × Speed

$$= \frac{\pi}{4} D^2 \times f \times N$$

$$= \frac{\pi}{4} (25)^2 \times 0.2 \times 400$$

$$= 39269.91 \text{ mm}^3$$

$$\text{Energy consumption} = \frac{39269.91}{452.4}$$

$$= 86.8 \text{ mm}^3/\text{watt minute}$$

Problem on Differential Indexing

Calculate the gear ratio, the indexing movement and the number of idler gears used for milling 119 teeth spur gear on a gear blank

Soln

There is no 119 hole circle is an index plate, the divisions can't be indexed by simple or plain indexing. So, differential indexing is selected

Assume $A = 120$

Note:

1. $\text{Gear ratio} = \frac{\text{Driving gear}}{\text{Driven gear}} = \frac{\text{Gear on spindle head}}{\text{Gear on bevel gear shaft}}$

$$= (A-N) \times \frac{40}{A}$$

Where, A — Assumed number of divisions that can be indexed by a plain or simple indexing

N — Required number of divisions to be indexed on the workpiece

2. Index crank movement $= \frac{40}{A}$

If $(A-N)$ is +ve, the index plate must be rotated in same direction

$(A-N)$ is −ve, the index plate must be rotated in opposite direction

To achieve these conditions,

a) If the gear train is simple and $(A-N)$ is +ve, only one idler gear is used

b) If the gear train is compound " " +ve, no idler gear is used

c) If the gear train is simple " " −ve, two idler gears used

d If the gear train is compound " " −ve, only one idler gear is used

Assume $A = 120$

a) Gear ratio $= \dfrac{\text{Gear on spindle head}}{\text{Gear on bevel gear shaft}}$

$$= (A-N) \times \frac{40}{A}$$

$$= (120-119) \times \frac{40}{A}$$

$$= 1 \times \frac{40}{120}$$

$$= \frac{1}{3} = \frac{1 \times 24}{3 \times 24} = \frac{24}{72}$$

(24 is the front side hole)

A simple gear train is used.

Gear on spindle will have 24 teeth.

Gear on bevel gear shaft will have 72 teeth

b) <u>Index crank movement</u>:

$$= \frac{40}{A} = \frac{40}{120} = \frac{1}{3}$$

$$= \frac{1 \times 8}{3 \times 8} = \frac{8}{24}$$

The index crank will have to be moved by 8 holes on 24 hole circle of each cut for 119 times.

c) <u>Number of Idlers</u>:

As $(A-N)$ is +ve, a simple gear train with one idler is used. The index plate will rotate in the same direction of the crank movement.

(6)

UNIT-IV

ABRASIVE PROCESS AND BROACHING

Abrasive Process:

Grinding is one of the abrasive processes.

Grinding is a metal removing process in which the metal is removed with the help of rotating grinding Wheel. Such wheels of made of fine grains of abrasive materials held together by a bonding material Called a bond.

Grinding Wheel:

Grinding wheels are madeup of small abrasive particles held together by bonding materials. Thus it forms a multi-edge cutter.

Grinding wheel abrasives:

Abrasive is a hard material. It is used to cut or wear away other materials. Small sizes of abrasives particles are used in grinding wheels. They are called abrasive grains. Abrasives may be classified in to the following two types.

i) Natural abrasives

ii) Artificial abrasives

i) Natural abrasives:

These are produced by uncontrolled forces of nature. These are obtained from mines. The following are the natural abrasives.

a) Sandstone or Solid quartz
b) Emery (50 to 60% crystalline Al_2O_3 + Iron oxide)
c) Corundum (75 to 90% crystalline Al_2O_3 + Iron Oxide)
d) Diamond

ii) Artificial abrasives:

These abrasives have better cutting properties and higher efficiency than natural abrasives.

The various manufactured abrasives are:

(a) Aluminium Oxide
(b) Silicon Carbide
(c) Artificial diamond
(d) Boron carbide
(e) Cubic boron nitride

Types of Bonds:

Bond is an adhesive substance which holds the abrasive grains together to form the grinding wheel. The bonds must sufficiently be strong to withstand the stresses of high speed rotating grinding wheel.

Bonds are classified in to the following two types.

Non organic bonds:

Metallic, Vitrified and Silicate bonds are non-organic.

Organic bonds:

Resinoid, rubber, shellac and oxychloride bonds are organic bonds.

Different types of bonds used in grinding are represented by different symbol as shown below.

Vitrified bond - V
Silicate bond - S
Resinoid bond - B
Rubber bond - R
Shellac bond - E
Oxychloride bond - O

Specification of Grinding wheel:

It refers actual size of abrasive particles.

1. Grit number and Grain Size:

Grain Size is denoted by the grit number. Grit number is equal to the number of meshes in 2.54 cm of a sieve through which the grains can pass through. Larger is the grit number, smaller will be the grain size and vice versa.

For rough grinding smaller grit number is used for finish grinding, large grit number are used

2. Grade:

Grade or hardness indicates the strength with which the bonding material holds the abrasive grains in the grinding wheel. It does not refer the hardness of abrasive grains.

The degrees of hardness are specified by the use of letters of the alphabet. 'A' indicates the softest grade whereas 'Z' indicates the ~~φ~~ hardest grade.

Soft → A to H
Medium → I to P
Hard → Q to Z

3. Structure of Wheels:

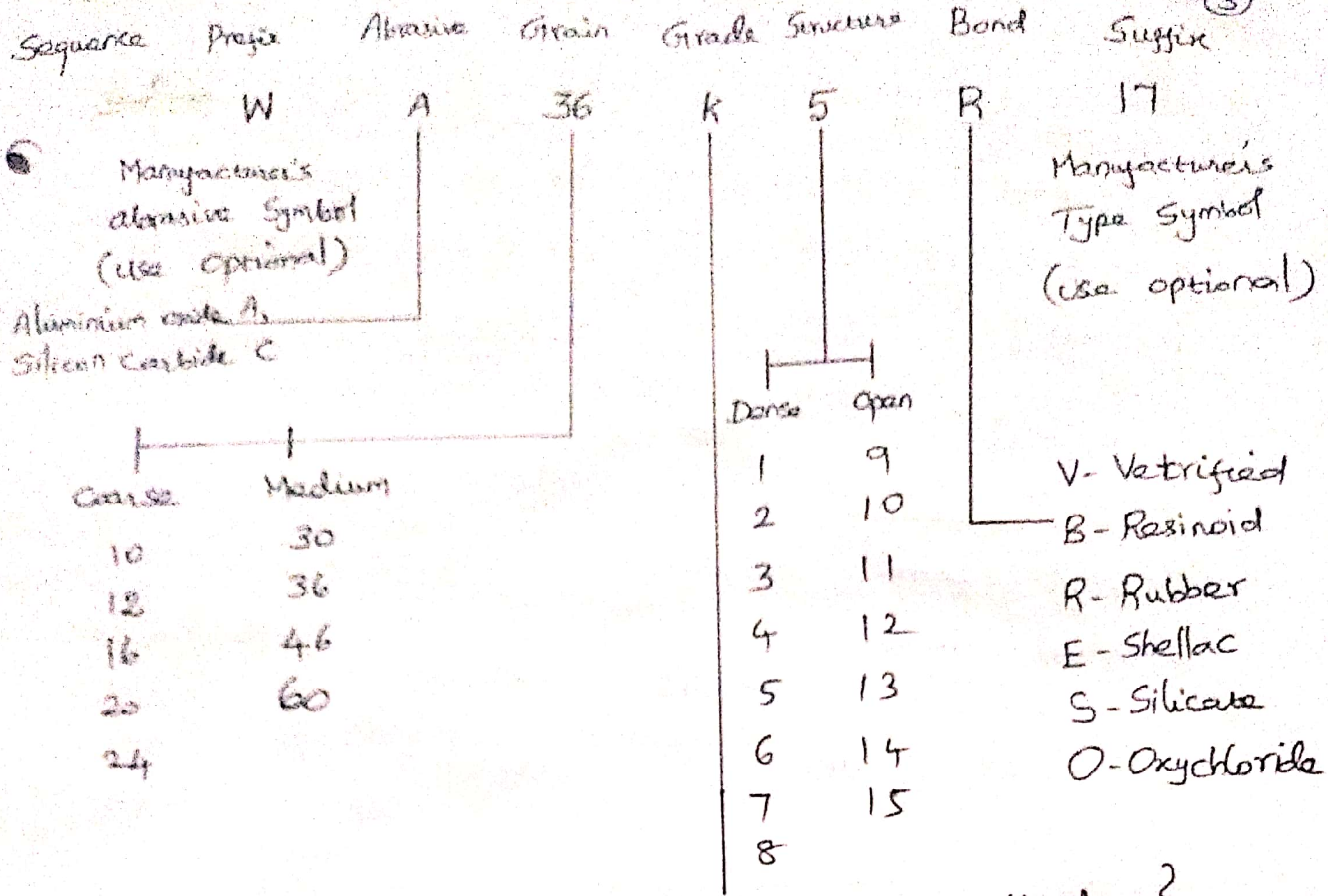
This term denotes the spacing between abrasive grains or in other words the density of the wheel. The structure of grinding wheel is designated by a number. Higher is the number, wider will be spacing. When the spacing is small, the structure is called dense structure. When the spacing is wide, the structure is called open structure.

Dense → 1 to 8
Open → 9 to 15 or more

Designation of Grinding Wheel:

An Indian Standard Institution (IS) 551-1954) has specified a standard system of marking the grinding wheels. According to this system, the following elements are represented in a definite order.

1. Type of abrasives.
2. Grain size or grit number
3. Grade of wheel
4. Structure
5. Type of bond
6. Manufacturer's code.



Grade { Soft ABCDEFGH   Medium IJKLMNOP   Hard QRSTUVWXYZ }

Selection of Grinding Wheel

The selection of proper grinding wheel is very important for getting the best results in grinding work.

It is necessary that the proper grain size, bond grade, Strength, Shape & Size of the wheel Should be selected to meet the Specific requirements.

The following are the factors upon which the above selection depends on

i) Constant factors
ii) Variable factors.

Constant Factors

1. Physical properties of material to be ground
2. Amount and rate of stock to be removed
3. Area of contact
4. Type of grinding machine

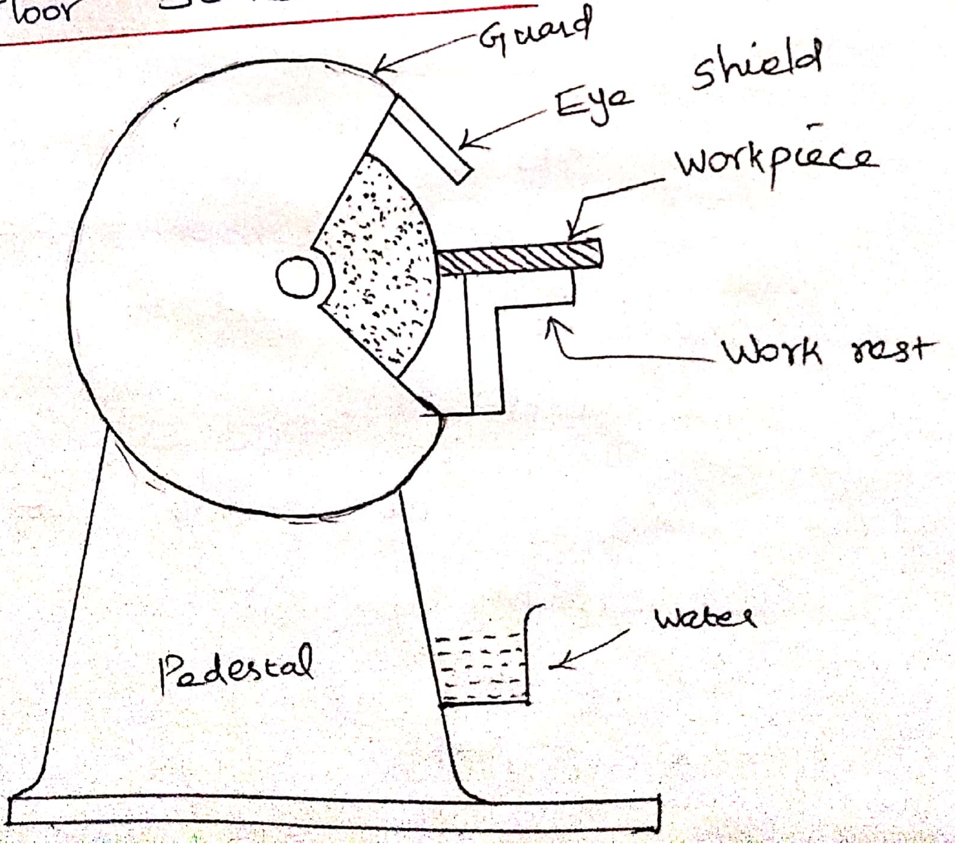
Variable Factors:

1. Work Speed
2. Wheel speed
3. Condition of the grinding machine
4. Personal factor

Rough Grinders:

Rough grinders are mainly used for removing large amount of metal from the workpiece. Therefore the surface finish and the accuracy in dimension are not high.

1. Floor Stand Grinder:



Precision Grinders:

Precision grinders are used to manufacture parts of accurate dimensions and good surface finish

Cylindrical Grinders:

There are four movements in a cylindrical centre type grinding.

i) Rotation of cylindrical workpiece about its axis
ii) Rotation of grinding wheel about its axis
iii) Longitudinal feed movement of the work past the wheel face.
iv) Movement of grinding wheel into the work ⊥r to the axis of the workpiece to give depth of cut

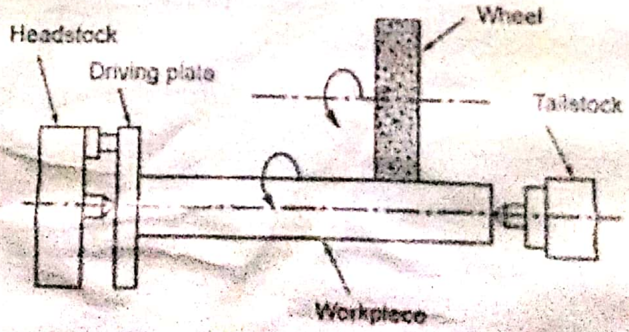


Fig. Longitudinal grinding

Two types of operations performed in cylindrical grinding.

i) Traverse grinding
ii) Plunge grinding

Traverse Grinding:

This method is used when the job length is more than the width of the grinding wheel. The job is held between two centres. The grinding wheel is made to rotate in a fixed position. The rotating work is made to traverse. The rotating work longitudinally moves in both directions. It is the longitudinal feed.

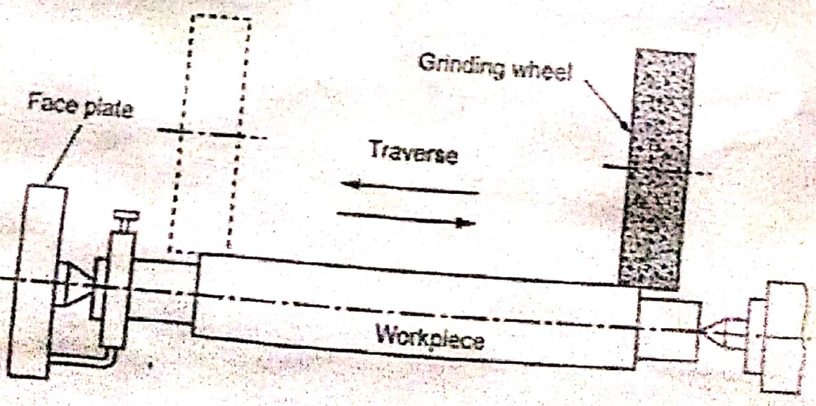


Figure 4.15 Traverse grinding

Plunge Grinding:

This method is used when the length of the workpiece is lesser than the width of the grinding wheel. Here, the workpiece need not longitudinally be fed. The grinding is done by giving only the cross feed to the grinding wheel. It is known as plunge grinding.

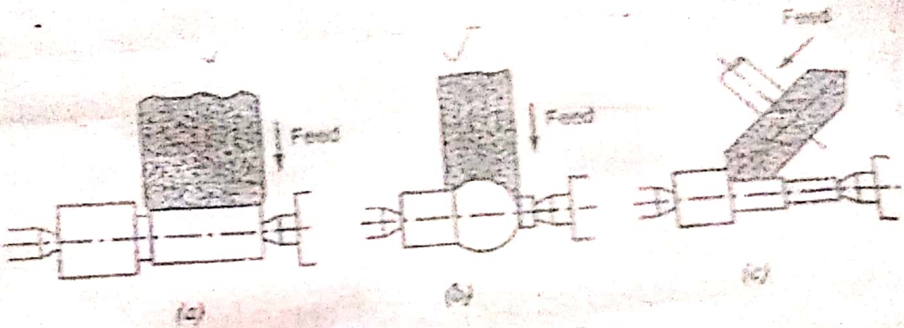


Figure 4.16 Plunge grinding

Types of Cylindrical Grinding:

i) Plain centre type cylindrical grinding machine:

The grinding machine consists of various parts.

1. Base
2. Table
3. Headstock
4. Tailstock
5. Wheelhead

~~6~~

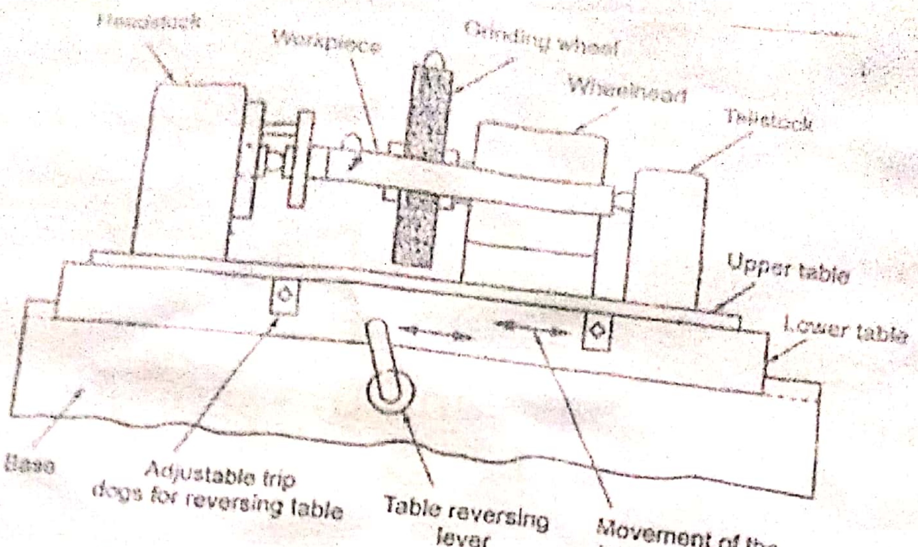


Figure 4.17 Plain centre type cylindrical grinding machine

ii) Centre type Universal grinding:

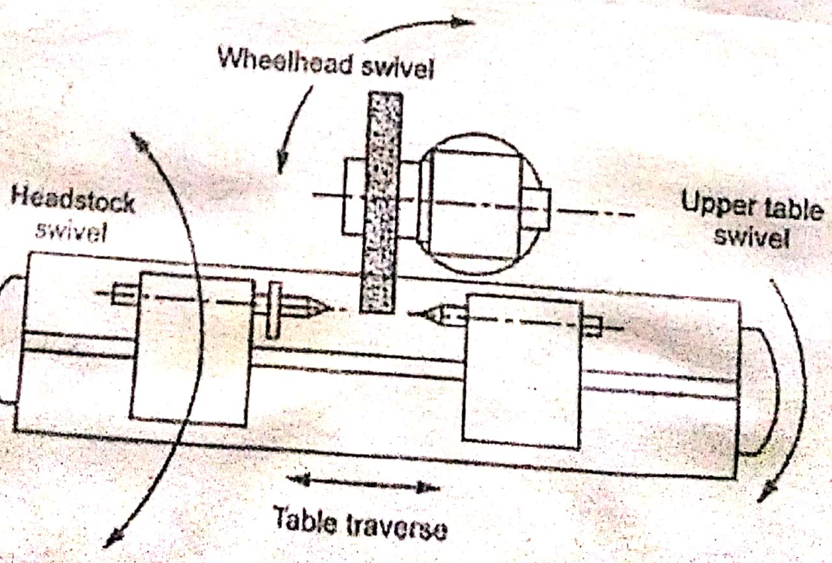


Figure 4.18 Universal grinder

The features of this machine are similar to those of plain grinders but in addition, it is provided with the following features.

1. The centre of the headstock spindle can be used as a live centre or dead centre. The work can be held and revolved by a chuck. It can also be held between centres and revolved.
2. The wheel head can be swivelled in a horizontal plane in any angle. The wheelhead can also be fed in the inclined direction.
3. The headstock can be swivelled to any angle in the horizontal plane.
4. The wheelhead can also be arranged for internal grinding.

Surface Grinders:

1. Horizontal Spindle Reciprocating Table Surface Grinder

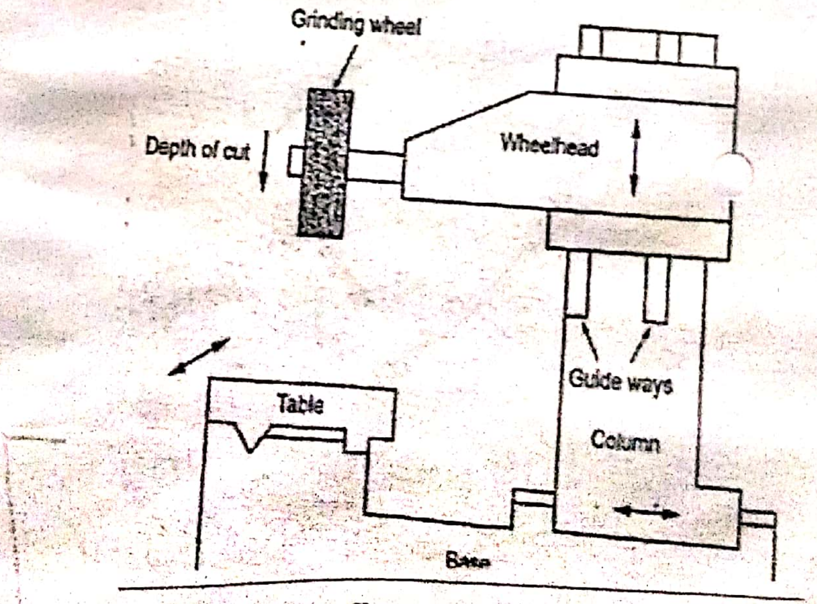


Fig. Horizontal spindle reciprocating table suface grinder.

2 Horizontal Spindle Rotary Table Surface Grinder

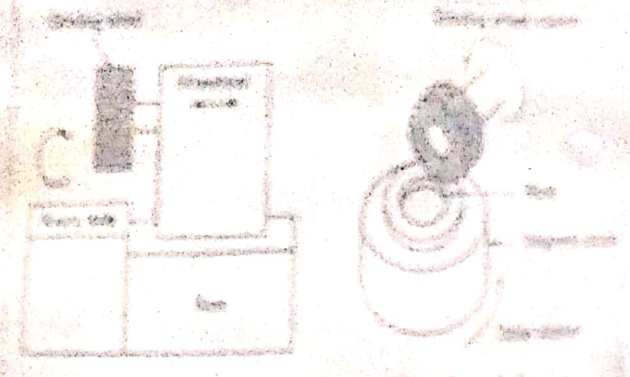


Figure 4.20 Horizontal spindle rotary table surface grinder

3 Vertical Spindle Reciprocating Table Surface Grinder

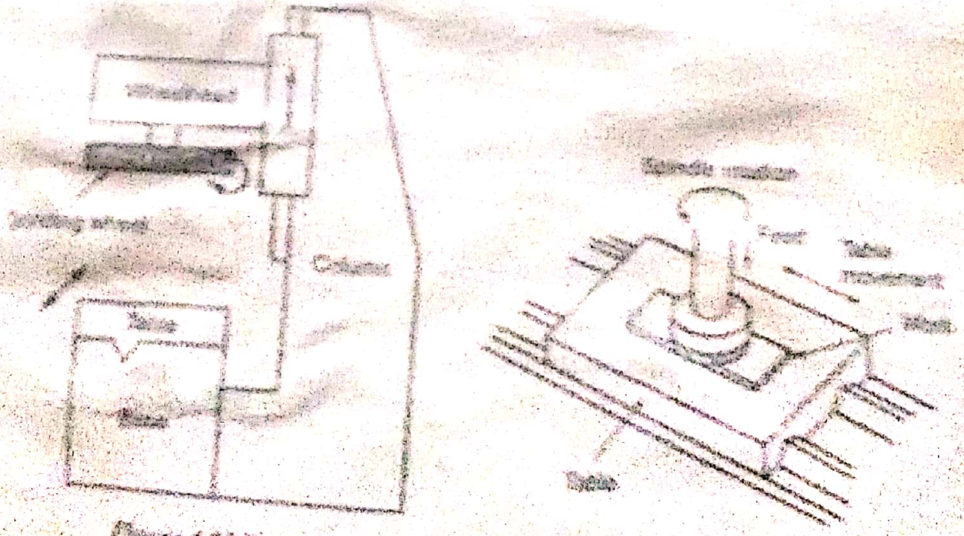


Figure 4.21 Vertical spindle reciprocating table surface grinder

4. Vertical Spindle Rotary Table Surface Grinder:

3.4. Vertical Spindle Rotary Table Surface Grinder

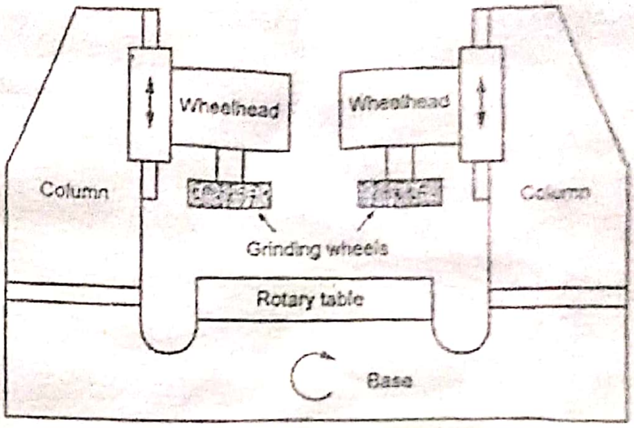


Figure 4.22 Vertical spindle rotary table surface grinder

Centreless Grinders:

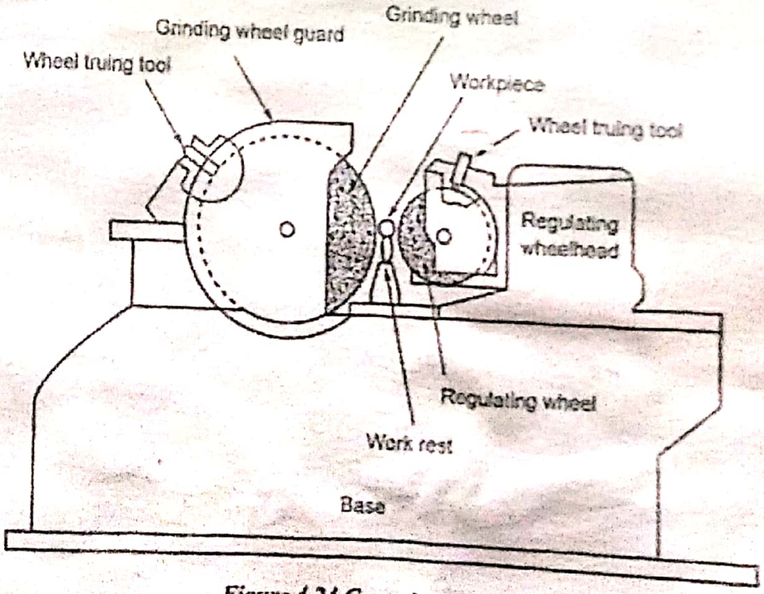
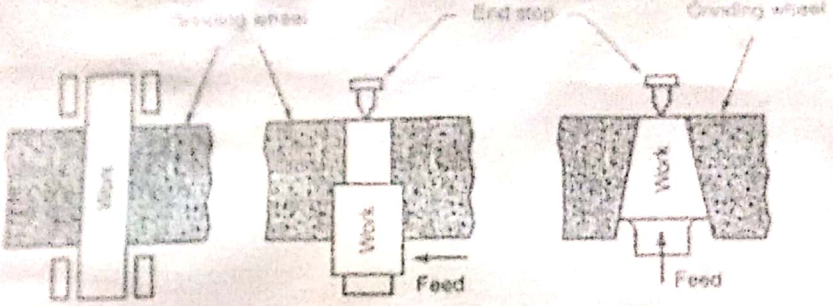


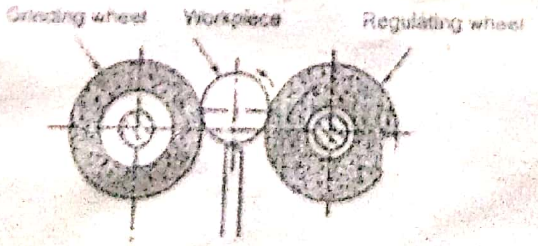
Figure 4.24 Centreless grinders

Centerless grinding is performed on cylindrical workpieces such as pistons, valves, rings, tubes, balls, wrist pins, drills, bushings, shapes etc.. Centreless grinding can be done on both external and internal cylindrical surfaces.

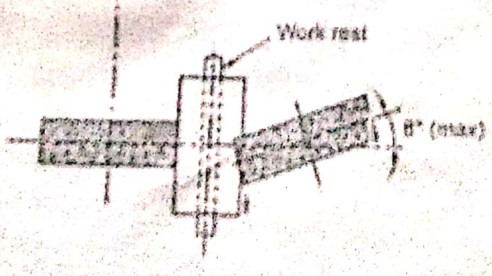


(a) Through-feed (b) In-feed (c) End-feed grinding

Figure 4.25 Methods of centreless grinding



(a) Front view



(b) Top view

Figure 4.23 Principle of centreless grinding

Internal Grinders:

Internal grinders are used to finish straight, tapered or formed holes to the correct size, shape and finish. There are three types of Internal grinders.

(a) Chucking type
(b) Planetary type
(c) Centreless type

Chucking Type Internal Grinders:

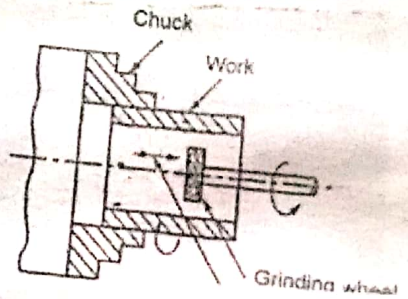


Fig. Chucking type internal grinders

Planetary Type Internal Grinders:

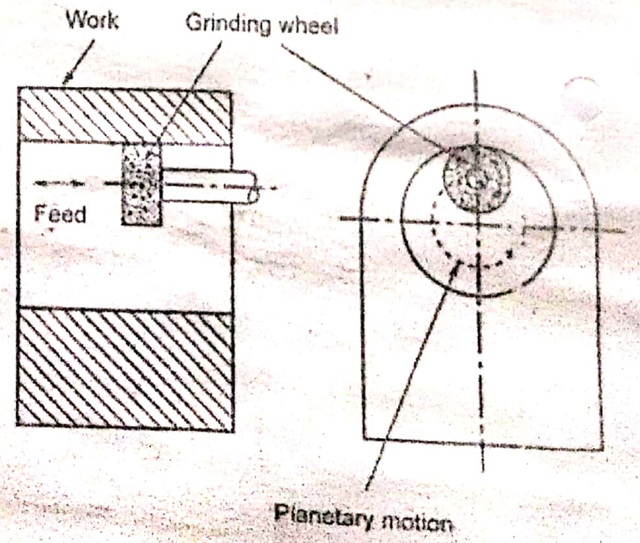


Figure 4.27 Planetary type internal grinders

3 Centreless Internal Grinding:

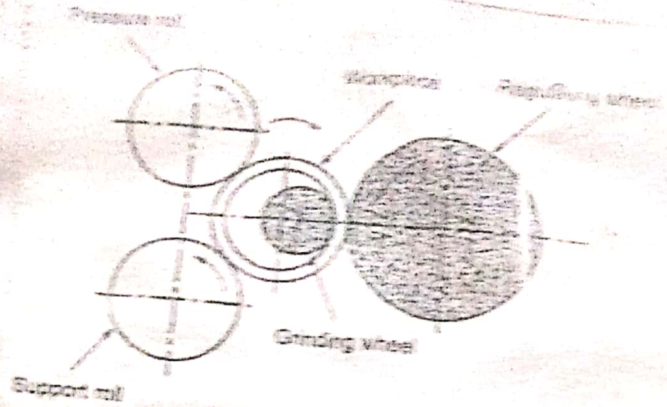
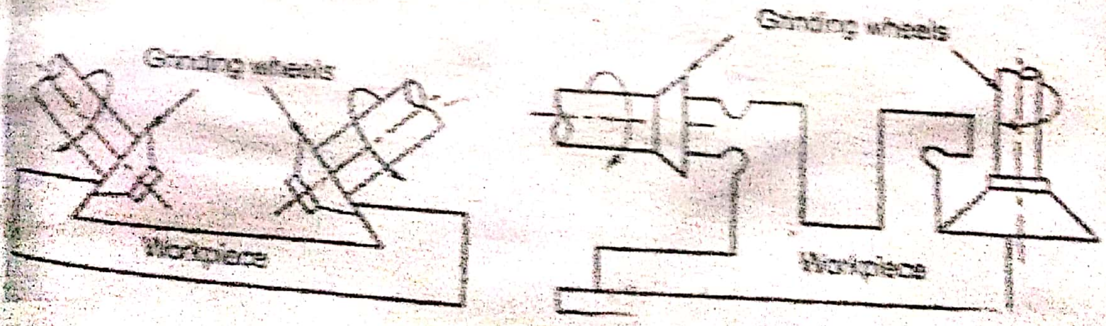


Figure 4.21 Centreless internal grinding

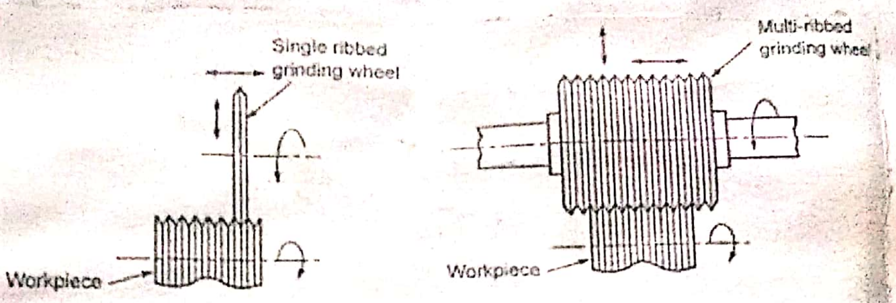
Typical Applications of Grinding Machines:

1. Gear Teeth Grinding (Refer unit - 3)
2. Form Grinders:

4.15.2. Form Grinders



3. Thread Grinding:



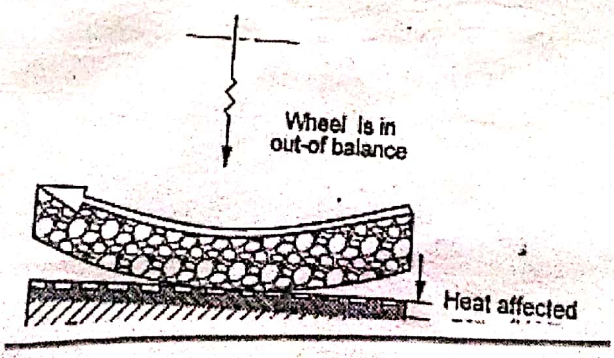
(a) Single ribbed thread grinding

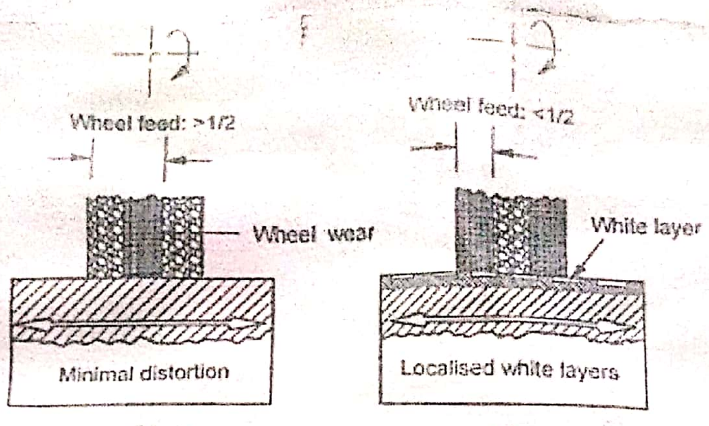
(b) Multi-ribbed thread grinding

Figure 4.30 Thread grinding

Concepts of Surface Integrity:

Surface Integrity is the Surface condition of a workpiece after completing a particular manufacturing process. In grinding process, the Surface Integrity of ground Specimen is analysed.





(b) (c)

Figure 4.33 Effect on machined surface integrity during surface grinding

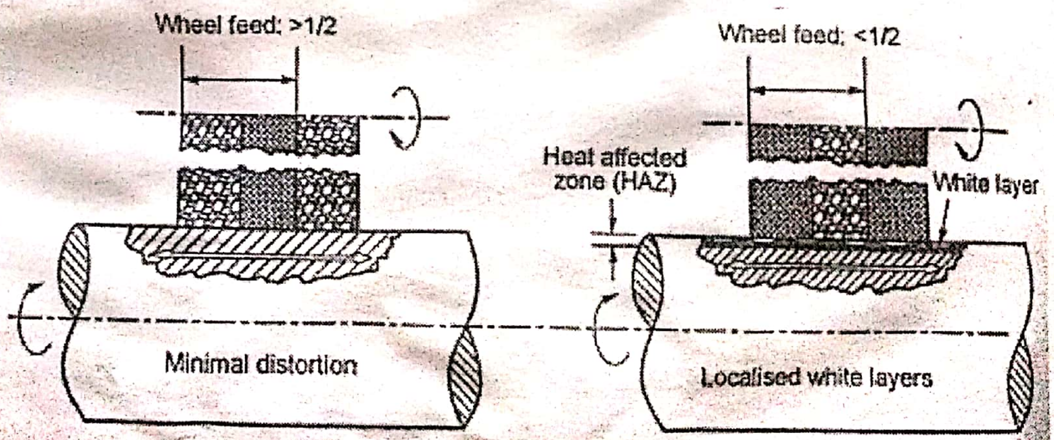
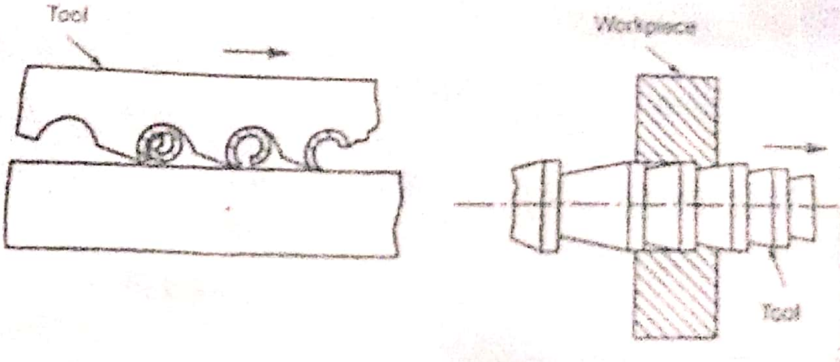


Figure 4.34 Effect on machined surface integrity during cylindrical grinding

UNIT - IV / PART - II

BROACHING

Broaching is a process of machining a Surface with a Special multipoint cutting tool called broach which has Successively higher cutting edges is a fixed path.



(a) External broaching (b) Internal broaching

Figure 4.35 Broaching principle

aching process

Horizontal Type Surface Broaching Machine : (internal)

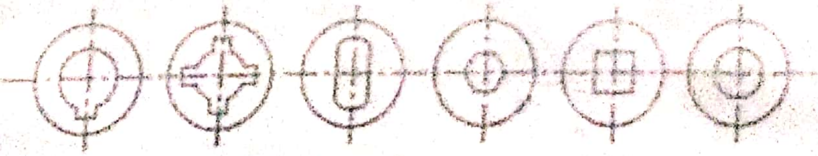


Figure 4.36 Typical internal broaching operations

Fig. shows the horizontal type surface broaching machine. Here the broach is pulled over the top surface of the workpiece hold in the fixture on the worktable.

Vertical Broaching Machines:

1. Push down Type Vertical Broaching Machine:

The push type vertical broaching machine is used in the surface broaching operation. It consists of a box shaped column, Slide and drive mechanism. Fig. shows the vertical push down type surface broaching machine.

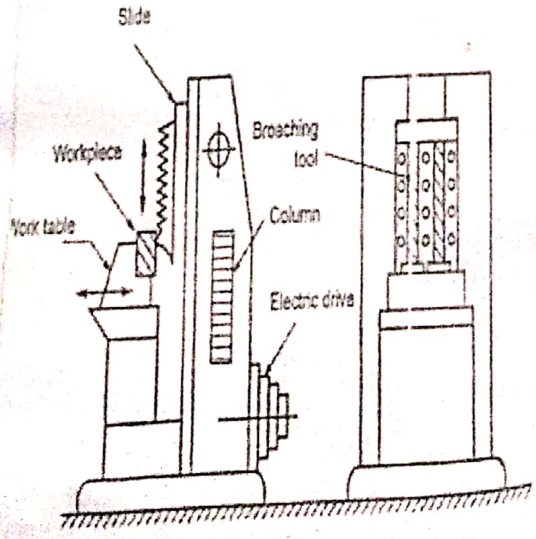


Fig. Push down type vertical broaching machine.

Pull Down type Vertical Broaching Machine:

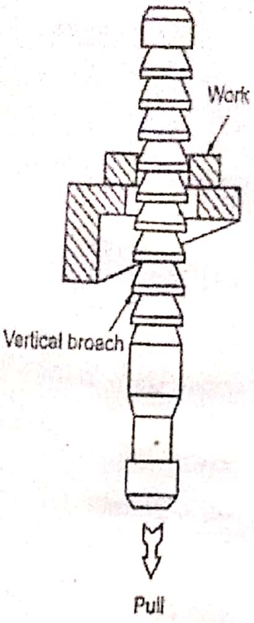


Figure 4.40 Pull down type

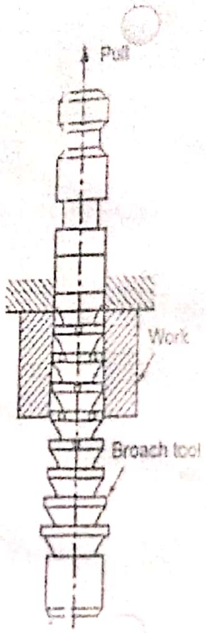


Figure 4.41 Pull up type

Continuous Broaching machines:

There are three types of continuous broachig machines as follows.

i) Horizontal

ii) Vertical

iii) Rotary type.

Horizontal Type Continuous Broaching Machine:

It is one type of surface broaching machine. The broaching machine has a driving unit which consists of two sprockets. They are connected by an endless chain as shown in fig. Fixtures are mounted at intervals on the chain for locating and holding workpieces.

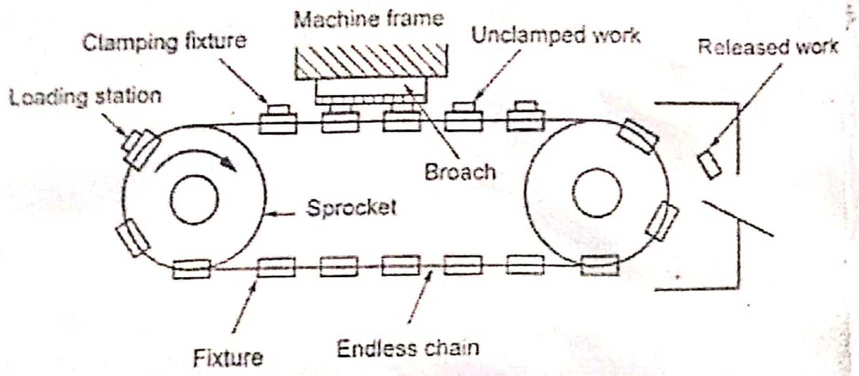


Figure 4.42 Horizontal type continuous broaching machine

Vertical Continuous Broaching Machine:

When the axes of two sprockets are vertical, it is called vertical broaching machine. The fixtures are mounted on the chain according to its movement. The operating principle is similar to a horizontal type continuous broaching machine. Here the broach is vertically placed on the frame of the machine.

3. Rotary type Continuous Broaching Machine

They are used for squaring distributor Shaft, slotting and facing the Small parts

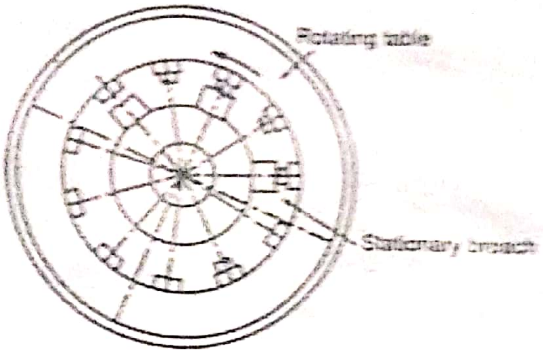
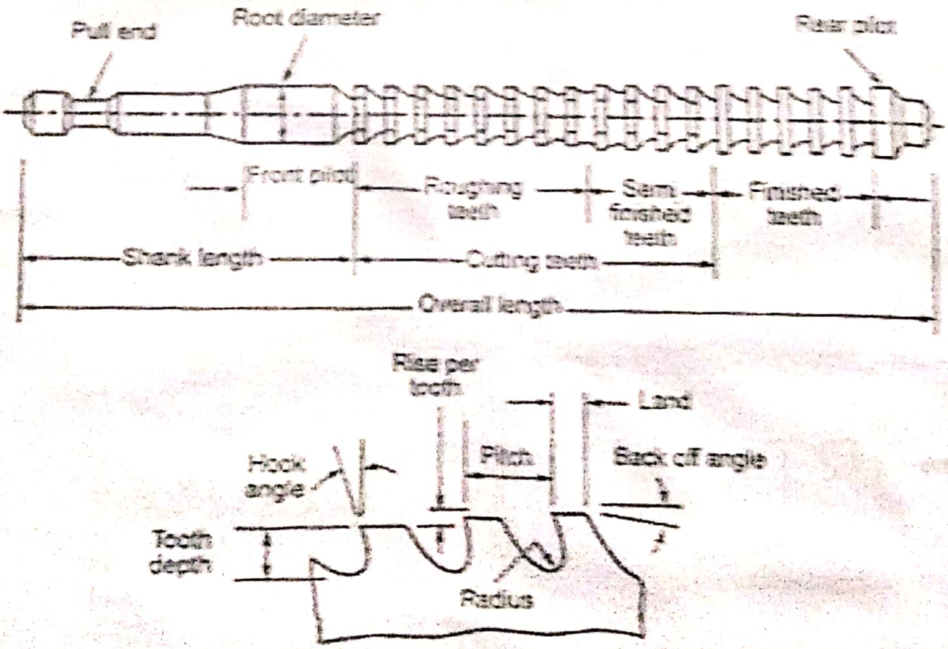


Figure 4.43 Rotary table continuous broaching machine

Broach Nomenclature:



Hook angle or rake angle or face angle 12° to 15°
back off or clearance angle 1° to 4°

Figure 4.44 Broach tool nomenclature

i) Roughing teeth:

These teeth have the highest rise per tooth and remove the bulk material.

ii) Semi-finishing teeth:

These teeth have slightly smaller rise per tooth than the previous one: Hence they remove a relatively smaller amount of material when compared to the roughing teeth.

iii) Finishing teeth:

The last set of teeth is called finishing or sizing teeth. Less amount of material is removed by these teeth. The necessary size is achieved by these teeth and hence, all teeth are of the same size which is finally required.

UNIT - V

CNC MACHINING

Numerical Control System:

NC machine tools are the machine tools operated by programmed commands in contrast to the manual control through the hand wheels or levers, or mechanically automated through Cams alone.

In NC machine tools, one or more of the following functions may be automatic.

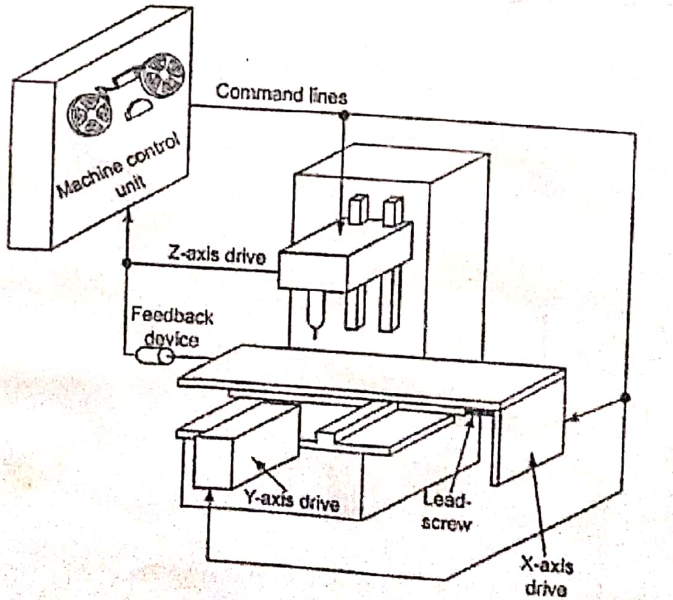
i) Starting and Stopping of machine tool spindle

ii) Controlling the Spindle Speed

iii) Positioning the tool tip at desired locations and guiding it along desired paths.

iv) Controlling the rate of movement of tool tip (feed rate)

v) Changing of tools in the spindle.



(a) NC Machine tool

Co-ordinates of NC Machine Tool:

As per manufacturing concepts is obtaining the degrees of freedom for the linear and transverse movements, each free body has six degrees of freedom. Among these degrees of freedom, three positive or negative translations are along X, Y & Z axis and three rotations clockwise or counter clockwise about these axes as shown in fig.

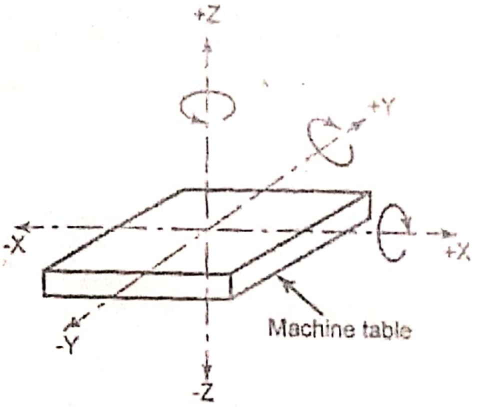


Figure 5.2 Coordinates of NC system

Absolute positioning and Incremental Positioning:

There are two ways of positioning the tool in relative to the workpiece as follows:

1. Absolute Positioning
2. Incremental positioning

Absolute Positioning:

It means that the tool locations are always defined in relation to zero point.

Incremental Positioning:

It means that the next tool location must be defined with reference to the previous tool locations.

Elements of NC system:

1. Program
2. Machine Control unit
3. Machine Tool

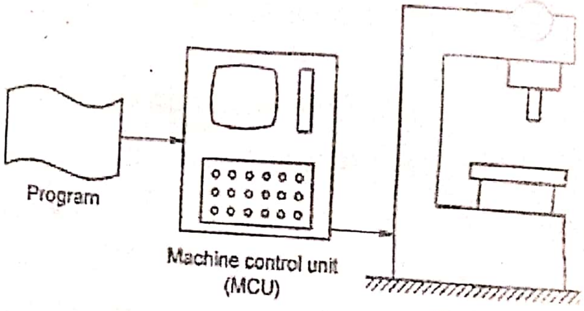
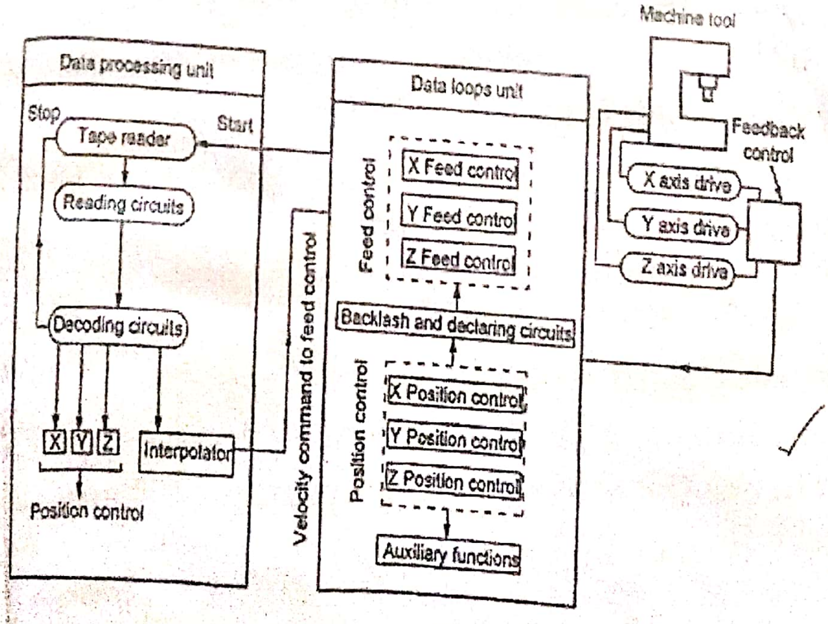


Figure 5.5 Basic components of an NC system



(b) Configuration of NC system

Figure 5.1 Numerical control machine tool system

Type of NC Systems based on the type of Machine Control:

NC systems are classified on the basis of types of machine control as follows.

(a) Traditional numerical Control (NC)
(b) Computer Numerical control (CNC)
(c) Distributed numerical Control (DNC)

(a) Traditional Numerical control (NC)

It has hardwired control where the control is proficient through the use of punched papers or plastic, tapes or cards. Tapes tend to wear and become dirty. It leads to misreading, some other limitations also noticed with the use of traditional NC tapes.

NC tapes should manually be reloaded for each new part. It lacks the program editing abilities which consumes more time.

(b) Computer Numerical Control:

CNC refers to a system which is locally linked with a computer to store all necessary numerical data. The main advantage of CNC systems is the flexibility allowed to edit the programs according to the need for executing cycles of machining commands.

(c) Distributed Numerical Control:

DNC system is almost similar to CNC except an isolated computer used to control a number of machines. A central computer is used to feed the data to local CNC computers.

Classification NC Machines based on the type of control system

NC systems are classified on the basis of type of control system used as follows.

(a) Open loop system
(b) Closed loop system

Open loop system:

In the open-loop control system, the programmed instructions are fed into the controller through an input device.

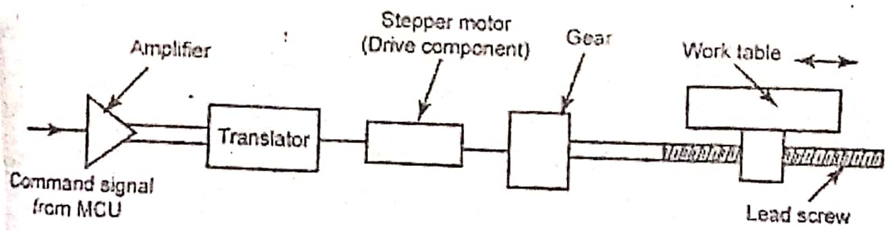


Figure 5.6 Open-loop machine slide control

The primary drawback of

The primary drawback of the open-loop system is that there is no feedback system to check whether the program position and velocity have been achieved. If the system performance is affected by load, temperature, humidity or lubrication, then the actual output could deviate from the desired output.

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. A position transducer acts as a feedback device. The feedback system could be either analog or digital.

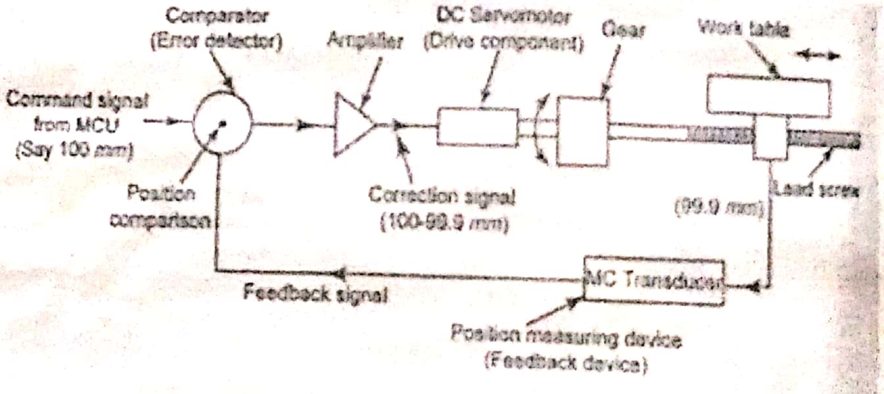


Figure 5.7 Closed-loop positioning control

Classification Of NC Machines based on Type of Motion Control:-

1. Point-to-Point NC system
2. Continuous path NC system
    a) Straight cut system
    b) Contouring system

Point to Point System

Some machine tools for examples drilling, boring and tapping machines etc require the cutter and the workpiece to be placed at certain fixed relative positions. These machines are known as point-to point machines. It moves the tool only in straight lines. The speed or path is not important in this system.

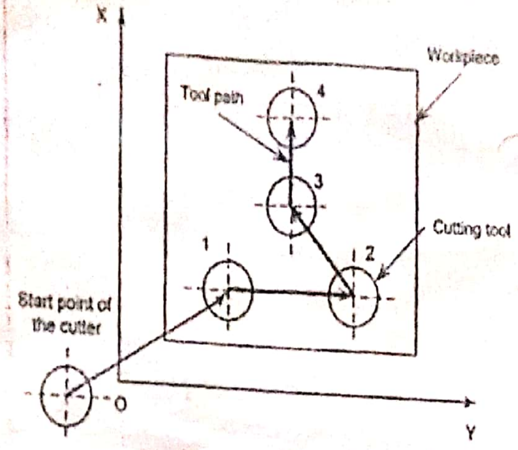


Figure 5.8 Point-to-point NC system

Continuous Path NC system:

i) Straight cut NC system:

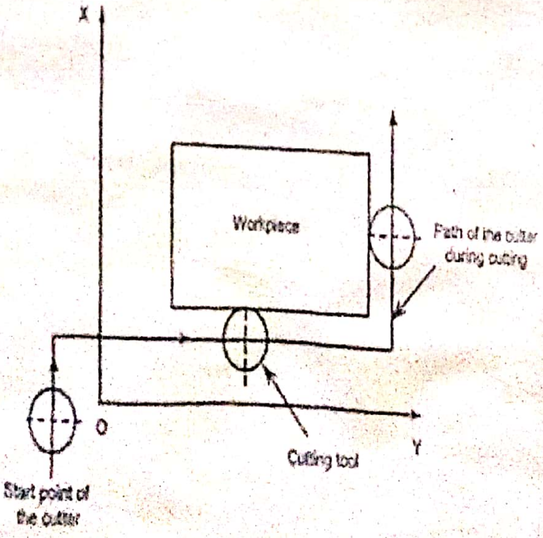


Figure 5.9 Straight-cut NC system

When continuous path control is utilized to move the tool parallel to only one of the major axes of the machine tool worktable, this is called straight cut NC system as shown in fig. It is performed for milling operations.

b) Contouring System:

When continuous path control is used for simultaneous control of two or more axes in machining operations, this is called contouring NC system. It is a complex and flexible method of tool control. It is capable of performing both point to point and straight cut operations.

machining centers.

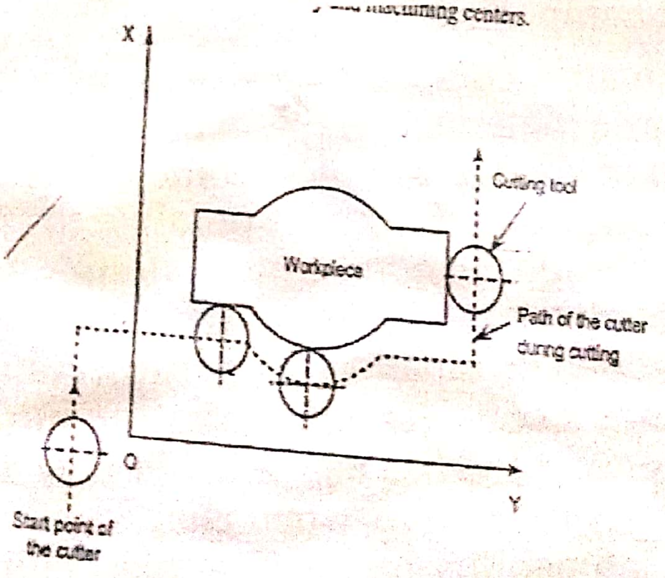


Figure 5.10 Contouring NC system

Applications:

1. NC lathe, either horizontal or vertical
2. NC boring mill, horizontal and vertical spindle.
3. NC drilling press
4. NC milling machine

Computer Numerical Control System (CNC)

Computer Numerical Control is a NC system that utilizes a stored program to perform the basic Numerical Control functions. A mini or micro computer based controller unit is used.

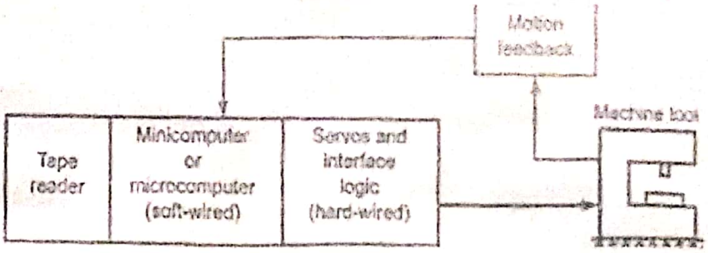


Figure 5.11 General configuration of CNC system

Classification of CNC Systems

a) According to the Structure of control system used

1. Analog system
2. Digital system

b) According to the type of Control loop used

1. Closed loop system
2. Open loop system

c) According to the type of tool motion control system used

1. Point to point system
2. Continuous path system

d) According to the programming mode used

1. Closed loop system
2. Open loop system

e) According to the controller design

1. Hybrid CNC
2. Straight CNC

Elements of CNC systems:

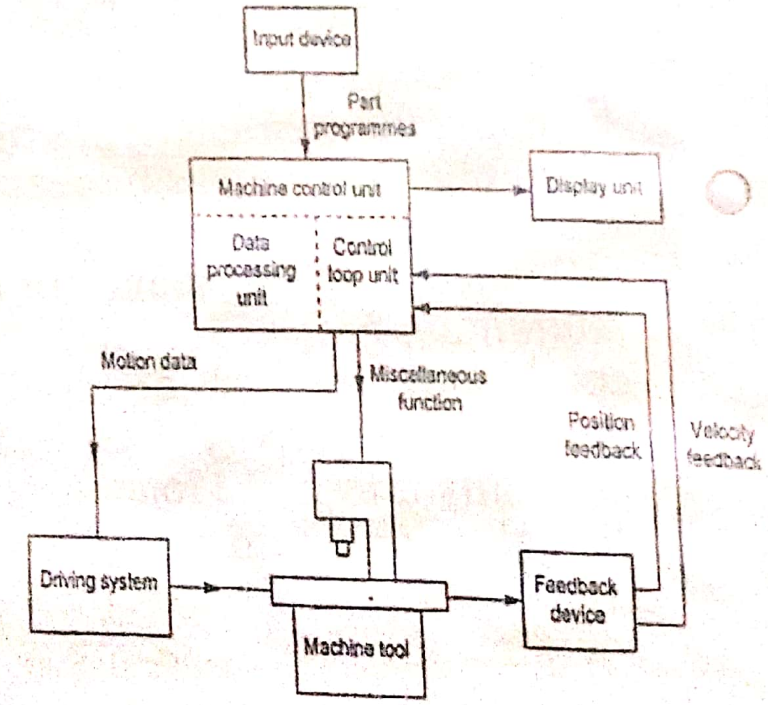


Figure 5.14 Elements of CNC systems

Constructional Features of CNC Machine Tool:

Some of Important parts of CNC machines tool are as follows:

(a) Machine Structure

(b) Slideways or Guide ways

(c) Spindle / spindle bearings

  1) Hydrodynamic

  2) Hydrostatic

  3) Antifriction

(d) Spindle drives

  1) Electrical drives

  2) Hydraulic drives

  3) Pneumatic drives

(e) Feed drives

  1) Servo motor

  2) Mechanical transmission system

(f) Measuring Systems.

  1) Direct

  2) Indirect

(g) Controls, Software and user interface

(h) Gauging

(i) Tool monitoring Systems

  1) Direct

  2) Inderect

Machining Centres:

A machining Centre is a highly automated machine tool capable of performing multiple machining operations under computer numerical control in one ~~St~~ Setup and it can work on more than one face of a component with minimal human attention.

Workers needed to load and unload component which usually takes considerable less time than the machine cycle time, So one worker may be able to tend more than one machine. Machining centres are among the most popular types of CNC machine tools these days

The following operations can be carried out on a machining centre.

1. Milling
2. Drilling
3. Reaming
4. Boring
5. Tapping

Features to reduce non-productive time:

1. Automatic tool changer
2. Automatic Component positioner
3. Atomatic pallet changer
4. Multiple operations in one setup.

Scanned with CamScanner

Classification of machining Centres:

According to the Spindle configuration, machining centres are classified as follows:

1. Horizontal spindle machining Centre
2. Vertical spindle machining centre
3. Universal machining centre

1. Horizontal Spindle machining Centre:

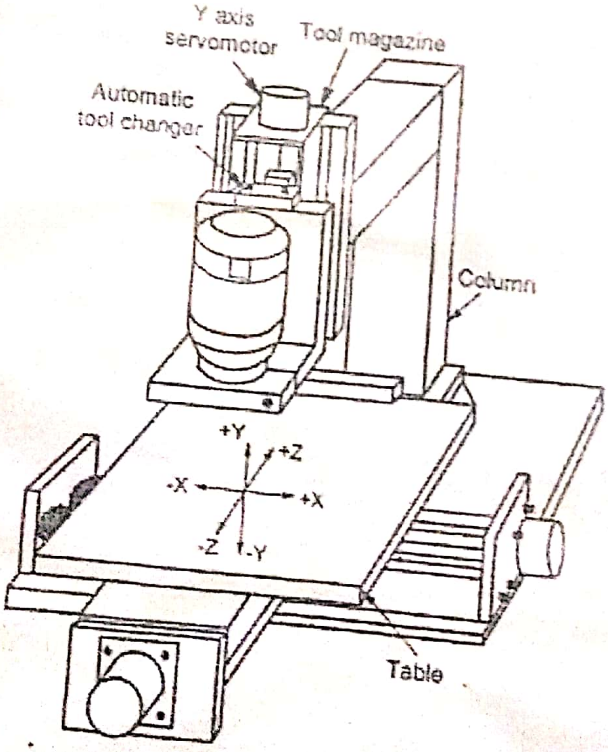


Figure 5.39 Horizontal machining centre

The axes of horizontal Spindle machining centre are given below.

X-axis ⇒ Table or Column motion left to right as viewed from the spindle

Y-axis ⇒ Spindle head motion up & down

Z-axis ⇒ Saddle / column / spindle head motion and away from the Spindle.

Vertical spindle Machining Centre:

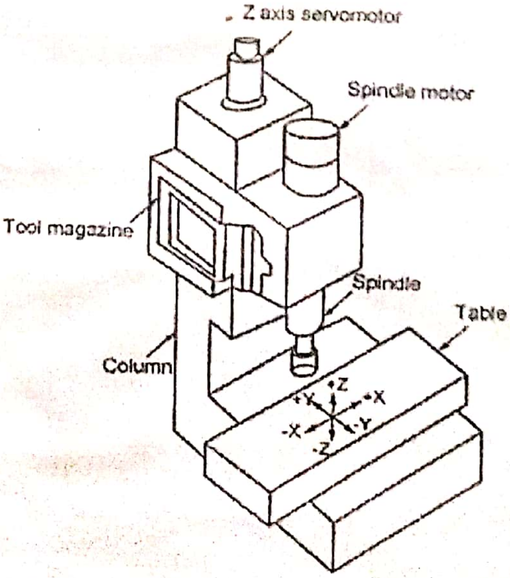


Figure 5.40 Vertical machining centre

Basic Vertical machining Centres will allow three directions or axes of motion as follows.

X axis ⇒ Table or Column motion left to right as
viewed from the spindle

Y axis → Saddle or table motion toward and
away from the spindle

Z axis → Spindle head or headstock motion
up and down

Universal machining Centre:

Universal machining Centres are similar to horizontal machining centres but it is with the spindle axis capable of tilting from horizontal to the vertical position. This feature allows ease of machining inclined surfaces. In some machines the table can tilted instead of spindle.

Features:

1. It has a single spindle
2. It has five axes of machine
3. The flexibility is more than other two types
4. Tool breakage detection is possible
5. Automatic loading and unloading of workpiece are possible.

Direct Numerical Control (DNC) System:

DNC can be defined as a manufacturing system in which a number of CNC machines are controlled by a single computer through direct connection and in real time.

The components of DNC are given below.

1. Central Computer
2. Bulk memory which stores the NC part programs
3. Telecommunication lines/Computer networks
4. NC/CNC machine tools

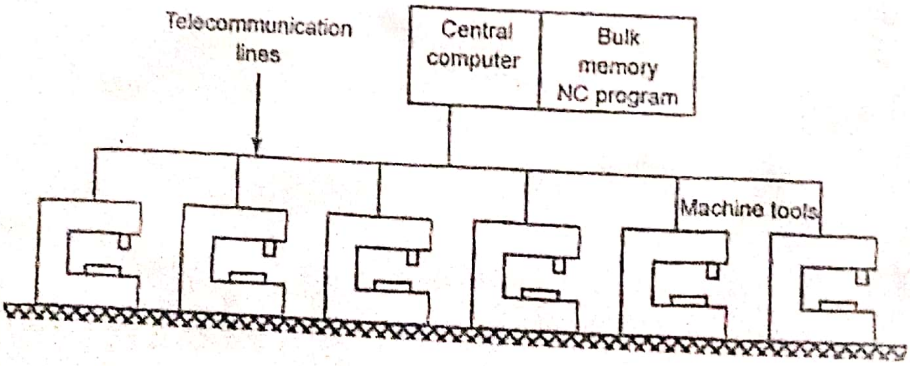
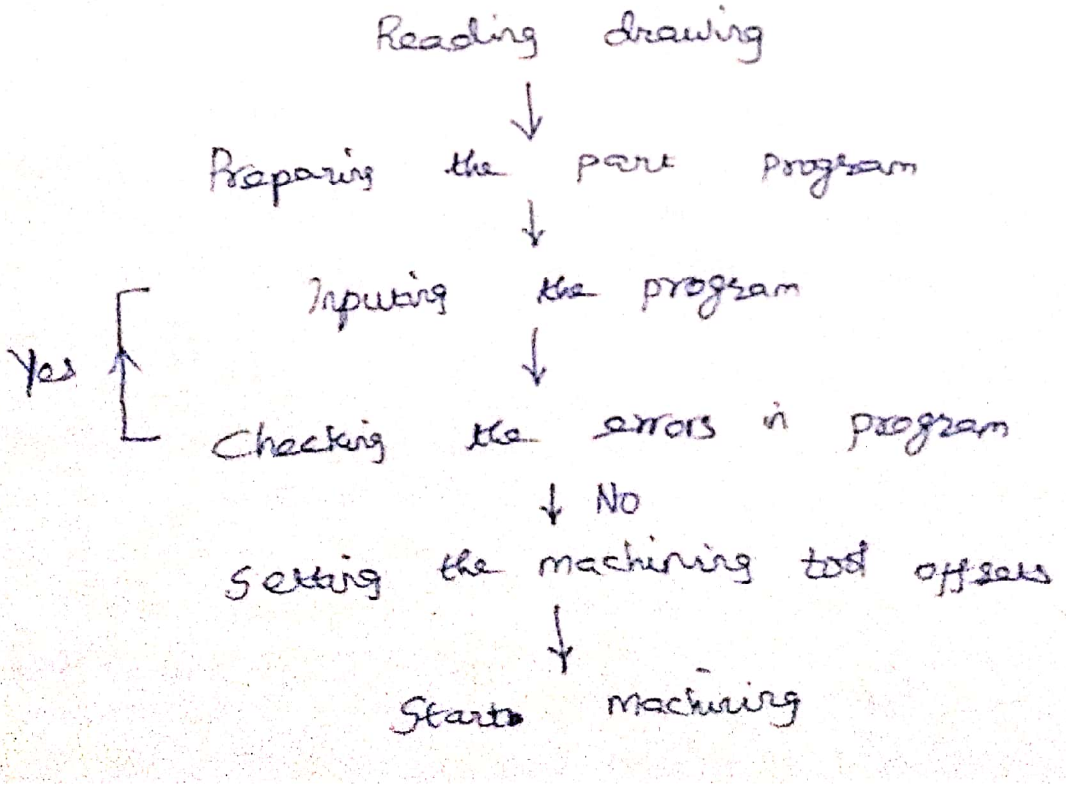


Figure 5.41 General configuration of DNC system

Comparison between CNC & DNC

S.No	CNC System	DNC System
1	CNC system can do operations on only one machine at a time	CNC system can do operations on multiple machines at a time.
2.	In CNC system, the program is fed to the machine through the computer	Part program is fed to the machine through the main computer
3.	The programs can be easily modified with the help of computer	In order to modify the program of many machines the single computer is used

Part Programming Fundamentals:



Part Program:

The part program is a set of instructions proposed to get the machined part starting with the desired blank and NC machine tool. Part programming contains geometric data about the part and motion information to move the cutting tool w.r. to the workpiece.

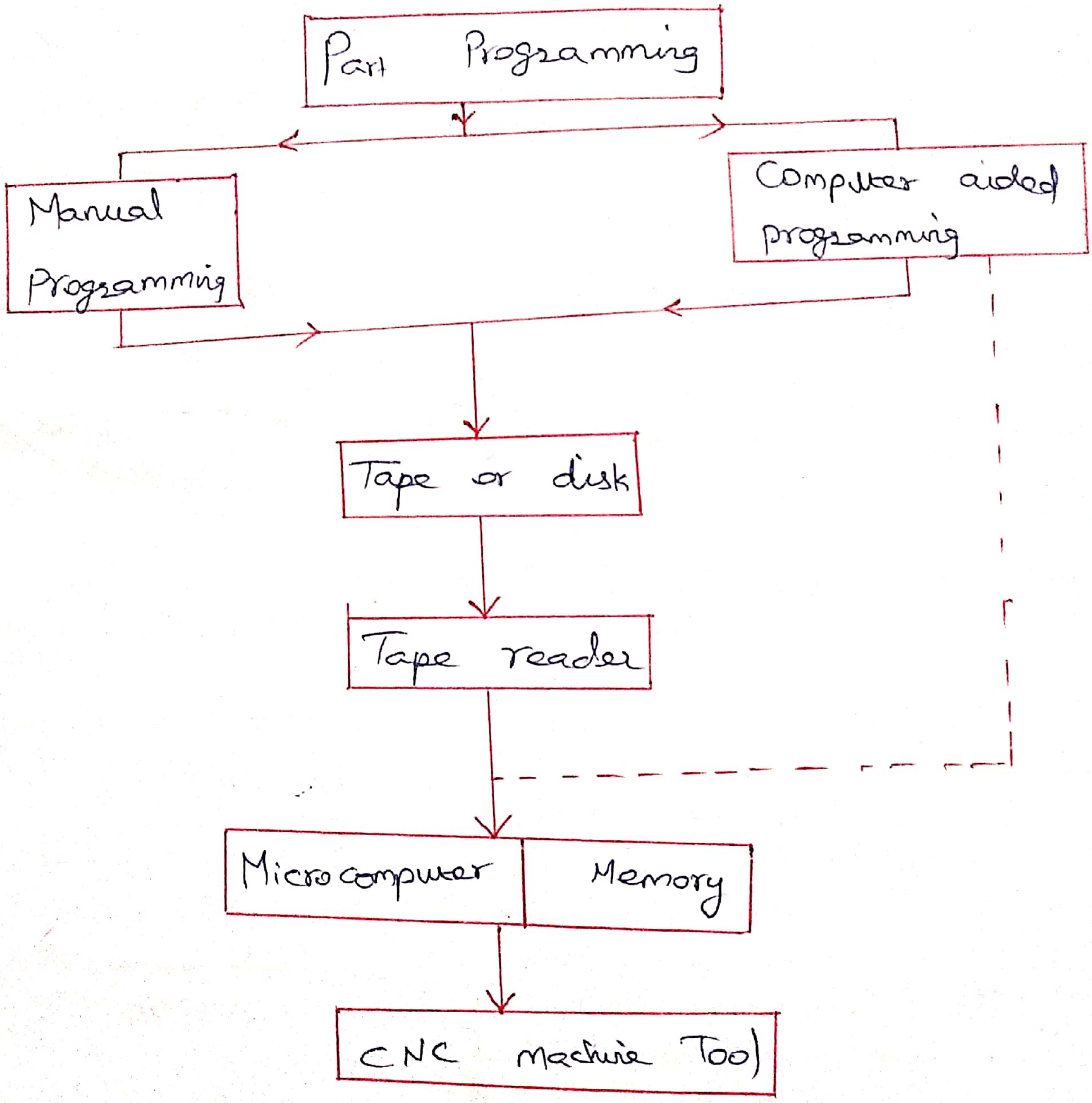


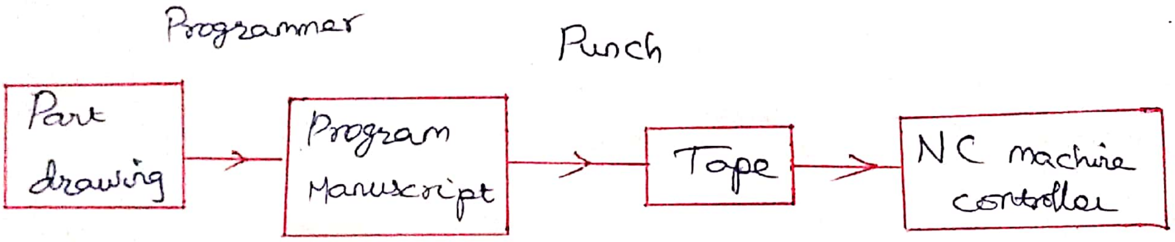
fig. Layout of part program. ~~Producer~~
Procedure

Methods of Creating Part Programming :

The following are various methods of creating part programming.

i) Manual part programming
ii) Computer assisted part programming
iii) Manual data Input
iv) NC programming using CAD/CAM
v) Computer automated part programming

Manual Part Programming :



The typical program format is.

N10 G01 X50 Y100 Z30 F100 S200 T02 M03

N10 → Block number

G01 → preparatory function

X50 Y100 Z30 → Target co-ordinates

F100 → Feed rate
S200 → Spindle speed
T02 → Tool number
M03 → Miscellan

Part Program for Drilling Process

1. Prepare a part program to drill the given component of 10mm thick shown in fig in a CNC drilling machine.

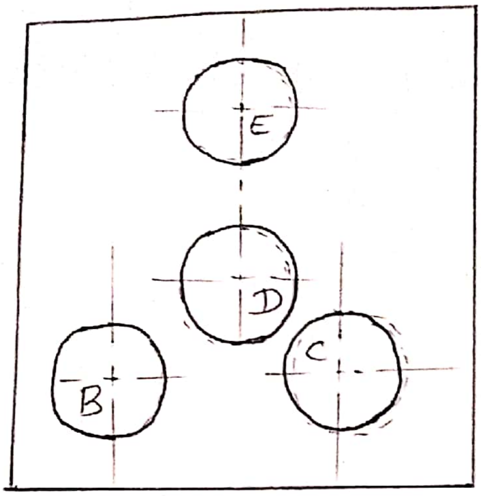


Fig. Tool positions for drilling

Program in absolute mode:

```
O0008
N00   G28   T00   U00   W00
N10   M06   T01
N20   G54   G90   S200   M03   T01
N30   M08
N40   G74   G98
N50   G00   X00   Y00   Z00
N60   G00   X10   Y10
N70   G01   Z-10  F5
N80   G00   X40   Y10   Z00
```

```
N90    G01   Z-10   F5
N100   G00   X25    Y25    Z00
N110   G01   Z-10   F5
N120   G00   X25    Y40    Z00
N130   G01   Z-32   F5
N140   G00   X00    Y00    Z00
N150   M05
N160   M09
N170   M02
N180   G28   Z00    M19
```

Part Program for Milling Process:

① Prepare a part program for manufacturing the given Component shown in fig. Profile milling process in a CNC milling machine by considering cutter compensation.

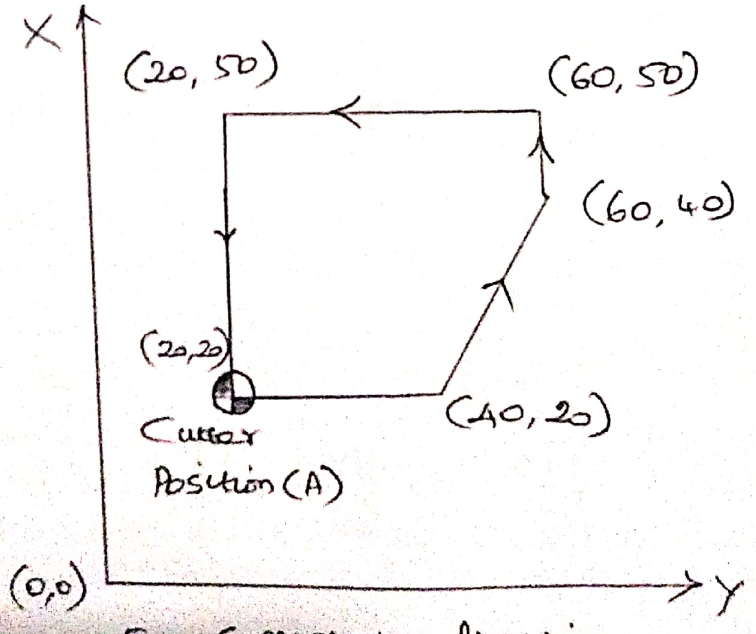


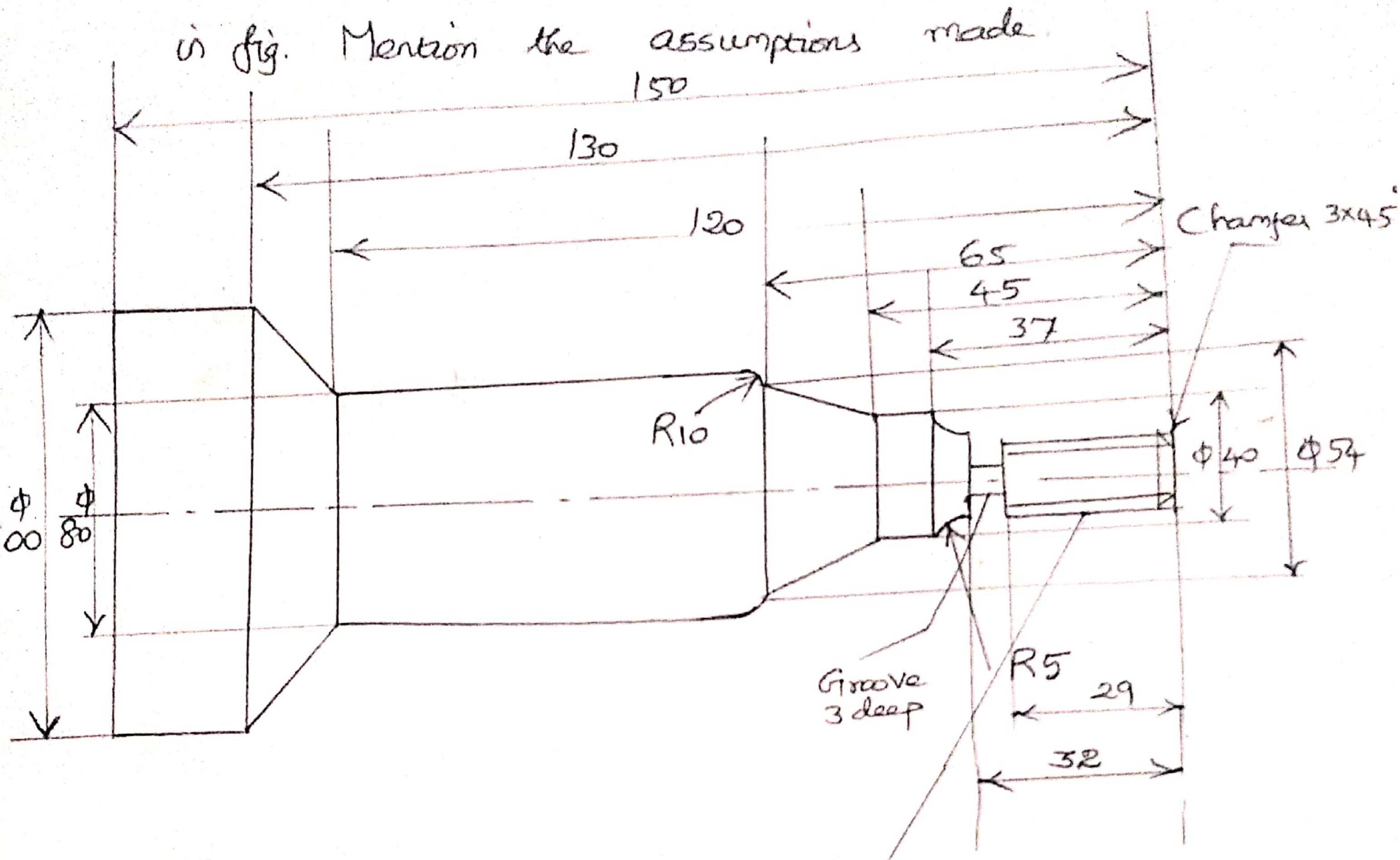
Fig. Component drawing

Part Program in incremental mode:

```
O0012
N00   G28   T00    U00    W00
N01   M06   T01
N02   G54   G91    S400   M03   T01
N03   M08
N04   G17
N05   G01   X00    Y00    Z00
N06   G00   X20    Y20
N07   Z-10  F0.8
N08   G42   D1
N09   G1    Y20    F199
N10   X40   Y20
N11   X60   Y40
N12   X60   Y50
N13   X20   Y50
N14   X60   Y50
N15   X00   Y00    Z00
N16   G40   D0
N17   M05
N18   M09
N19   M02
N20   G28   Z00    M19
```

Part Program on Lathe Operation:

① Write CNC part program for the Component Shown in fig. Mention the assumptions made.



Billet diameter = 110 mm

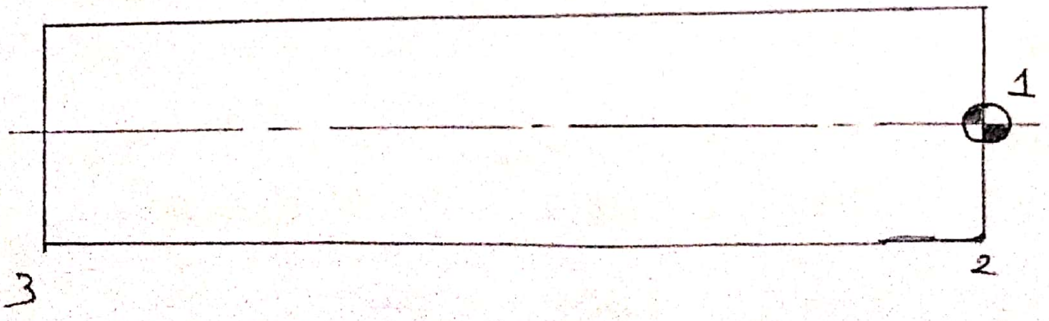
Billet length = 200 mm

M 20 = 2.5 Thread

Root diameter = 16.25

All Dimensions are in mm

Soln:-



$$\text{Depth of cut per place} = \frac{\text{Depth of cut}}{\text{Number of passes}} = \frac{0.65 \times 2.5}{10} = 0.1625\text{m}$$

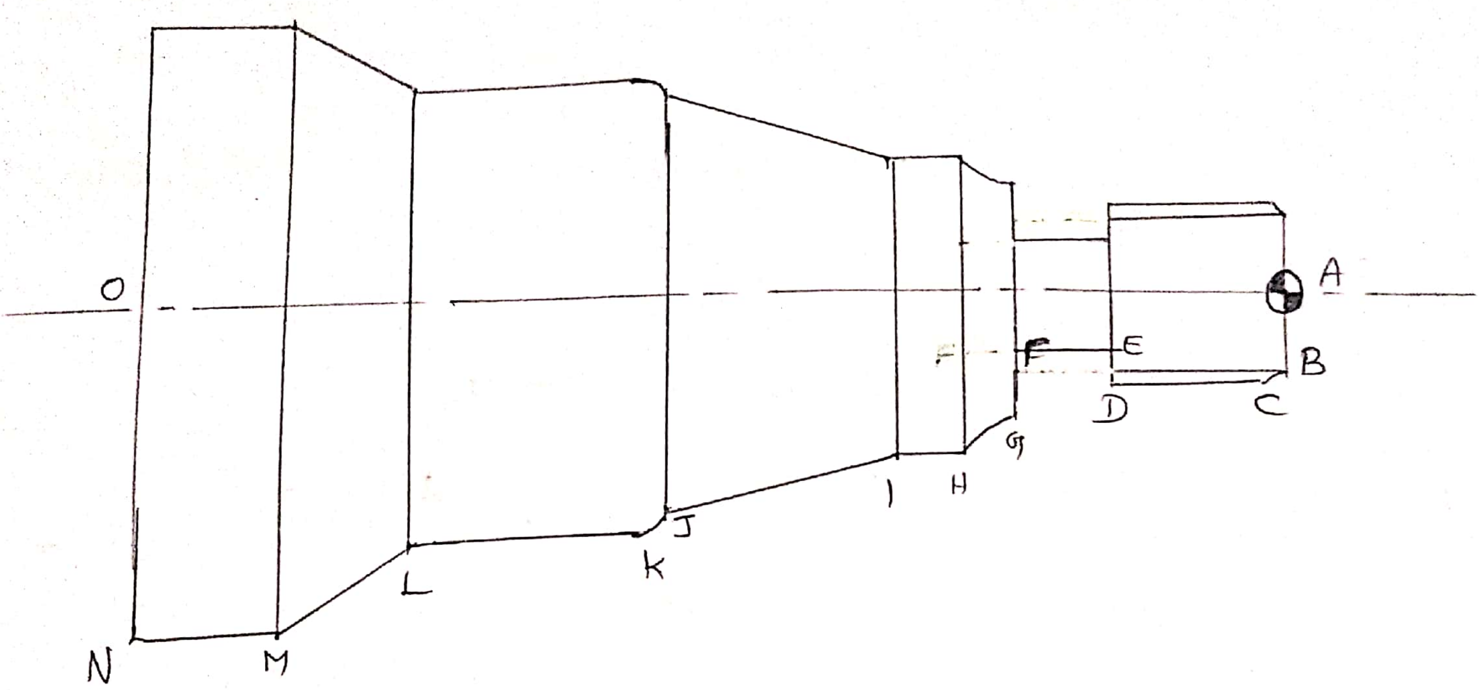


Fig. Tool Positions for the Component

Programming in Incremental mode :

```
O0024

N01   G28   T00   U00    W00

N02   M06   T01

N03   G54   G91   S1500  M03   T01

N04   M08

N05   G00   X00   Y00

N06   G01   X00   Y-50   F80

N07   G01   X-153 Y00

N08   G00   X00   Y00
```

```
N09   G01   X00    Y-7
N10   G01   X-3    Y-3
N11   G01   X-26   Y00
N12   G01   X00    Y3
N13   G01   X-3    Y00
N14   G01   X00    Y-8
N15   G03   X-5    Y-5
N16   G01   X-8    Y00
N17   G01   X-20   Y-7
N18   G02   X-10   Y-10
N19   G01   X-45   Y00
N20   G01   X-10   Y-10
N21   G01   X-20   Y00
N22   G01   X00    Y50
N23   G00   X150   Y00
N24   G54   G91    S100     M03   T02
N25   G74   Y0.123   L10
N26   G84   X-19   Y16.25   K2   D1.23   F0.08   A60
N27   G80
N28   G00   X00    Y00   Z00
N29   M09
N30   M05
N31   M02
N32   G28   X00   ~~Y~~ M19
```

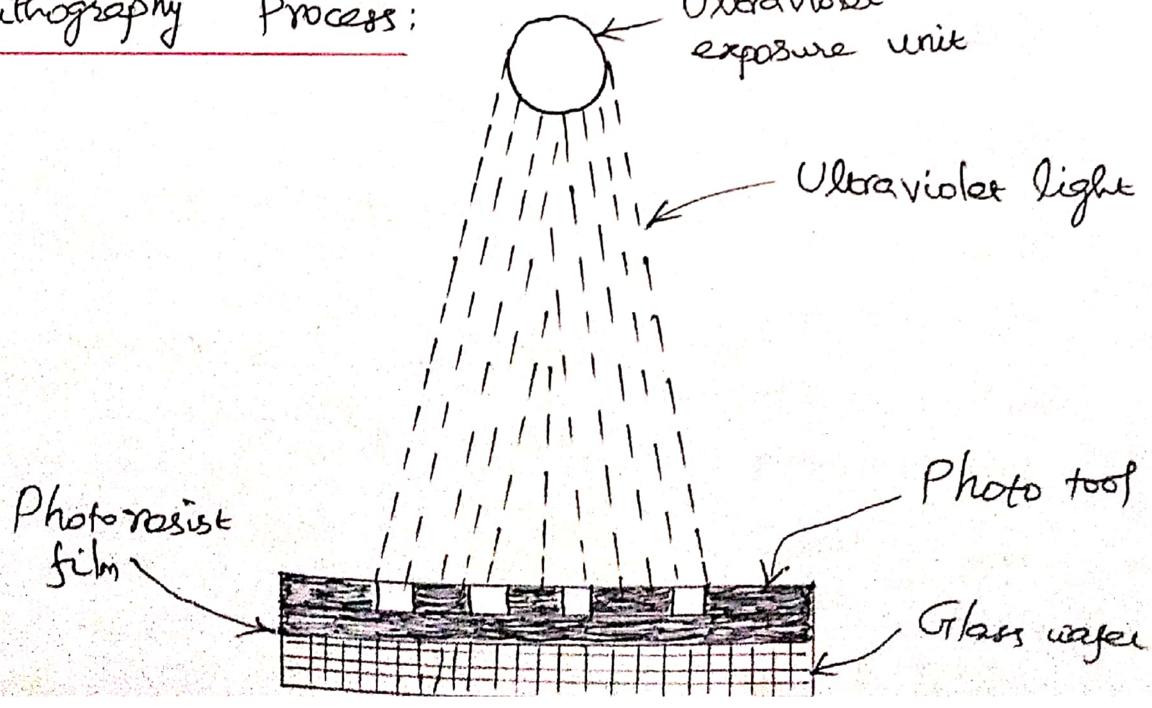
Surface Micromachining or Wafer Machining:

In the surface micromachining process, the structures are created on top of a substrate. In this case, a silicon wafer is selectively etched to produce structures. In this machining, the microstructures are built by deposition and etching of different structural layers on top of the substrate. This layers are selectively etched by photolithography and either can combine the chemical etching with physical etching, or ion bombardment of the material. Generally, Poly-Silicon is used as one of the layers and silicon dioxide is used as a sacrificial layer. The purpose is to remove or etch voids.

Micromachining Processes:

1. Lithography based Micromachining:

i) Photolithography Process:



i) Photolithography Process:

A photo polymer called photoresist (PR) is the basis for photolithography. If a layer needs to be patterned, if we want to remove the material from a layer selectively, we need to create a masking layer to define the windows through which to etch. The mask is usually a glass plate with a chromium pattern.

The photoresist layer is then exposed to UV light through a mask. The UV exposed regions of the photoresist change properties via depolymerization. Then the photoresist layer is developed. It is done by spraying a solution called photoresist developer.

ii) Etching

iii) LIGA

iv) Thin film deposition

v) Planarization

Non-Lithography Based Micromachining:

1. Mechanical micromachining
2. Micro-EDM
3. Abrasive machining
4. Laser ablation process
5. Focused Ion Beam milling