STELLA MARY'S COLLEGE OF ENGINEERING

(Accredited by NAAC, Approved by AICTE - New Delhi, Affiliated to Anna University Chennai)

Aruthenganvilai, Azhikal Post, Kanyalumari District, Tamilnadu - 629202.

ME8451 MANUFACTURING TECHNOLOGY II

(Anna University: R2017)

Prepared By

Mr. F. X. Aruno Xavier

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

STELLA MARY'S COLLEGE OF ENGINEERING

(Approved by AICTE, New Delhi, Affiliated to Anna University, Chennai
Aruthenganvilai, Kallukatti Junction Azhikal Post, Kanyakumari District-629202, Tamil Nadu.

DEPARTMENT OF MECHANICAL ENGINEERING

COURSE MATERIAL

REGULATION	**2017**
YEAR	**II**
SEMESTER	**04**
COURSE NAME	**MANUFACTURING TECHNOLOGY II**
COURSE CODE	**ME8451**
NAME OF THE COURSE INSTRUCTOR	**Mr. F. X. ARUNO XAVIER**

SYLLABUS:

UNIT I THEORY OF METAL CUTTING 9

Mechanics of chip formation, single point cutting tool, forces in machining, Types of chip, cutting tools– nomenclature, orthogonal metal cutting, thermal aspects, cutting tool materials, tool wear, tool life, surfacefinish, cutting fluids and Machinability.

UNIT II TURNINGMACHINES 9

Centre lathe, constructional features, specification, operations – taper turning methods, thread cutting methods, special attachments, machining time and power estimation. Capstan and turret lathes- tool layout – automatic lathes: semi automatic – single spindle : Swiss type, automatic screw type – multi spindle:

UNIT III SHAPER, MILLING AND GEAR CUTTING MACHINES 9

Shaper - Types of operations. Drilling ,reaming, boring, Tapping. Milling operations-types of milling cutter. Gear cutting – forming and generation principle and construction of gear milling ,hobbing and gear shaping processes – finishing of gears.

UNIT IV ABRASIVE PROCESS AND BROACHING 9

Abrasive processes: grinding wheel – specifications and selection, types of grinding process– cylindrical grinding, surface grinding, centreless grinding and internal grinding- Typical applications – concepts of surface integrity, broaching machines: broach construction – push, pull, surface and continuous broaching machines

UNIT V CNCMACHINING 9

Numerical Control (NC) machine tools – CNC types, constructional details, special features, machining centre, part programming fundamentals CNC – manual part programming – micromachining – wafermachining.

TEXT BOOKS :

1. Hajra Choudhury, "Elements ofWorkshopTechnology", Vol.II., Media Promoters 2014
2. Rao. P.N “Manufacturing Technology - Metal Cutting and Machine Tools", 3rd Edition, Tata McGraw-Hill, ewDelhi, 2013.

REFERENCES:

1. Richerd R Kibbe, John E. Neely, Roland O. Merges and Warren J.White “Machine Tool Practices”, Prentice Hall of India, 1998
2. Geofrey Boothroyd, "Fundamentals of Metal Machining and Machine Tools", Mc Graw Hill, 1984
3. HMT, "ProductionTechnology", Tata McGrawHill, 1998.
4. Roy. A.Lindberg, “Process and Materials of Manufacture,” Fourth Edition, PHI/Pearson Education 2006.

Course Outcome Articulation Matrix

	Program Outcome												*PSO*		
Course Code / CO No	***1***	***2***	***3***	***4***	***5***	***6***	***7***	***8***	***9***	***10***	***11***	***12***	***1***	***2***	***3***
ME8451 / C212.1	3	2	3	2	3	2	0	0	2	2	2	3	3	2	3
ME8451 / C212.2	3	2	3	0	3	2	0	0	2	2	2	3	3	2	3
ME8451 / C212.3	3	0	3	0	3	2	0	0	2	2	2	3	3	2	3
ME8451 / C212.4	3	2	3	0	3	2	0	0	2	2	2	3	3	2	3
ME8451 / C212.5	3	3	3	2	3	2	0	1	2	2	2	3	3	2	3
Average	3	2	3	1	3	2	0	0	2	2	2	3	3	2	3

Unit-1

Theory of metal cutting

Cutting Tools
- → Single Point Tool (Lathe, Slotting, Shaper tool)
- → Multi point tool (Milling Cutters, Broaching tool)

Factors that have a predominant influence on metal cutting:-

1. Cutting tool Geometry → Rake angle, → Clearance angle
2. work & cutting Tool material
3. cutting speed 4. feed
5. Depth of cut 6. Cutting fluids used.

Rake angle & Clearance angle:-

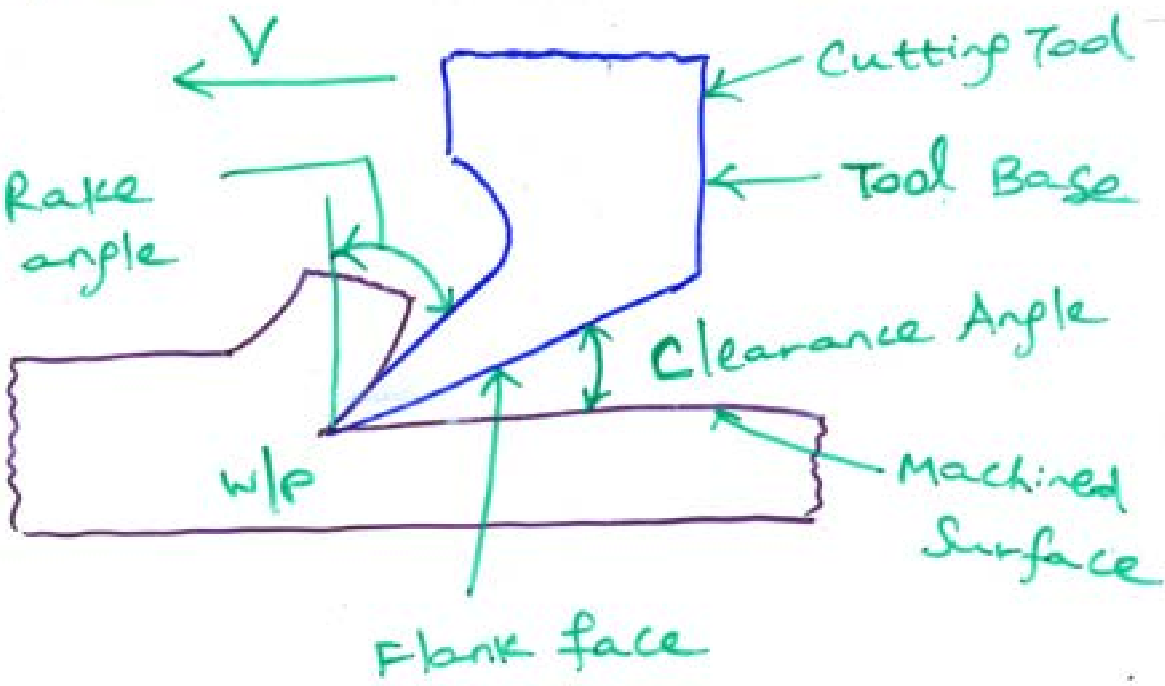
* "Rake angle" is the angle b/w the Rake face of the cutting tool & the line ∥" to the base of the tool (or) It is the angle b/w the Rake face and the line normal to the machining direction.

Significance of the Rake angle :-

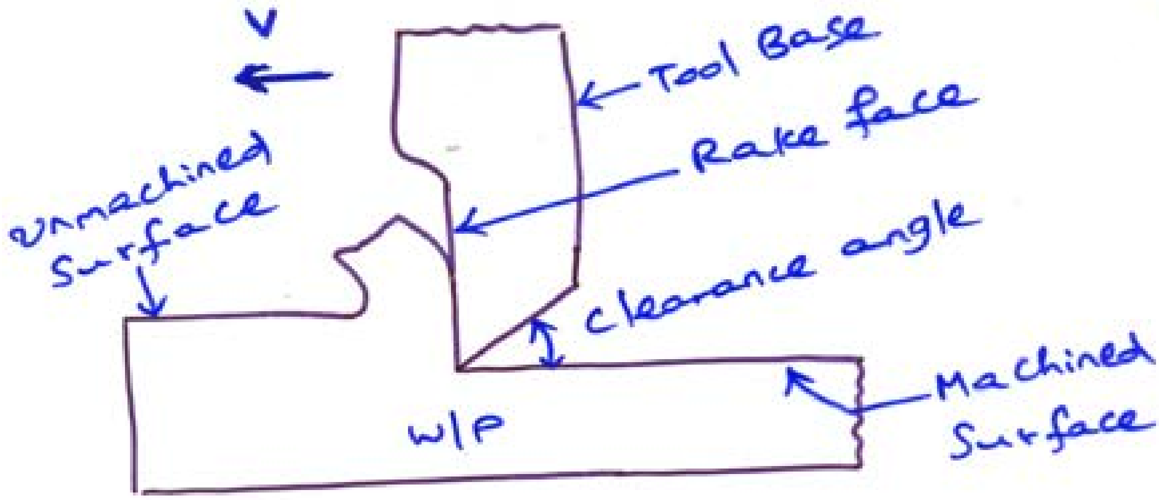
1. It specifies the ease with which a metal is cut.
2. Higher the rake angle, better is the cutting and less are the cutting forces.

Note :-

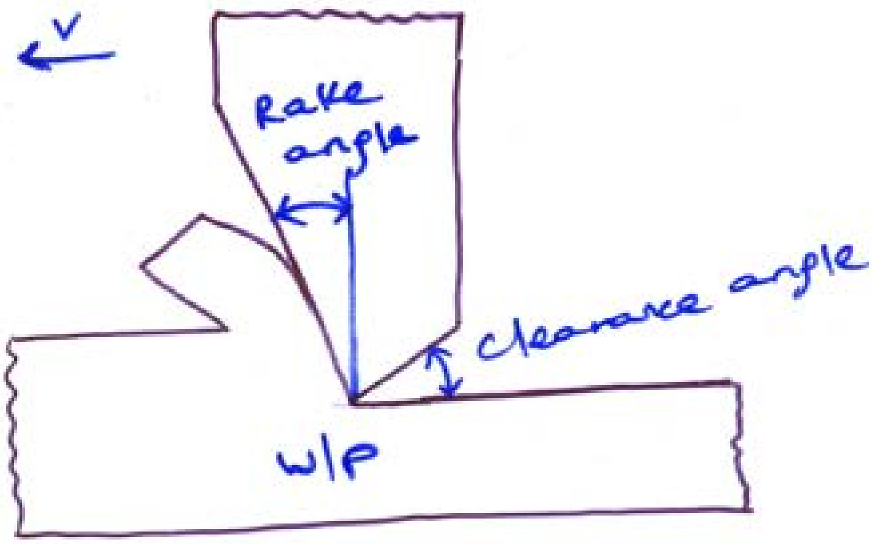
* the maximum limit for the rake angle is 15°
* zero & negative rake angle is used for giving extra strength to the tool tip. these tools are used for m/cing highly hard material such as Carbides (or) Diamond.



(a) Positive Rake angle



(b) Zero (or) Neutral Rake angle



(c) Negative Rake angle

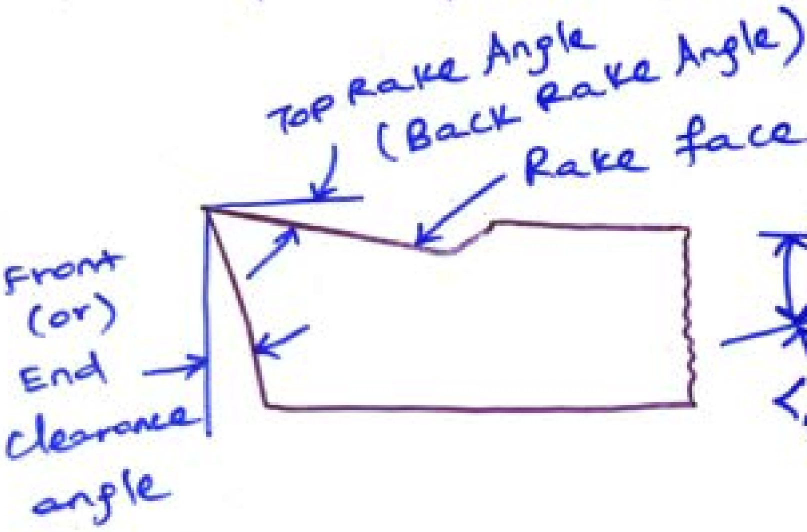
"Clearance Angle" is the angle b/w the machined surface and the flank face of the tool.

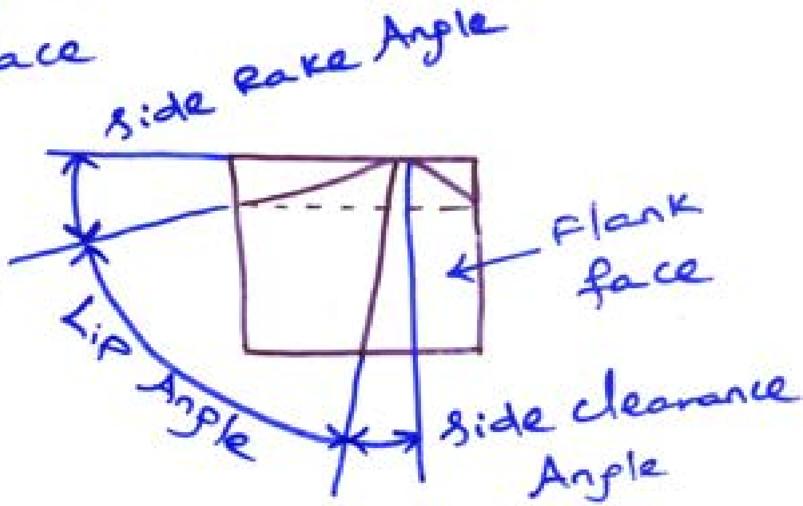
Significance of Clearance Angle:-

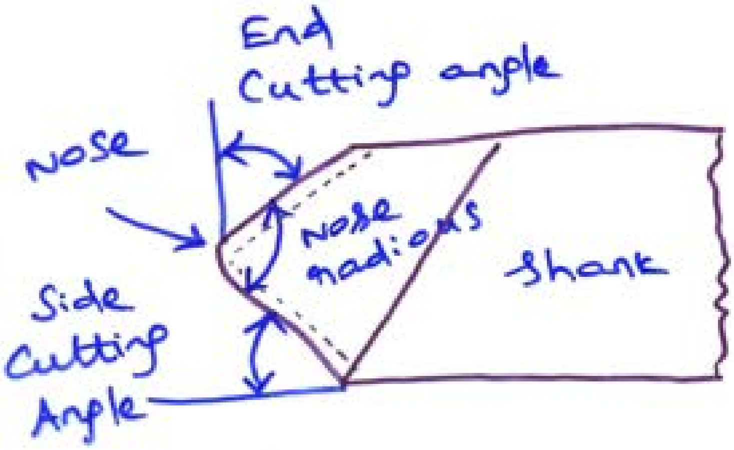
1. It is provided on the tool to avoid the rubbing of the tool on the w/p when the machining takes place. Hence it prevents the spoiling of the machined surface

2. It is used to increase the cutting forces.

Single Point Cutting Tool Nomenclature:-







① Shank:-

Portion of the tool bit which is not ground to form the cutting edges & is rectangular in cross-section.

② Rake face:-

Surface against which the chip slides

③ Flank face:-

Surface which faces the w/p.

④ Nose:-

The point of intersection of side & End cutting edges. Nose radious increases the tool life & surface finish.

⑤ Cutting edge:-

It is the junction of the Rake face & the flank face.

a) Side cutting edge

b) End Cutting edge.

⑥ Various angles in the cutting Tool:-

① Rake Angle → Top Rake Angle
→ Side Rake Angle

② Clearance Angle (or) Relief Angle

a) End (or) Front Relief angle

b) Side clearance angle

③ Cutting edge Angle

a) Side cutting edge Angle

b) End Cutting edge Angle

④ Nose radious (or) Nose angle.

Top Rake angle:-

* Also called as "Back Rake angle".
* It is the angle b/w the Rake face of the tool & the line ॥" to the base of the tool. This is measured along the length of the tool.

Side Rake angle:-

It is also the angle b/w the Rake face & the base of the tool. But it is measured along the width of the tool.

End (or) Front Relief angle:-

It is the angle b/w the front flank face & the line ⊥r to the base of the

Unit-1

Theory of metal cutting

Cutting Tools
- → Single Point Tool (Lathe, Slotting, Shaper tool)
- → Multi point tool (Milling Cutters, Broaching tool)

Factors that have a predominant influence on metal cutting:—

1. Cutting tool Geometry → Rake angle, → Clearance angle
2. Work & cutting Tool material
3. Cutting speed
4. feed
5. Depth of cut
6. Cutting fluids used.

Rake angle & Clearance angle:—

* "Rake angle" is the angle b/w the Rake face of the cutting tool & the line ॥" to the base of the tool (or) It is the angle b/w the Rake face and the line normal to the machining direction.

tool. The purpose of providing this angle is to avoid the rubbing of the tool on the w/p when the tool is fed along the length of the Job.

Side Relief (or) Clearance angle:-

It is the angle b/w the Side flank face and the line ⊥r to the base of the tool. The purpose of providing this angle is to avoid the rubbing of the tool on the w/p when the tool is fed Cross wise to the w/p.

Side & End cutting edge angle:-

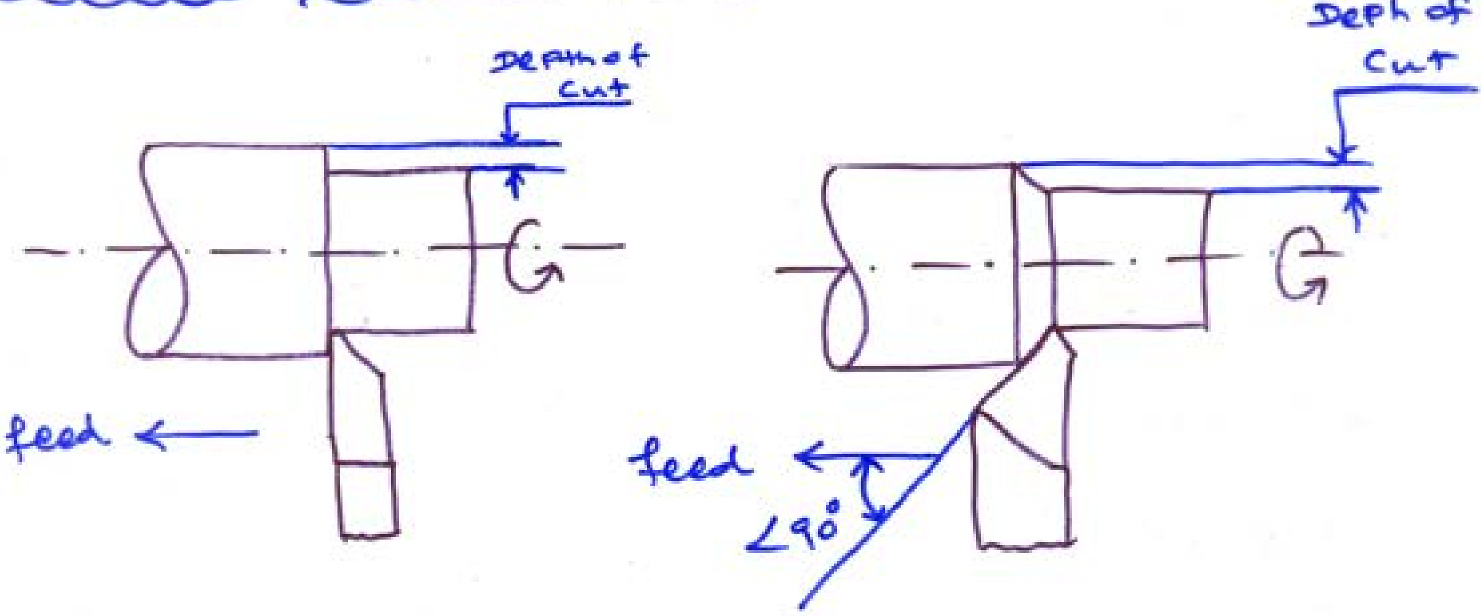
* Side cutting edge angle is the angle b/w the side cutting edge and the longitudinal axis of the Cutting Tool whereas the end cutting edge angle is the angle b/w the End cutting edge and the line ⊥r to the longitudinal axis.

* these angles will allow only a Smaller Portion of the cutting edge to contact with the w/p and thus prevents the Vibration and chatter. Normally this angle Varies from 5° to 15°.

Nose angle (or) nose Radious:-

* It is the angle b/w the Side cutting edge and End cutting edge
* It is used to increase the tool life and surface finish of the w/p.

Orthogonal & oblique cutting:-



The metal cutting processes are mainly classified into 2 types

(i) Orthogonal cutting [2-D cutting]

(ii) Oblique cutting process [3-D cutting]

* If the cutting edge is at 90° to the line of action (or) Path of the tool, it is known as "oblique cutting".

* If the cutting edge of the tool is at an acute angle [less than 90°] to the line which is normal to the cutting velocity vector it is known as "oblique cutting".

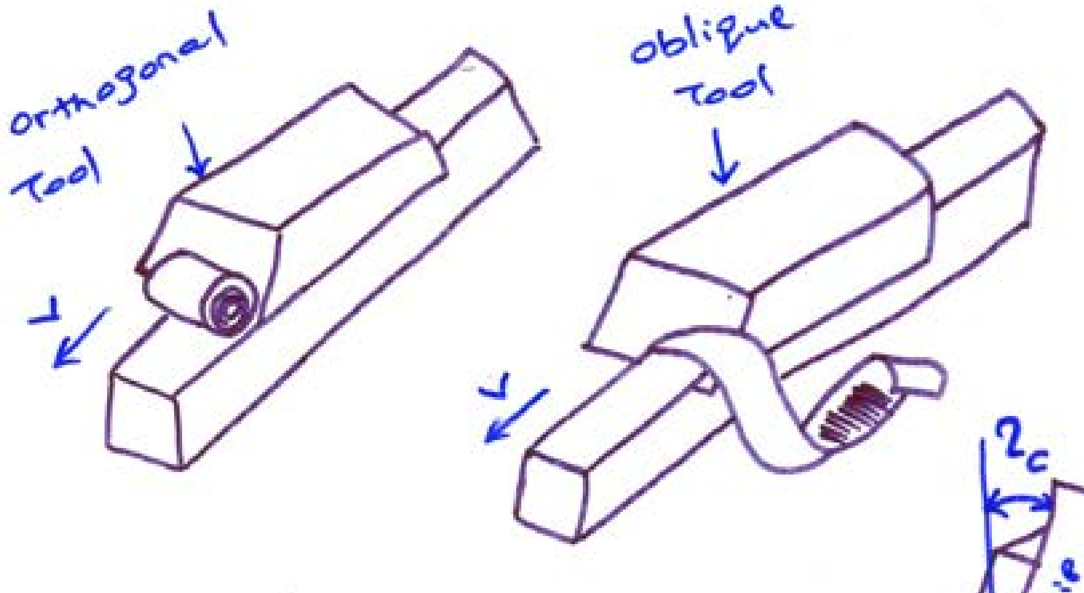
Points to be noted:-

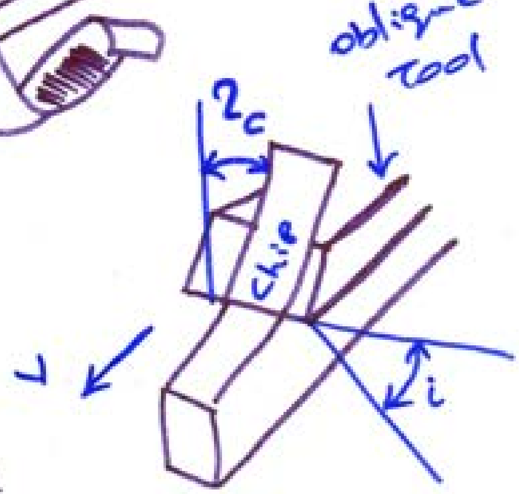
* For the same depth of cut & feed,

a) the force which cuts (or) shears the metal acts on a larger Area in case of oblique tool and the oblique tool will have a longer life as the heat developed per unit Area due to friction along w/p-tool interface is small.

b) The oblique tool will remove more metal compared to orthogonal tool.

Chip flow in orthogonal & oblique cutting:—





η_c → Chip flow Angle

- Angle b/w the direction of chip flow and the line normal to the cutting edge

i → Inclination Angle

- Angle b/w the cutting edge and the line normal to the cutting velocity vector.

* In oblique cuttings, the cutting edge is inclined at an angle "i" and the chip flows in a sideways in the form of long curl.

* In orthogonal cuttings the inclination angle "i" is zero and the chips flow like "tight, flat spiral" and hence η_c is also zero. (ie) Here $i=0$ & $\eta_c=0$

Example for orthogonal & oblique cutting:-

Orthogonal cutting – Sawing, Broaching, Slotting, parting-off oprns.

oblique cutting – turning opreation in Lathe, Milling, drilling opreations etc,

Comparison b/w orthogonal & oblique cutting:-

S.no	Orthogonal cutting	oblique cutting.
1.	Cutting edge of the tool is $\perp^r$ to the line of action (or) path of the tool.	cutting edge of the tool is at an acute angle to line which is normal to the cutting Velocity Vector.
2.	the Chip flows like a tight flat spiral.	The chip flows like long curl.
3.	Only one cutting edge Contacts with the w/p during oprn.	Frequently, more than one edge are in action during the opreation

S.No	Orthogonal cutting	oblique cutting.
4.	Only 2 components of forces are there during opreation [feed & cutting force)	3 components of forces are there during opreation [feed, cutting & thrust force]
5.	For the same depth of cut, the frictional force/unit Area is high as less area contacts with the w/p during the opn and hence tool life is comparatively less	For the same depth of cut, the frictional force/unit Area is comparatively low as more area contacts with the w/p during the opn and hence the tool life is comparatively more
6.	examples:— Sawing, Broaching, slotting and Lathe Parting-off opns, shaping opreations.	examples:— Lathe turning opn, Milling, drilling etc.,

Chip formation:—

Various types of Chips are formed during metal cutting. the type of Chip formed during metal cutting depends on the machining condition & material to be cut.

Factors affecting the formation of Chip:—

(i) Mechanical properties of the material to be cut & the Cutting tool.

(ii) Depth of cut, Cutting speed, feed rate.

(iii) Geometry of the Tool.

(iv) Type of cutting fluid.

(v) M/cing temperature in the cutting region.

(vi) Co-eff of friction b/w w/p & Tool.

Types of Chips:—

1. Continuous Chips (or) Ribbon type Chip.
2. The discontinuous Chip (or) Segmental Chip.
3. The Continuous Chip with Built-up edge (BUE)

① Continuous Chip (or) Ribbon type Chip:-

* During cutting ductile material, a continuous ribbon like chips is produced.
* these chips are in the form of Long curl and have same thickness throughout its length.

Advantages:-

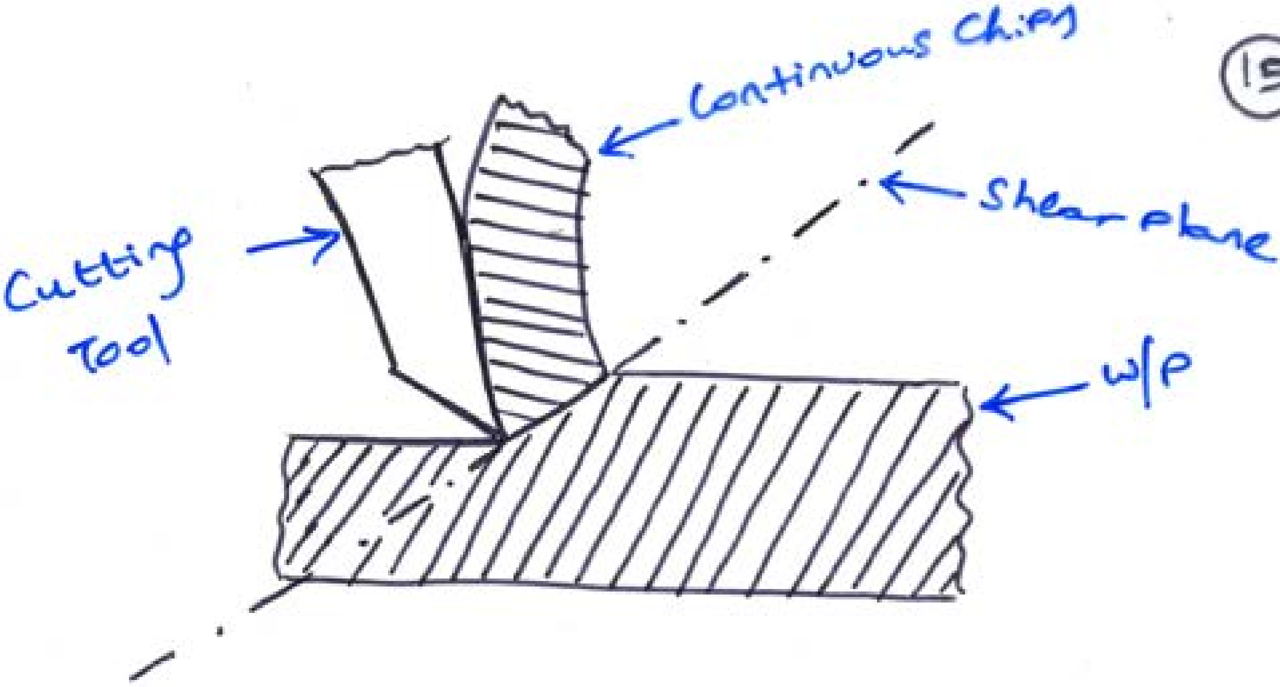
Good surface finish, Less power consumption, More tool life

DisAdvantages:-

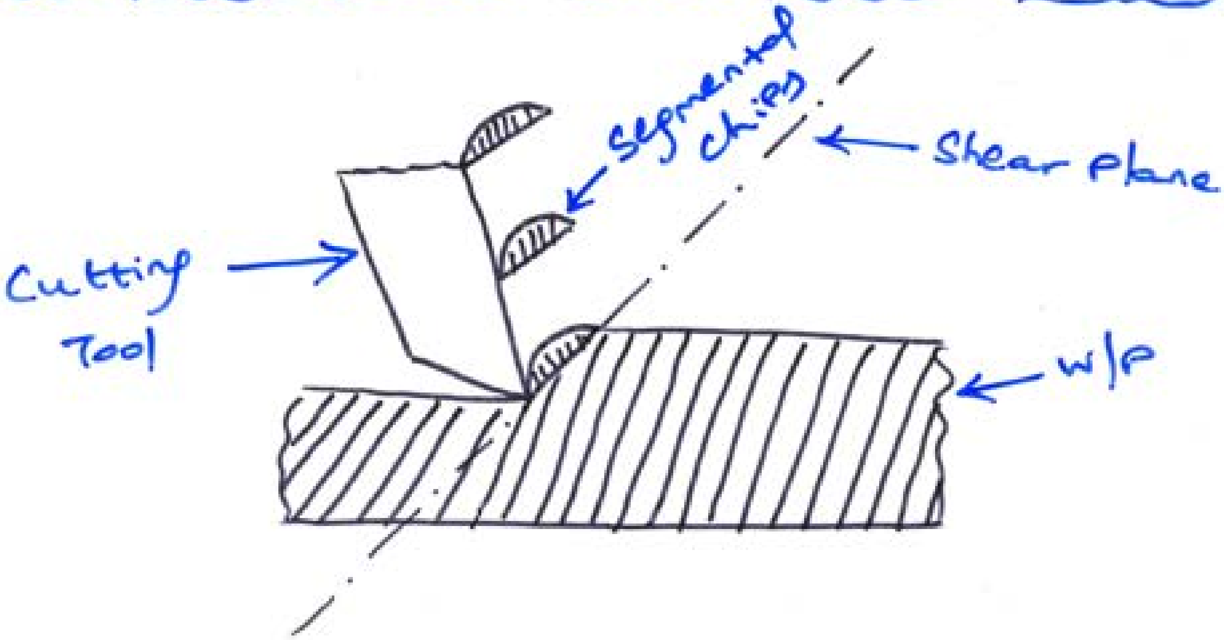
a) chip disposal is not easy
b) the surface finish of finished goods gets affected

Favourable conditions:-

① Ductile material such as low carbon steel, aluminium, copper etc.,
② Smaller depth of cut
③ High cutting speed ④ Large Rake angle.
⑤ proper cutting fluid.
⑥ Low friction b/w tool & chips.



2 Discontinuous Chips (or) Segmental Chips:-



* This type of Chips are produced while machining brittle materials such as bronze, brass & cast Iron.

* Here, Chips are produced in the form of small segmental separate pieces.

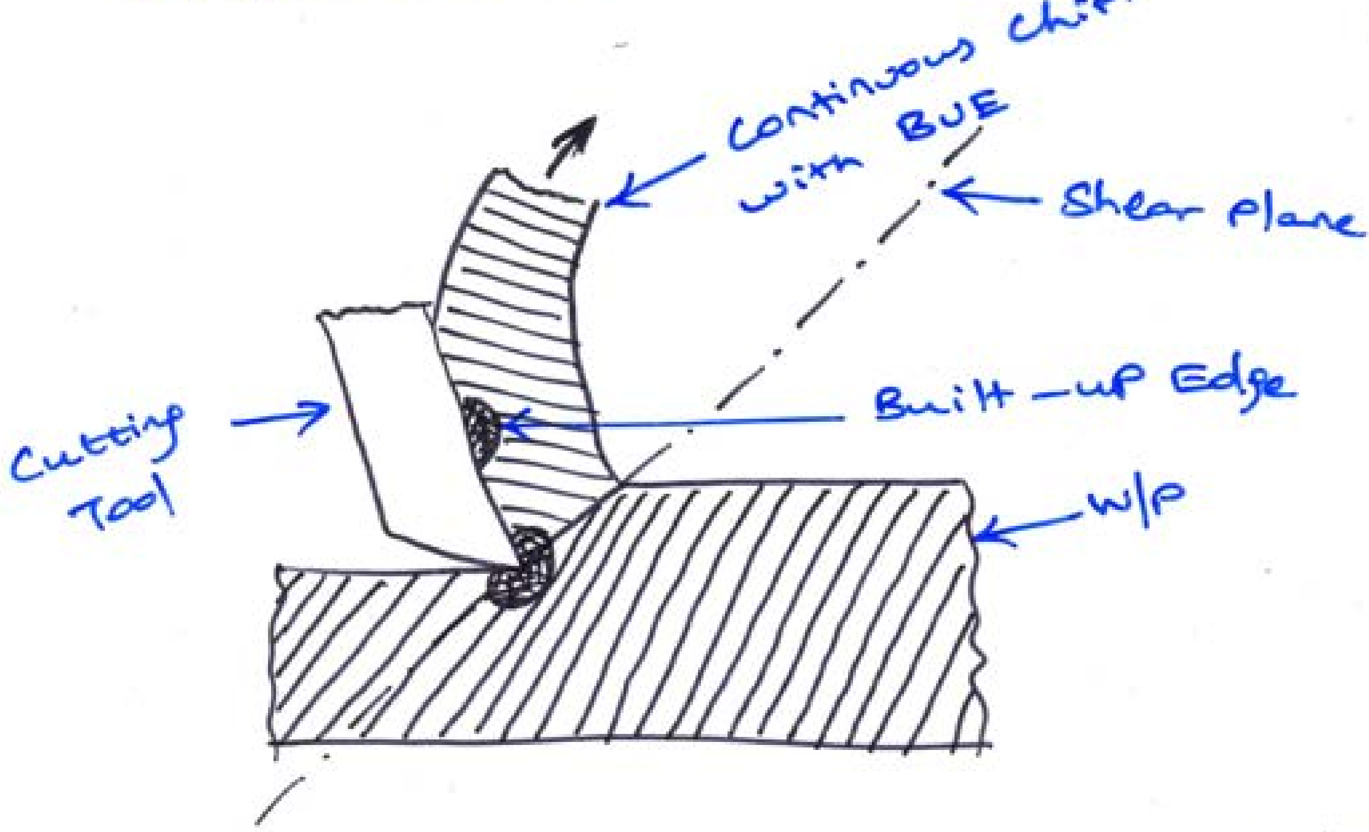
* The discontinuous chips can also be (16)
produced in cutting of ductile metals at
very low feed, high cutting speed, larger depth of
cut & high friction.

* If the discontinuous chips are produced from
the brittle material, then the surface finish
is fair, power consumption is low & tool
life is reasonable. However if it is
produced from ductile metals, then finish is
poor & the tool wear is excessive.

Conditions promoting the formation of discontinuous
chips:-

1. Machining the brittle materials
2. For ductile metals
   a) Greater depth of cut
   b) High cutting speed
   c) low feed
   d) Smaller rake angle
   e) High friction b/w w/p & tool.

(3) Continuous Chips with Built-up Edge (BUE):-



Here, the chips are continuous with Built-up edge

Built-up edge:-

During the cutting process of ductile metals, the interface temperature and pressure are quite high and hence there is a high friction b/w the tool & chip interface. It causes the chip to weld itself to the cutting edge of the tool and it will act as a false cutting edge of the tool. This extra welded metal on the cutting edge is called as "the Built-up Edge (BUE)".

The BUE is a highly strain hardened brittle material. So when the chip flows over the tool face, it will easily break & some part of it will be carried away with the chip and the remaining part of it will adhere to the work piece leading to a poor surface finish. (17-1)

Disadvantages of BUE:-

1. Poor surface finish.
2. the rake angle of the tool is altered & hence the cutting force of the tool will get affected.

Advantages of BUE :-

1. the Rake face of the tool is protected from wears due to the moving chips over the tool face. It may result in increasing the tool life

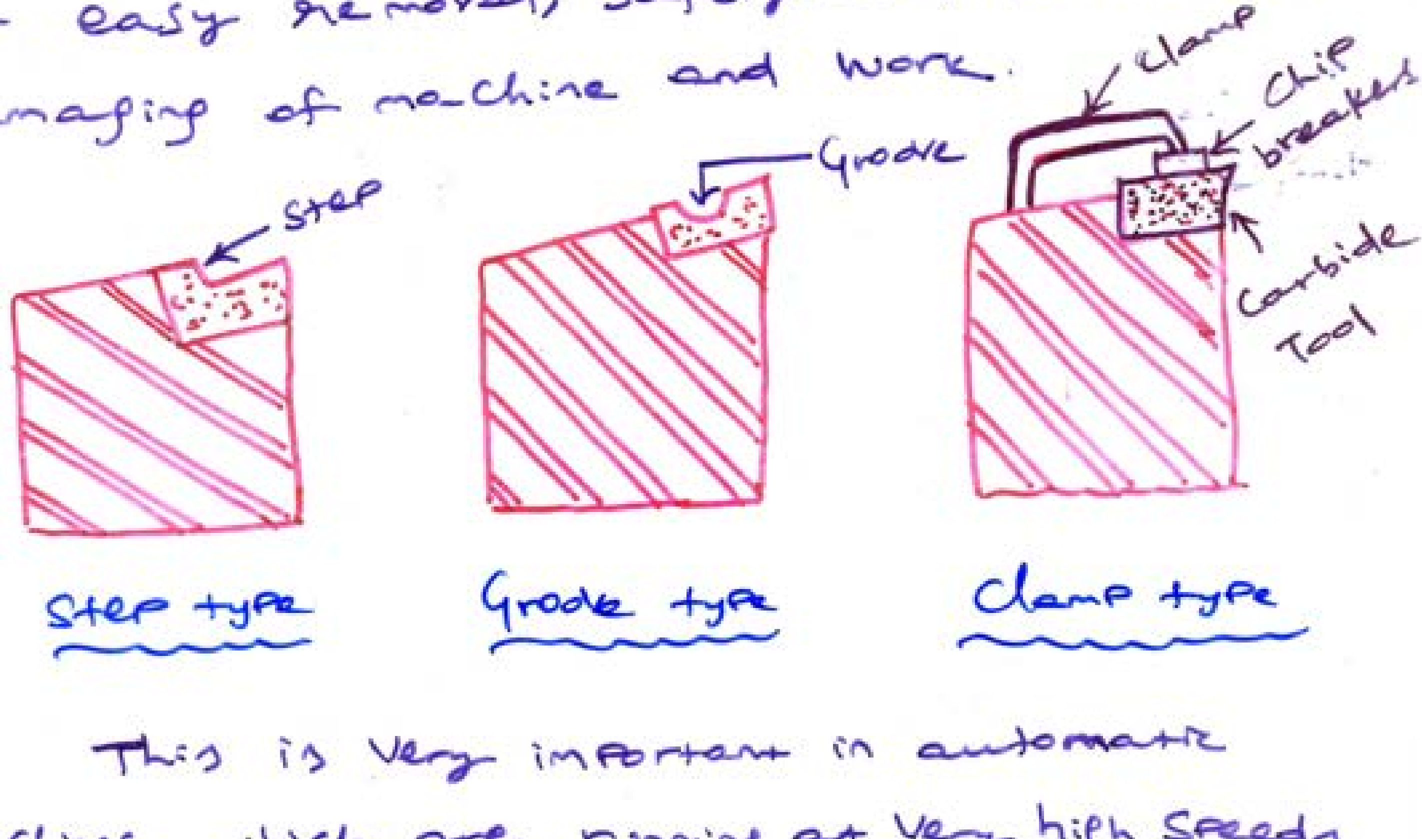
Conditions Promoting the formation of Built-up Edge (BUE):—

(17-2)

1. Low cutting speed
2. Excessive feed
3. Larger depth of cut.
4. Lack of Lubricant
5. Smaller Rake angle
6. High friction b/w the tool & w/P.

CHIP BREAKERS:-

During machining of ductile material, long and continuous chip that are formed at high cutting speed will affect the machining. It will spoil tool, work and machine. It will be difficult to remove this types of chip and also it is dangerous to safety. Chip breakers are used to break the chips into small pieces for easy removal, safety and to prevent damaging of machine and work.



Step type   Groove type   Clamp type

This is very important in automatic machines which are running at very high speeds. The chip breaker is provided on the cutting edge as shown in the figure.

The different type of chip breaker used on a cutting tool are.

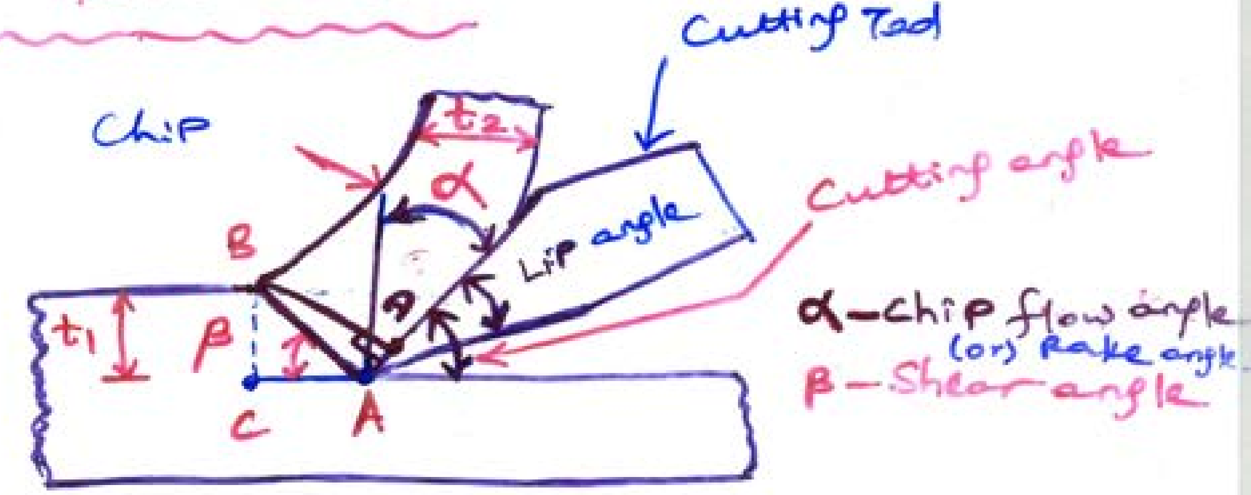
1. Step type 2. Groove type 3. Clamp type.

In Step type, a Step is ground on the tool face behind the cutting edge.

In Groove type, a groove on the tool face behind the cutting edge will break the chip.

In Clamp type, a thin chip breaker is clamped (or) screwed on the face of the tool.

Mechanism of Metal cutting (or) Mechanism of Chip formation :—



* During machining, the cutting tool exerts a shear force on the workpiece.
* The material of the w/p is stressed beyond its yield point under this shear force.
* This causes the material to deform plastically and finally get sheared off.
* The plastic flow takes place in a localized region called shear plane (or) shear zone which extends from the cutting point to

the uncut surface

* The sheared material begins to flow along the cutting tool face in the form of small pieces called chips.

* when the chip flows over the tool face, high heat is produced b/w the chip & Tool face. This temperature rise in the tool tends to soften the cutting edge and leads to failure of the cutting edge

Geometry of chip formation:-

* Generally, the chip thickness is larger than the thickness of material to be removed. Also, the metal prior to cut is longer than the length of the chip. Hence volume of metal prior to and after the cut is same

Let,

t_1 = chip thickness prior to cut

t_2 = chip thickness after cut.

* The ratio of chip thickess before cutting to chip thickness after cutting is called as 'Chip thickness ratio (r)'.

(ie) $\boxed{\text{Chip thickness ratio, } r = \frac{t_1}{t_2}}$

21

Its value is always less than one. A ratio of 1:2 yield a good result.

* The reciprocal of the chip thickness ratio is called "chip reduction coefficient" (K).

$$\boxed{\text{(ie)} \quad K = 1/r}$$

* $\left.\begin{array}{l}\text{Volume of metal}\\ \text{to be removed}\end{array}\right\} = \text{Volume of chip}$

$$t_1 \times l_1 \times b_1 = t_2 \times l_2 \times b_2$$

$$t_1 \times l_1 = t_2 \times l_2 \quad (\because b_1 = b_2)$$

$$\boxed{\therefore \frac{t_1}{t_2} = \frac{l_2}{l_1} = r = \text{Chip thickness ratio}}$$

Shear angle (β):—

From Δ ABC,

$$\sin\beta = \frac{t_1}{AB} \Rightarrow AB = \frac{t_1}{\sin\beta} \quad \text{———} \quad (1)$$

From $\triangle ABD$;

$$\sin[90-\beta+\alpha] = \frac{BD}{AB} = \frac{t_2}{AB}$$

$$\therefore \quad AB = \frac{t_2}{\sin(90-\beta+\alpha)} \quad \text{———} \quad (2)$$

Comparing (1) & (2)

$$\frac{t_1}{\sin\beta} = \frac{t_2}{\sin(90-\beta+\alpha)}$$

$$\frac{t_1}{t_2} = \frac{\sin\beta}{\sin(90-\beta+\alpha)}$$

$$\boxed{\begin{aligned} r &= \frac{\sin\beta}{\cos(\beta-\alpha)} \\ \therefore r &= \frac{\sin\beta}{\cos\beta\cos\alpha+\sin\beta\sin\alpha} \end{aligned}}$$

$$\frac{r[\cos\beta\cos\alpha+\sin\beta\sin\alpha]}{\sin\beta} = 1$$

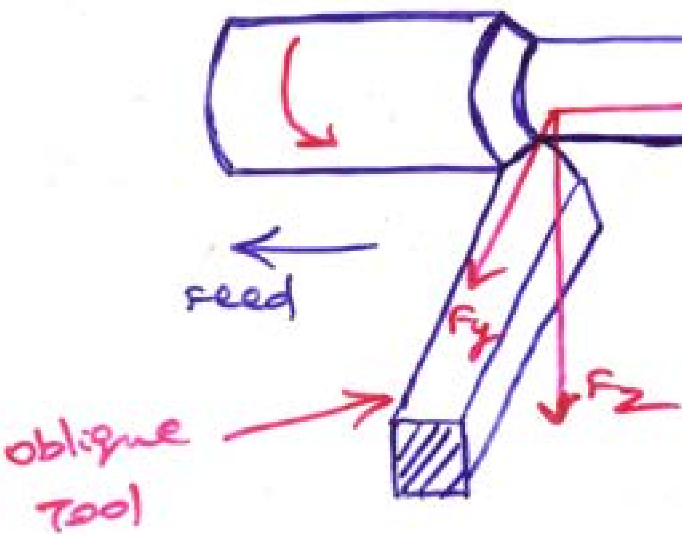
$$\frac{r\cos\alpha}{\tan\beta} + r\sin\alpha = 1$$

$$\frac{r\cos\alpha}{\tan\beta} = 1 - r\sin\alpha$$

$$\tan\beta = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

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

Cutting forces in Metal cutting:—



$F_x \rightarrow$ feed force

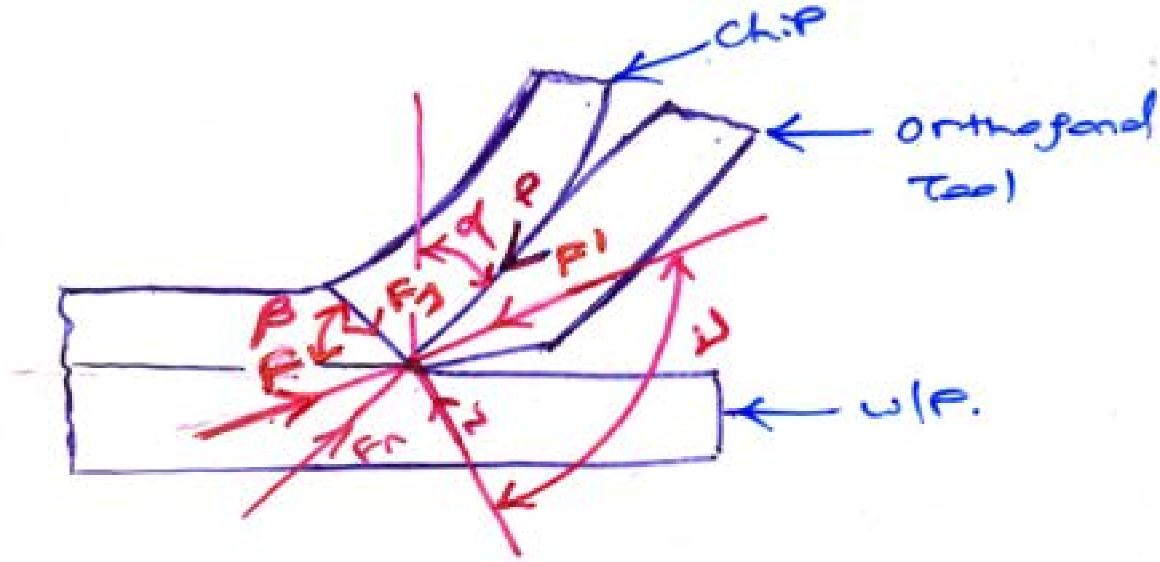
$F_y \rightarrow$ Thrust force

$F_z \rightarrow$ Cutting force

In orthogonal cuttings only two forces are there (ie) F_x - feed force & F_z - Cutting force

* Feed force (F_x) acts in a direction opposite to the feed.
* Thrust force (F_y) acts in a direction $\perp^r$ to the generated surface.
* F_z acts in a direction of cutting velocity.

The various forces acting on the tool and workpiece other than the cutting forces in orthogonal cutting:—



F, F' = Resultant forces for F_n, F_s & P, N respectively

α, β, ν = Rake angle, Shear angle, friction angle respectively.

F_s = Shear Resisting force

F_n = Back-up force on the Chip by the workpiece.

P = Frictional force along the rake face

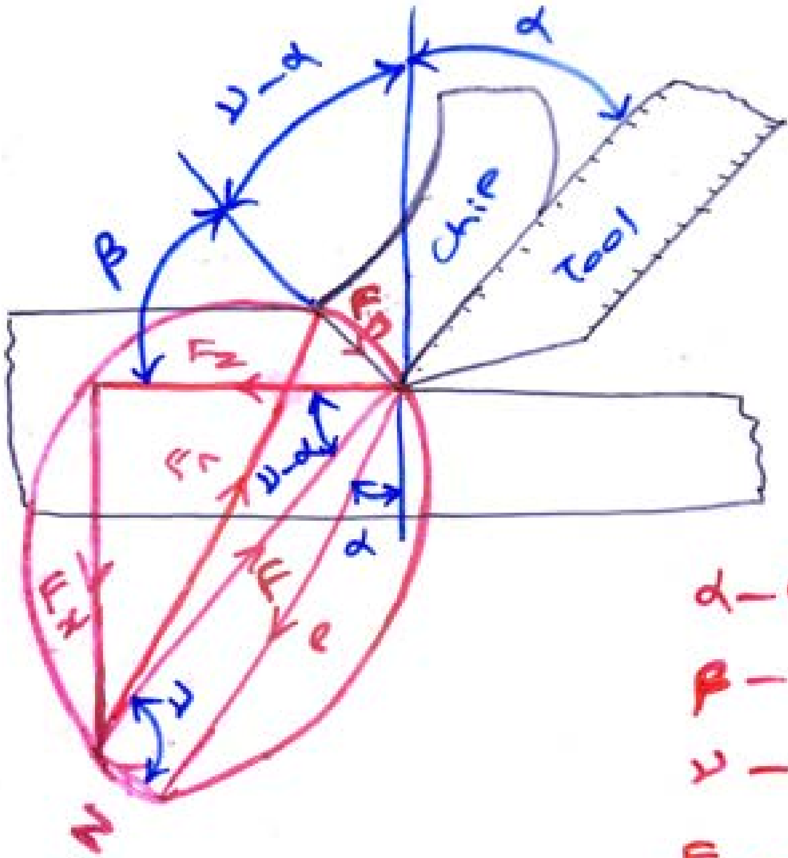
N = Normal force acting on the tool when the chip starts flowing over its rake face.

Merchant Circle Diagram of forces:-

This diagram is used to find the relation b/w various forces and angles.

The following assumptions are made while drawing the merchant circle Diagram.

1. The tool is perfectly Sharp and has no contact along the clearance face.
2. The Surface where Shear is occurring is considered as a plane.
3. The cutting edge is $\perp^r$ to the cutting Velocity Vector & generates a plane Surface as the work moves past it.
4. The chip doesn't flow to either side (or) no side spread.
5. Uncut chip thickness is constant.
6. The width of the tool is greater than the width of the w/p.
7. A continuous chip is produced without any Build-up edge (BUE).
8. The stresses on a shear plane are uniformly Distributed.



α – Rake angle

β – Shear angle

ν – Friction angle

F_z – cutting force

F_x – Feed force

* Two force triangles is combined together. their resultant will be given by the resultant force 'F'.

* Feed force (F_x) & cutting force (F_z) are found out by tool force dynamometer & is drawn to the suitable scale. their resultant will be given by the resultant force 'F'.

* Merchant circle is drawn with the diameter "F" & it is passing through the tool point.

Co-eff of friction (μ):—

when the chip slides over the tool face under pressure, there will be some friction b/w these two. Therefore Kinematic Co-eff of friction can be expressed as

$$\boxed{\mu = P/N = \tan\lambda}$$

Shear angle (β):—

$$\boxed{\tan\beta = \frac{r\cos\alpha}{1 - r\sin\alpha}}$$

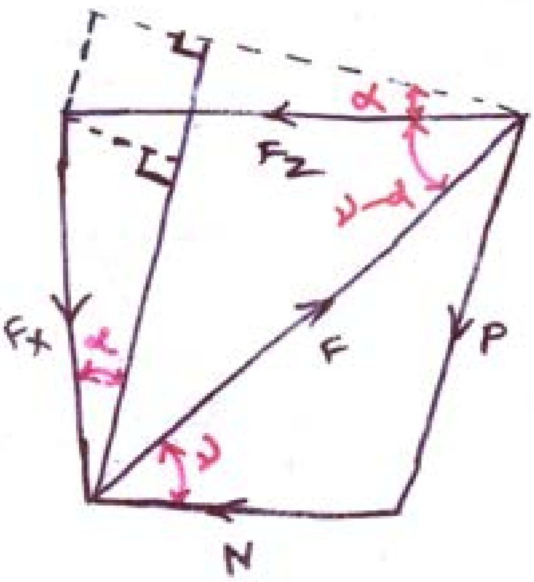
Chip-thickness ratio (r):—

$$\boxed{r = \frac{t_1}{t_2}}$$

$t_1 \Rightarrow$ uncut chip thickness

$t_2 \Rightarrow$ Chip thickness which was cut.

Frictional force (P) & Normal force (N):—



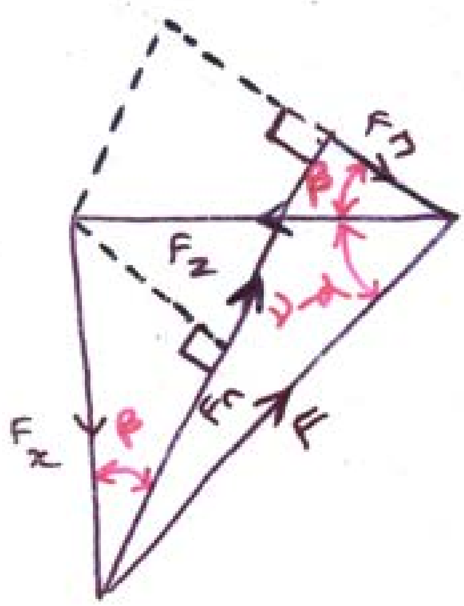
$P = F_x \cos\alpha + F_z \sin\alpha$

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

F = Resultant force

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

Shear Resisting force (F_s) & Back-up force (F_n):—



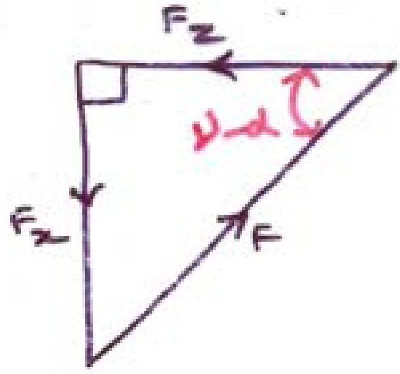
$F_s = F_z \cos\beta - F_x \sin\beta$

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

Cutting force (F_z):-

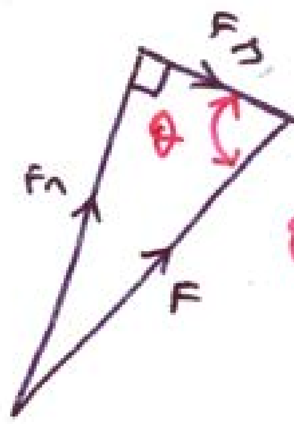
$$\cos(\nu-\alpha) = \frac{F_z}{F}$$

$$\boxed{F_z = F\cos(\nu-\alpha)}$$



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

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



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

Substituting 'F' in F_z equation

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

$$\boxed{F_z = \frac{F_s\cos(\nu-\alpha)}{\cos(\nu-\alpha+\beta)}}$$

Coeff of friction (μ):-

$$\mu = \tan\nu = F/N$$

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

$\div$ by $\cos\alpha$,

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

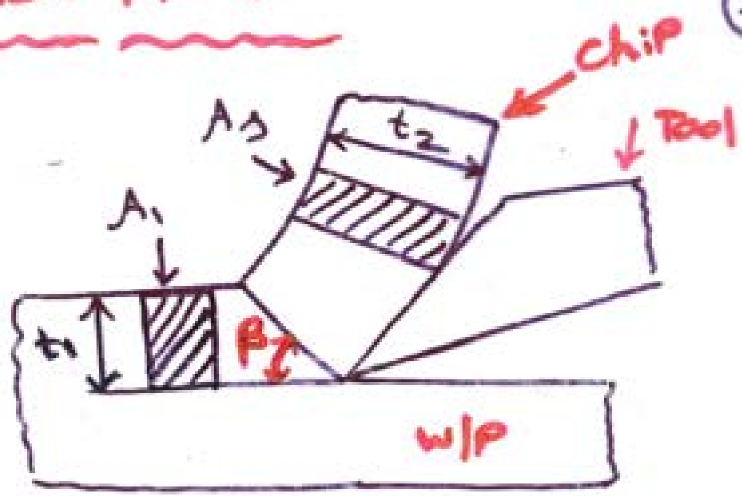
Shear Stress in Shear plane:—

Let

A_1 – Area of uncut chip

A_s – Shearing Area

τ – Shear Stress



$$\text{Shear stress, } \tau = \frac{\text{Shear force}}{\text{Area}} = \frac{F_s}{A_s}$$

$$\sin\beta = \frac{A_1}{A_s} \Rightarrow A_s = \frac{A_1}{\sin\beta}$$

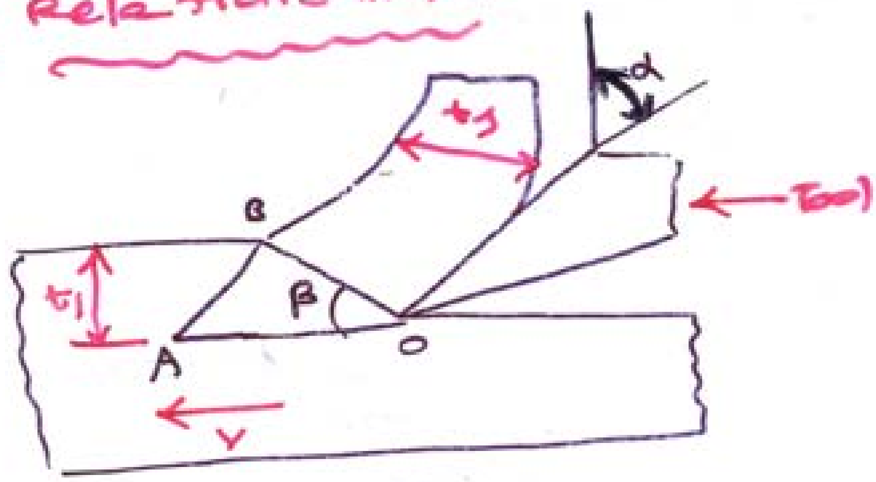
$$\therefore \tau = \frac{F_s \sin\beta}{A_1}$$

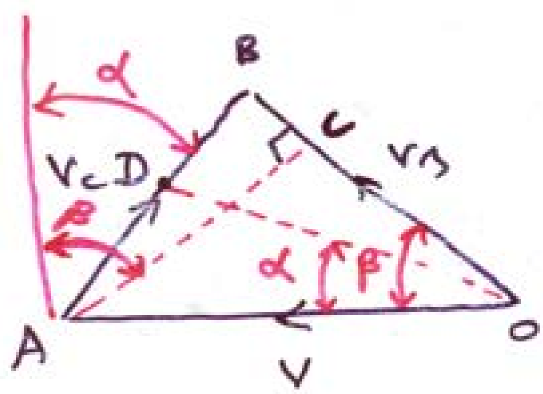
$$\tau = \frac{(F_z \cos\beta - F_x \sin\beta)\sin\beta}{A_1}$$

$$\boxed{\left.\begin{array}{l}\text{Shear}\\ \text{stress}\end{array}\right\} \quad \tau = \frac{F_z \cos\beta \sin\beta - F_x \sin^2\beta}{A_1}}$$

Velocity Relationship:—

(31)





The above figure shows the velocity relationship in orthogonal cutting. The various velocities are

1. Cutting Velocity (V)
2. Chip Velocity (V_c)
3. Shear Velocity (V_s)

* The chip velocity is the velocity of chip relative to the tool & is directed along the tool face.

* Shear velocity v_s is the velocity of chip relative to the w/p and is directed along the shear plane.

32

From OCA

$$\sin\beta = \frac{AC}{OA}$$

$$\therefore AC = OA \sin\beta$$

$$\boxed{AC = V \sin\beta} \quad \text{--- (1)}$$

From BAC

$$\cos(\beta-\alpha) = \frac{AC}{AB}$$

$$\boxed{\therefore AC = v_c \cos(\beta-\alpha)} \quad \text{--- (2)}$$

From (1) & (2),

$$V \sin\beta = v_c \cos(\beta-\alpha)$$

Chip Velocity

$$\boxed{\therefore V_c = \frac{V \sin\beta}{\cos(\beta-\alpha)}}$$

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

Similarly,

From ODA,

$$\cos\alpha = \frac{OD}{OA} = \frac{OD}{V}$$

$$\boxed{OD = V \cos\alpha} \quad \text{--- (3)}$$

From ODB,

$$\cos(\beta-\alpha) = \frac{OD}{OB} = \frac{OD}{V_s}$$

$$OD = V_s \cos(\beta - \alpha) \quad \text{--- (4)}$$

From (3) & (4),

$$V \cos\alpha = V_s \cos(\beta - \alpha)$$

Shear velocity

$$\boxed{\therefore V_s = \frac{V \cos\alpha}{\cos(\beta - \alpha)}}$$

Formulae used in metal cutting:–

(1) Chip thickness ratio, $r = t_1/t_2 = l_2/l_1 = \frac{\sin\beta}{\cos(\beta-\alpha)}$

(2) Chip reduction co-efficient, $K = 1/r$

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

(4) Co-eff of friction, $\mu = \tan\nu = P/N$

(5) Friction force, $P = F_z \sin\alpha + F_x \cos\alpha$

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

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

Back-up force, $F_n = F_x \cos\beta + F_z \sin\beta$

(7) Shear stress, $\tau = \frac{F_s \sin\beta}{A_1}$ $A_1 \rightarrow$ Area of uncut chip

(8) cutting force, $F_z = F \cos(\nu - \alpha)$

$$= \frac{F_s \cos(\nu - \alpha)}{\cos(\nu - \alpha + \beta)}$$

(9) feed force $F_x = F \sin(\nu - \alpha)$

$$= \frac{F_s \sin(\nu - \alpha)}{\cos(\nu - \alpha + \beta)}$$

(10) velocity of chip $V_c = \dfrac{v \sin\beta}{\cos(\beta - \alpha)}$

(11) Shear velocity $v_s = \dfrac{v \cos\alpha}{\cos(\beta - \alpha)}$

(12) shear strain, $e = \cot\beta + \tan(\beta - \alpha)$

(13) $$\begin{bmatrix}\text{Total work}\\ \text{done in}\\ \text{metal cutting}\end{bmatrix} = \begin{bmatrix}\text{work done}\\ \text{in shearing}\\ \text{of material}\end{bmatrix} + \begin{bmatrix}\text{work done}\\ \text{against}\\ \text{friction}\end{bmatrix}$$

$$F_z \times v = (F_s \times v_s) + (P \times v_c) - \left(\frac{N-m}{sec}\right)$$

(14) Material removal rate for turning operation [M.R.R]

$$= \begin{bmatrix}\text{Feed} \\ \text{in mm/rev}\end{bmatrix} \times \begin{bmatrix}\text{Depth of} \\ \text{cut in} \\ \text{mm}\end{bmatrix} \times \begin{bmatrix}\text{Cutting speed} \\ \text{in mm/min}\end{bmatrix}$$

$$- \text{mm}^3/\text{min}$$

(15) Machining constant $(C_m) = 2\beta + \gamma - \alpha$

In metal cutting,

Feed in mm/rev = uncut chip thickness

$= t_1$

Depth of cut = width of cut.

M.R.R is defined as the volume of material removed from the workpiece per unit time. It is measured in terms of mm^3/min.

Problems on metal cutting:-

① During an orthogonal turning opreation of C20 steel, the following data were recorded

Rake angle $=10^\circ$; Chip thickness $=0.48$mm;

width of cut $=2.0$ mm; Feed $=0.25$mm/rev;

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

Find the value of shear force at the shear plane; find also the kinematic co-eff of friction at the chip-tool interface.

Given Data:-

$\alpha = 10^\circ$; $t_2 = 0.48$mm;

$b = 2.0$mm; $f = 0.25$ mm/rev;

$F_z = 1200$N; $F_x = 300$N;

$V = 2.5$m/sec;

To find:-

$F_s = ?$ $\mu = ?$

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

Co-eff of friction $\Big\}$ $\mu = \tan\nu = \frac{F}{N} = \frac{\text{frictional force}}{\text{Normal force}}$

Shear angle (β):-

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

Chip thickness ratio $\Big\}\; r = \dfrac{t_1}{t_2}$

$$= \frac{0.25}{0.48}$$

$$\boxed{r = 0.5208}$$

$$\beta = \tan^{-1}\left[\frac{0.5208 \times \cos 10^{\circ}}{1 - (0.5208 \times \sin 10^{\circ})}\right]$$

$$\boxed{\beta = 29.41^{\circ}}$$

Shear force (F_s):-

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

$$= 1200\cos(29.41) - 300\sin(29.41)$$

$$\boxed{F_s = 898.03\ N}$$ -Ans-

Frictional force (F):-

$$F = F_z \sin\alpha + F_x \cos\alpha$$

$$= 1200(\sin 10^{\circ}) + 300(\cos 10^{\circ})$$

$P = 503.82 N$

Normal force (N):—

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

$$= 1200 \cos 10 - 300 \sin 10$$

$$= 1129.67 N$$

Coeff of friction (μ) :—

$$\mu = P/N = \frac{503.82}{1129.67}$$

$$\boxed{\mu = 0.44}$$

—Ans—

Friction angle (ν):—

$$\mu = \tan\nu$$

$$\therefore \nu = \tan^{-1}(\mu) = 24.03^\circ$$

---

(2) 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 = 1290N

Vertical component of cutting force = 1650N

From Merchant's theory, calculate the various components of the cutting forces and the coefficient of friction at the chip-tool interface. (39)

Given Data:-

chip thickness ratio, $r = 0.3$

Horizontal component force, $F_z = 1290N$

Vertical component cutting force, $F_x = 1650N$

Find:-

frictional force $(F) = ?$ normal force $(N) = ?$

shear force $(F_s) = ?$ Back-up force $(F_n) = ?$

co-eff of friction $(\mu) = ?$

Same as the previous problem.

---

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

(i) Find the shear angle & shear strength

(ii) If $\mu = 0.5$, what is machining constant $C_m = ?$

Given :-

40

width, $b = 2.5mm$; uncut chip thick, $t_1 = 0.25mm$;

Chip thickness, $t_2 = 1.25mm$; Rake angle, $\alpha = 0°$;

Cutting force, $F_z = 900N$; Thrust force, $F_x = 810N$;

co-eff of friction, $\mu = 0.5$

Find:-

Shear angle, $\beta = ?$ Shear Strength, $\tau = ?$

Machining constant, $C_m = ?$

Soln:-

① Chip thickness ratio, $r = \frac{t_1}{t_2} = \frac{0.25}{1.25} = 0.2$;

② Shear angle, $\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1-r\sin\alpha}\right]$

$$= \tan^{-1}\left[\frac{0.2\cos 0}{1-0.2\sin 0}\right]$$

$$\boxed{\beta = 11.31°}$$

-Ans-

③ Shear Strength (τ):-

$$\tau = \frac{F_s}{A_1} \times \sin\beta$$

$A_1 = b \times t_1$

$$\boxed{A_1 = 2.5 \times 0.25 \ mm^2}$$

A_1 = Area of uncut Chip

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

$$= 900\cos(11.31^\circ) - 810\sin(11.31^\circ)$$

$$F_s = 723.66\,N.$$

$$\therefore \tau = \frac{F_s}{A_1} \times \sin\beta$$

$$= \frac{723.66}{(0.25 \times 2.5)} \times \sin(11.31^\circ)$$

$$\boxed{\tau = 227.07\ N/mm^2}$$

Machining constant, $C_m = 2\beta + \nu - \alpha$

Friction angle, $\nu = \tan^{-1}(\mu)$

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

$$\boxed{\nu = 26.56^\circ}$$

$$\therefore C_m = 2(11.31^\circ) + 26.56^\circ - 0^\circ$$

$$\boxed{C_m = 49.18^\circ}$$

Tool Signature

Tool angles given in a definite pattern is called "Tool Signature". The Tool angles have been standardized by the American Standards Association (ASA). As per ASA, the tool signature is given by the following order.

(i) Back Rake angle
(ii) Side Rake angle
(iii) End relief angle
(iv) Side relief angle
(v) End cutting edge angle
(vi) Side cutting edge angle
(vii) nose radious.

example :- A tool having 8, 8, 5, 5, 6, 6 and 1 as signature in ASA System will have

Back rake angle = 8°
Side rake angle = 8°
End relief angle = 5°
Side relief angle = 5°
End cutting edge angle = 6°
Side cutting edge angle = 6°
Nose Radious = 1 mm.

8. Orthogonal Test is conducted on mild steel tube of size 225mm diameter and 5mm thickness. The velocity of cutting is 105m/min and feed rate is 0.21mm/rev. The following observations were made (43)

Cutting force = 2300N; Feed force = 825N;

Chip thickness = 0.3mm; Rake angle = -10°;

Determine the shear strain and energy per unit volume.

Given:-

$D = 225mm$; $t = 5mm$; $V = 105 m/min$; $F = 0.21 mm/rev$;

$F_z = 2300N$; $F_x = 825N$; $t_2 = 0.3mm$; $\alpha = -10°$

Soln:-

① Chip thickness ratio, $r = \frac{t_1}{t_2} = \frac{0.21}{0.3} = 0.7$

② Shear angle, $\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1 - r\sin\alpha}\right] = 31.57°$

③ Shear strain, $e = \cot\beta + \tan(\beta - \alpha) = 2.513$

④ Shear velocity, $V_s = \frac{V\cos\alpha}{\cos(\beta-\alpha)} = \frac{105\cos(-10)}{\cos[31.57-(-10)]}$

$V_s = 138.2 m/min$

⑤ Shear energy / unit volume $= \frac{\tau \times V_s}{V \times \sin\beta}$

[shear stress] $= \frac{761.67 \times 138.2}{105 \times \sin 31.57}$

$= 1914.87 N/mm^2$

(9) Mild steel is being machined orthogonally & the following results are obtained

Cutting force = 1000N; feed force = 625N;

cutting velocity = 165 m/min; rake angle = 10°; (4-4)

shear angle = 19°;

Determine the following (i) shear velocity (ii) chip flow velocity (iii) work done per min in shearing the metal and work done against friction (iv) show that the work input is equal to sum of work done in shearing and against friction.

Given:-

$F_z = 1000N$; $F_x = 625N$; $v = 165\ m/min$;

$\alpha = 10°$; $\beta = 19°$

Find:- v_s, v_c, W_s, W_f

Soln:-

① shear velocity, $v_s = \dfrac{v \cos\alpha}{\cos(\beta-\alpha)} = 164.5\ m/min$

② chip velocity, $v_c = \dfrac{v \sin\beta}{\cos(\beta-\alpha)} = 54.38\ m/min$

③ Shear force, $F_s = F_z \cos\beta - F_x \sin\beta$

$F_s = 742N$

④ work done in shear, $W_s = F_s \times v_s = 742 \times 164.5$

$W_s = 122 \times 10^3\ N\text{-}m/min$

$\therefore \boxed{W_s = 2033.33 \text{ watts}}$

(5) Frictional force, $F = F_2 \cos\alpha + F_z \sin\alpha$ (45)

$$\boxed{F = 789.15N}$$

(6) Workdone in friction, $W_f = F \times V_c$

$$= 715.23 \text{ watts}$$

(7) Work input, $W = F_z \times V$

$$= 1000 \times 165$$

$$= 165000 \text{ N-m/min}$$

$$= \frac{165000}{60} = 2750 \text{ watts}$$

$$\boxed{W = 2750 \text{ watts}}$$

(8) $W_s + W_f = 715.23 + 2033.33$

$$\boxed{W_s + W_f = 2748.56 \text{ watts}}$$

$\therefore \boxed{W \simeq W_s + W_f}$ Hence Proved.

(10) While turning a C15 steel rod of 160mm dia at 315rpm, 2.5mm depth of cut and feed rate of 0.16 mm/rev by tool geometry 0, 10, 8, 9, 15, 75, 0, the following observations were made. (46)

Tangential component (P_z) of the cutting force = 500N

Radial component (P_x) of the cutting force = 200N

Chip thickness = 0.48 mm

(a) Draw schematically the merchant's circle diagram for the cutting forces in the present case.

(b) Determine the dynamic yield strength of the present work material.

Given:-

$D = 160mm$; $N = 315rpm$;

depth of cut = width of uncut chip

$$\boxed{b = 2.5mm}$$

feed rate = thickness of uncut chip

$$\boxed{t_1 = 0.16\ mm}$$

Rake angle, $\alpha = 0°$ [from tool geometry]

$F_z = 500N$; $F_x = 200N$

Chip thickness, $t_2 = 0.48mm$;

Soln:-

Dynamic yield Strength, $\tau = \dfrac{F_s \times \sin\beta}{A_1}$ (47)

where β = Shear angle, A_1 = Area of uncut chip

F_s = Shear force

① Chip thickness ratio (r):-

$$r = \frac{t_1}{t_2} = \frac{0.16}{0.48} = 0.33$$

② Shear angle (β):-

$$\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1-r\sin\alpha}\right] = 18.43^\circ$$

③ Shear force (F_s):-

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

$$\boxed{F_s = 411\,N}$$

④ Yield strength (τ):-

$$\tau = \frac{F_s \times \sin\beta}{A_1} = \frac{411 \times \sin(18.43)}{b_1 \times t_1}$$

$$= \frac{411 \times \sin(18.43)}{2.5 \times 0.16}$$

$$\boxed{\tau = 324.92\ N/mm^2}$$

(11) During machining C 20 steel with a carbide cutting tool having a tool geometry given by (48)

0 − 5 − 6 − 6 − 8 − 75 − 1 mm ORS, the following forces have been recorded by a two dimensional dynamometer:

Cutting force = 1300 N

Feed force = 800 N

Find (1) Frictional force (2) Normal force (3) μ

Here Rake angle, $\alpha = 5^\circ$

(12) The following data from an orthogonal cutting test is available

Rake angle = 5°

Chip thickness ratio = 0.383

Uncut chip thickness = 0.5 mm

width of cut, b = 3 mm;

Yield stress of material in shear = 280 N/mm²

Average coeff of friction on the tool face = 0.7

Determine the normal & tangential forces on the tool face.

Given:—

$\alpha = 15°$; $r = 0.383$; $t_1 = 0.5mm$; (49)

$b = 3mm$; $\tau = 280 N/mm^2$; $\mu = 0.7$

Find:—

Tangential force (F_z) = ? normal force (F_x) = ?

Soln:—

$$\beta = \tan^{-1}\left[\frac{r\cos\alpha}{1 - r\sin\alpha}\right] = 22.3°$$

$$\tau = \frac{F_s \times \sin\beta}{A_1} = 280 N/mm^2;$$

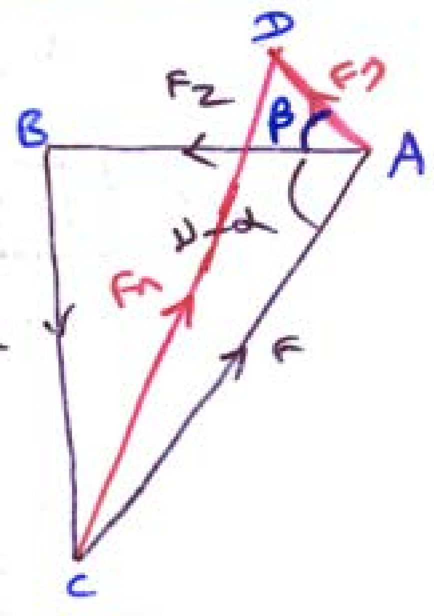
$$\boxed{F_s = 1105.63 N}$$

$\mu = 0.7$ (Given)

$$\boxed{\nu = \tan^{-1}(\mu) = 35°}$$

Tangential force (or) cutting force (F_z):—

From ΔABC



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

$$\boxed{F_z = F\cos(\nu - \alpha)}$$

From ΔADC

$$\cos(\nu - \alpha + \beta) = \frac{F_s}{F}$$

$\therefore$ Resultant force, $F = \dfrac{F_y}{\cos(\nu - \alpha + \beta)}$

$= \dfrac{1105.63}{\cos(35 - 15 + 22.3^\circ)}$

$\boxed{F = 1494.83\,N}$

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

$= 1494.83 \cos(35 - 15)$

$\boxed{F_z = 1404.68\,N}$ —Ans—

Feed force (F_x) (or) normal force :—

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

$\therefore F_x = \sqrt{F^2 - F_z^2}$

$= \sqrt{1494.83^2 - 1404.68^2}$

$\boxed{F_x = 511.26\,N}$

Tool Life:—

* It is defined as the interval of time for which tool works satisfactorily b/w two consecutive resharpenings (Grindings).

* Tool life is used to calculate the tool material performance and machinability of the workpiece material.

* Tool life can be expressed in the following way.

✓ Time period in minutes b/w two consecutive resharpenings.

✓ Number of components machined b/w two consecutive Grindings

✓ **Volume of material removed b/w two consecutive Grindings.**

Tool life in terms of Volume of material removal:—

WKT $\left.\begin{array}{r}\text{Volume of material} \\ \text{removed / min}\end{array}\right\} = \pi D N t f \ mm^3/min$

where,

D – w/p diameter in mm

t – Depth of cut in mm

f – feed rate in mm/rev

N – No of revolution/min

If Tool life (T) in minutes, then

$$\left.\begin{array}{l}\text{total volume of material}\\ \text{removed for tool failure}\end{array}\right\} = \pi D N t f T \ mm^3$$

(or)

$$= (V \times 1000)\, t f T$$

$$\therefore \text{Tool life } (T) = \frac{\text{Volume of material removed for Tool failure}}{(V \times 1000)\, t f}$$

Factors affecting the Cutting Tool life:—

(a) Cutting speed (b) feed & Depth of cut

(c) Tool Geometry (d) Tool material

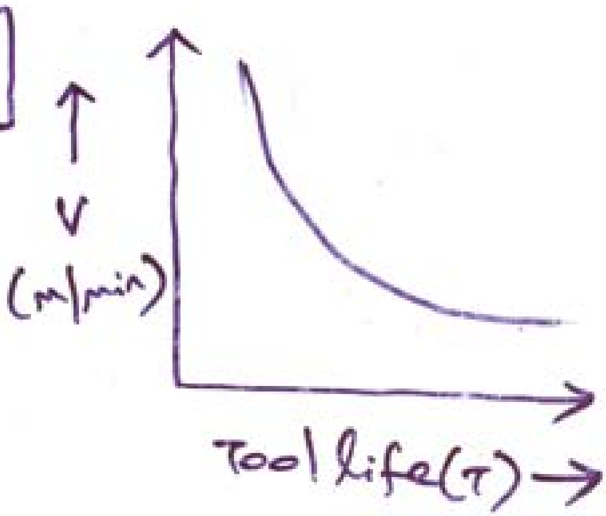
(e) work material (f) Nature of cutting

(g) Rigidity of m/c Tool & work (h) Cutting fluids

(a) Cutting Speed:-

* Major factor affecting the tool life.
* It varies inversely with the tool life which leads to the generation of parabolic curve.
* the relation b/w the tool life and cutting speed is invented by F.W. Taylor & hence Sometimes this equation is called as "Taylor's tool life equation".

$$\boxed{VT^n = C}$$



where

V – Cutting speed (m/min)

T – Tool life

C – Constant

n – Tool life index, which depends on tool material.

n = 0.1 for HSS, 0.2-0.25 for Carbide Tools, 0.4-0.55 for Ceramic tools

(b) Feed & Depth of cut:-

* Important parameter affecting the tool life
* They are inversely proportional to tool life.

$$\boxed{V = \frac{257}{T^{0.19} \times f^{0.36} \times t^{0.8}} \text{ m/min}}$$

V – Cutting speed in m/min

T – Tool life in min

f – feed in mm/rev

t – Depth of cut in mm.

V can also expressed as

$$\boxed{V = \frac{c}{f^{a} \times t^{b}}}$$

f – feed in mm/min

t – Depth of cut in mm.

c – Constant.

a, b – Constant which depends on mechanical properties of workpiece material.

(C) Tool Geometry:–

* Tool angles of the cutting Tool affect the tool performance & Tool life.
* If the rake angle is increased in a positive direction, then the cutting force and heat generation IS reduced which increases

the tool life whereas if it is too large, then it reduces the mechanical strength of the tool and hence tool life.

* Hence the rake angle should be in between −5 to +10° where − sign indicates negative rake angle

55

* Relief (or) clearance angle are used to prevent the rubbing of the tool flank against the w/p & if it is too large, weakening of the tool occurs. Generally it varies from 5° to 8°.

* Cutting edge angles also affect the tool life in the similar way. Hence it is kept in between 5° to 8°.

* Nose radious increases the abrasion which helps in improving the surface finish, tool strength & hence tool life

(d) Tool material:-

* Tool material which can withstand maximum cutting temperature without losing its mechanical properties & geometry will ensure maximum Tool life

* Hence higher the mechanical properties (mainly hardness and toughness) in the tool material, longer be the tool life.

(e) Work material:-

Higher the hardness of the work material, Greater will be the tool wear and hence shorter tool life

(f) Nature of Cutting:-

* Tool life is also affected by cutting nature (ie) whether it is continuous (or) intermittent.

* In continuous cutting, the tool life is more than the intermittent cutting.

(g) Rigidity of the m/c tool & work:-

* M/c Tool & w/p both should remain rigid while machining operation.
* If they are not rigid, then vibrations are developed and hence cutting tool will be subjected to intermittent cutting instead of continuous cutting and hence tool life will be shorter.

(f) Cutting fluids:-

* for efficient performance of the opreation, cutting fluids are used.
* cutting fluids reduces the heat b/w the tool & the work, reduces the friction, improves surface finish, helps in removal of chips & hence increases the tool life.

① The Taylorian tool-life equation for machining C-40 steel with a 18:4:1 H.S.S cutting tool at a feed of 0.2 mm/min and a depth of cut of 2 mm is given by $VT^n = C$, where n and c are constants. The following V and T observations have been noted.

V (m/min)	25	35	(58)
T (min)	90	20	

Calculate:-

(i) n and c

(ii) Hence recommend the cutting speed for a desired tool life of 60 minutes.

Soln:-

$$V_1 = 25 \text{ m/min} ; \quad V_2 = 35 \text{ m/min}$$

$$T_1 = 90 \text{ min}; \quad T_2 = 20 \text{ min};$$

Taylor's equation:- $VT^n = \text{Constant}$

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

$$25 (90)^n = 35 (20)^n$$

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

$$(4.5)^n = 1.4$$

$$n \log 4.5 = \log 1.4$$

59

$$n = \frac{\log 1.4}{\log 4.5} = 0.223$$

$$\boxed{n = 0.223}$$ — Ans —

Constant, $C = V_1 T_1^n = V_2 T_2^n$

$$C = 25(90)^{0.223}$$

$$\boxed{C = 68.19}$$ — Ans —

Cutting speed (v) for T = 60 minutes :—

$$v T^n = C$$

$$v(60)^{0.223} = 68.19$$

$$\boxed{v = 27.36 \text{ m/min}}$$ — Ans —

(2) During straight turning of a 24mm diameter steel bar at 300 r.p.m with an H.S.S. tool, a tool life of 9 min was obtained. when the same bar was turned at 250 r.p.m, the tool life increased to 48.5 min what will be the tool life at a speed of 280 r.p.m?

60

Given:-

Diameter of bar, $D = 24mm$;

Speed of bar, $N_1 = 300 r.p.m$;

Tool life, $T_1 = 9 min$

Speed of bar $N_2 = 250 r.p.m$;

Tool life, $T_2 = 48.5 min$;

Find:-

Tool life $T = ?$ when $N = 280 r.p.m$

Soln:-

(1) Speed in m/min (V):-

For $N_1 = 300 r.p.m$, $\boxed{V_1 = \frac{\pi D N_1}{1000} m/min}$

$\boxed{V_1 = 22.61 m/min}$

For $N_2 = 250 r.p.m$, $\boxed{V_2 = 18.84 m/min}$

For $N = 280 r.p.m$, $\boxed{V = 21.11 m/min}$

(2) To find n and c in Taylor's eqn:-

Taylor's eqn:- $\boxed{VT = C}$

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

$$(22.61)(9) = (18.84)(48.5)^n$$

$$\left(\frac{9}{48.5}\right) = \frac{18.84}{22.61}$$

$$(0.1855)^n = 0.8332$$

$$n \log(0.1855) = \log(0.8332)$$

$$\boxed{\therefore n = 0.1\ 83}$$

Constant, $c = V_1 T_1^n$

$$= 22.61\,(9)^{0.1083}$$

$$\boxed{c = 28.68}$$

(3) Tool life (T) when N = 280 r.p.m:-

$$VT^n = c$$

$$21.11\,(T)^{0.1083} = 28.68$$

$$\boxed{T = 16.93\ \text{min}}$$

$$T^{0.1083} = (28.68\ \ 21.11)$$

$$T = (28.68/21.11)^{1/0.1083} = 16.93\ \text{min}$$

(3) A cutting Tool when used for machining w/p at a cutting speed of 50 m/min lasted for 100 minutes. Taking $n = 0.26$ in the Taylor's tool life equation, determine (1) the life of the tool for an increase in cutting speed by 25%. (2) the cutting speed to obtain a tool life of 180 minutes.

62

Soln:-

$$V = 50\ m/min\ ;\ T = 100\ min;\ n = 0.26$$

$$\therefore\quad V T^n = \text{constant (Taylor's eqn)}$$

$$50\,(100)^{0.26} = 165.56$$

$$\boxed{C = 165.56}$$

(1) Tool life (T) when V = 1.25 (50):-

$$V = 62.5\ m/min$$

$$62.5\,(T)^{0.26} = 165.56$$

$$T = (165.56/62.5)^{1/0.26}$$

$$\boxed{T = 42.38\ min}$$

(2) Cutting Speed (V) when T = 180 min:-

$$V\,(180)^{0.26} = 165.56$$

$$\boxed{V = 42.91\ m/min}$$

④ The following equation for Tool life is given for a turning operation

$$VT^{0.13} f^{0.77} d^{0.37} = C$$

63

A 60 minute tool life was obtained while cutting at $V = 30\,m/min$, $f = 0.3\,mm/rev$; $d = 2.5\,mm$

Determine the change in Tool life if the cutting speed, feed, and depth of cut are increased by 20%. Individually and also taken together

Soln:—

$$VT^{0.13} f^{0.77} d^{0.37} = C$$

$$30\,(60)^{0.13}\,[0.3]^{0.77}\,(2.5)^{0.37} = C$$

$$\boxed{C = 28.37}$$

① Tool life if V is increased by 20% :—

$$V = 1.2\,(30) = 36\,m/min$$

$$\therefore\ 36\,(T)^{0.13}\,(0.3)^{0.77}\,(2.5)^{0.37} = 28.37$$

Tool life, $\boxed{T = 14.74\,min}$

② Tool life if f is increased by 20% :—

$$f = 1.2\,(0.3) = 0.36\,mm/rev$$

$$30\,(T)^{0.13}\,(0.36)^{0.77}\,(2.5)^{0.37} = 28.37$$

$$\boxed{T = 20.36 \text{ min}}$$

64

③ Tool life (T) if d is increased by 20%:-

$$d = 1.2\,[2.5]$$

$$\boxed{d = 3\text{mm}}$$

$$30\,(T)^{0.13}\,(0.3)^{0.77}\,(3)^{0.37} = 28.37$$

$$\boxed{T = 35.68 \text{ min}}$$

④ Tool life (T) if V, f, d are increased by 20%:-

$$36\,(T)^{0.13}\,(0.36)^{0.77}\,(3)^{0.37} = 28.37$$

$$\boxed{T = 2.98 \text{ min}}$$

Surface finish:-

Generally, Surface finish of any Component produ-ced by m/cing depends on the following main factors.

a) Cutting Speed b) feed
c) Depth of cut d) M.R.R

a) Cutting Speed:-

* It is defined as the rate at which the cutting edge of the tool passes over the Surface of the w/p in unit time.

* It is expressed in m/min.

* If the cutting Speed is too high, then the tool gets over heated & the Cutting edge may blunt quickly which affects the Surface finish.

* At the Same time, if it is too low, then m/cing time is more & hence productivity decrease

b) Feed:-

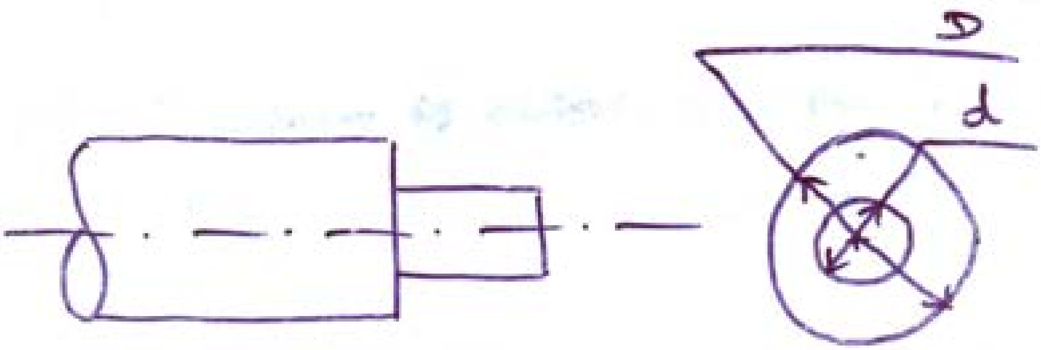
It is defined as the distance travelled by the tool along (or) into the w/p for each pass of tool point.

* Feed should be within certain range to get a good surface finish.

(66)

c) Depth of cut:—

* It is the penetration of the cutting edge of the tool into the material of the w/p, which is measured ⊥r to the machined surface.



$$\boxed{\text{Depth of cut} = \frac{D-d}{2}}$$

* If the depth of cut is high, it will affect the surface finish of the w/p.

d) M.R.R:—

* It denotes the volume of material removed per unit time.

M.R.R = feed × depth of cut × velocity

$= f \times t \times V \quad mm^3/min$

Machinability:-

" The ease with which the given material can be worked with cutting tool is "Machinability".

When it is stated that the material 'P' is more machinable than material 'Q', it means that

* Less power is required to machine material 'P'.
* Less tool wear is obtained with material 'P'.
* Better surface finish can be achieved with material 'P'.

Parameter affecting the machinability :-

M/c Variables

* cutting speed, feed & depth of cut
* Tool material.
* Tool Geometry.
* Cutting fluid
* Cutting type [Intermittent (or) Continuous]

Workpiece material variables

* Hardness of the material
* Chemical composition of material

* Tensile strength
* Strain hardening
* Shape & size of the w/p.
* Rigidity of the w/p.

General criteria for evaluating the machinability are as follows.

1. Tool life:-

Longer tool life at a given cutting speed indicates better machinability.

(2) Surface finish:-

Better the surface finish, higher is the machinability

(3) Power consumption:-

If the power consumption is low, it indicates better machinability.

(4) Cutting force:-

Lesser the cutting force for removing the material, higher is the machinability.

5 Shear angle:-

Larger shear angle gives better machinability.

Machinability Index:-

* It is used for comparing machinability of different material.

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

Cutting Tool material:-

[P.no: 71]

Characteristics that the tool material should have

1. Hot hardness
2. Wear resistance
3. Toughness
4. Cost & easiness in fabrication

Types of cutting Tool materials:— [P. no: 71]

(1) Carbon steels (2) Medium alloy steels

(3) High speed steels (H.S.S)

(4) Stellites (5) Cemented carbides

(6) Ceramics (7) Diamonds (8) Abrasives.

Cutting fluids:— [P. no: 76]

Purpose:—

1. To cool the tool & w/p
2. To lubricate & reduce friction
3. To improve surface finish
4. To protect the finished surface from corrosion
5. To cause the chip break up into small parts
6. To wash away the chip from the tool.

Properties of Cutting fluids:—

1. High heat absorption
2. Good lubricating qualities
3. High flash point
4. Neutral 5. odourless
6. Harmless to the Skin of the opreators
7. Non-corrosive to the work/Tool/machine
8. Transparency
9. Low Viscosity 10. Low Cost.

Choice of cutting fluid:—

1. Type of opreation 2. M.R.R
3. material of w/p & Tool.
4. Surface finish requirement.
5. Cost of cutting fluid.

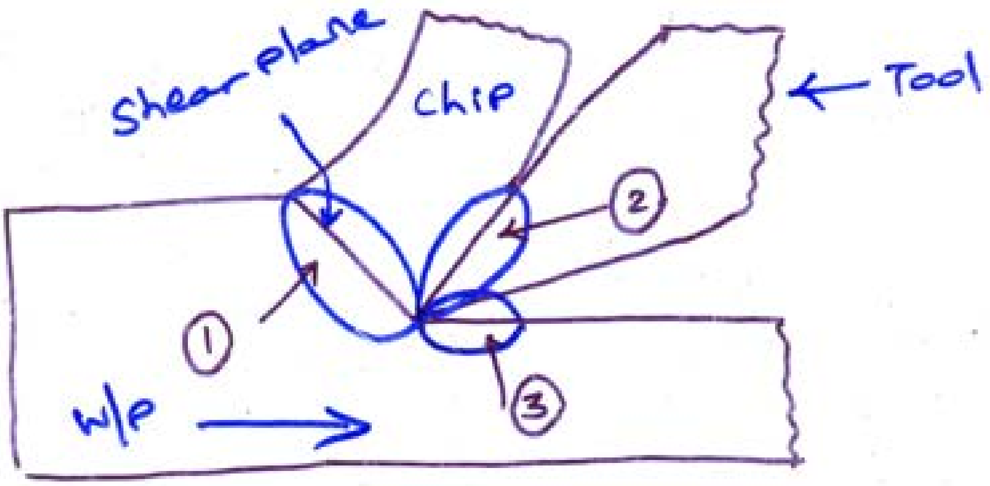
Types of Cutting fluids:—

1. Water 2. Soluable oils 3. Straight oils
4. Mixed oil 5. Solid lubricants etc.,

Sources of Heat in metal cutting:-

There are 3 regions in the metal cutting where heat is generated.

1. Around Shear Zone
2. Tool-Chip Interface
3. Tool - w/p Interface



1. Shear zone:-

* Called as "primary deformation zone" in which Shearing of the metal (or) plastic deformation of the metal takes place.

* Heat produced in this zone is carried by the chip & workpiece

② Tool-Chip Interface:—

* Called as "Secondary deformation zone" inwhich the heat is generated due to the friction between the Chip & tool face.

* The Heat produced in this zone is Carried by the Chip & the tool.

③ Tool-W/P Interface:—

* this is the Contact Area b/w the flank face of the tool & the workpiece.

* Here, the heat is generated due to the friction b/w the flank face of the tool & the workpiece

* when the clearance angle is not Sufficiently provided on the tools the heat generated in this Area will be more.

Tool failure:-

If the tool is not giving the satisfactory performance, then it is an indication of the "tool failure" which causes the following adverse effects.

* Poor surface finish of the w/p.
* High power consumption.
* Incorrect work dimension
* overheating of the cutting tool & the workpiece.

The main causes of the failure of the tool are as follows.

(a) thermal cracking & softening
(b) Mechanical chipping
(c) Tool wear → a) Crater wear, b) Flank wear, c) Chemical wear.

(a) thermal cracking & softening:-

* During metal cutting, high heat is generated at the cutting edge & the tip of the tool. Due to this, plastic deformation takes place

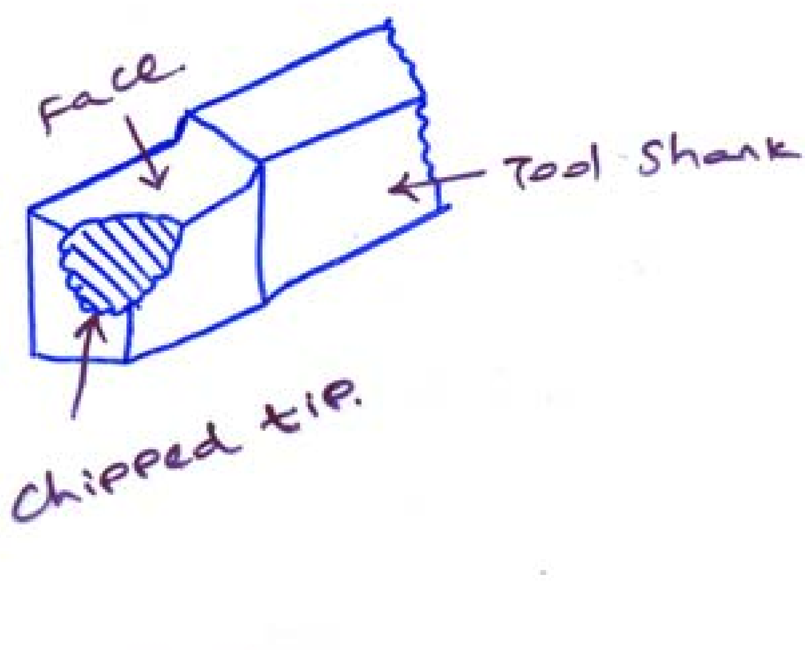
at the cutting edge & the tool tip. Due to this, the tool cutting edge & its tips will become soft & it will lose its cutting ability. This is known as "softening".

75

* Sometimes, due to the high heat, there may be a chance of the cracks which is developed on the cutting edge of the tool. This is known as "thermal cracking".

(b) Mechanical chipping:—

It denotes the breakage of the tool nose due to the high cutting pressure, mechanical impact, excessive wear & high vibration etc.,

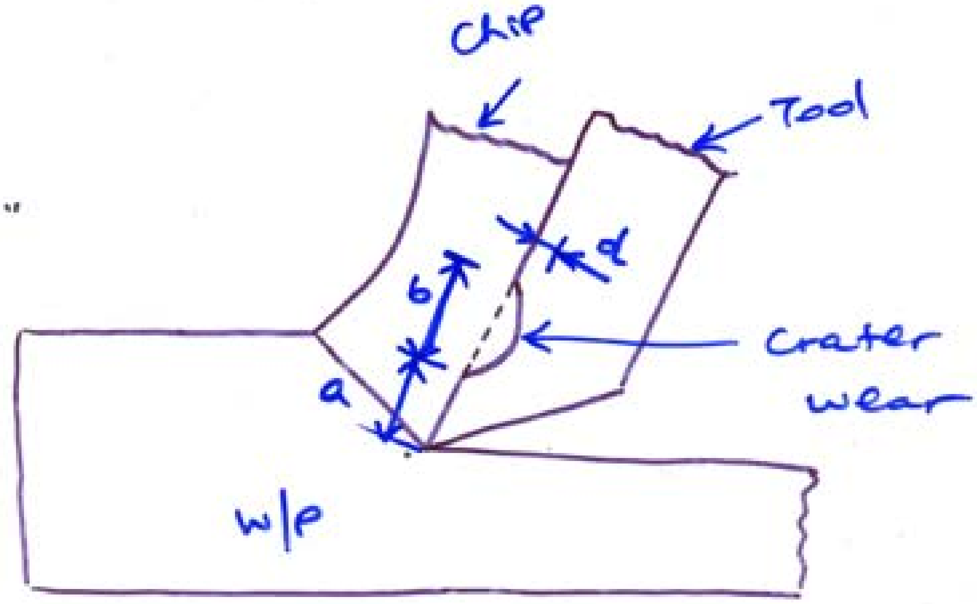


Tool Wear:-

Sometimes, when the tool is in use, it will loose its weight, which means it has lost some material from it due to wear which are of the following types.

(i) Crater Wear:-

depression width - "b"
Depth - "d"
Distance - "a" from the tool tip.



* The depression on the tool face is known as "Crater wear". This type of wear takesplace during machining of ductile material inwhich Continuous Chip will be formed.

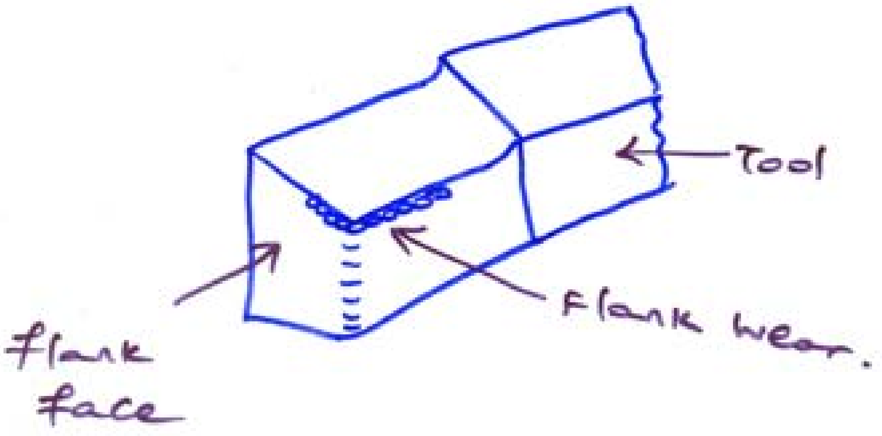
* The main cause for this depression (or) Crater is due to the pressure by the hot chip sliding on the tool face.

(ii) Flank wear:-

* The flank face portion below the cutting edge is the region where an appreciable amount of wear occurs.

* "Flank wear" is due to the abrasion b/w the tool flank face & the w/p.

* This type of wear mainly occur when machining brittle material.



(iii) Chemical wear:-

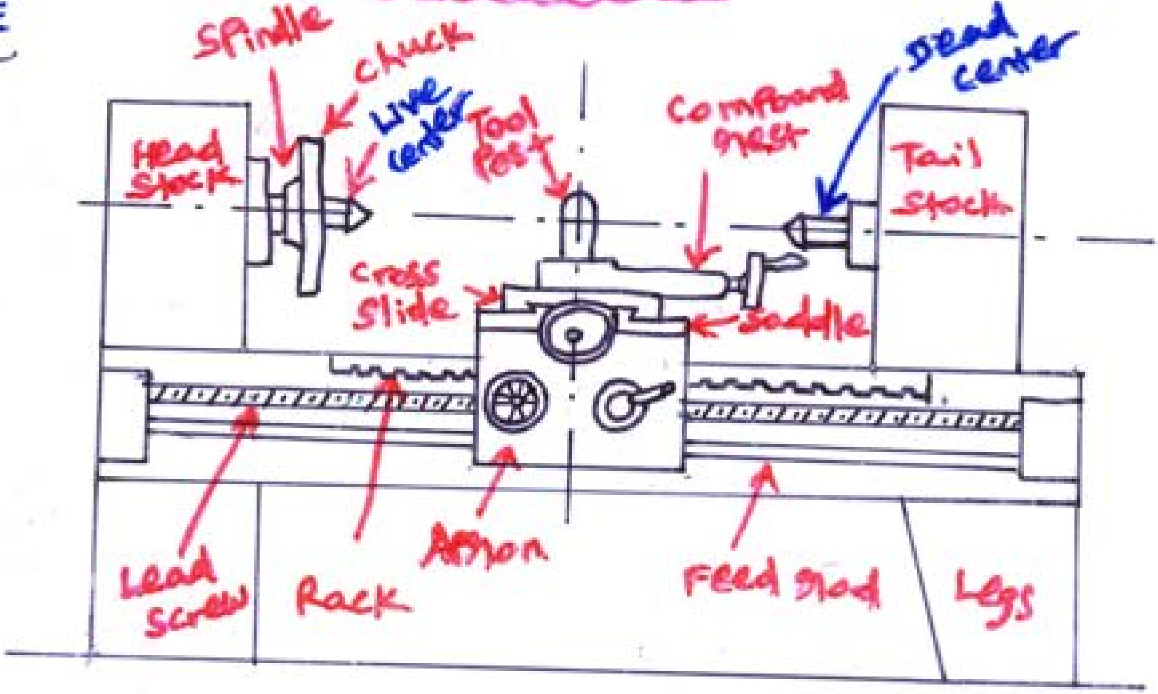
* This type of wear occurs when a cutting fluid used is chemically reactive.

* Due to this, the chemical reaction takes place b/w the cutting tool & cutting fluid & the erosion takesplace on the cutting tool.

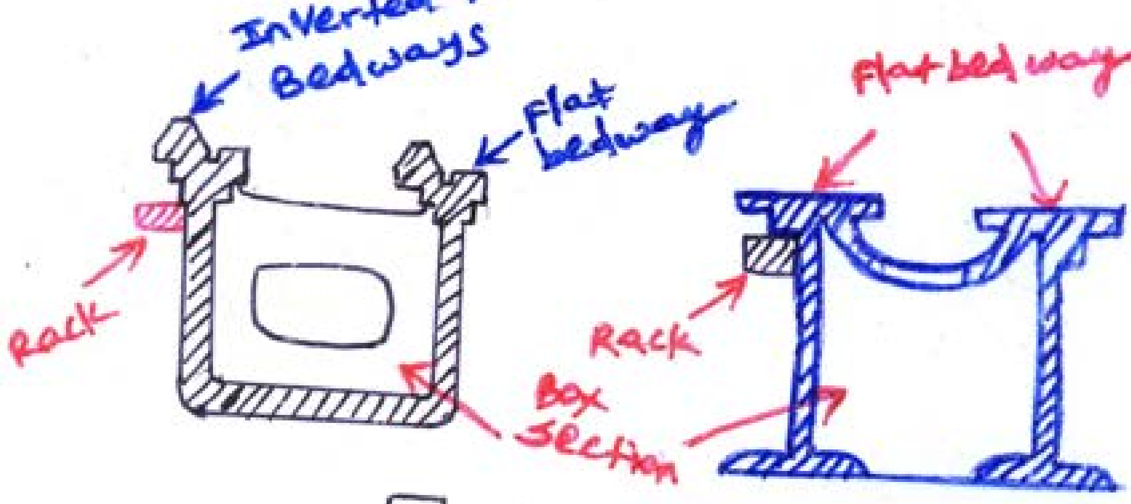
Unit-2

Turning Machines

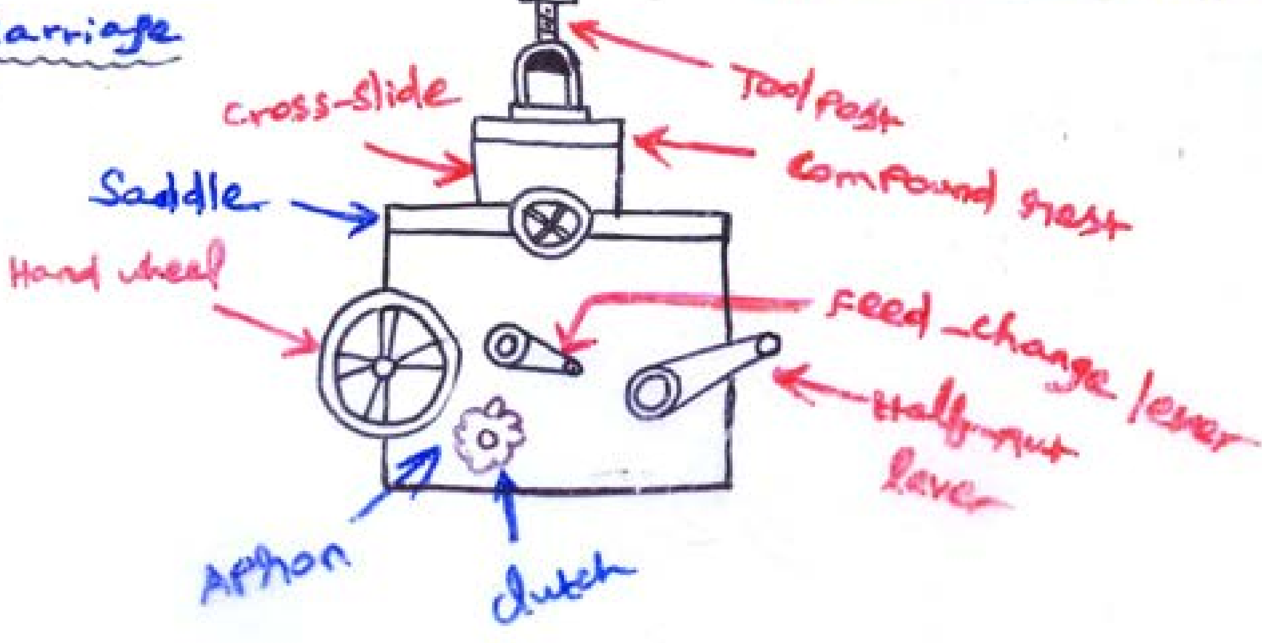
LATHE



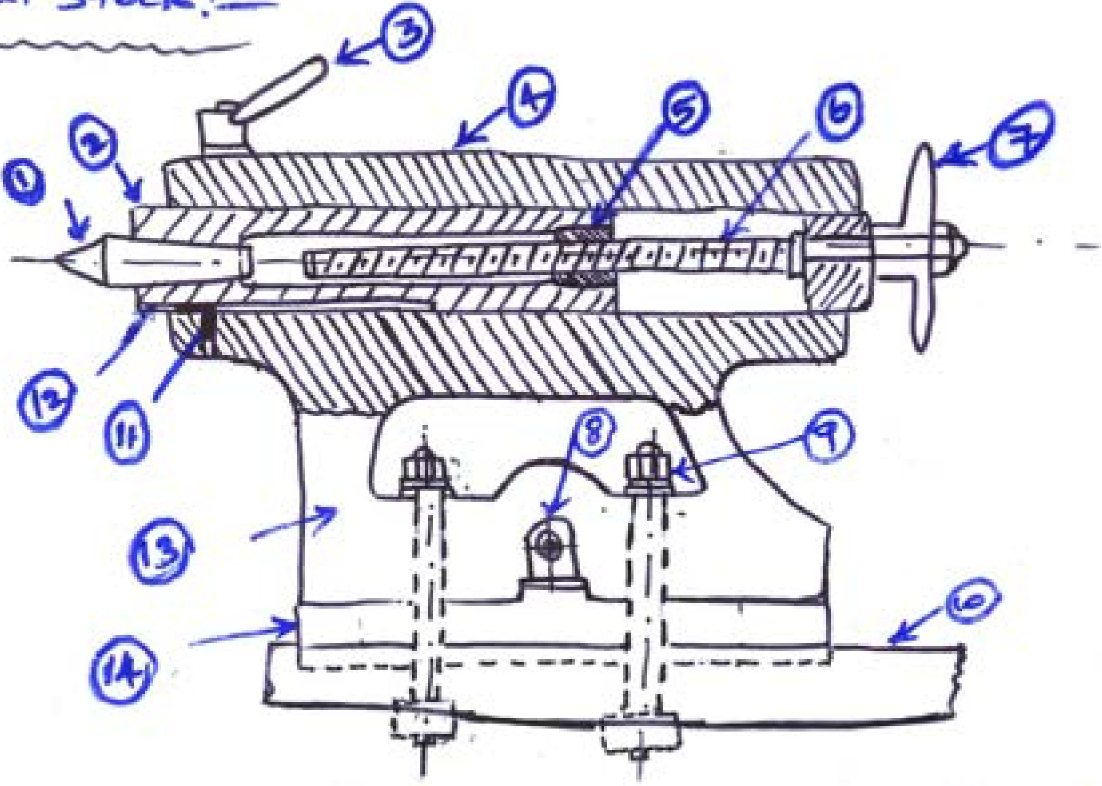
Bed



Carriage

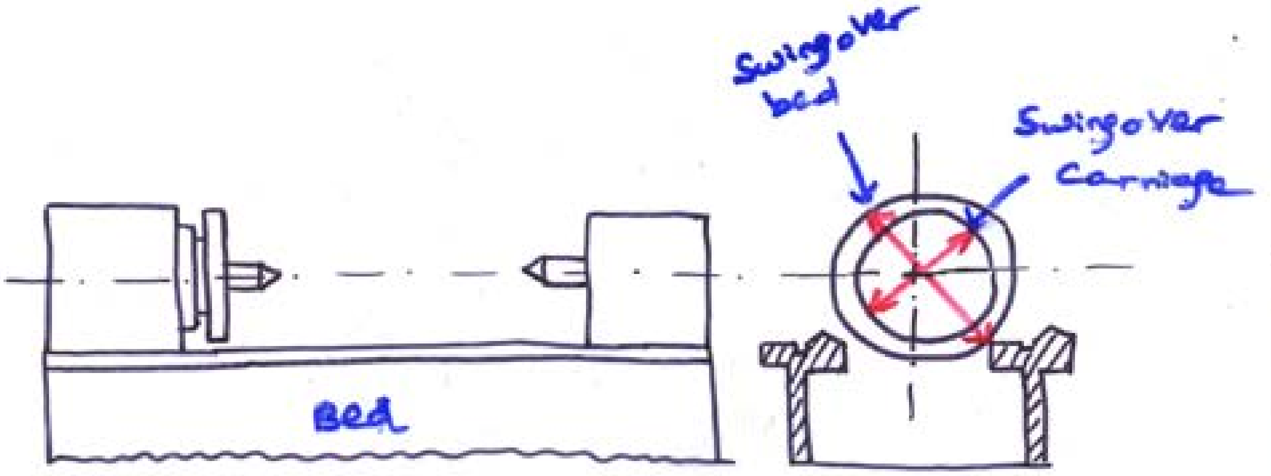


Tail stock:—



1 Dead center 2 Spindle 3 Spindle clamp 4 Barrel
5 Bush 6 Screw 7 Hand wheel. 8 Set over screw
9 Tail stock clamping bolt 10 Lathe bed ways.
11 key 12 keyway 13 Top body 14 Lower body

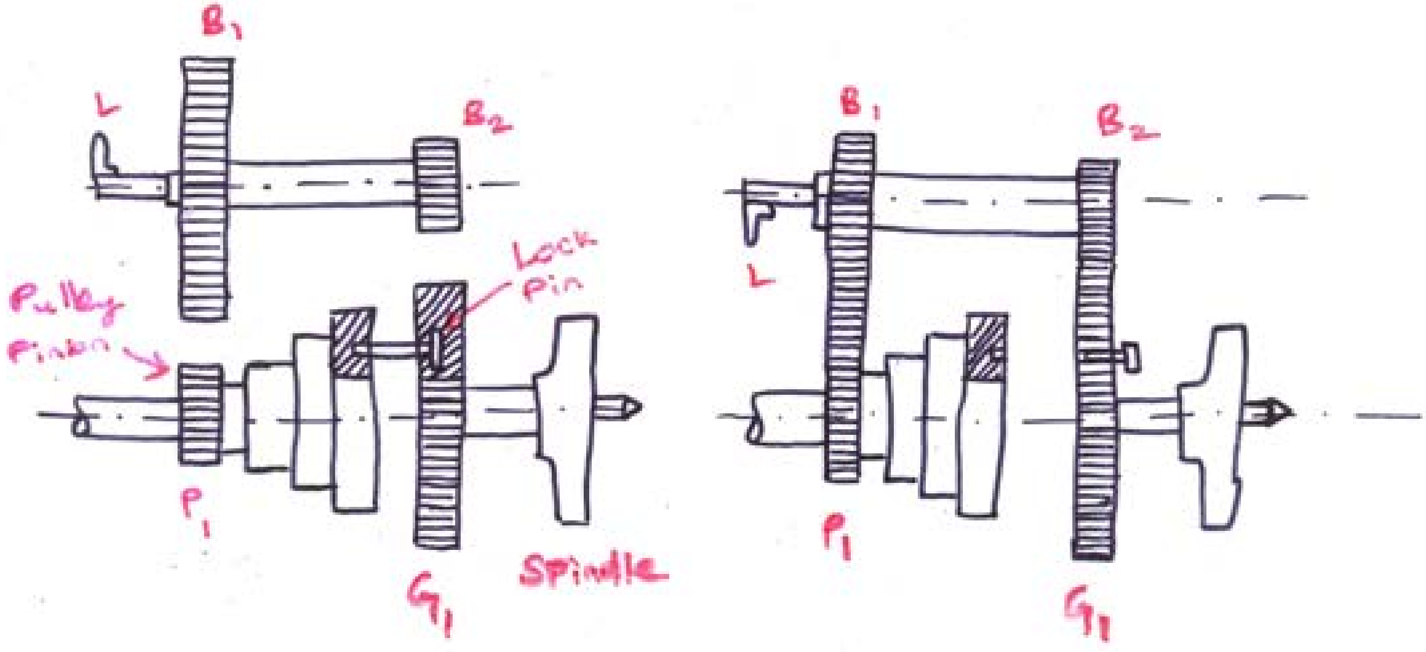
Specification of Lathe:—



Head Stock Mechanism:—

1. Back Geared Head Stock
2. All Geared Head Stock.

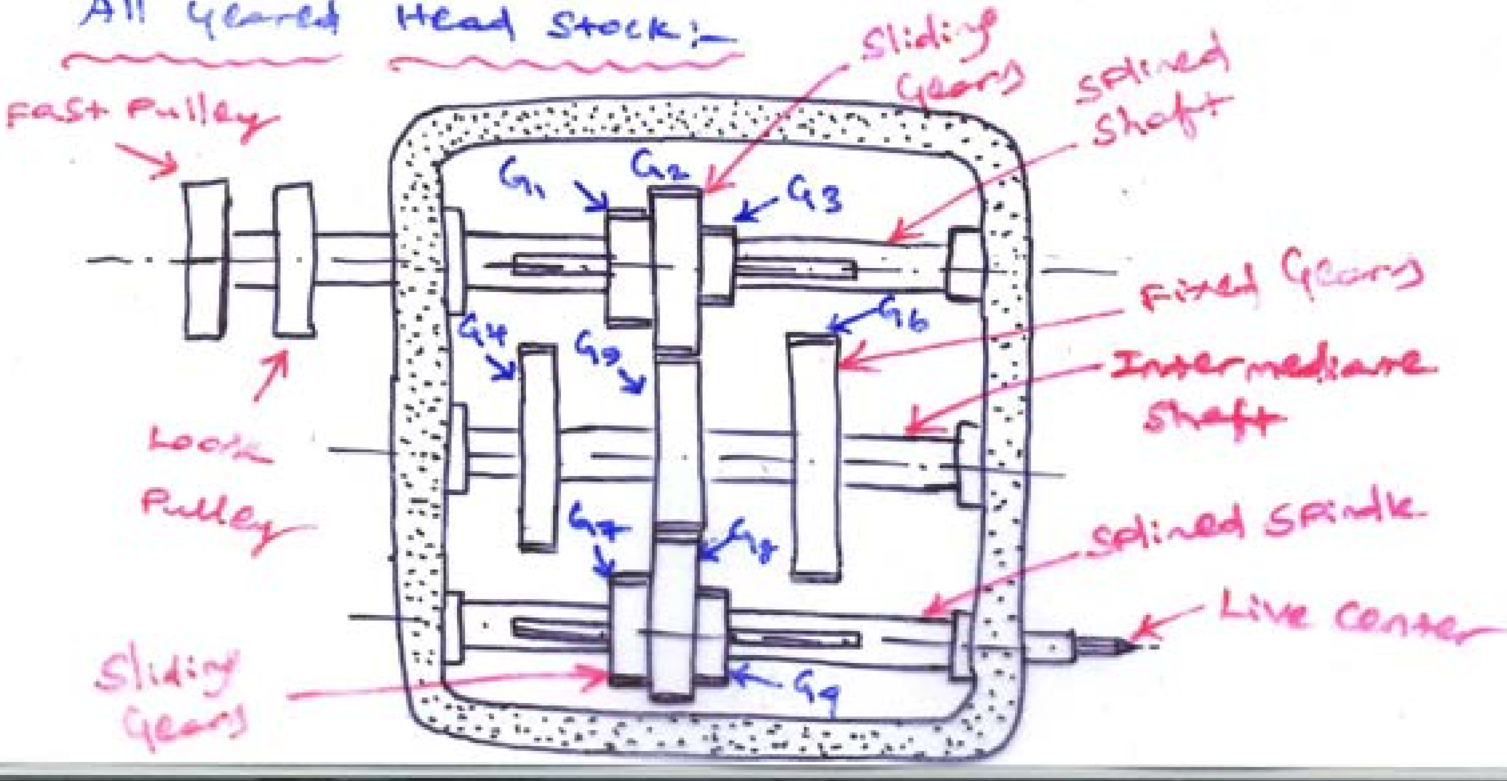
Back Geared Head Stock:—



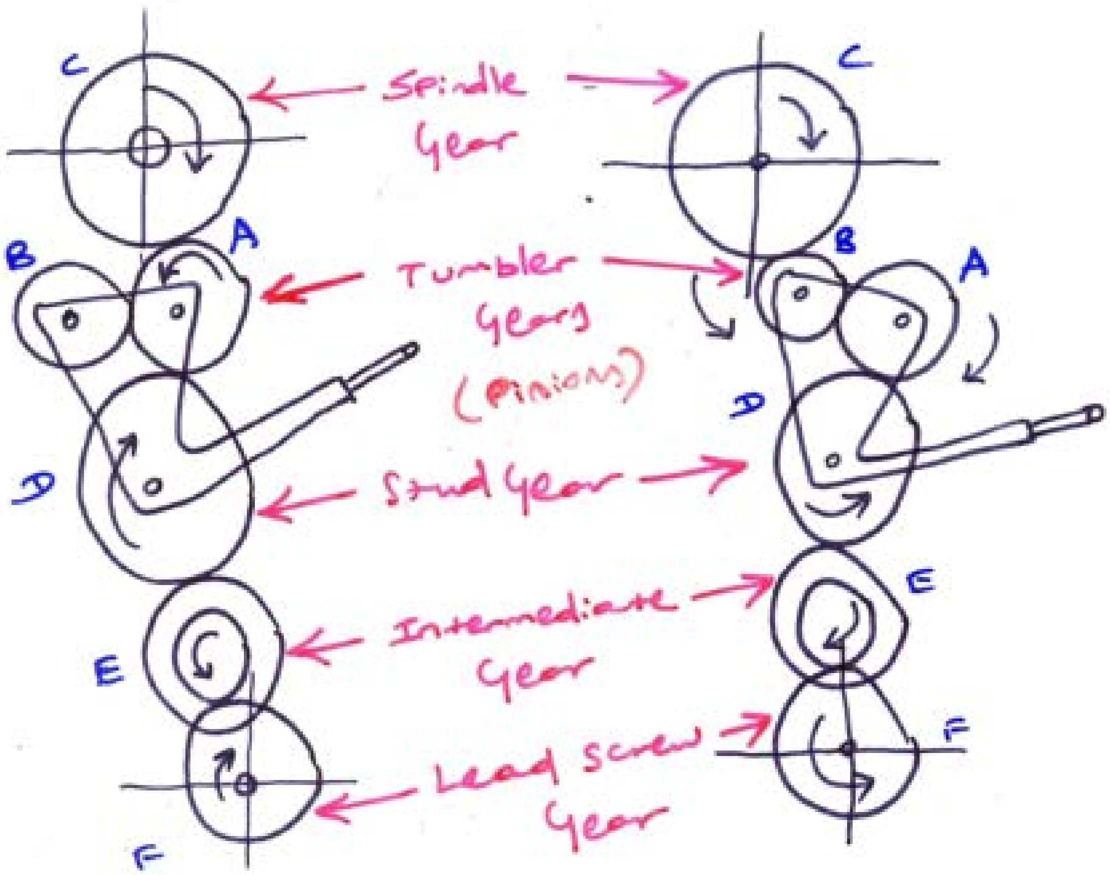
(a) Direct Speed [Lock pin Engaged]

(b) Indirect Speed [Lock pin Disengaged]

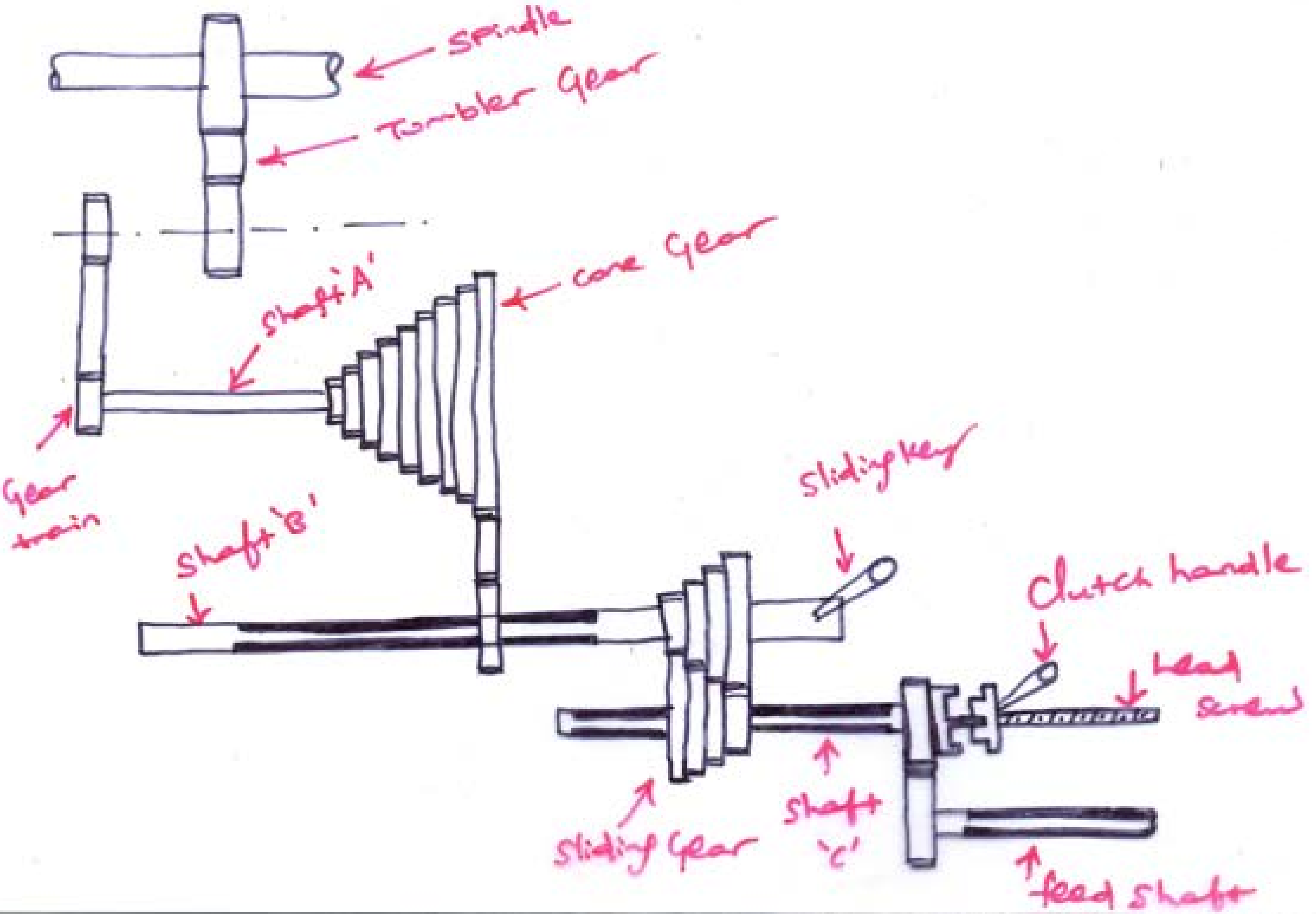
All Geared Head Stock:—



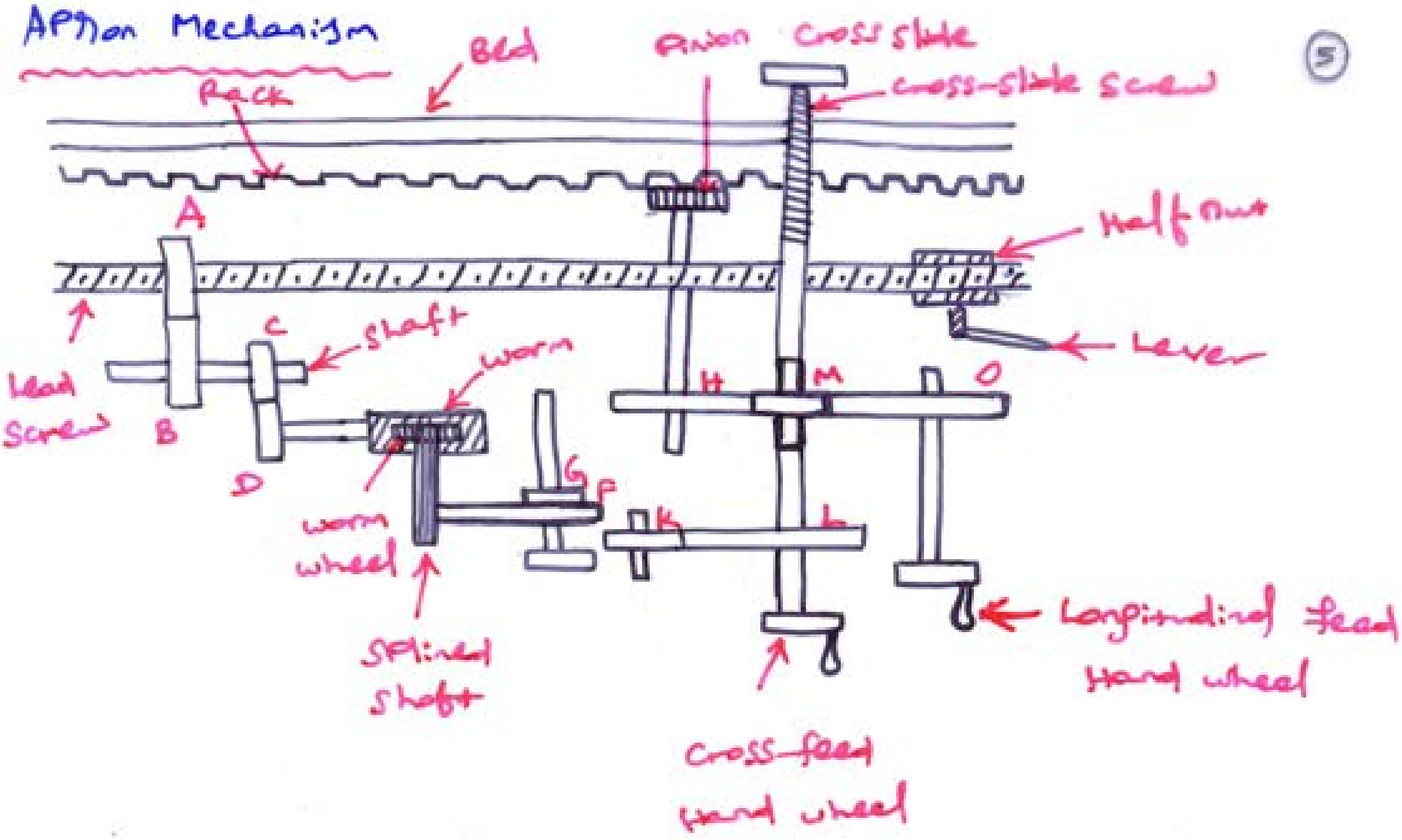
Tumbler Gear Reversing Mechanism



Quick change Gear box:-



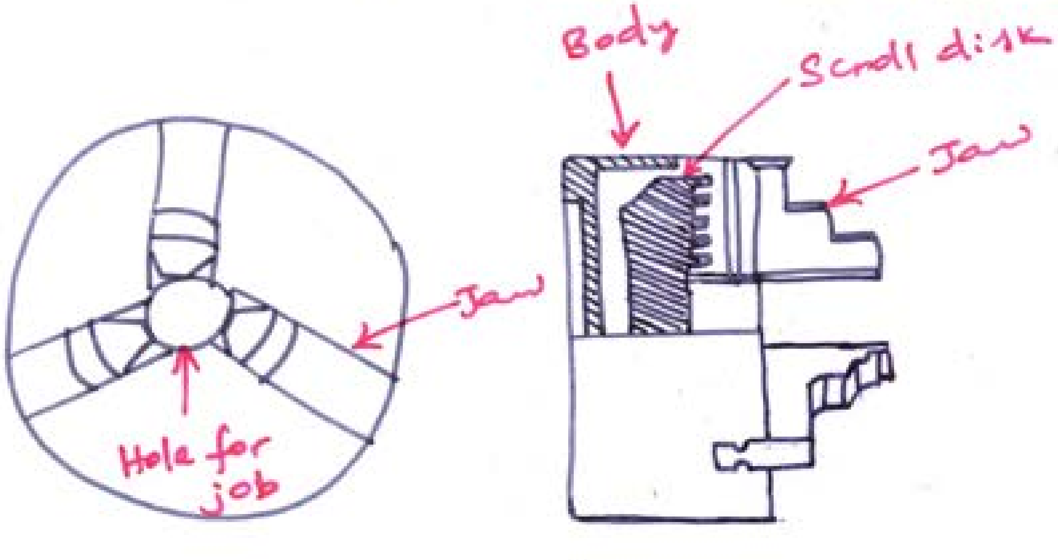
Apron Mechanism



Work Holding Devices:—

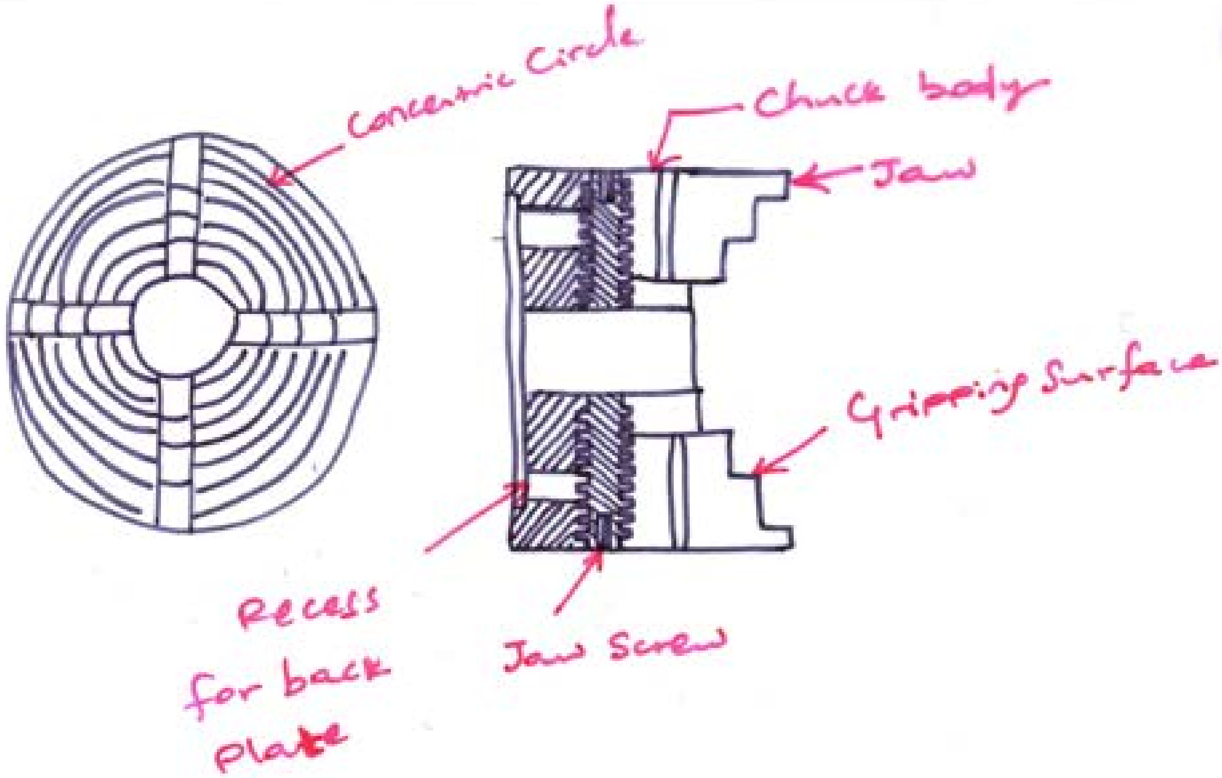
① chucks ② centers ③ faceplate ④ Angle plate
⑤ Mandrels ⑥ Steady & follower rest

① Chucks = ⓐ Three jaw (or) self centering
ⓑ four jaw Chuck. ⓒ Magnetic chuck

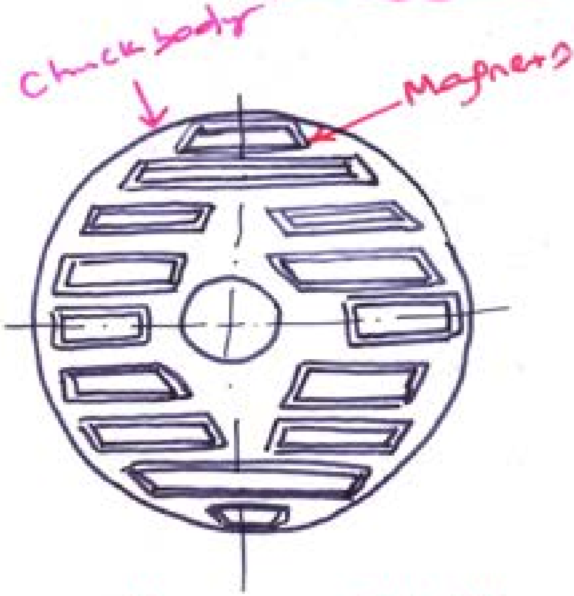


L < 40

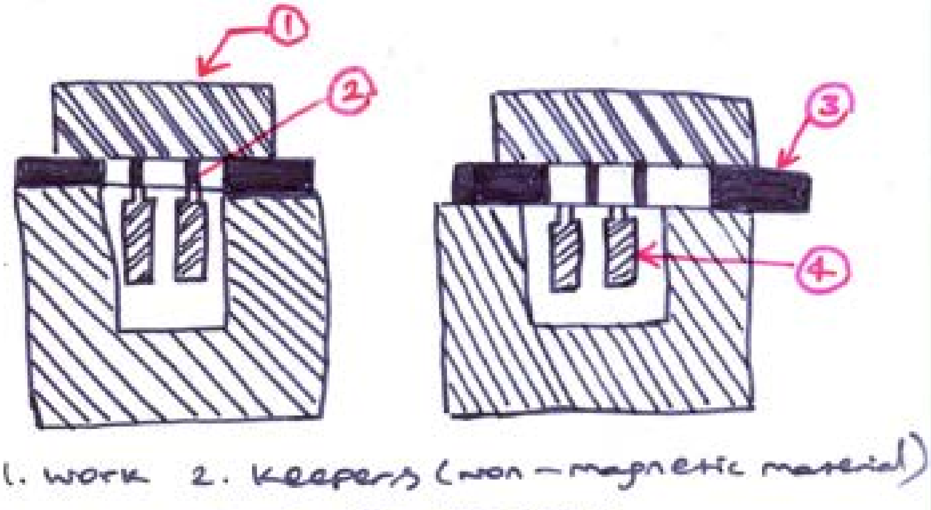
Three jaw (or) Self-centering Chuck



Four Jaw Independent Chuck



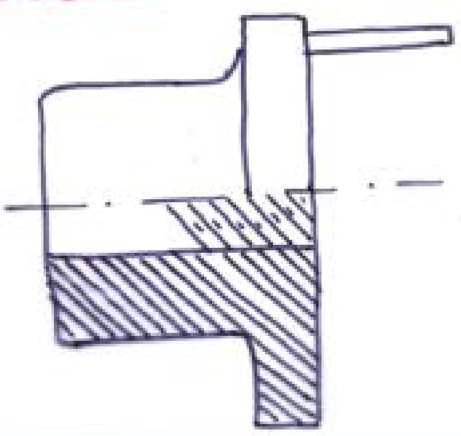
Magnetic Chuck



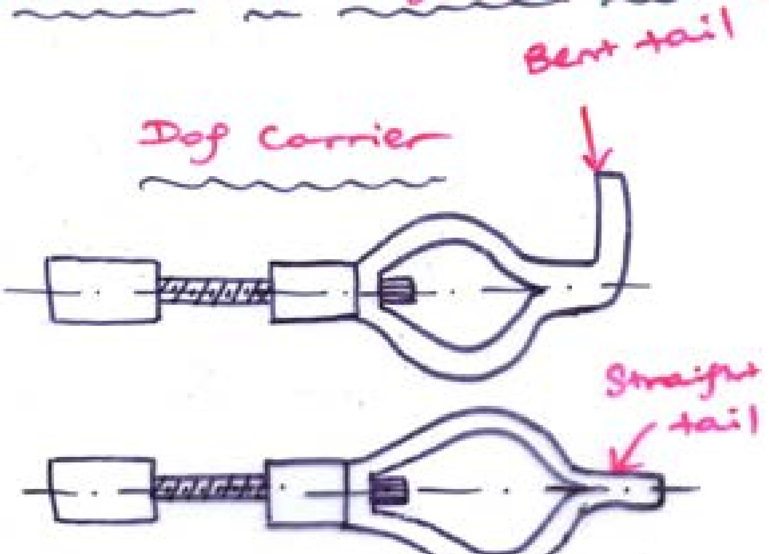
1. work 2. Keepers (non-magnetic material)
3. Face plate (4) Magnets.

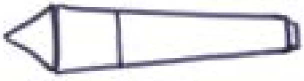
Principle of Magnetic Chuck

(2) Centers :-

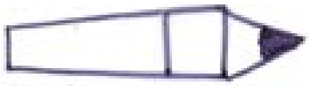


Catch Plate

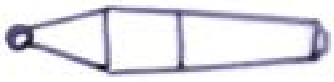




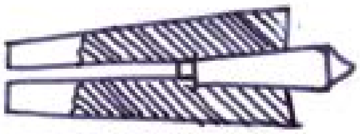
ordinary center



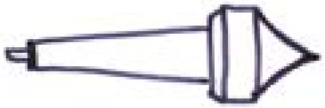
Tipped center



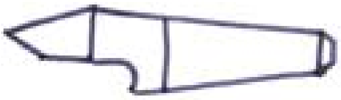
Ball center



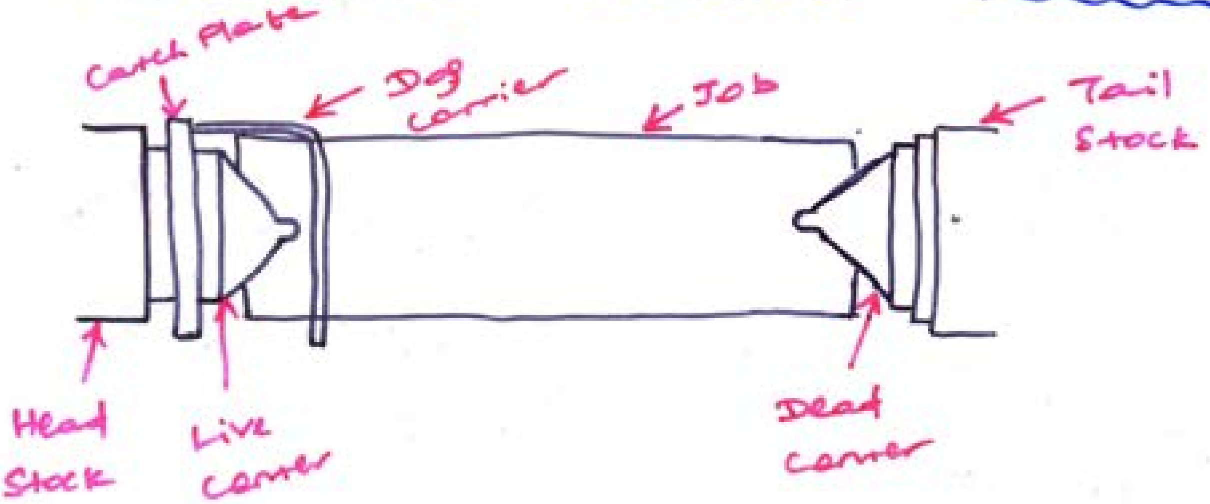
Insert type center



Pipe center

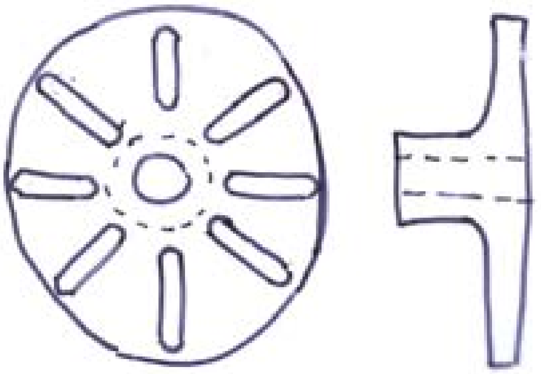


Half center

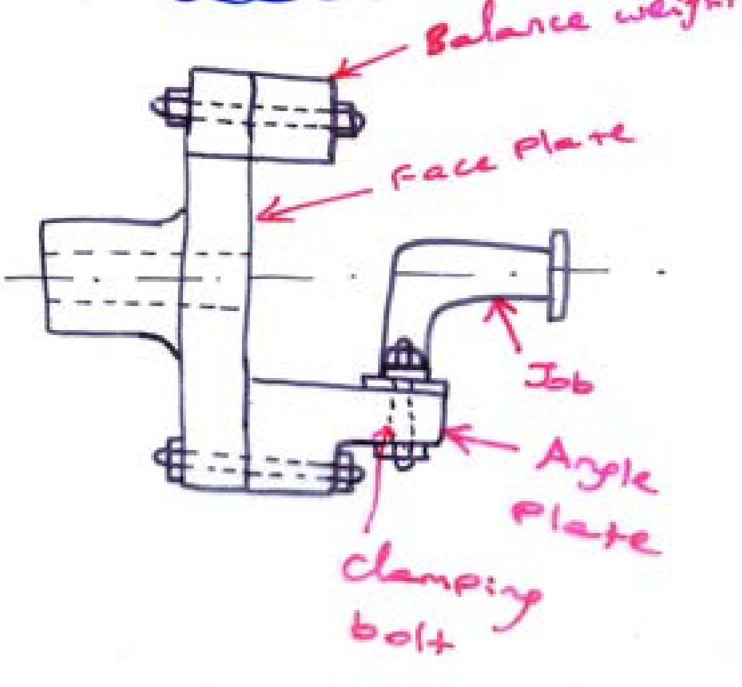


Work Held b/w Centers

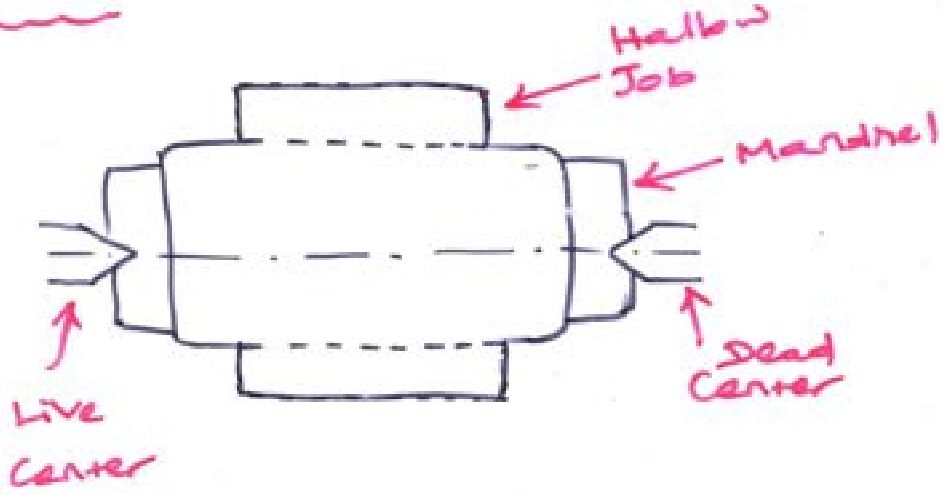
Face Plate:—



Angle plate



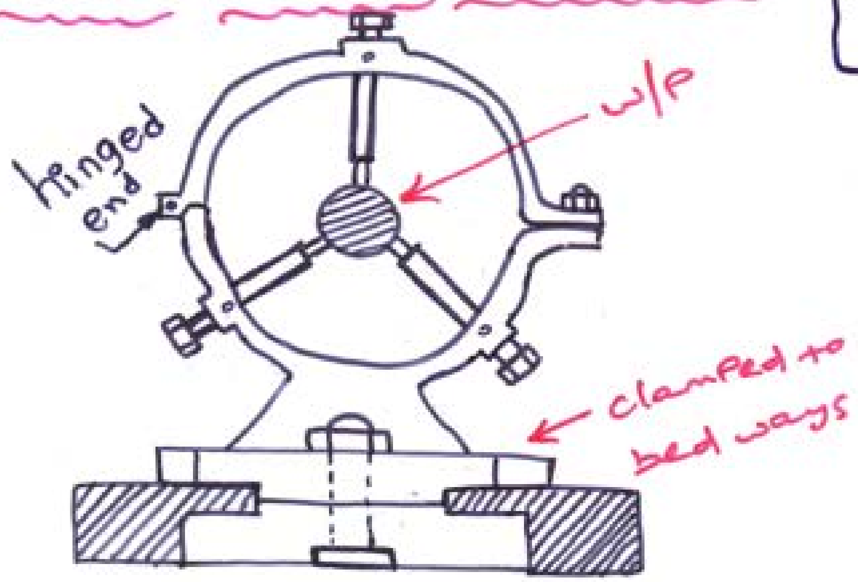
Mandrels:-



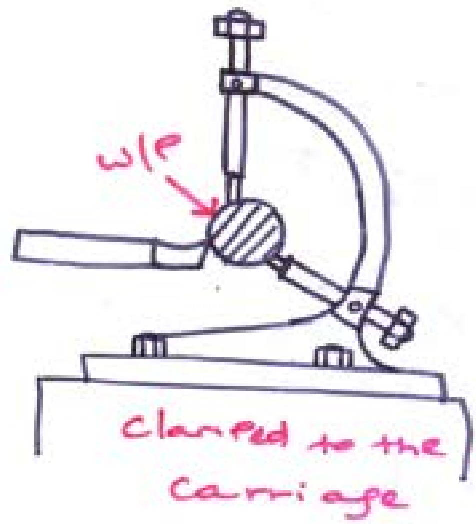
Plain Mandrel

Steady and Follower Rest:-

$$\left[\frac{L}{D} > 10 \text{ (or) } 12\right]$$



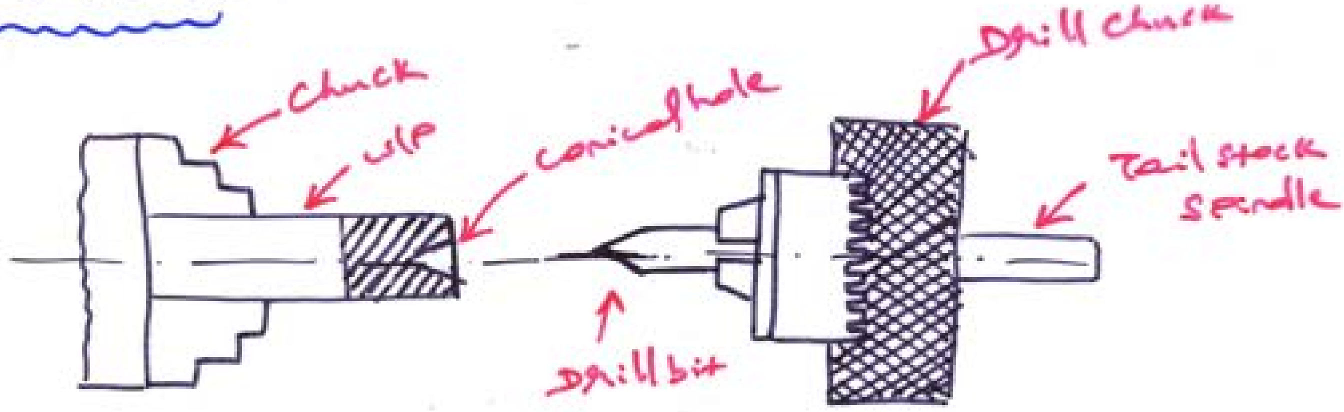
Steady Rest



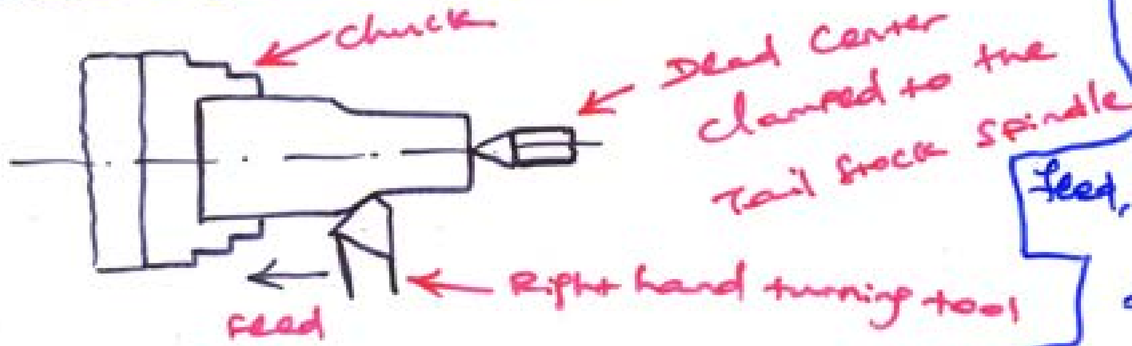
Follower Rest

Various operations Performed on a Lathe:-

① Centering:-



② Straight (or) Plain turning:-



Rough turning

F ↑ d ↑ S ↓

feed, f = 0.3 – 1.5 mm/rev

d = 2m – 5mm

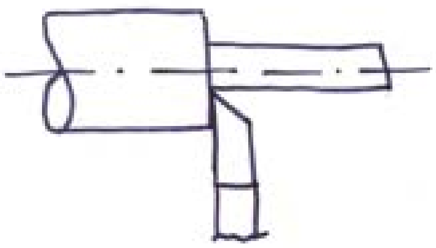
Finish turning

F ↓ d ↓ S ↑

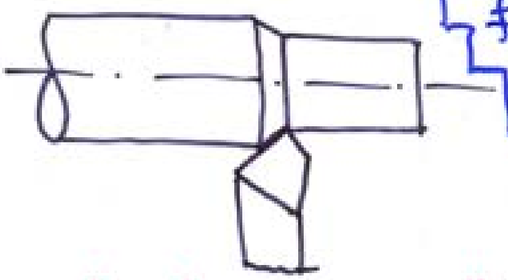
f = 0.1 – 0.3

d = 0.5 – 1

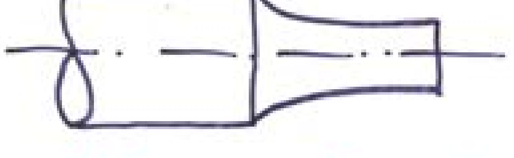
③ Shoulder turning:-



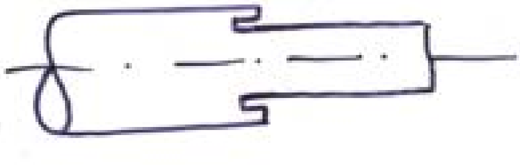
Square Shoulder



Angular (or) Bevelled Shoulder

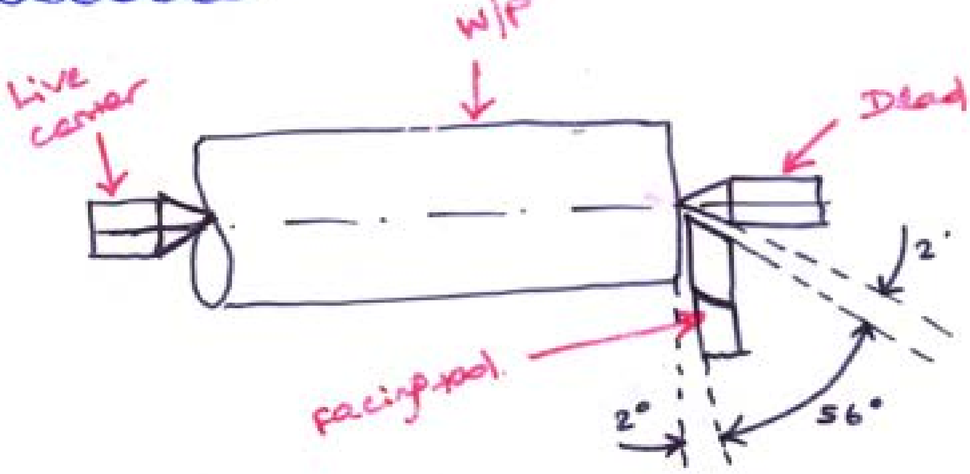


Radius (or) filleted Shoulder

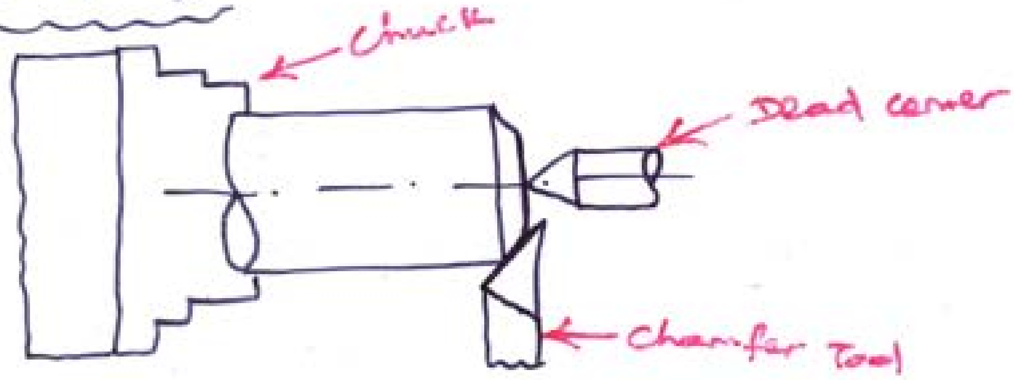


Undercut Shoulder

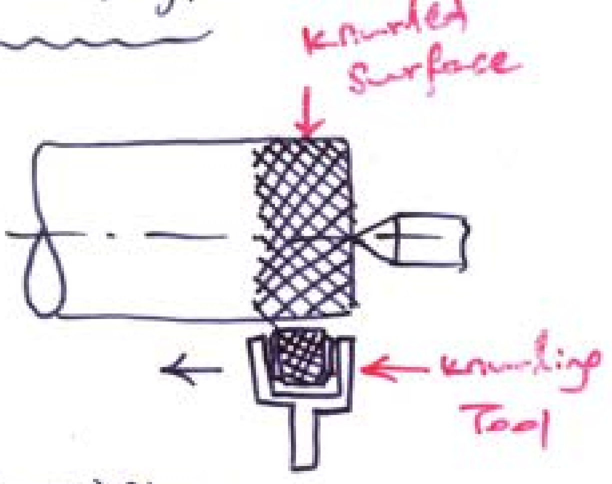
④ Facing :—



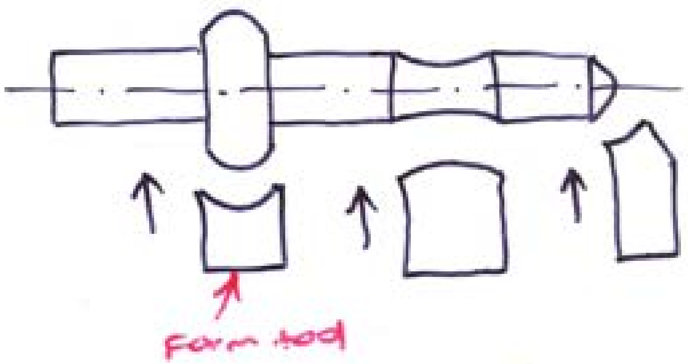
⑤ Chamfering :—



⑥ Knurling :—

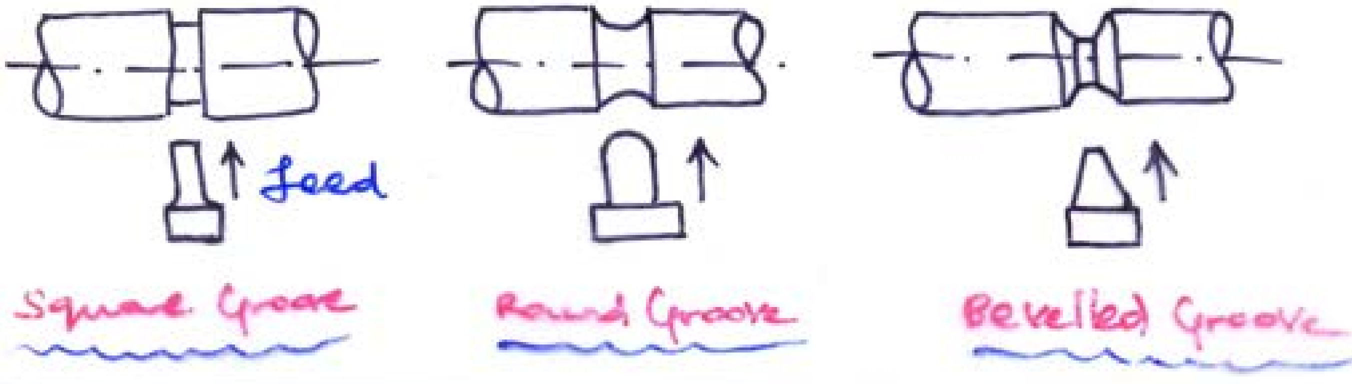


⑦ Forming :—

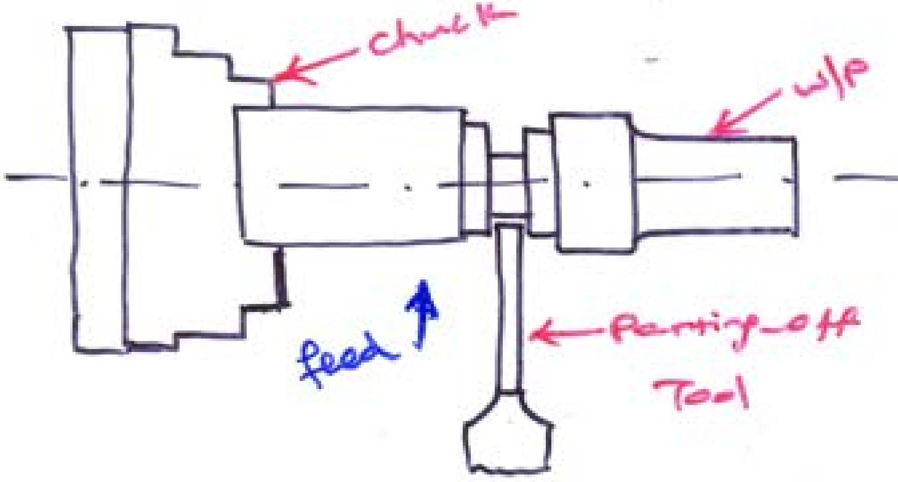


⑧ Grooving :—

(Recessing, undercutting, necking)



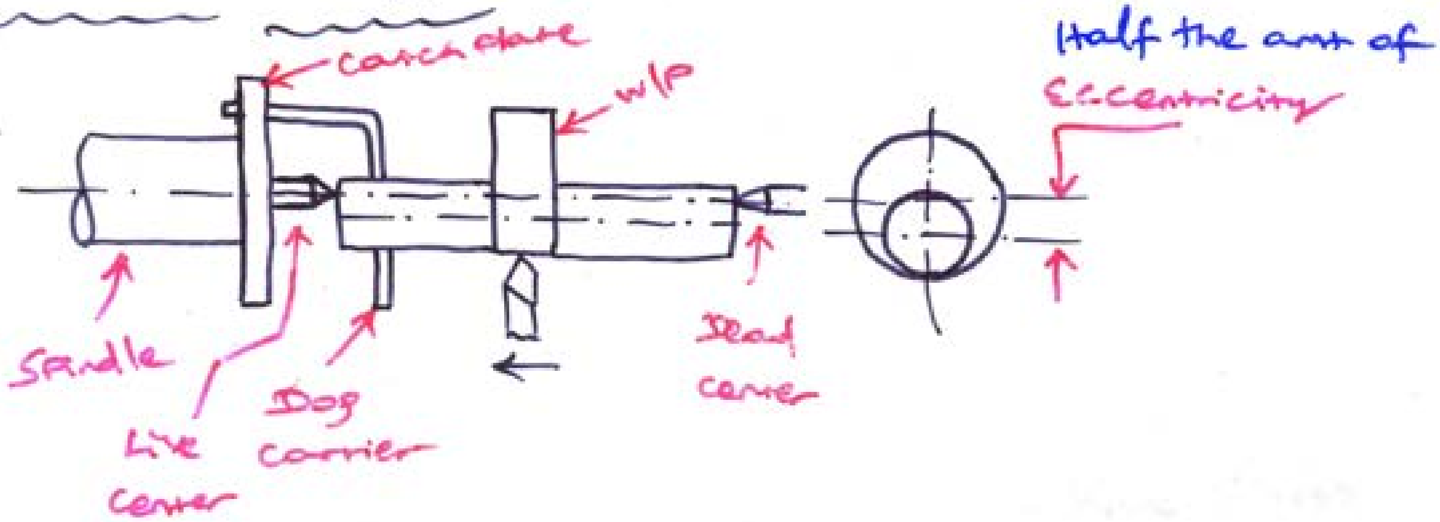
(10) Parting-off:—



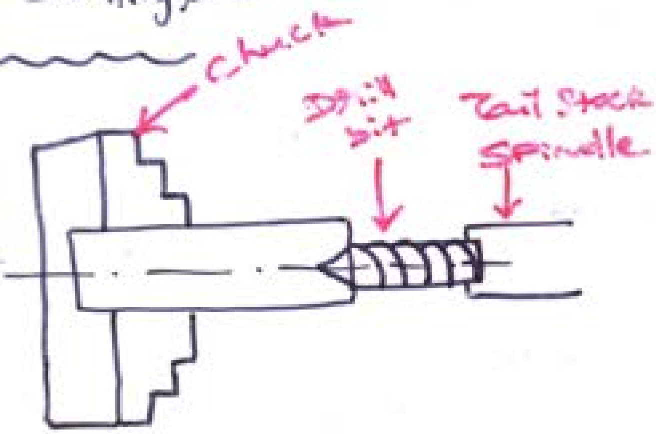
(1) Spindle speed = $\frac{1}{2}$ (speed of turning)

(2) Slow cross-feed.

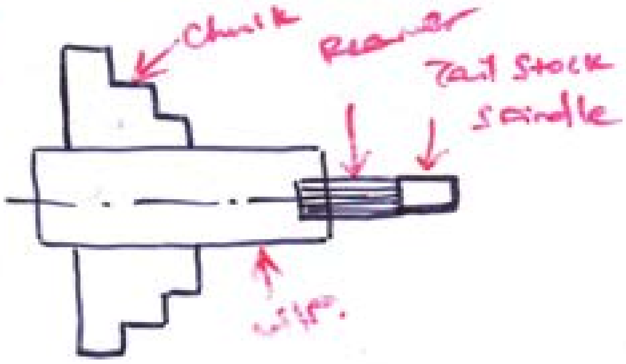
(11) Eccentric turning:—



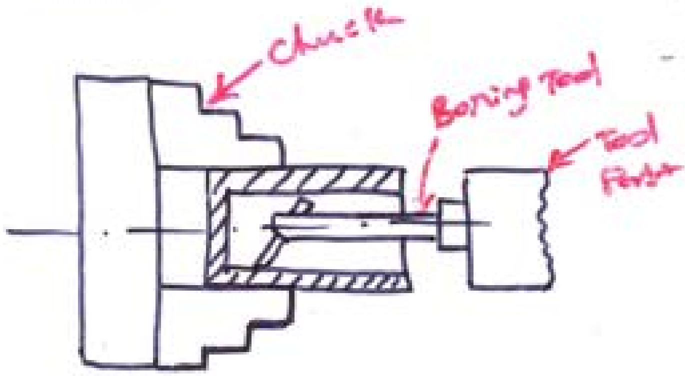
(12) Drilling:—



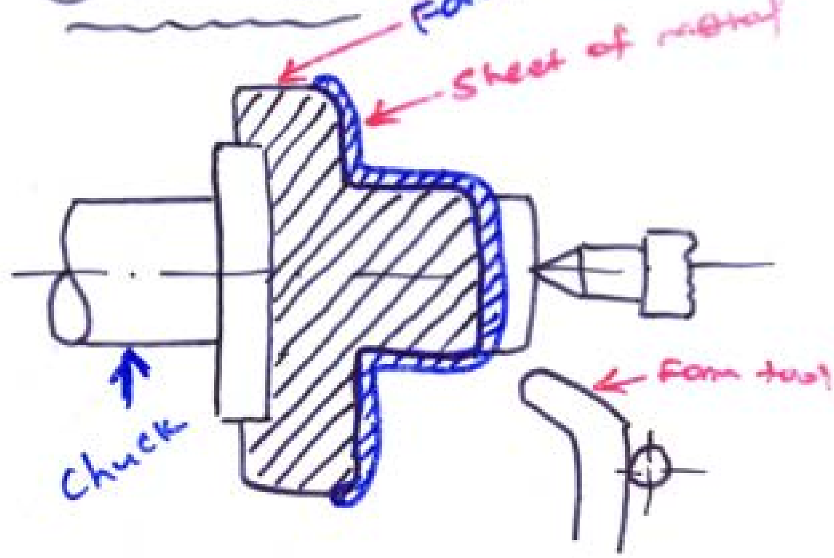
(13) Reaming:—



(14) Boring:-



(15) Spinning:-

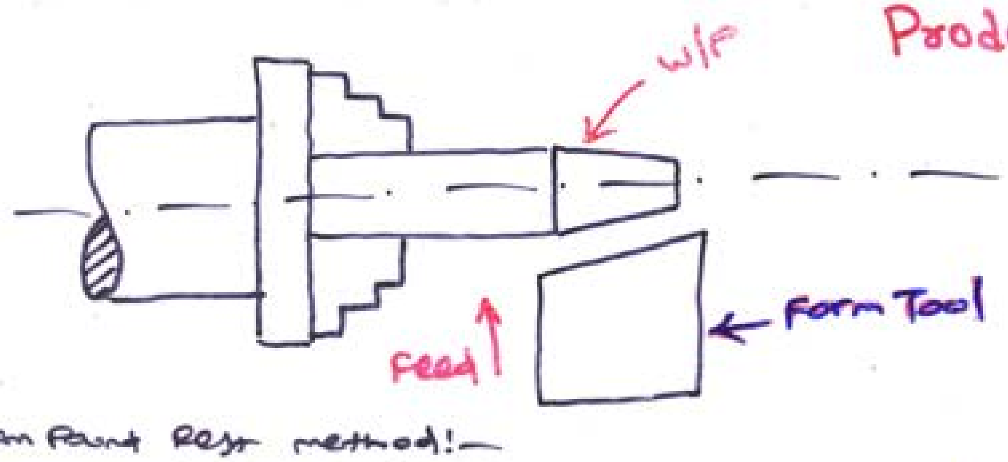


Taper turning Methods:-

(a) Form Tool method (b) Tailstock set over method (c) Compound Rest method (d) Taper turning Attachment method (e) by combining feeds.

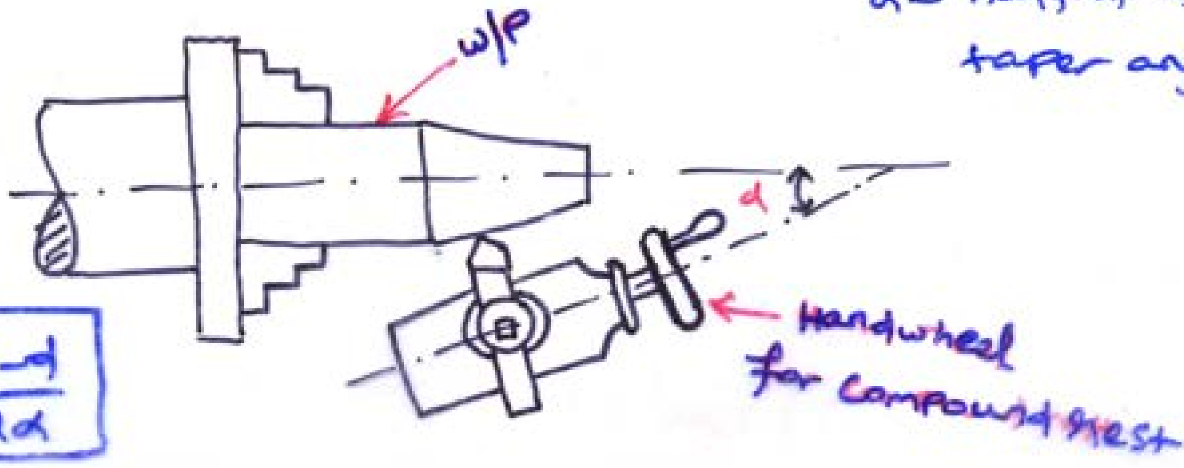
(a) Form Tool Method:-

Produce Short taper



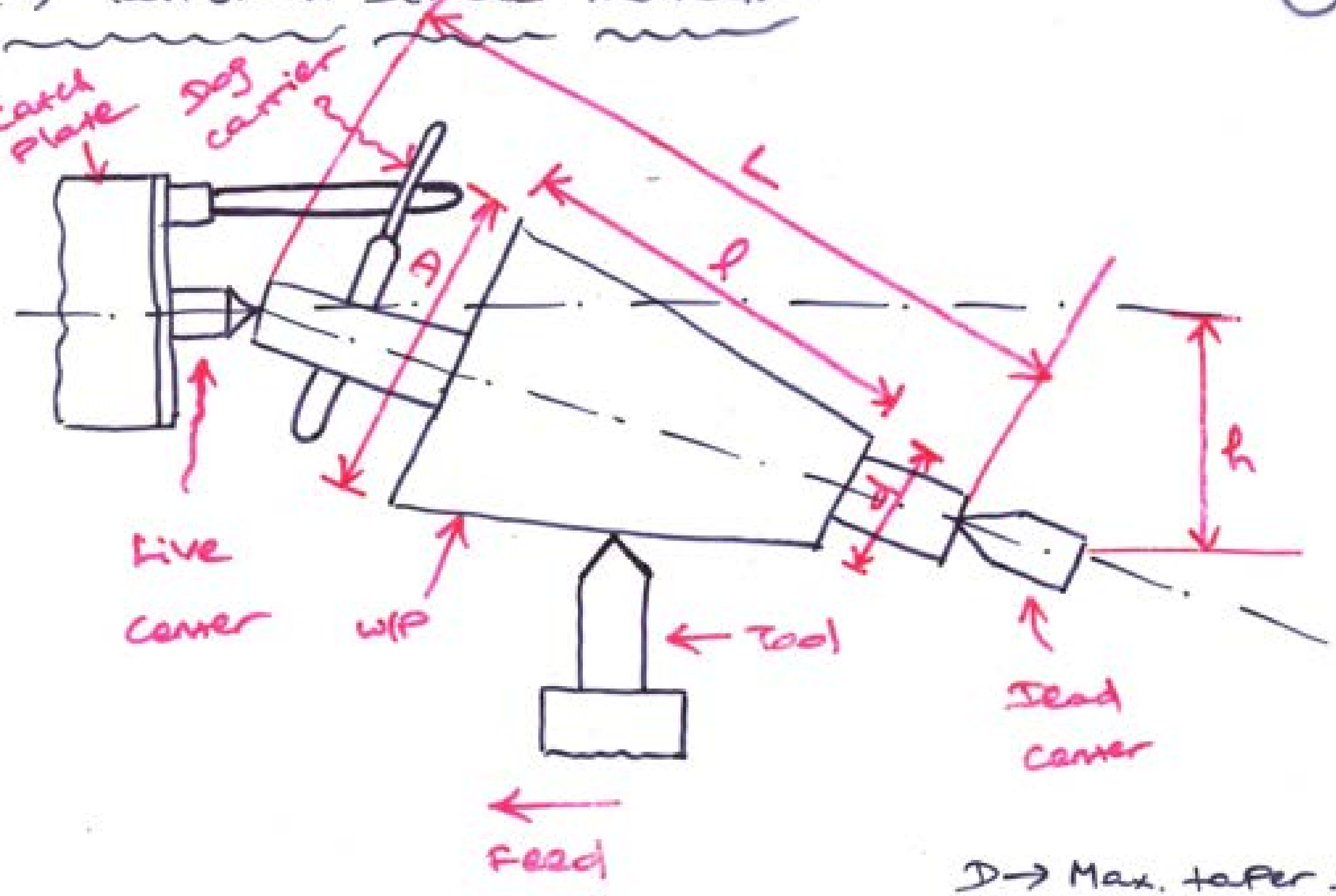
(b) Compound Rest method:-

α - Half of the taper angle



$$\boxed{\tan\alpha = \frac{D-d}{2L}}$$

(c) Tail Stock set over method:—



D → Max. taper Dia

d → Min taper Dia

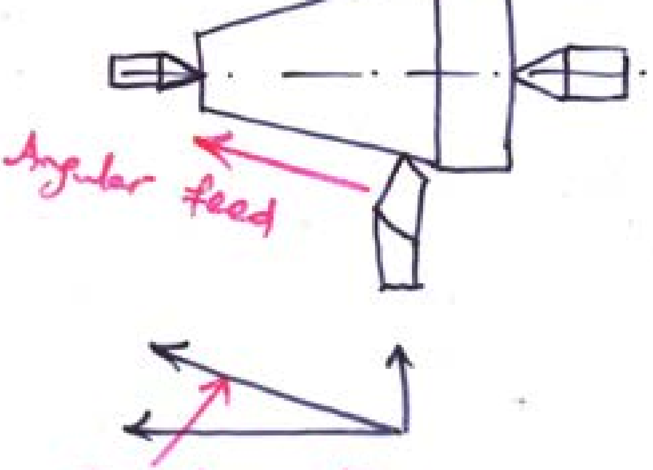
l → Taper length

L → Length of w/P

α → Half taper angle

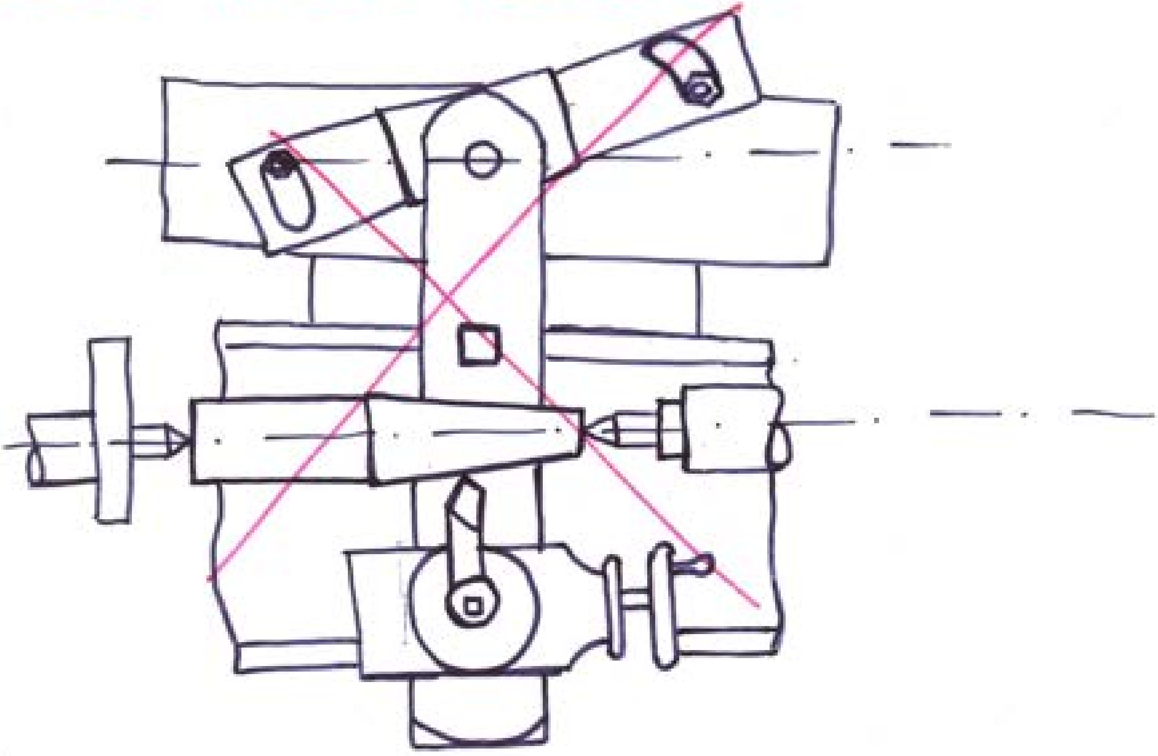
$$\text{set over} \left.\right\} h = \frac{D-d}{2l} \times L = L \tan\alpha$$

(d) Taper turning by giving combined feed:—

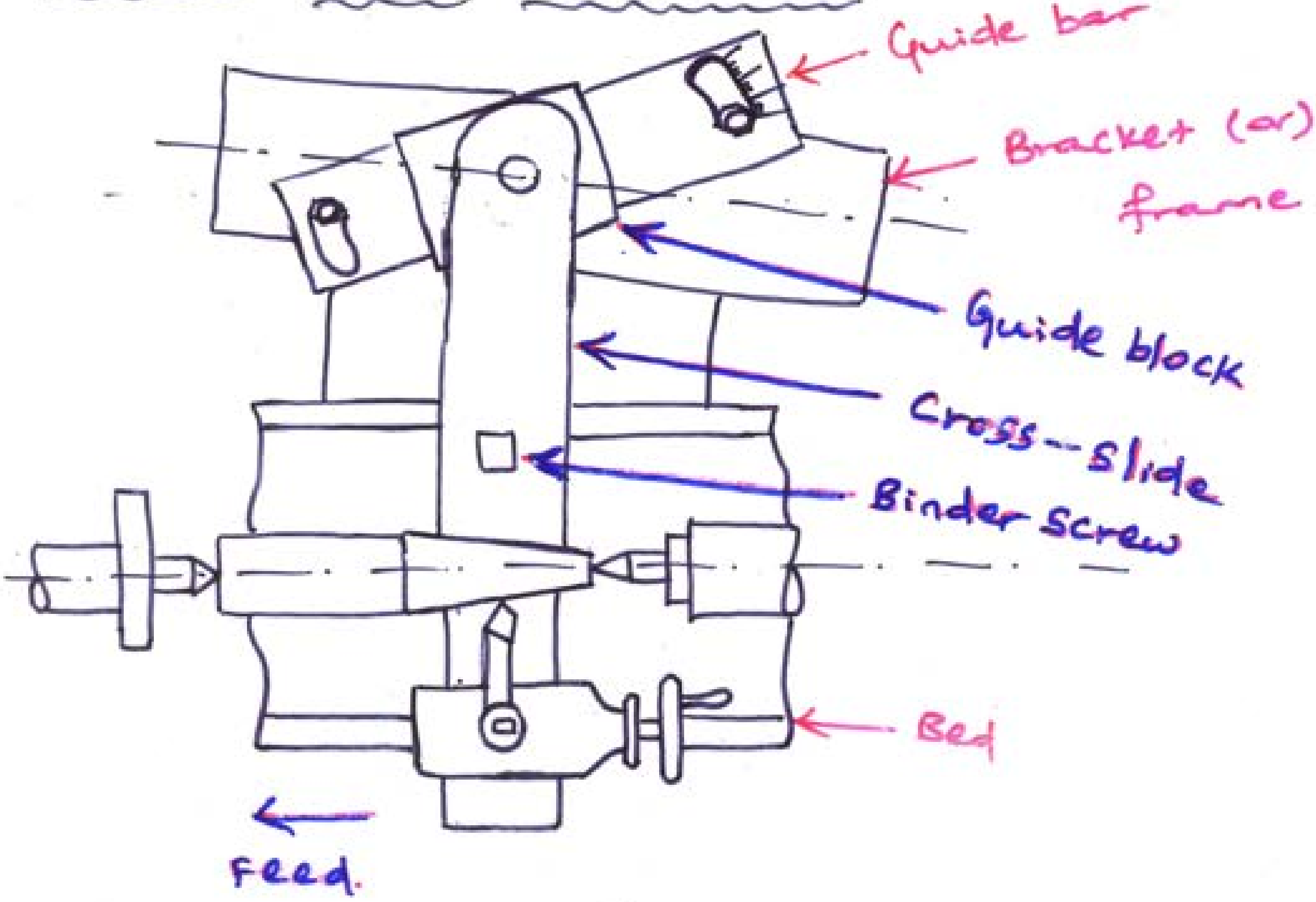


[Combination of cross & longitudinal feed]

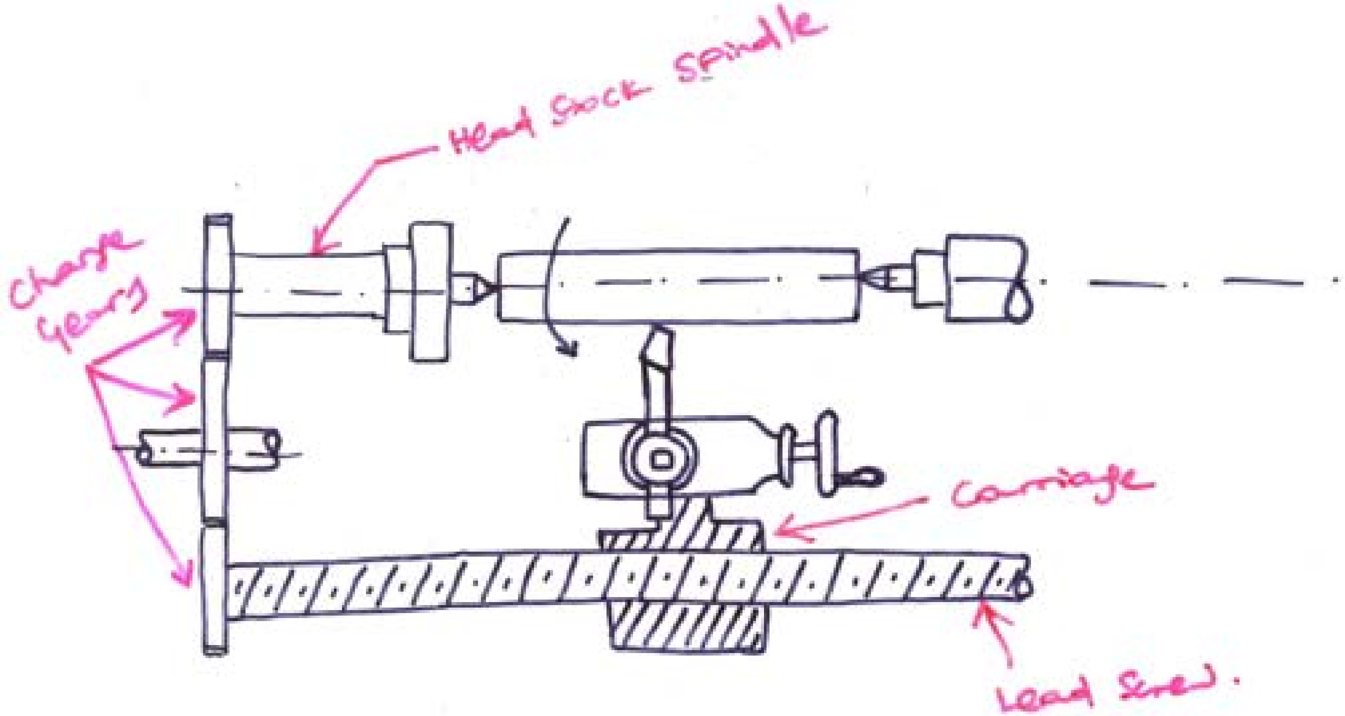
(e) Taper turning Attachment:-



(e) Taper turning Attachment:-



thread cutting:-



formula for calculating the change wheels (gears):-

① $$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{lead Screw speed}}{\text{Spindle speed}} = \frac{\text{Pitch of Screw to be cut}}{\text{Pitch of the lead Screw}}$$

In english measurements

② $$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{thread per inch [t.p.i] of w/p}}{\text{thread per inch [t.p.i] of leadscrew}}$$

Here, $$\text{pitch} = \frac{1}{\text{no of thread/inch}}$$

Metric thread on English leadscrew:—

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch on the work}}{\text{Pitch on the lead screw}}$$

$$= \frac{P}{\left[\frac{1}{n} \times \frac{127}{5}\right]}$$

$$\boxed{\frac{127}{5} = 25.4\,mm}$$

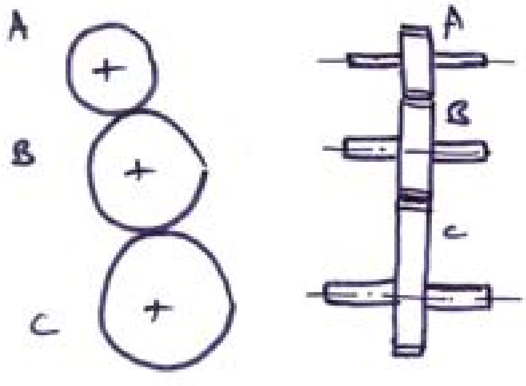
$$\boxed{\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{5Pn}{127}} \quad (3)$$

1 inch $= 25.4\,mm$

$$= \frac{25.4 \times 5}{5} = \frac{127}{5}$$

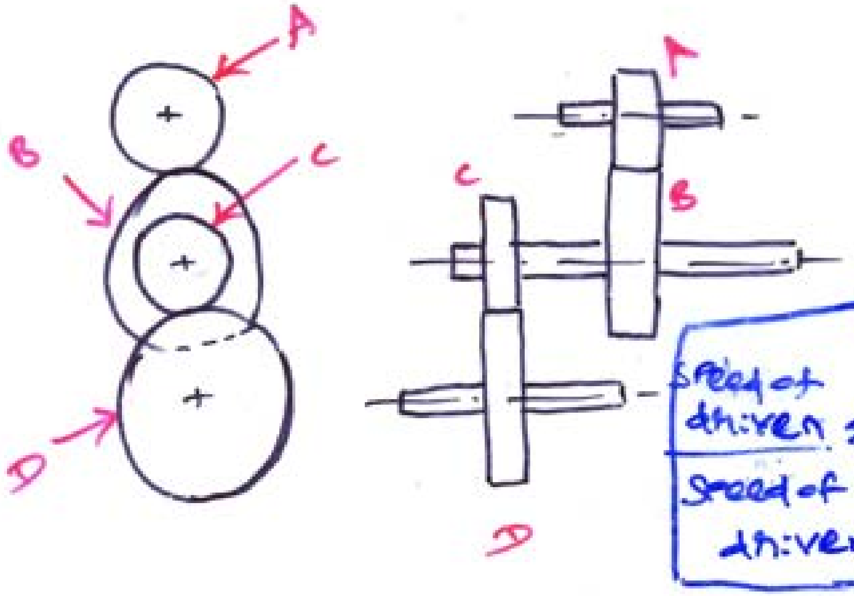
* So, the cutting of metric thread on a Lathe with a English lead screw may be carried out by introducing a translating gear of 127 teeth.

Simple and Compound Gear train:—



B → Intermediate Gear

Simple Gear train



A, C $\Rightarrow$ Driver

B, D $\Rightarrow$ Driven

$$\boxed{\frac{\text{Speed of driven}}{\text{Speed of driver}} \Rightarrow \frac{A}{B} \times \frac{C}{D}}$$

driven – lead screw

driver – Spindle

Compound Gear train

Often engine lathe are equipped with a set of gears ranging from 20 to 120 teeth in a steps of 5 teeth and one gear with 127 teeth

Problems:–

① The pitch of a lead screw is 6mm & the pitch of the thread to be cut 1 mm. find Change gears

$$= \frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{pitch of work}}{\text{Pitch of lead screw}}$$

$$= \frac{1}{6} = \frac{1 \times 20}{6 \times 20} = \frac{20}{120} \left[\text{Simple Gear train} \right]$$

So, Driver gear should have 20T & Driven gear should have 120 teeth

(2) Pitch of lead screw – 6 mm and pitch of thread to be cut is 1.25 mm. Find the change wheels

Soln:–

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of work}}{\text{Pitch of lead screw}} = \frac{1.25}{6}$$

$$= \frac{1.25 \times 4}{6 \times 4} = \frac{5}{24}$$

$$= \frac{5 \times 5}{24 \times 5} = \frac{25}{120} \quad \left[\text{Simple Gear train}\right]$$

(or)

$$= \frac{5 \times 1}{6 \times 4} = \frac{5 \times 10}{4 \times 10} \times \frac{1 \times 20}{6 \times 20}$$

$$= \frac{50}{40} \times \frac{20}{120} \left[\frac{A}{B} \times \frac{C}{D}\right]$$

[Compound Gear train]

∴ Driver A & C = 50T & 20T

Driven B & D = 40T & 120T

(3) It is required to cut a screw having 7 mm pitch on a lathe having a leadscrew of 4 threads per inch. Calculate gears.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{5p_n}{127} = \frac{140}{127}$$

$$= \frac{70 \times 2}{127} = \frac{70 \times 2 \times 20}{127 \times 20} = \frac{70}{127} \times \frac{40}{20}$$

Steps involved in thread cutting opreation:—

1. Remove the excess material from the w/p to make its diameter equal to the major diameter of the thread.

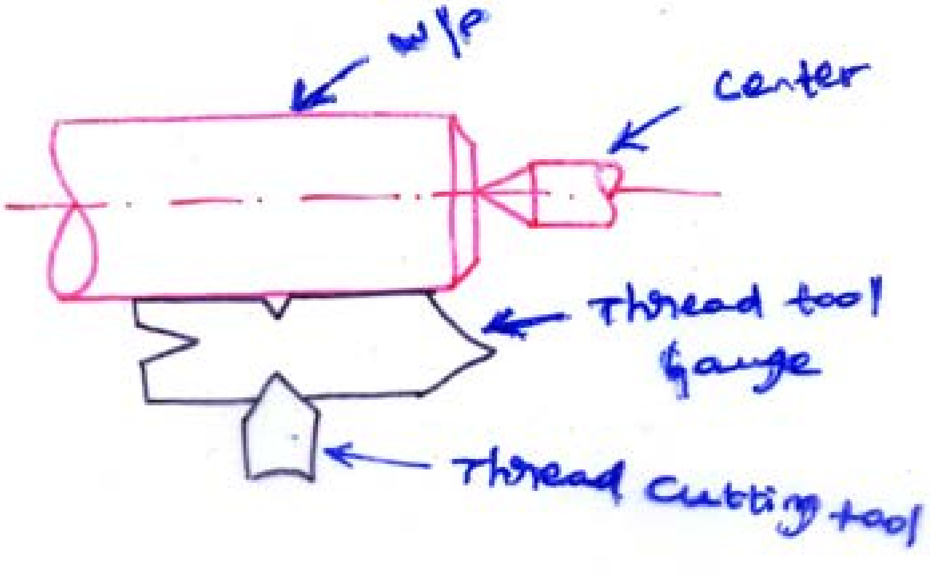
2. Change gears of correct size are then fitted b/w the spindle and the Lead Screw.

3. Choose the cutting tool based on the shape (or) form of the thread.

4. For a metric thread, the included angle b/w the cutting edge of the tool is 60°.

5. The nose of the tool should be set at the same height as the center of the w/p.

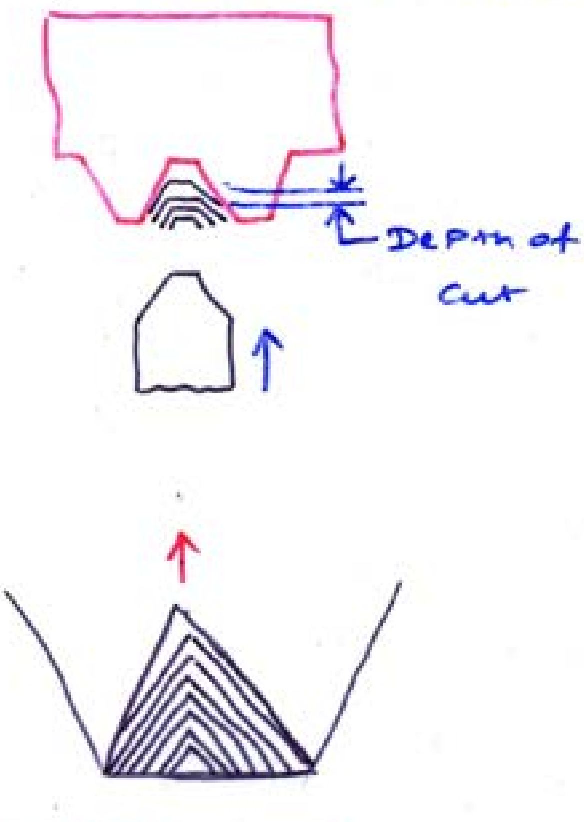
6. A thread tool gauge is usually used against the turned surface to check the cutting tool so that each face of the tool is equally inclined to the centerline of the workpiece



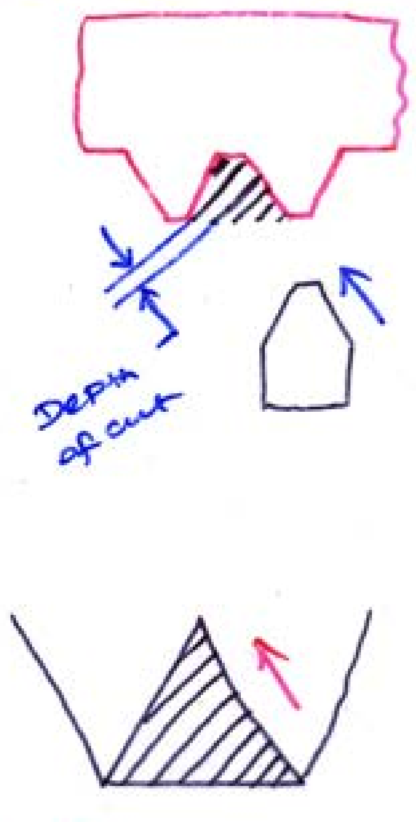
7. The speed of the spindle is reduced by one-half to one-fourth of the speed required for turning according to the type of material being machined.

8. The depth of cut which usually varies from 0.05 – 0.2 mm is applied by advancing the tool $\perp^r$ to the axis of the work (or) at an angle equal to one-half of the angle of the thread and 30° in case of metric thread by swivelling the compound rest.

Different Methods of Applying Depth of cut



Applying Depth of cut $\perp^r$ to the lathe axis by Cross-Slide



Applying Depth of cut inclined to the axis by compound rest

(9) After the tool has produced "helical groove" upto the end of the work, this is quickly withdrawn from the cross-slide and the tool is brought back to the starting position to give a fresh cut.

(10) Before giving a fresh cut, it is necessary to ensure that the tool is at the starting point of the thread. otherwise the job will be spoiled. Several cuts are necessary before the full depth of thread is reached.

(11) Making the tool being at the start point of the thread when the fresh cut is given is called "pick-up of thread".

(12) Based on the "pick-up of thread", thread cutting methods can be classified as

1. Thread cutting by reversing the m/c
2. Thread cutting by marking the lathe parts
3. thread cutting using chasing dial (or) thread indicator
4. thread cutting by thread chaser

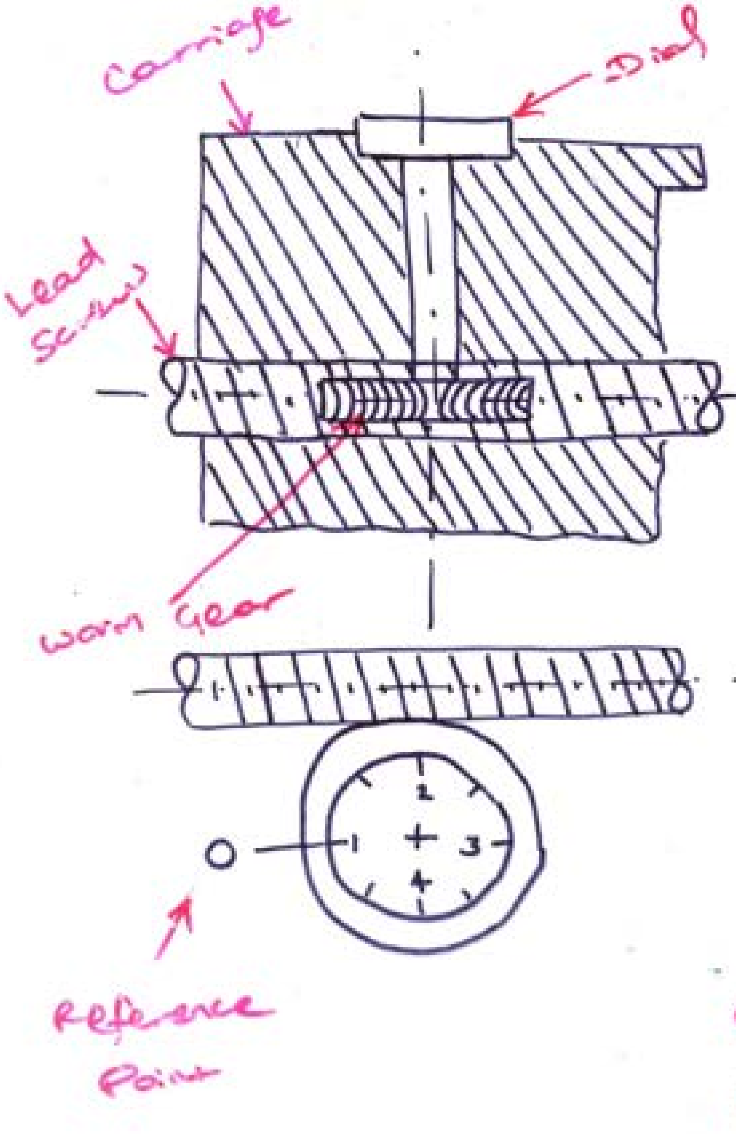
① Reversing the m/c:—

After the end of one cut, the tool is brought back to the starting position by reversing the machine keeping half-nut permanently engaged. This method requires considerable time.

② Marking the lathe part:—

Marking lead screw, Gear wheels starting position of the carriage on the bed.

③ Using Chasing dial (or) thread Indicator:—



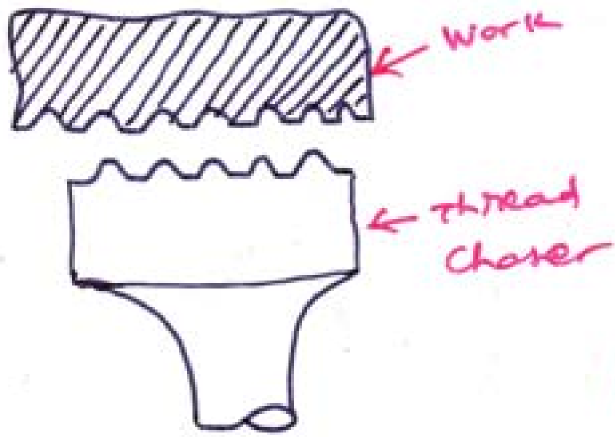
① Chasing dial (or) Thread Indicator consists of a worm gear which is in mesh with a lead screw.

② With a worm Gear attached a vertical shaft connecting the dial gauge.

③ When the lead screws rotates, the dial gauge will also rotate.

④ By using this, the start point of the tool can be easily identified.

(4) Thread cutting by using thread chaser:-



(1) A chaser is a multi point threading tool having the same form and pitch of the thread to be chased.

(2) It is used to finish a partly cut thread to the size & shape required

(3) Thread chasing is done $\frac{1}{3}$ to $\frac{1}{2}$ of the speed of the turning.

Classification of threads:-

(1) Based on the standards

(1) Whitworth [British Standard]

Depth = 0.6403 × P

Angle = 55° to the Lathe axis

(2) British Association [BA]

Depth = 0.6 × P

Angle = 47.5°

(3) Metric thread [ISO]

Max Depth = 0.7035 × P

Min Depth = 0.6855 × P

Angle = 60°

(4) ACME [American Standards]

Height of thread = 0.5P + 0.254 mm

Angle = 29°

(2) Based on the pitch of the lead screw & work

(1) Even thread

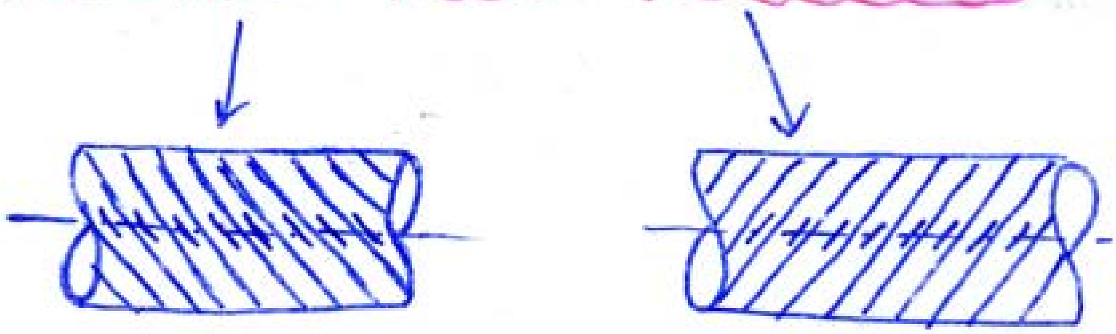
- pitch of work is an multiple of the pitch of lead screw.

(2) Odd thread

- pitch of work is not an multiple of pitch of lead screw.

(3) Based on the thread position

Right hand & Left hand thread



(4) Single Start & Multi-Start threaded Screw

Single Start threaded Screw :-

For one complete turn round the screw, if there is a movement of one thread, the screw is called single start thread.

* "Lead" is the distance a screw thread advances along its axis in one turn.

* So, In single start thread, Lead = Pitch

$$\boxed{\text{Lead} = \text{No. of Start} \times \text{Pitch}}$$

Multi-Start thread:-

* For one complete turns when there is a movement of more than one thread it is called multi Start thread.

* In case of, say, a three-Start thread, for one complete turn, the thread advances three time as if it was a single thread.

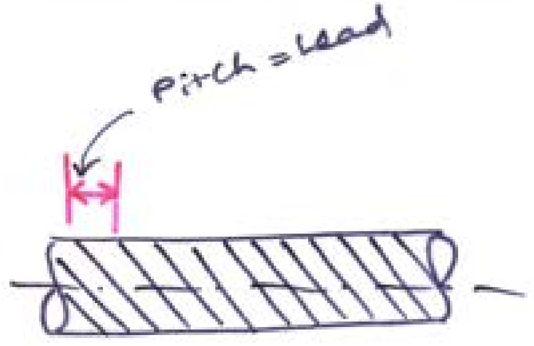
Here $\boxed{\text{Lead} \neq \text{pitch}}$

$\boxed{\text{Lead} = 3P}$ [for three Start thread]

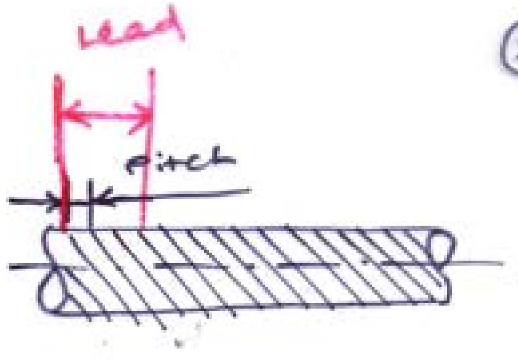
Application:-

1. House hold faucet's & taps
2. Milk bottles & water bottles
3. Medicines. etc.

$$\boxed{\frac{\text{Driver Gear}}{\text{Driven Gear}} = \frac{\text{Lead of the work}}{\text{Pitch of the Lead Screw}}}$$

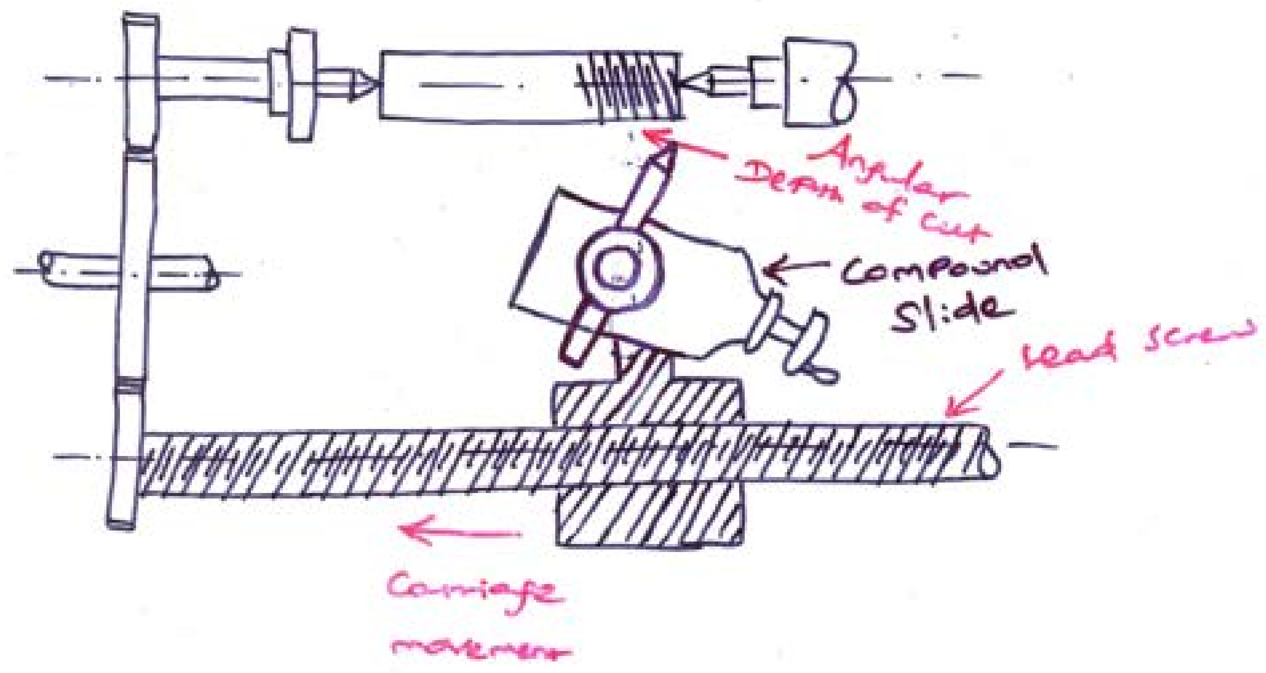


Single Start thread



Three Start thread

Thread cutting using Compound Slide

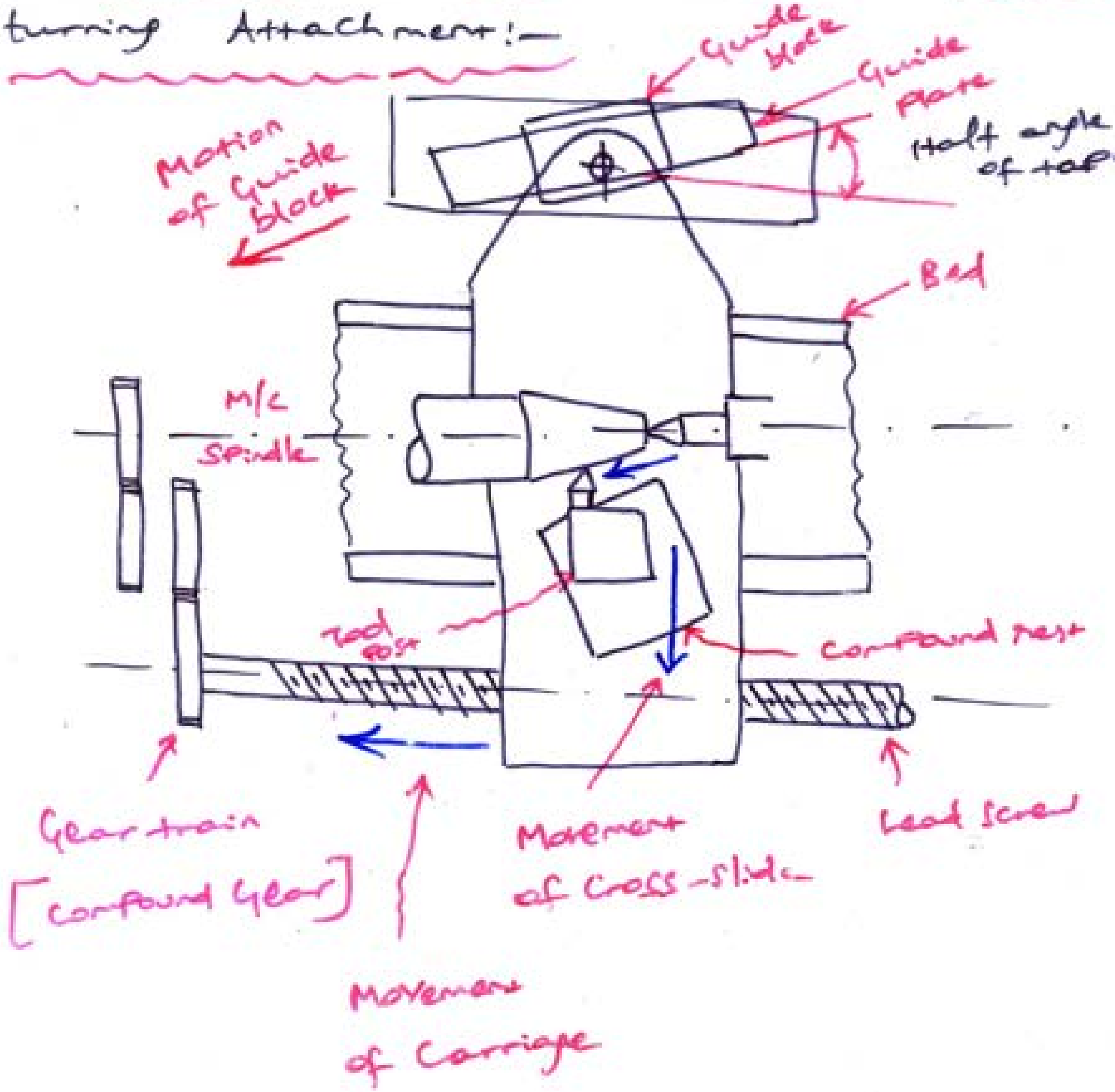


* when cutting Right hand thread, the carriage must move towards the tailstock.

* For Left hand thread, the Carriage must move away from the Head stock and towards the tail Stock.

* The job moves as always in anticlockwise direction when viewed from the tailStock.

Thread cutting on a tapered Surface using taper turning Attachment:-



Checking a Screw Cutting Set-up:-

1. The Gear train must be correct.
2. Tumbler gears must give the carriage the movement in the right direction
3. The Spindle should be arranged to give required low Cutting speed.
4. The feed shaft must be disengaged
5. The apron feed mechanism must be at neutral
6. Finally Half-nut must be engaged to the lead Screw.

Special Attachments:-

① Milling Attachment:-

ⓐ For cutting grooves & keyways

w/p – Cross slide; Milling cutter – Chuck

Depth of cut – Provided by vertical slide in the Attachment.

ⓑ For cutting multiple grooves & gear wheels:-

w/p – held b/w centers;

Attachment – Cross-slide; [Driven by separate motor]

No of grooves – by rotating the work

Gear – Universal Dividing head attached to rear end of the spindle.

② Grinding Attachment:-

* Abrasive wheel called as "Grinding wheel".

* work – b/w centers for external grinding.

* work – Chuck (or) face plate

* Depth of cut – by moving cross-slide

Calculation of Machining time & Power for cutting:—

(3.)

Formulaes:—

Turning operations

① Cutting speed, $V = \frac{\pi D N}{1000}$ m/min

V - Cutting speed in m/min
D - Diameter of w/p in mm
N - Rotational speed of w/p in rpm.

② Diameter of w/p, $D = \frac{D_1 + D_2}{2}$ in mm

D_1 - Max (or) Blank Diameter
D_2 - Diameter after m/cing
(or) Minimum Diameter of w/p

③ M/cing Time for single pass, $t = \frac{L + L_0 + L_1}{fN}$ min

L = Length of job
L_0 = Over travel beyond the length of the job to help in setting of the tool in mm
L_1 = Tool Approach Distance in mm
f = feed rate in mm/rev
N = Rotational speed in rpm.

④ Power required for cutting, $P = K \times d \times f \times v$ watts

K = Constant depending on the work mat'l

Material being cut	K (N/mm^2)
Steel, 100 – 150 BHN	1200
Steel, 150 – 200 BHN	1600
Steel, 200 – 300 BHN	2400
Steel, 300 – 400 BHN	3000
Cast Iron	900
Brass	1250
Bronze	1750
Aluminium	700

d = depth of cut in mm;

f = feed rate in mm/rev;

v = cutting speed in m/sec;

Problem :-

AU – May – 2012

(1) A Blank 180 mm long and 70 mm diameter is to be machined in a lathe to 175 mm long and 60 mm diameter. The workpiece rotates at 450 r.p.m, the feed is 0.3 mm/rev; and the maximum depth of cut is 2 mm. For turning operation, the approach plus over travel distance is 6 mm. Assuming that the facing operation is done after the turning, calculate the machining time

Soln:-

Given:-

Length of w/p, $L = 180mm$;

Dia of blank, $D_1 = 70mm$;

Dia of w/p after machining, $D_2 = 60mm$;

feed rate, $f = 0.3$ mm/rev

Depth of cut, $d = 2mm$;

Approach + over travel $(L_0+L_1) = 6mm$;

Turning.

Dia of w/p, $D = 70mm$;

Length of Tool travel, $L = D/2 + (L_0+L_1)$

$= 35 + 6$

$$\boxed{L = 41mm}$$

Cross feed rate for facing is not Given

So, Assume, $f = 0.3$ mm/rev;

Soln:-

M/c'ing time for turning opn:-

M/c'ing time, $t = \dfrac{L + L_0 + L_1}{fN}$ minutes

$$N = \frac{1000V}{\pi D} \text{ rpm} = 450 \text{ r.p.m}$$

$\therefore$ Machining time for turning $\Big\}$ $t = \dfrac{180+6}{0.3 \times 450}$

$\boxed{t = 1.37 \text{ minutes}}$ for one pass

$\boxed{\text{No of passes required} = 3}$

M/cing time for facing opern $\Rightarrow$ $\dfrac{(D_1 - D_2)/2}{\text{Depth of cut}}$

$$t = \frac{L}{fN} \text{ minutes}$$

$$L = \frac{60}{2} + (l_0 + l_1)$$

$$= \frac{60}{2} + 6 = 36 \text{ mm}$$

$$\therefore t = \frac{36}{0.3 \times 450} = 0.26 \text{ minutes} \simeq 0.3 \text{ minutes}$$

$\boxed{\text{No of passes required} = 3}$

$\therefore$ ~~Total~~ m/cing time for one pass $\Big\}$ $= 1.37 + 0.3 = \underline{1.67 \text{ minutes}}$

Total M/cing time for complete m/cing $\Big\}$ $= (1.37 \times \underset{\text{No of passes}}{3}) + (0.3 \times 3)$

$= 4.11 + 0.9$

$= \underline{5.01 \text{ minutes}}$

Formulaes:-

(4) Number of passes required $= \dfrac{[D_1 - D_2]/2}{\text{Depth of cut } (d)}$

D_1 – Max (or) Blank dia (mm)

D_2 – Minimum Diameter (mm)

(5) Total Machining time $= \begin{bmatrix} \text{M/cing time for} \\ \text{Single pass} \end{bmatrix} \times \begin{bmatrix} \text{no of passes} \\ \text{required} \end{bmatrix}$

(6) For facing operation

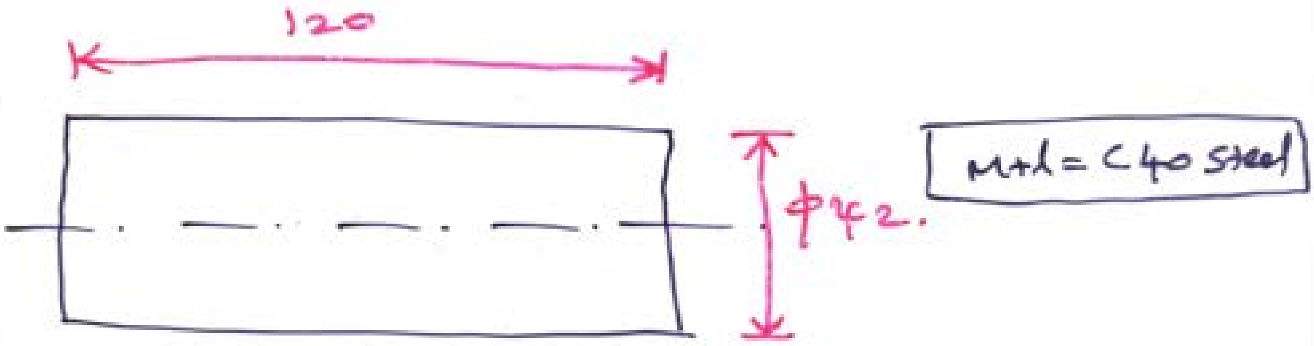
Diameter of w/p = Dia of blank diameter (or) Dia of finished w/p

Length of Tool travel $= D/2 + (L_0 + L_1)$

$(L_0 + L_1)$ = Approach + over travel

Problem:- 2

Estimate the actual machining time required for a component shown in figure. The available spindle speeds are 70, 110, 176, 280, 440, 700, 1100, 1760, and 2800. Use a roughing speed of 30m/min and finish speed of 60m/min the feed for roughing is 0.24 mm/rev, while that for finishing is 0.10 mm/rev. the maximum depth of cut for roughing is 2 mm. Finish allowance may be taken as 0.75 mm. Blank to be used for machining is 50 mm diameter



Soln:-

$$\text{Stock to be removed} = \frac{50-42}{2} = 4\,mm$$

$$\text{Finish allowance} = 0.75\,mm$$

Roughing:-

$$\text{Roughing stock} = 4 - 0.75 = 3.25\,mm$$

$$\therefore \text{No of passes} = \frac{3.25}{2} = 2 \text{ passes}$$

Given cutting speed, $v = 30$ m/min

$$\text{Avg Diameter, } D = \frac{50+42}{2} = 46\,mm$$

$$\therefore \text{Spindle speed, } N = \frac{1000 \times 30}{\pi \times 46} = 207.59 \text{ rpm}$$

Nearest rpm from the list, $N = 176$ r.p.m

$$\text{M/c time for one pass} = \frac{120+2}{0.24 \times 176} = \underline{\underline{2.898 \text{ mts}}}$$

Finishing:-

Given cutting speed, $v = 60$ m/min

$$\text{Spindle speed, } N = \frac{1000 \times 30}{\pi \times 42} = 439.05 \text{ rpm}$$

Nearest rpm from the list $= 440$ rpm

$$\text{M/cing time for one pass} = \frac{120+2}{0.10 \times 440} = 2.77 \text{ mts}$$

$$\therefore \text{Total m/cing time} = (2 \times 2.888) + (2.77 \times 1)$$

$$= 8.546 \text{ mts}$$

Power required for roughing cycle:-

Given feedrate, $f = 0.24$ mm/rev

Depth of cut, $d = 2$ mm

$$\text{Cutting speed, } v = \frac{\pi \times 176 \times 46}{1000} = 25.43 \text{ m/min}$$

The value of K from Table } $K = 1600$ N/mm^2

$$\therefore P = \frac{1600 \times 25.43 \times 0.24 \times 2}{60} = 325.5 \text{ watts}$$

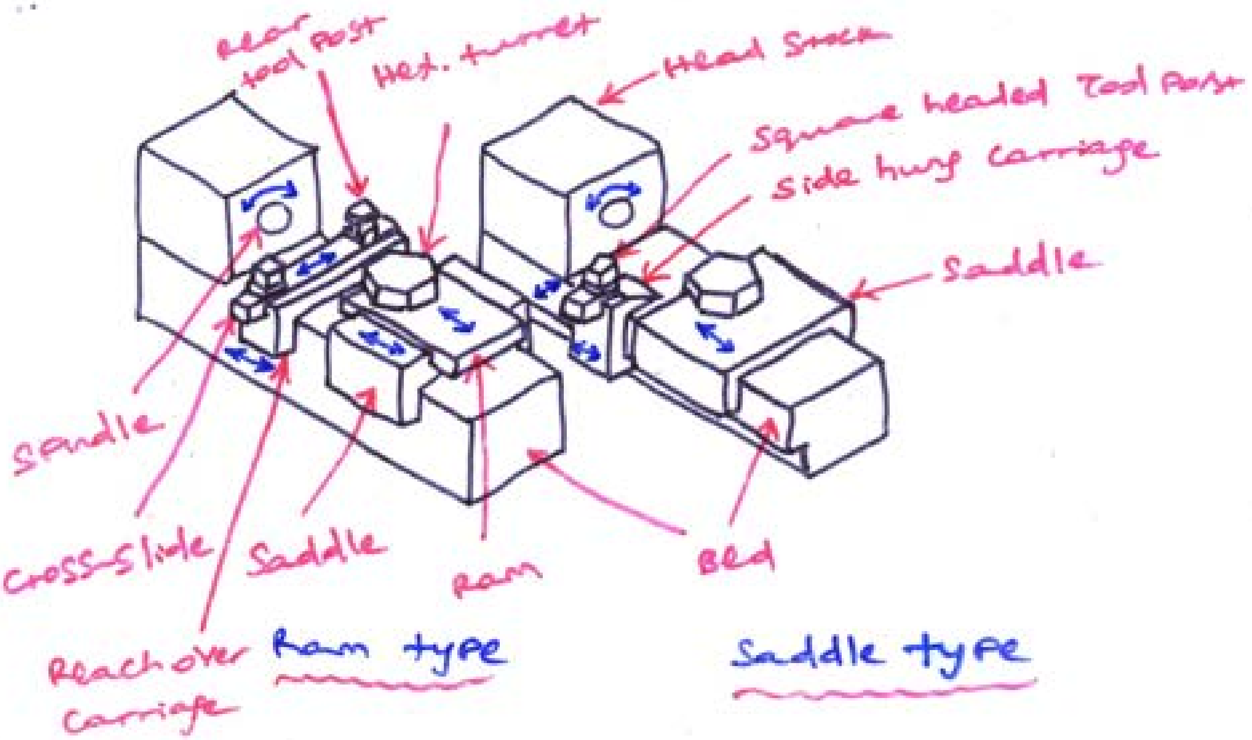
$$\boxed{P = 0.326 \text{ kW}}$$

$$\boxed{P = K \times d \times f \times v \text{ watts}}$$

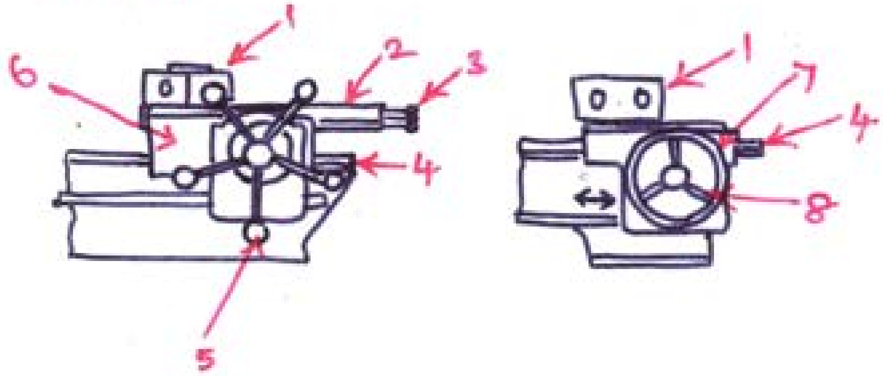
CAPSTAN & TURRET LATHE:-

* Development of Engine Lathe
* Semi-Automatic
* Production Lathe
* Developed in U.S.A by Pratt & Whitney in 1860

Parts:- Bed, All Geared Head Stock, Saddle, Carriage, Hexagonal turret, Adjustable stoppers.



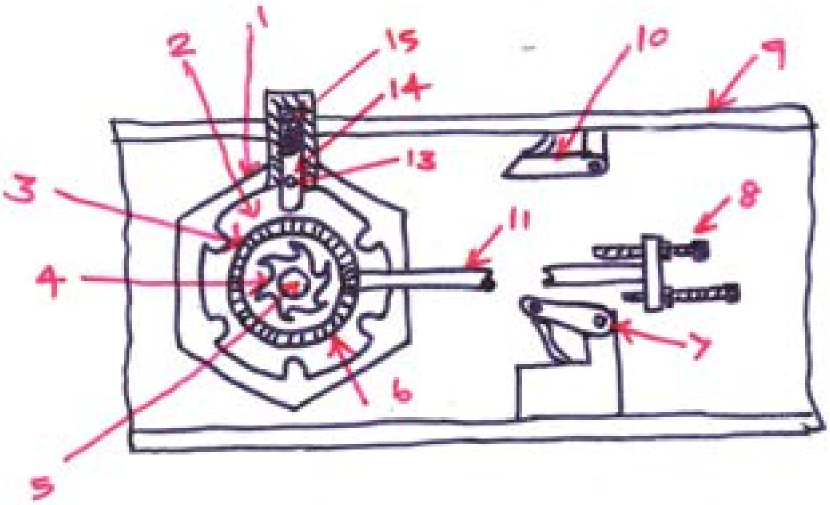
Ram type   Saddle type



Capstan & turret Lathe to show their difference

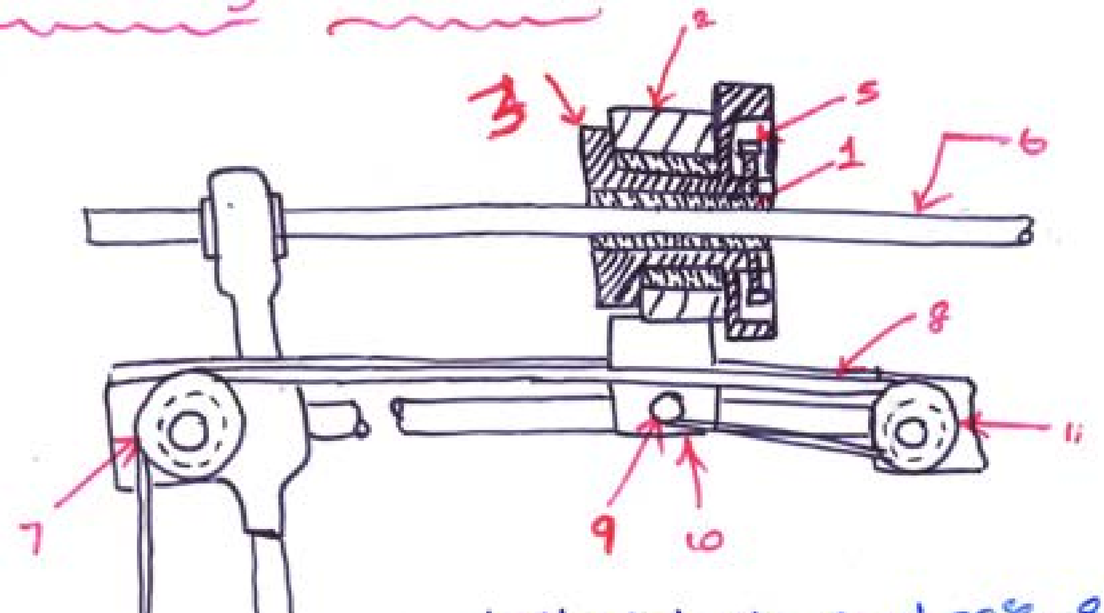
① Hexagonal turret ② Ram ③ Adjustable stoppers ④ Bed ⑤ Handwheel for Ram ⑥ Saddle ⑦ Turret Saddle ⑧ Handwheel for Saddle.

Turret Indexing Mechanism:-



1. Hexagonal turret 2. Index plate 3. Bevel gear 4. Indexing ratchet
5. Turret Spindle 6. Beveled pinion 7. Indexing pawl
8. Screw type stop rod 9. Lathe bed 10. Cam 11. pinion shaft
13. Plunger pin 14. Plunger 15. Plunger Spring.

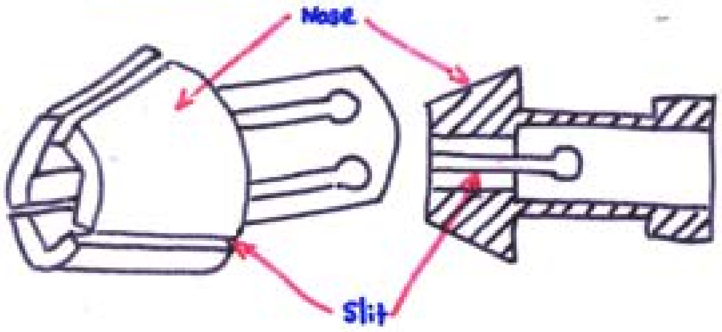
Bar feeding Mechanism:-



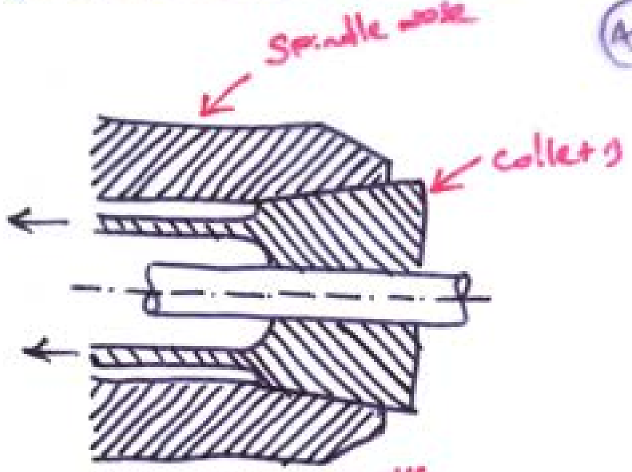
1-Chuck bush 2- Loose Sleeve
3- Bar Chuck 4- weight
5. Bar Chuck Set Screw 6. Bar
7, 11 - Pulley 8. Chain
9- pin on Sliding bracket
10- Sliding bracket

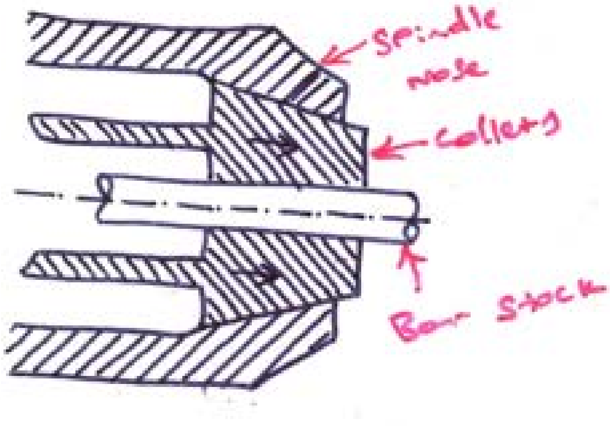
Work Holding Devices:-

① Collets

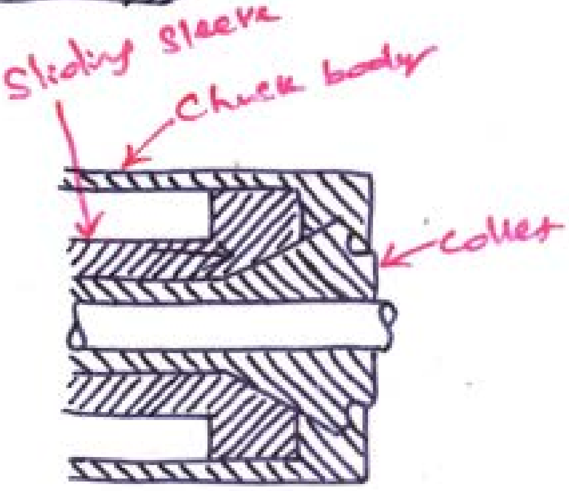


Draw back collet

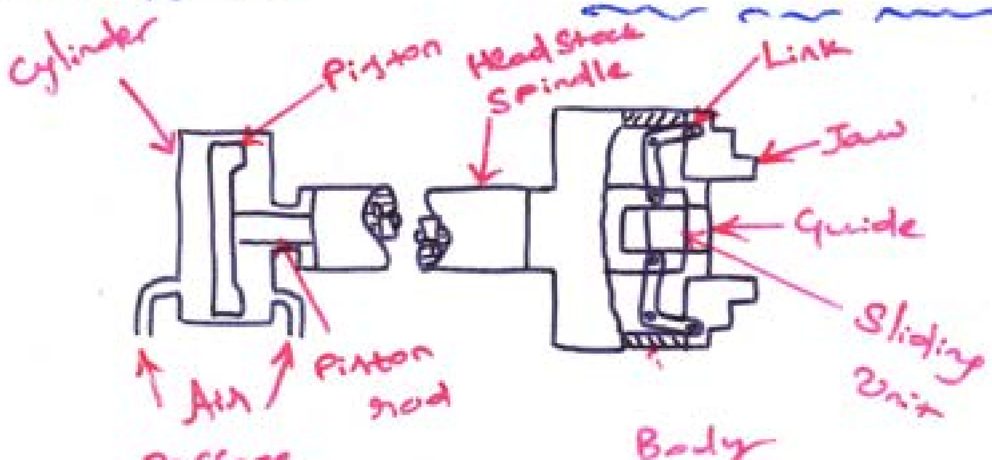




Push out collet



Dead length collet



Power Chuck

Tool Layout:-

(A1)

Planning of Sequence of opreation & preparation of turret & capstan Lathe as per the Sequence of oprn to be carried out on the work.

It includes three stages.

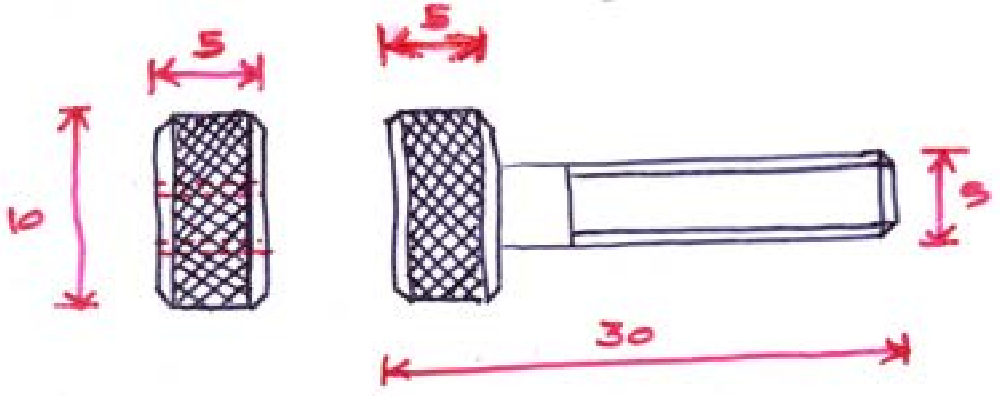
1. Preparation of opreation sheet (or) Process planning sheet.
   * component drawing
   * Sequence of oprn
   * selection of Tool
   * selection of V, f, d for each oprn
   * Tool Travel length
2. Tool layout
3. Tool Sequence Chart

Feed, depth of cut, cutting speed for various oprn:-

S.no	Opreation	Mild steel			Aluminium		
		V	f	d	V	f	d
1.	Plain turning	50	0.10	2	200	0.10	2
2.	Form turning	40	0.05	–	200	0.05	–
3.	Thread cutting	7	0.05	2	7	0.05	2
4	Drilling	40	0.05	–	200	0.05	–

V – m/min
f – mm/rev
d – mm

Draw the tool layout for manufacturing knurled screw and nut as shown in figure on a turret lathe (42)



Soln:-

Stage:-1 [Preparation of operation sheet]

1. The component drawing is drawn
2. The total length of the work is calculated & 14mm is provided for clearance

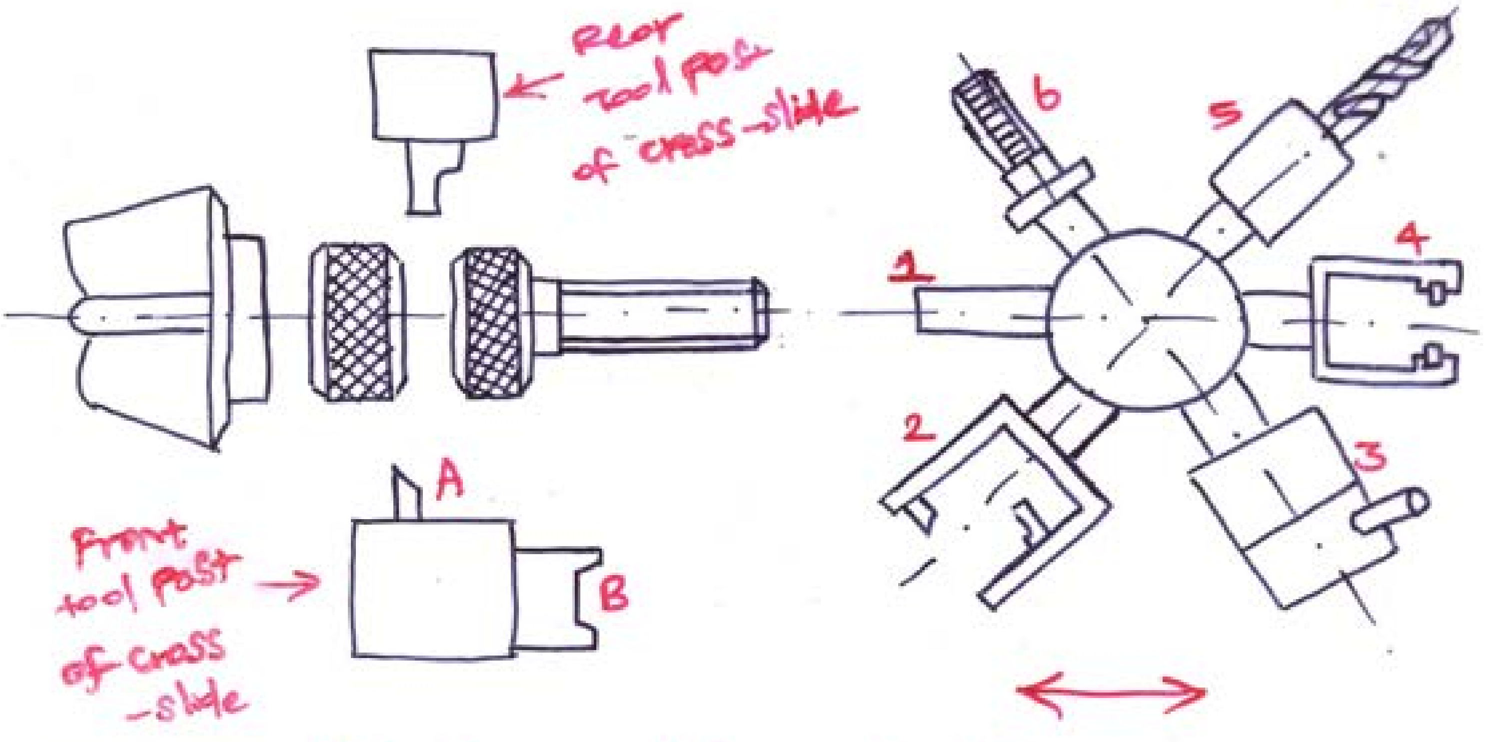
   4+4 = Clearance for parting tool

   6 = clearance b/w chuck & parting tool.

3. Number of operation is listed.
4. Sequence of operation is listed.
5. Material - Mild steel.
6. Tool is selected & Tool length is adjusted by stopper

Stage:-2 [Tool-layout]

Tool layout is drawn as shown in figure



Stage:-3 [Tool Scheduling Chart]

Machine:- turret lathe

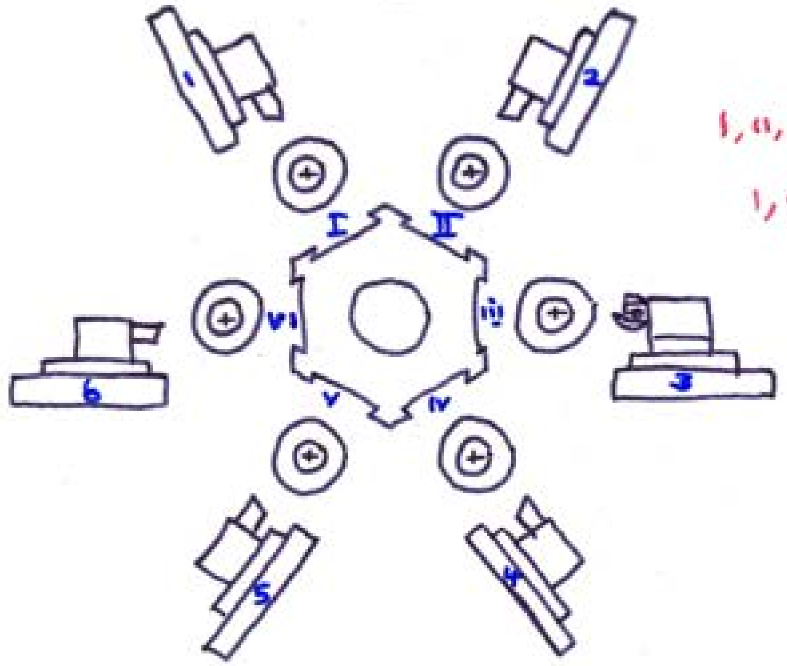
Material:- Mild steel

Operation sequence	Description of operation	Tool position	Tools	V (m/min)	f (mm/rev)	d (mm)
1.	Holding the required bar in the collet & setting the required length of 52 mm [35+14+3]	Turret position 1	Bar stop	–	–	–
2.	Turn to 10mm dia to a length of 46mm from right end	Front tool post [Position A]	Turning tool	50	0.1	2

Operation Sequence	Description of operation	Tool Position	Tools	V [m/min]	f [mm/rev]	d [mm]
3.	Turn to 5mm dia & from the right end of the bolt for a length of 26.5mm	Turret Position 2	Roller steady box Turning tool	50	0.1	2
4.	Facing the right end to a length of 1.5mm	Front Tool Post (Position A)	Turning tool	50	0.1	0.5
5.	External thread Cutting of 5mm dia to a length 23mm from the right end	Turret Position 3	Self opening die head with thread Chaser	7	0.05	2
6.	Knurling on the required length	Turret Position A	Knurling tool	7	0.05	0.5
7.	Chamfering the bolt head	Front Tool Post B	Chamfering Tool (Form Tool)	40	0.05	-
8	Chamfering the end of bolt	A	Turning tool	50	0.1	2
9	Parting off the bolt	Rear Tool Post	Parting off Tool (In inverted Position)	40	0.05	-
10	Facing the right side of the nut to 1.5 mm	Front Tool Post Position 'A'	Turning tool	50	0.1	0.5
11.	Drilling the nut	Turret Position '5'	Drilling Tool	40	0.05	-

Operation Sequence	Description of operation	Tool Position	Tools	v	f	d
12	Threading by Tap	Turret Position '6'	Tapping Tool	40	0.05	–
13	Parting off the nut	Rear Tool Post	Parting off Tool	40	0.05	–

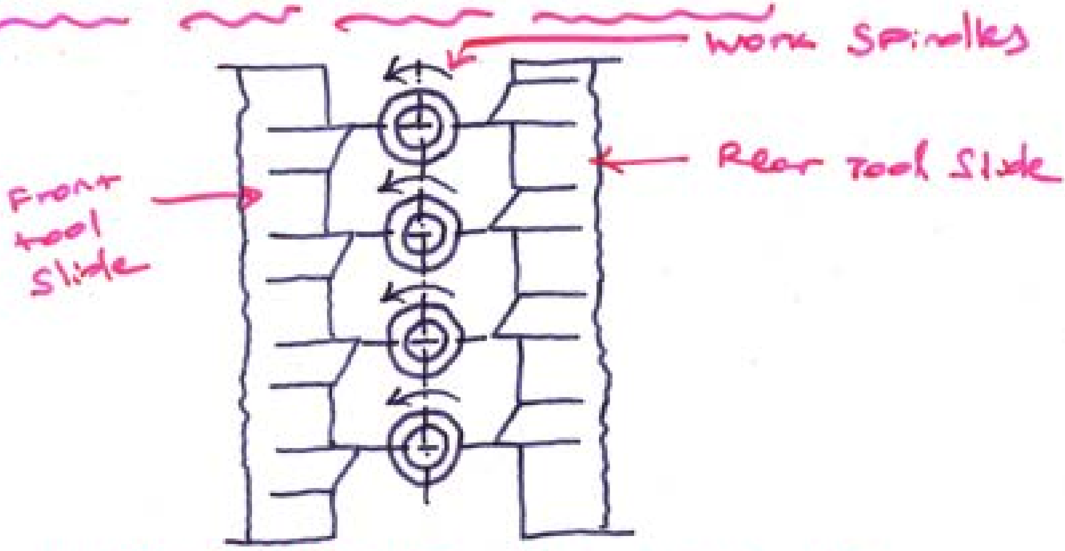
Multi-Spindle Automatic Lathes:—



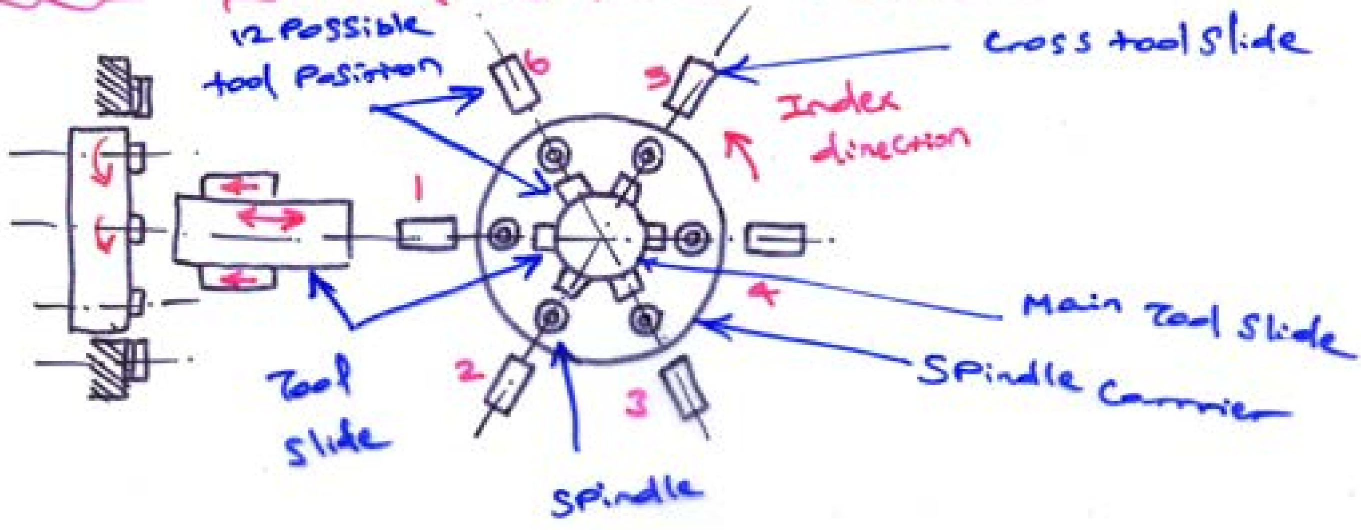
I, II, III, IV, V, VI – Spindles

1, 2, 3, 4, 5, 6 – Cross slides

Parallel Action Multi Spindle Automats



Progressive Action Multi Spindle Automats



Comparison of Single Spindle & multi Spindle Auto Lathe:-

Sl no	Single Spindle	multi-Spindle
1.	one Spindle	2, 4, 5, 6 (or) 8 Spindle
2.	Only 1 w/p machined at a time.	no of workpieces machined at a time.
3.	Production rate is low	High
4.	Accuracy is high	Lower.
5.	Tool setting time is less	More
6.	Tooling cost is less	more
7.	It is more economical for shorter as well as longer runs	It is more economical for longer runs only.
8.	The time required to produce one component is sum of the turret operation times.	Time of longest cut in any Spindle
9.	Tools in turret are indexed.	w/p held in Spindle are indexed. (progressive action type)

Comparison b/w Parallel Action & Progressive Multi Spindle Lathe:—

Sl no	Parallel Action Machine	Progressive Action Machine
1.	Same opreations in all spindles	Differen oprns on jobs one after the another
2.	In one cycle, no of components produced simultaneously is equal to no of spindles	For every indexing of component, one component is produced.
3.	Rate of production is high	Moderate.
4.	If Anything goes wrong in one station, the production in that particular station is only affected.	The production is completely affected in all station.
5.	Small parts of simple shapes are produced.	Parts of complicated shapes

Unit-II

SHAPER, MILLING & GEAR CUTTING Machines

Introduction:-

① The shaper is a reciprocating type of machine tool intended primarily to produce flat surfaces. These surfaces may be horizontal, vertical (or) inclined.

② Feed of a shaper is measured in terms of mm/stroke. It is defined as the movement of the cutting tool (or) workpiece per stroke of the cutting tool.

③ The cutting type of a shaper is a intermittent type as the cutting tool cuts the workpiece periodically during its stroke.

Classification of a Shaper:-

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

a) Crank type b) Geared type c) Hydraulic type

② According to the position & travel of the ram

a) Horizontal type b) Vertical type

c) Travelling head type

(2)

③ According to the movement given to the table

a) Standard shaper b) Universal shaper

④ According to the type of cutting stroke

a) Push type b) Draw type.

Crank Shaper:-

This is the most common type of shaper in which a single point cutting tool is given a reciprocating motion equal to the length of stroke desired while the work is clamped on an adjustable table. This shaper employs a crank mechanism to change a circular motion of a large gear called "bull gear" incorporated in a machine to reciprocating motion of the ram. The bull gear receives power from an electric motor.

Geared type:-

Here, the reciprocating motion of the ram is obtained by a rack & pinion. The rack teeth which are cut below the ram mesh with a spur Gear. This type of shaper is not widely used.

(3) Hydraulic Shaper:-

Here, the reciprocating motion to the ram is obtained by Hydraulic power. Oil under high pressure is pumped into the hydraulic cylinder fitted with a piston. The end of the piston rod is connected to the ram. The high pressure oil first acts on one side of the piston and then on the other causing the piston to reciprocate and the motion is transmitted to the ram. The piston speed is changed by varying the amount of liquid delivered by the pump.

Advantages of a hydraulic Shaper:-

(1) It offers great flexibility in speed control.

(2) It Eliminates shock by permitting slowing up of motion when the cutting tool is overloaded, thus protecting the parts (or) the tools from breakage

(3) The machine does not make any noise and opreates very quietly.

(4) Horizontal Shaper:-

In a horizontal shaper, the ram holding the tool reciprocates in a horizontal axis. This shaper is mainly used to produce flat surfaces.

(5) Vertical Shaper:-

In a Vertical Shaper, the ram holding the tool reciprocates in a vertical axis. It may be crank driven, rack driven (or) hydraulic driven. The work table of a vertical shaper can be given cross, longitudinal & rotary movement. The tool used in a vertical shaper is entirely different from Horizontal Shaper. Vertical Shaper is very convenient for machining

a) Internal surfaces such as keyways, slots (or) Grooves.

b) Large Internal & external Gears by indexing arrangement of Rotary table

6 Travelling head shaper:-

1. In this shaper, the ram carrying the tool while it reciprocates moves cross-wise to give the required feed.

2. Heavy jobs which are very difficult to hold on the table of a standard shaper and fed past the tool are held on the basement of this shaper while the ram reciprocates and supplies the feeding movements.

7 Standard shaper:-

A shaper is termed as standard (or) plain when the table has only two movements for table.

1. Vertical feed 2. Horizontal feed.

8 Universal shaper:-

In universal shaper, in addition to the two movements provided on the table of a standard shaper, the table can be swivelled about an axis parallel to the ram ways and an axis perpendicular to the ram ways.

As the work mounted on the table can be adjusted in different planes, the machine is most suitable for different types of work and is given the name "universal". A universal shaper is mostly used in tool room work (6)

(9) Push type shaper:-

this is the most general type of shaper used in common practice. Here, the metal is removed when the ram moves away from the column (ie) pushes the work.

that is, here forward stroke is a cutting stroke and the return stroke is an idle stroke.

(10) Draw cut shaper:-

(1) In this shaper, the metal is removed when the ram moves towards the column of the machine, (ie) draws the work towards the machine. Here, the forward stroke is an idle stroke and the return stroke is a cutting stroke

② Here, the tool is set in a reversed direction to that of a standard shaper. ⑦

③ Here, the ram is generally supported by an overhead arm which ensures rigidity and eliminates deflection of the tool.

④ In this shaper, the cutting pressure acts towards the column which relieves the cross-rail & other bearings from excessive loading and allows to take deep cuts. Vibration in these machines is practically eliminated.

Quick return mechanism:—

In shaper, the ram moves at a comparatively slower speed during the forward cutting stroke whereas during the return stroke, the ram moves at a faster speed to reduce the idle return time. This is called as "Quick return motion" and the mechanism used to obtain this is called as "Quick return mechanism". Generally there are 3 types of Quick return mechanisms used in a shaper. They are

a) Crank & slotted link mechanism

b) Whitworth quick return mechanism

c) Hydraulic mechanism.

Principle of Shaper:-

In Shaper, the metal is cut by the reci-procating movement of the cutting tool over the workpiece which is fixed on the table. The following feeds are available in a shaper.

① Cross feed:-

It denotes the movement of the table perpendicular to the ram movement per stroke of the cutting tool. It is obtained by moving the saddle over the guide ways of the cross-rail.

② Vertical feed:-

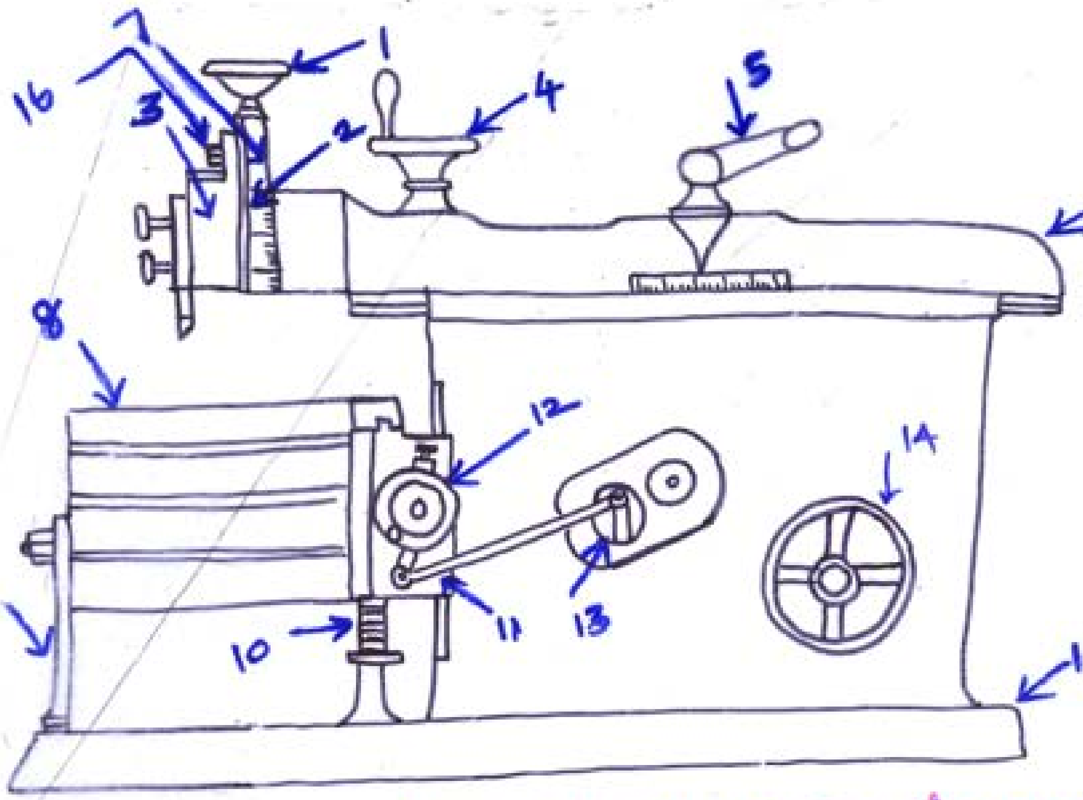
It denotes the movement of the table in a vertical direction per stroke of the cutting tool. It is obtained by moving the cross-rail over the vertical guide ways of the column by rotating the elevating screw (or) by moving the tool head vertically by rotating the down feed screw. the vertical feed of the shaper is used to produce vertical plane surfaces.

(3) Angular feed:-

It denotes the movement of the cutting tool in a vertically angular direction per stroke of the cutting tool. It is obtained by first tilting the tool head to the required angle & then feeding the cutting tool in a vertical direction by rotating the down feed screw. the angular feed is used to produce angular surfaces.

Note:-

the angular feed & the vertical feed of a shaper is given only after tilting the apron to some angle. this is done to give the clearance b/w the cutting tool & the work piece to avoid the rubbing of the cutting tool with the work piece during machining.



(1) Down feed handwheel (2) Swivel Base (3) clapper box
(4) Stroke position handwheel (5) Ram Locking handle (6) Ram
(7) Tool Head (8) Table (9) Table Support (10) Elevating Screw
(11) Cross-rail (12) Pawl & Ratchet mechanism (13) Feed disc
(14) Driving pulley (15) Base (16) Apron clamping bolt

Fig 3.1 - Construction of a Shaper

Construction of a shaper:-

A shaper consists of the following principal parts.

(1) Base (2) column (3) Cross-rail. (4) Saddle
(5) Table (6) Ram (7) Tool Head

① Base:-

It is necessary bed (or) support required for all machine tools. It may be rigidly bolted to the floor (or) on the bench according to the size of the machine. It is so designed that it can take up the entire load of the machine and the forces set up by the cutting tool over the work. It is made of Cast Iron to resist vibration and take up high compressive load.

② Column:-

It is a box like casting mounted upon the base. It encloses the ram driving mechanism. Two accurately machined guideways are provided on the top of the column on which the ram reciprocates. It has two vertically machined guideways on its front face, which serves as the guide ways for the cross-rail for its vertical movement. The other side of the column contains levers, handles etc., for operating the machine.

③ Cross-rail:-

It is mounted on the vertical guideways of the column. It has a accurately machined

Guide ways on its top, which is perpendicular to the ram axis. The table may be given a vertical feed by rotating an elevating screw. A Horizontal cross feed screw which is fitted within the cross-rail and parallel to the top guideways of the cross-rail actuates the table to move in a cross-wise direction.

(4) Saddle:-

(12)

The Saddle is mounted on the cross-rail which holds the table firmly on its top. Cross wise movement of the saddle by rotating the cross-feed screw by hand (or) power causes the table to move sideways.

(5) Table:-

The table which is bolted to the Saddle receives cross wise and Vertical movements from the saddle & cross-rail. the table has T-slots both on the top and sides for clamping the work. In a universal shaper, the table can be tilted about an axis parallel to the ram ways and an axis perpendicular to the ram ways.

⑥ Ram:-

The ram is a reciprocating member of a shaper. This is semi-cylindrical in form. It slides on the accurately machined dove-tail guideways on the top of the column and is connected to the reciprocating mechanism contained within the column. It houses a screwed shaft for altering the position of the ram with respect to the work and it holds the tool head at its front end.

⑦ Tool head:- [Refer figure 3.2 in P.no:15]

① The tool head of a shaper holds the tool rigidly, provides the vertical and angular feed movement of the tool and allows the tool to have an automatic relief during its return stroke.

② The vertical slide of the tool head has a swivel base which is held on a circular seat on the ram. The swivel base is graduated in degrees, so that the vertical slide may be set perpendicular to the work surface (or) at any desired angle. By rotating the down feed screw handle, the vertical slide carrying the tool executes down feed (or) angular feed movement while machining vertical (or) angular surface. the amount of feed (or) depth of cut may be measured by a micrometer dial on the top of the down feed screw.

14

3 The Apron consisting of clapper box, clapper block & the tool post is clamped upon the vertical slide by a screw. By releasing the apron clamping screw, the apron may be swivelled either towards right (or) left with respect to the vertical slide. This arrangement is necessary to provide relief to the tool while making vertical (or) Angular cuts.

4 The two vertical walls on the apron called clapper box houses the clapper block which is connected to it by means of a hinge pin. The tool post is mounted on the clapper block. on the forward cutting stroke, the clapper block fits securely to the clapper box to

make a rigid tool support. On the return stroke, a slight frictional drag of the tool lifts the clapper block out of the clapper box a sufficient amount preventing the tool cutting edge from dragging & consequent wear. The work surface is also prevented from any damage due to dragging.

(15)

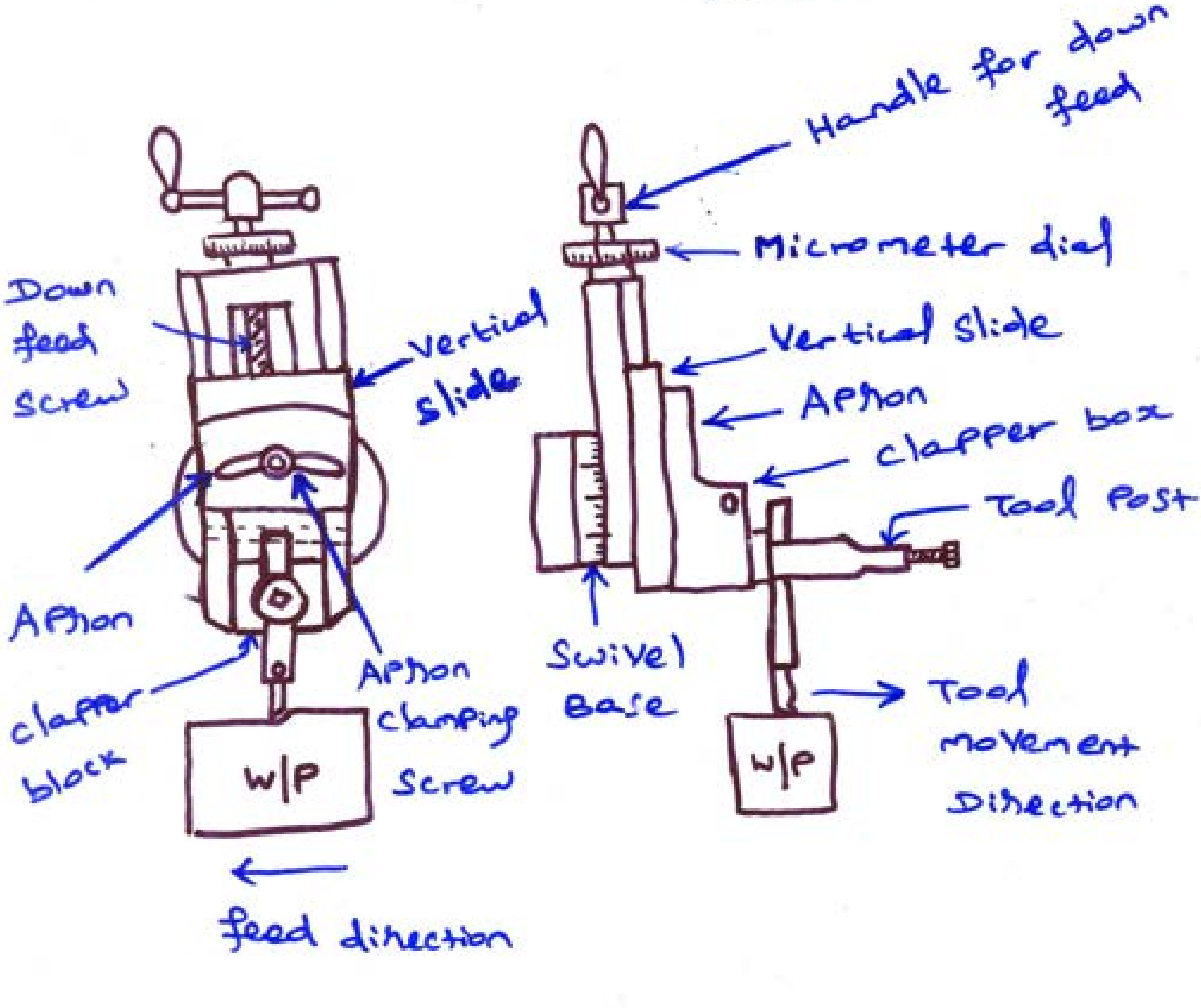


Fig 3.2 – Tool Head of a Shaper

SHAPER MECHANISMS:-

1. In a shaper, rotary movement of the drive is converted into the reciprocating motion of the ram by the mechanism contained within the column of the machine.

2. In a standard shaper, the metal is removed during the forward cutting stroke and the return stroke is an idle stroke as no metal is removed during the return stroke.

3. To reduce the total machining time, it is necessary to reduce the time taken by the return stroke.

4. Thus the shaper mechanism is so designed

that it can allow the ram holding the tool to move at a comparatively slower speed during the forward cutting stroke whereas during the return stroke, it can allow the ram to move at a faster rate to reduce the idle return time. This mechanism is known as "quick return mechanism".

(5) There are 3 types of quick return mechanism used in the shaper.

(1) Crank & slotted link mechanism
(2) whitworth quick return mechanism
(3) Hydraulic drive mechanism

Crank & slotted link mechanism

The motion (or) power is transmitted to the bull gear through a pinion which receives its motion from an individual motor through a stepped cone pulley & belt drive (or) a set of gear train. Speed of the bull gear can be changed by shifting the belt to various pulley in case of belt drive (or) by various gears in case of gear drive.

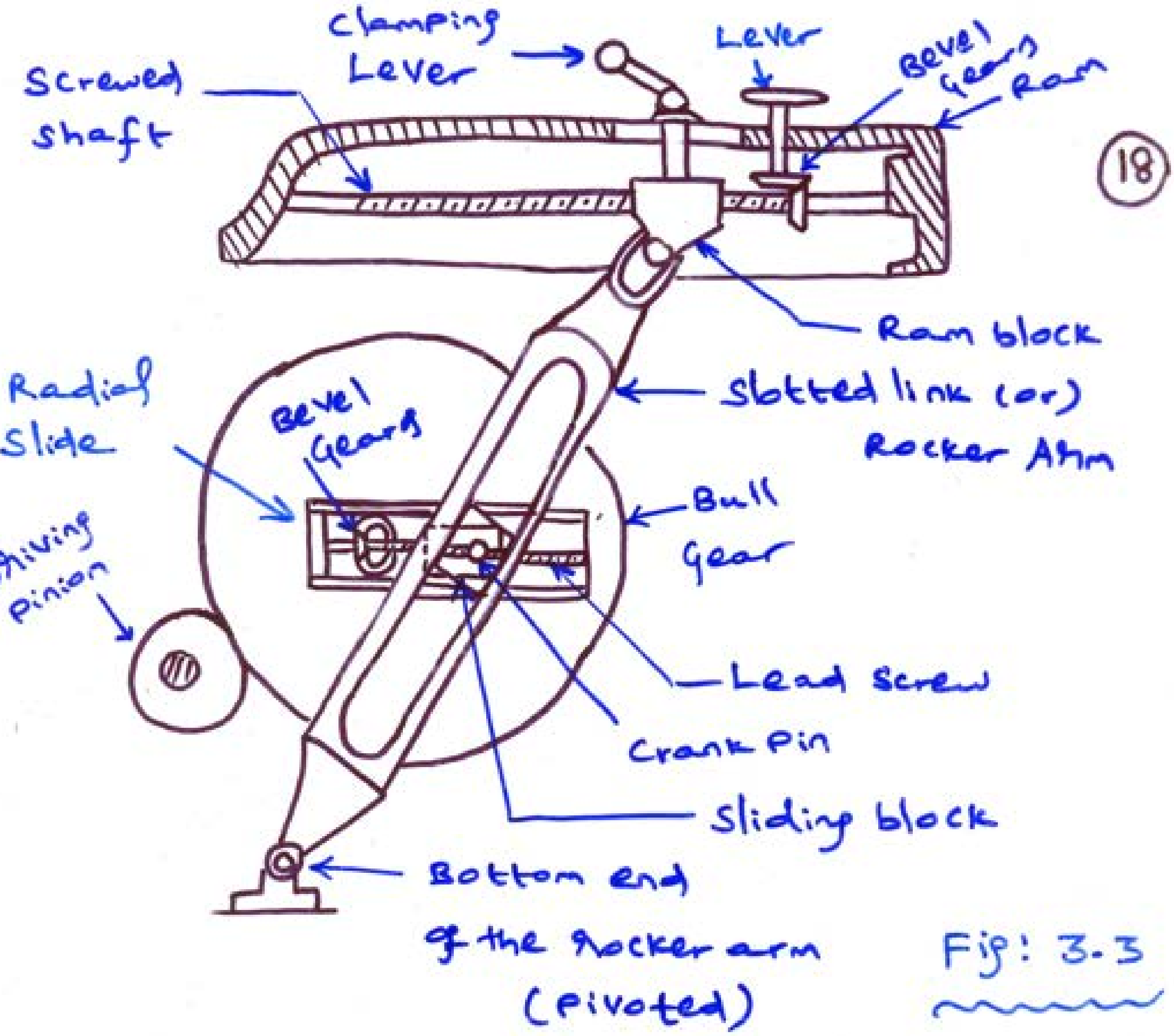


Fig: 3.3

Bolted to the center of the bull gear is a radial slide which carries a sliding block into which the crank pin is fitted. Rotation of the bull gear will cause the crank pin to rotate at a uniform speed. Another sliding block is also there into the slot provided in the slotted link (or) rocker arm. It is fitted to the sliding block in the radial slide by a crank pin. The rocker arm is pivoted at its bottom end and is attached to the frame of the column. The upper end of the rocker arm is forked & connected to the ram block by a pin. As the bull gear rotates, the crank pin will rotate on a crank pin circle. As the sliding block is connected with a crank pin, it also rotates along crank pin circle, which will give the rocker arm a rocking movement. As the rocker arm is connected to the ram by ram block, the ram gets a reciprocating motion from the rocking movement of the rocker arm. Thus the rotary motion of the bull gear is converted to the reciprocating movement of the Ram.

To get quick return motion from crank & slotted link mechanism :- [Refer 3.4 on page no: 21]

From figure 3.3.3, it can be understood that when the link is in the position PM, the ram will be at the extreme backward position of its stroke & when it is at PN, the extreme forward position of the ram will have been reached.

PM & PN are tangent to the crank pin circle at the extreme position of the forward & Return stroke. The forward cutting stroke takes place when the crank pin rotates through the angle C_1KC_2 and the return stroke takesplace when the crank rotates through the angle C_2LC_1. It is evident that the angle C_1KC_2 made by forward (or) cutting stroke is greater than the angle C_2LC_1 made by the return stroke. As the angular velocity of the crank pin is constant, the return stroke is completed within a shorter time.

$$\boxed{\frac{\text{Cutting time}}{\text{Return time}} = \frac{\text{Angle of } C_1KC_2}{\text{Angle of } C_2LC_1}}$$

Cutting time to return time ratio usually varies between 2:1 and 3:2. From the figure 3.3.2, it can be understood that, the cutting speed is minimum at the extreme position of the ram & is maximum when the rocker arm is vertical. The only disadvantage of this method, cutting speed and return speed is not constant throughout the stroke.

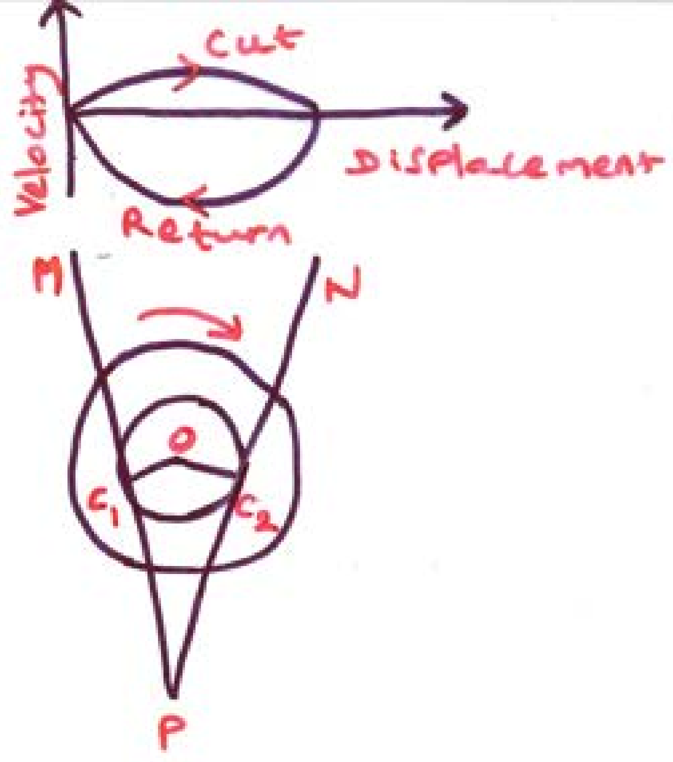


Fig: 3.4 - Quick return motion from Crank & slotted link mechanism

Adjusting the length of the stroke:-

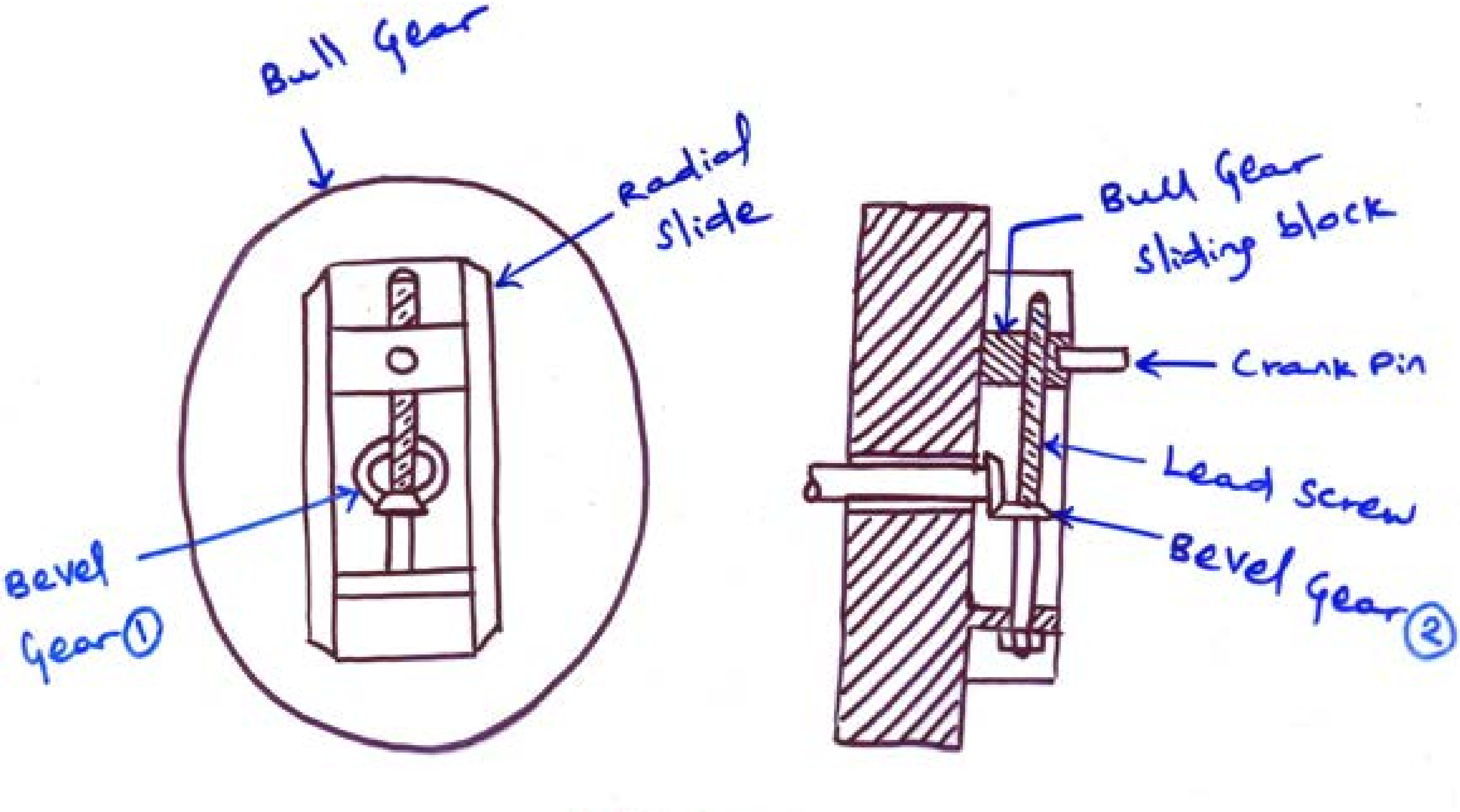


Fig: 3.5

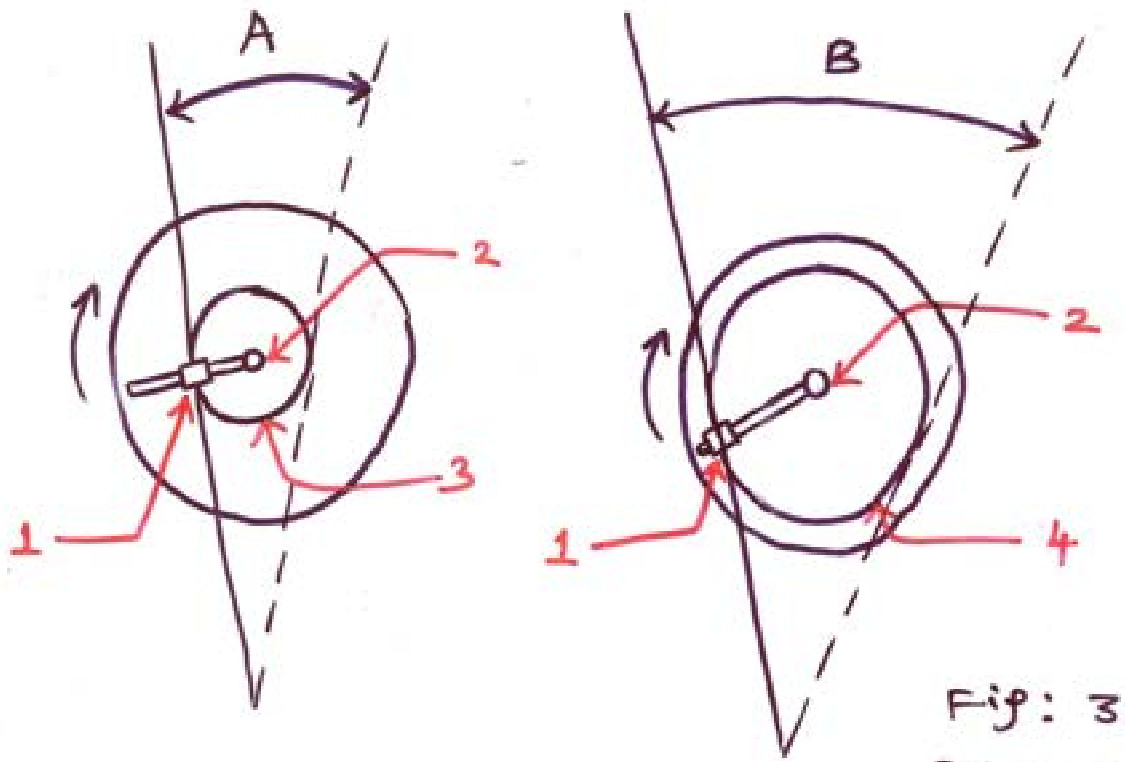


Fig: 3.6

① Position of the sliding block ② Center of bull gear

③ Shorter Crank pin circle ④ larger Crank pin circle

A → Shorter Stroke length

B → Larger stroke length

The stroke length of the ram can be adjusted by moving the bull gear sliding block towards (or) away from the center of the bull gear. The bevel gear ① is placed at the center of the bull gear. It can be rotated by a handle. when the bevel gear ① is rotated, the bevel gear ② will rotate causing the lead screw to rotate. Thus the bull gear sliding block which engages with the lead screw can be moved towards (or) away from the center of the bull gear based on the direction of rotation of the lead screw. The closer the crank pin is brought to the center of the bull gear, the smaller will be the crank radious and hence the smaller will be the stroke length. The farther the crank pin is brought away from the center of bull gear, the bigger will be the crank radious & hence the larger will be the stroke length. Maximum stroke length is obtained when the crank pin is shifted towards the farthest end of the slide

Adjusting the position of the stroke:— [See Fig: 3.3]

Inside the ram of a shaper, there are screwed shaft, Ram block, Bevel gears. The position of the ram with respect to the work piece can be adjusted by rotating the screwed shaft of the ram. For rotating the screwed shaft of the ram, first the clamping lever for the ram block is loosened & then the lever for the bevel gear is rotated. Now, the screwed shaft starts rotating.

As the ram block which acts as a nut is a stationary one, the ram of the shaper will move forward (or) backward with respect to the ram block based on the direction of rotation of the screwed shaft. Thus the position of the ram with respect to the workpiece is adjusted.

Whitworth Quick return mechanism:-

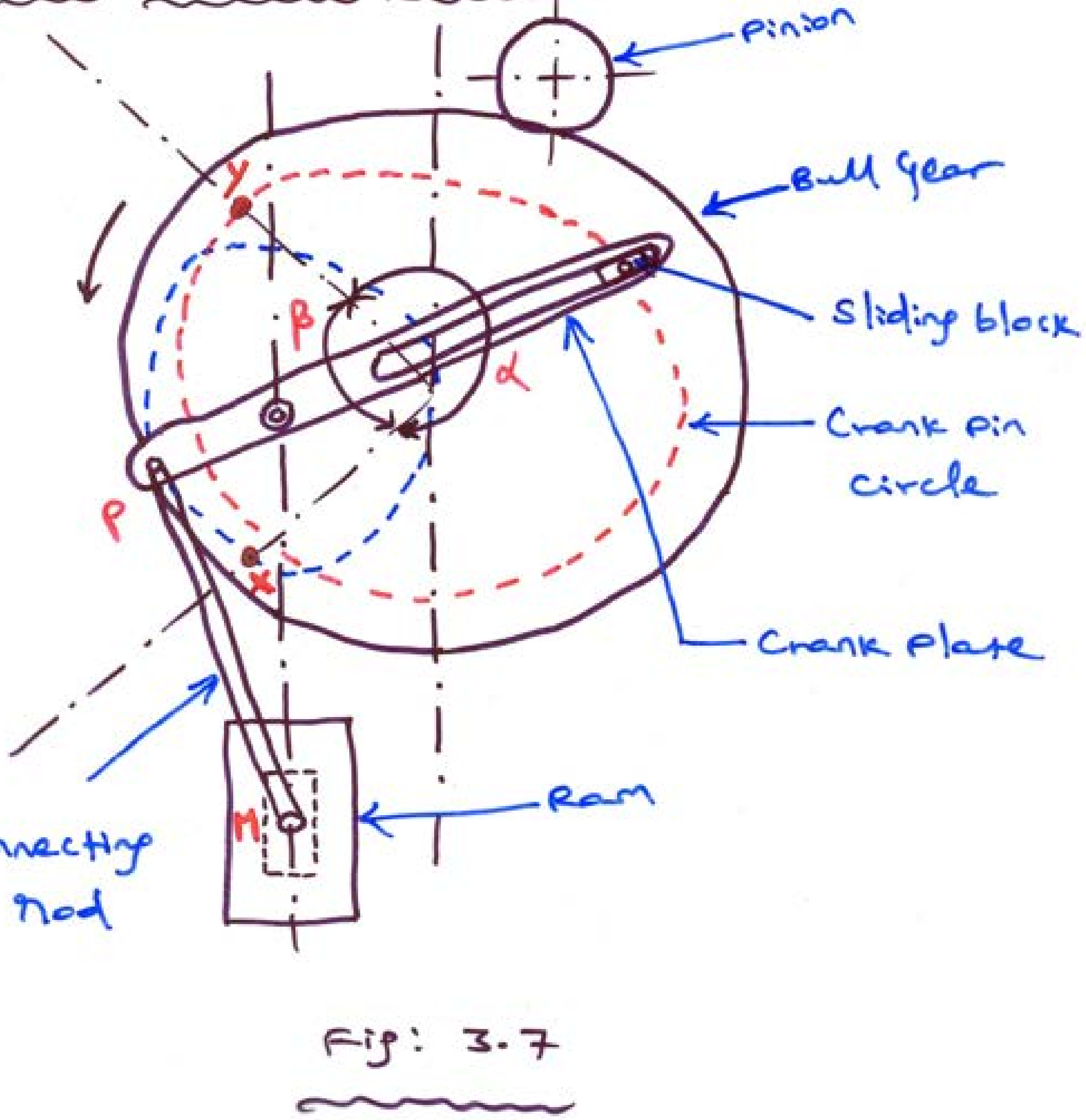


Fig: 3.7

Figure 3.7 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 sliding block which is fitted on the bull gear by a crank pin. the slotted crank plate is eccentrically pivoted on the bull gear at point "S". the sliding block in the bull gear slides into the slot of the crank plate. A connecting rod connects the crank plate by a pin at P on one end and the ram at the other end M. when the pinion rotates, the bull gear is also rotated along with the sliding block which slides into the slot of the crank plate. It makes the ram to move up & down [Reciprocating motion] by the connecting rod.

position of the crank pin 'A' :—

① when the crank pin A is at x, the ram will be in forward stroke. At that time, the bull gear rotates in the Anti clock wise direction at an angle "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 of β is less than α. So, the time taken for the return stroke is reduced. (2b)

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

Reason for making the stroke length greater the work piece length in a horizondal shaper:— 2 Mark

(1) To allow sufficient time for giving cross-feed.

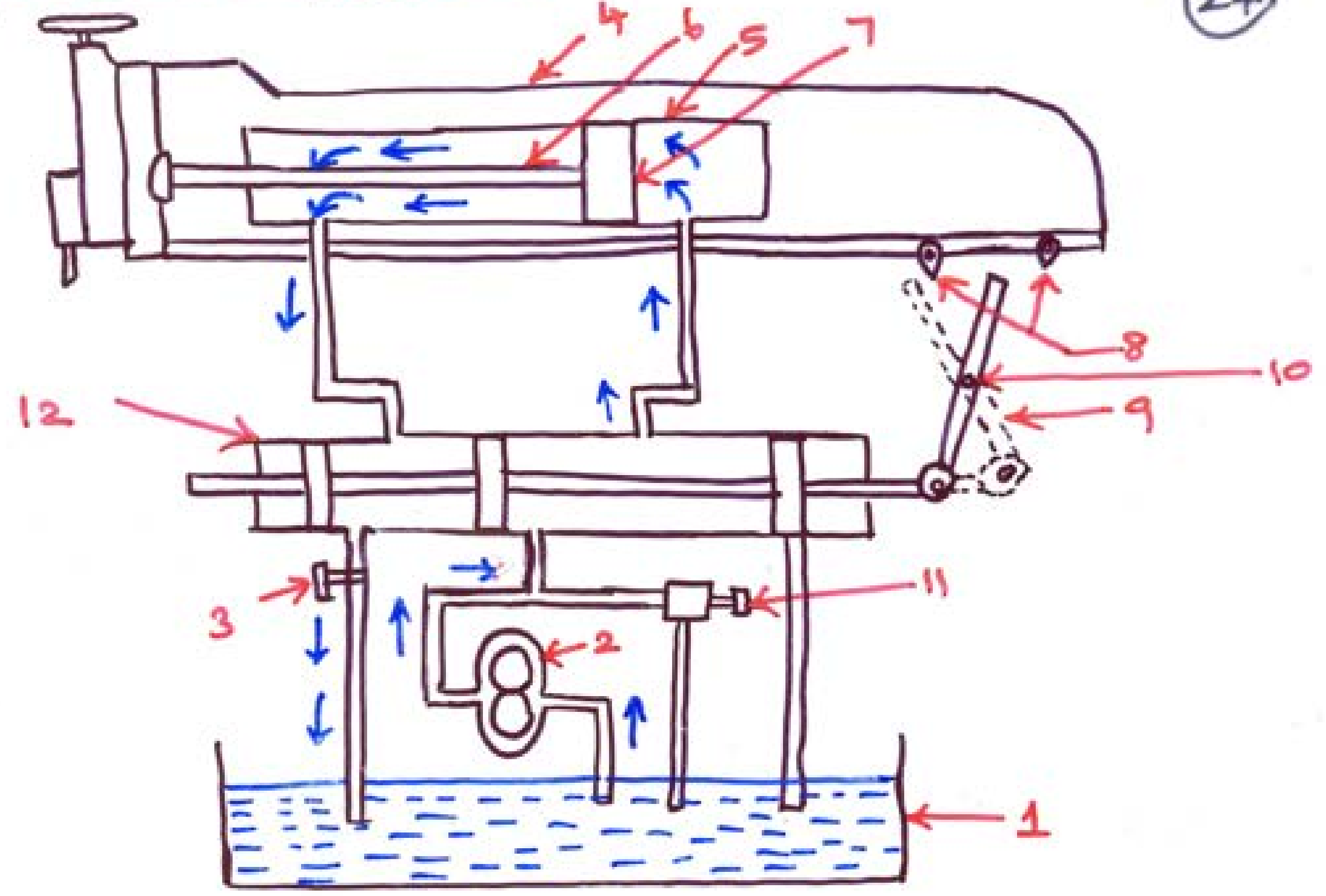
(2) To allow sufficient time for the clapper box to attain its proper seat for cutting.

(3) To allow the ram for obtaining the proper cutting speed.

(4) To avoid the rubbing of the tool on the work surface during the return stroke.

(5) To obtain the good surface finish.

Hydraulic drive :-



① oil reservior ② Oil pump ③ throttle valve
④ Ram ⑤ Cylinder ⑥ Piston rod ⑦ Piston
⑧ Reversing dog ⑨ Reversing lever ⑩ Reversing lever pivot ⑪ Relief valve ⑫ Direction Control valve.

① In a hydraulic shaper, the ram is moved forward & backward by a piston rod attached with a piston which is moving inside a hydraulic cylinder placed inside the ram.

② This mechanism consists of

ⓐ constant oil discharge pump.

(b) Relief Valve (c) Direction control valve [DCV]

(d) Hydraulic cylinder (28)

(3) The Position of the DCV is changed by a reversing dog at the end of each stroke. At one position of DCV, the oil flows through the piston side of the cylinder making forward stroke of the ram taking place. At another position of DCV, the oil flows through the piston rod of the cylinder making the return stroke of the ram taking place

(4) The purpose of oil pump is to suck the oil from the reservior & discharge it to the cylinder through DCV at high pressure

(5) The volume of the cylinder in the piston rod side is less than the volume of the cylinder in the piston side. So, when the same amount of oil flows through the piston rod side, the pressure developed will be comparatively high, which in turn increases the speed of the ram during the return stroke. Thus, the quick

return motion of the Ram is achieved.

6. The speed of the ram can be increased (or) decreased by controlling the amount of oil discharged to the tank. This is done by adjusting the throttle valve. When the oil flows through one side of the cylinder, the oil in another side of the cylinder will be discharged to the tank through the throttle valve. For example, when the throttle valve is partially closed, the speed of the cylinder will be reduced. At the same time, the outlet pressure will be increased. In this time, the relief valve will open and discharge excess oil to the tank, thus maintaining the constant pressure in the system.

Advantages of the Hydraulic drive:-

1. The cutting & return stroke speeds are practically constant throughout the stroke. This permits the cutting tool to work uniformly during the cutting stroke

(2) the reversal of the ram is obtained quickly without any shock as the oil on the other end of the cylinder provides cushioning effect. (30)

(3) Flexibility in obtaining various cutting speeds.

(4) the relief valve ensures safety to the tool & the machine when the machine is overloaded.

(5) The opreation is smooth & silent.

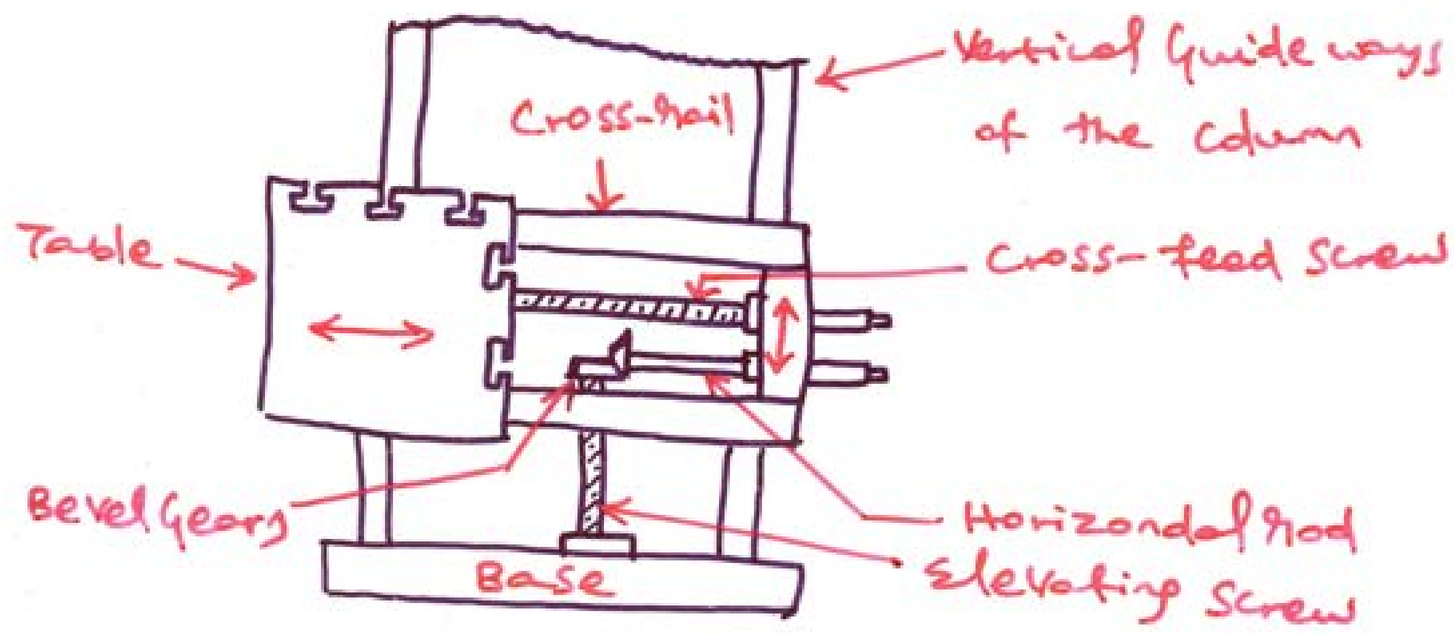
Feed Mechanism :-

(1) Hand feed mechanism
- → Vertical feed by the tool head
- → Angular feed by the tool head
- → Vertical feed for the table
- → Horizondal (or) cross-feed for the table

(2) Auto-feed mechanism → Horizondal (or) Cross-feed for the table.

Unlike lathe, the feed movements in a shaper are provided intermittently & is given at the end of return stroke only.

Hand feed mechanism for the table of the shaper:—



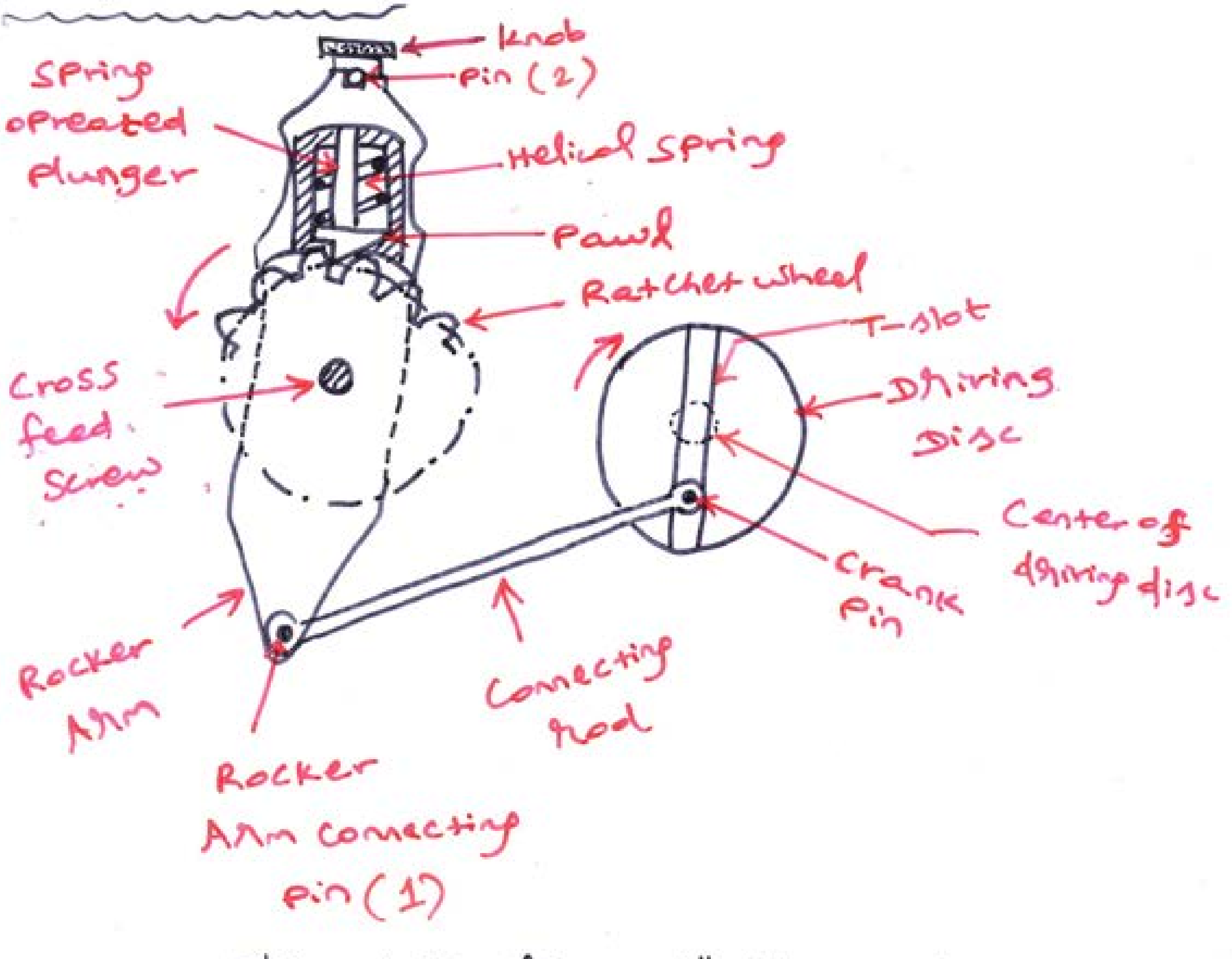
① Vertical Hand feed:—

When the horizonal rod is rotated by a key, the power will be transmitted to the elevating screw through a set of bevel gears. The rotation of the elevating screw make the cross-rail of the shaper move along the vertical guide ways of the column, thus giving the vertical feed to the table.

② Horizondal (or) cross feed by hand:—

When the cross-feed screw is rotated by a keys the cross-feed screw will rotate. The rotation of the cross-feed screw make the saddle move along the horizondal flat guideways of the cross-rail, thus giving the horizondal (or) cross feed to the table of the shaper.

Auto Cross-feed Mechanism for a Shaper:- (Pawl & Ratchet)



The above figure illustrates the automatic cross-feed mechanism of a shaper. The driving disc is connected to the bull gear. So, when the bull gear rotates, the Driving disc will also rotate. The driving disc has a T-slot along its diameter. The one end of the connecting rod is connected to the driving disc eccentrically by a crank pin. The other end of the connecting rod is connected to the bottom end of the rocker arm by a pin(1). The rocker arm is fulcrumed at the center of the ratchet wheel which is keyed to the cross-feed screw. The top portion of the rocker arm has a spring opreated pawl which is straight on one side and slant (or) Bevelled on the other side. The pawl is into the space between the teeth of the ratchet wheel. When the driving disc rotates through half of the revolution in a clockwise direction, the rocker arm top portion starts rocking in a clockwise direction as it is connected with the driving disc by connecting rod. During the clockwise direction of the rocking arm, the pawl in the rocker arm just slips over the the teeth of the ratchet wheel imparting no movement to the table of a shaper. When the driving disc rotates through another half in a clockwise direction, the rocker arm top portion will start

33

rocking in the Anti clockwise direction. In this condition, the straight side of the pawl engages with the teeth of the ratchet wheel & rotate it. As the ratchet wheel is keyed to the cross feed screw, cross-feed screw will also rotate when the ratchet wheel rotates. Thus, the cross-feed is given to the table automatically. This cross-feed is given during the return stroke only.

34

The following points is worth noted.

1. To reverse the direction of the cross-feed of the table, the pin (2) of the rocker arm is lifted from the slot & rotated through 180° so that the position of the pawl is reversed & the pin is again put into the slot. Now, the direction of Auto cross feed is reversed.

2. The amount of cross feed may be altered by shifting the position of the crank pin with respect to the center of the driving disc. Greater the throw of Eccentricity, more will be the rocking

movement for the rocker arm and the Pawl will rotate three (or) four teeth on the ratchet wheel at a time imparting greater feed movement. 35

Work holding devices of a shaper:—

The top & sides of the table of a shaper have T-slots for clamping the work. The work may be supported on the table by the following methods depending on the nature of the w/p.

1. clamped in a vise
2. clamped on the table
3. clamped to the angle plate
4. clamped on a V-block
5. Held between the centers.

1. Shaper vises:—

* A vice is a quick method of holding and locating relatively small & regular shaped workpieces.

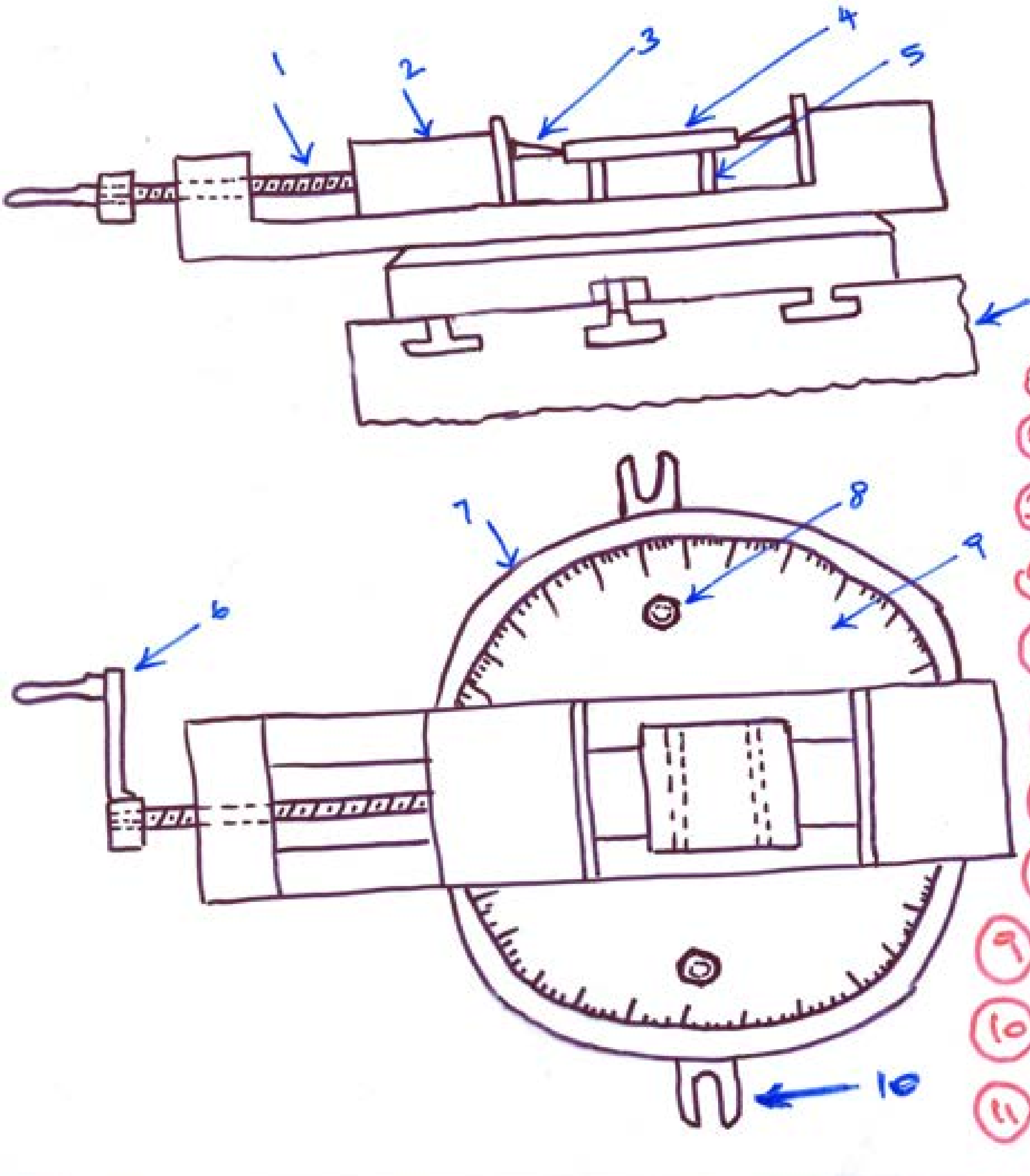
* It consists of base, screw, fixed

and movable jaws.

* The base has a projection (or) tongue which fits into the slot of machine table.

* For properly securing it to the table lugs are provided for clamping the vice by T-bolts.

* The work is clamped b/w the fixed & movable jaws by a screw.



① Screw
② Movable jaw
③ Hold down
④ Work
⑤ Parallels
⑥ Handle
⑦ Base
⑧ Clamping bolt
⑨ Swivel base
⑩ Lugs
⑪ Table of Shaper

A machine vice is classified into 3 types

1. plain vice → Single Screw, Double Screw
2. Swivel vice
3. Universal vice

Plain vice:-

It is the most Simple of all the types. The vice may have a single screw (or) double Screws for actuating the movable jaw. The double Screws add gripping strength while taking deeper cuts (or) handling heavier jobs

Swivel vice:-

In this vice, the base is graduated in degrees and the body of the vice may be Swivelled at any desired angle on a horizontal plane. The Swivelling arrangement is useful for machining horizontal bevelled Surface of the workpiece.

Universal vice:-

In this vice, the body of the vice can be Swivelled on a horizontal plane like a Swivel vice

In addition to that, the body can be tilted in a vertical plane upto 90° from the horizontal plane. It is used for vertical angular machining. 38

Parallels:-

* Parallels are square (or) rectangular bars of steel (or) cast iron which is hardened and ground with opposite sides parallel.

* When the height of the job is less than the height of the jaws of the vice, parallels are used to raise and seat the work piece above the jaws.

* Parallels are available in various sizes for seating the workpiece of different heights and are always used in pairs.

Hold-downs:-

* Hold-down is a hardened wedge shaped piece with the taper angle of 5°.

* Hold-downs are placed b/w the two jaws of vice & the workpiece.

* It is used for holding thin pieces of work in a shaper vise. It is also used for holding the work of smaller height than the vice jaws where

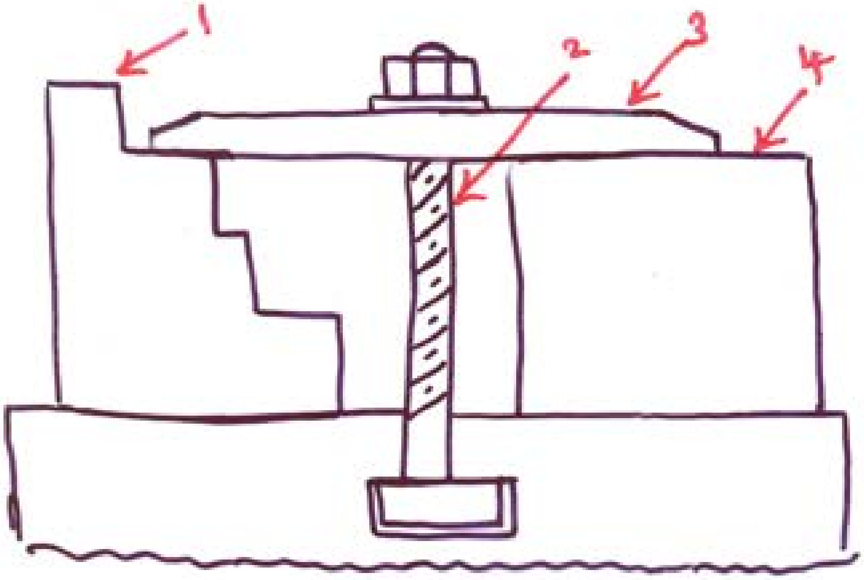
Suitable parallels are not available.

2 clamping work on the table:-

when the workpiece is too large to be held in a vise, it must be fastened directly on the shaper table. The different methods employed to clamp different types of work on a shaper table are

1. T-bolts & clamps
2. Stop pins
3. Stop pins & toe dogs
4. Strip & stop pins.

1 T-bolts & clamps:-



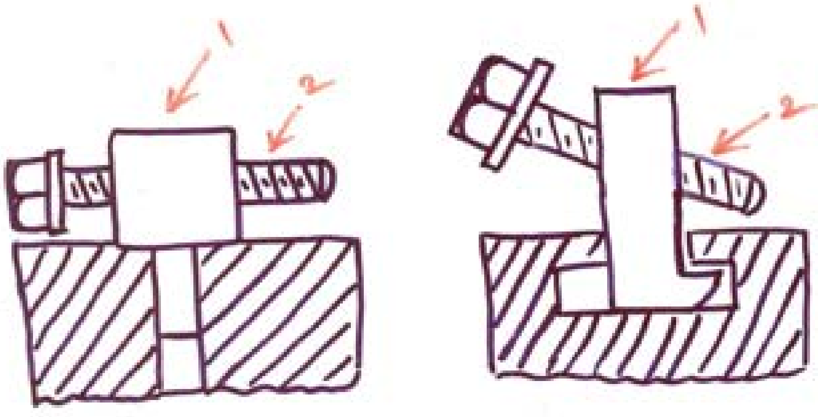
1 step-block 2 T-bolt
3 clamp 4 work

* Figure illustrates the use of T-bolts and clamps for holding the work.
* T-bolts having are fitted on the T-slots of the table. the length of the threaded portion is sufficiently long in order to accomodate different heights of work.

* The clamps are made of steel having slots at the center for fitting the bolt. One end of the clamp rests on the side of the work while the other end rests on a block.

* To hold a large work on the table, a series of clamps and T-bolts are used all round the work.

② Stop pins:—



① Body ② Screw

* A stop pin is a one leg screw clamp
* It is used to prevent the work to get pushed out of its position during forward cutting stroke of the tool.

* The body of the stop pin is fitted in the hole (or) slot on the table & the screw is tightened till it forces against the work.

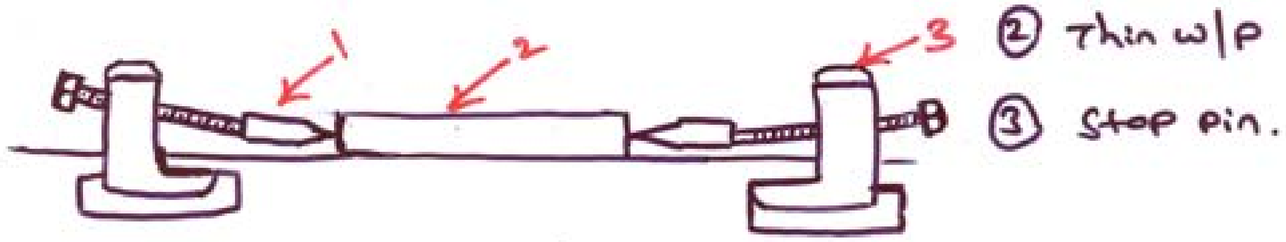
③ Stop pins & toe dogs:—

* while holding the thin work pieces on the table, stop pins in conjuction with toe dogs are

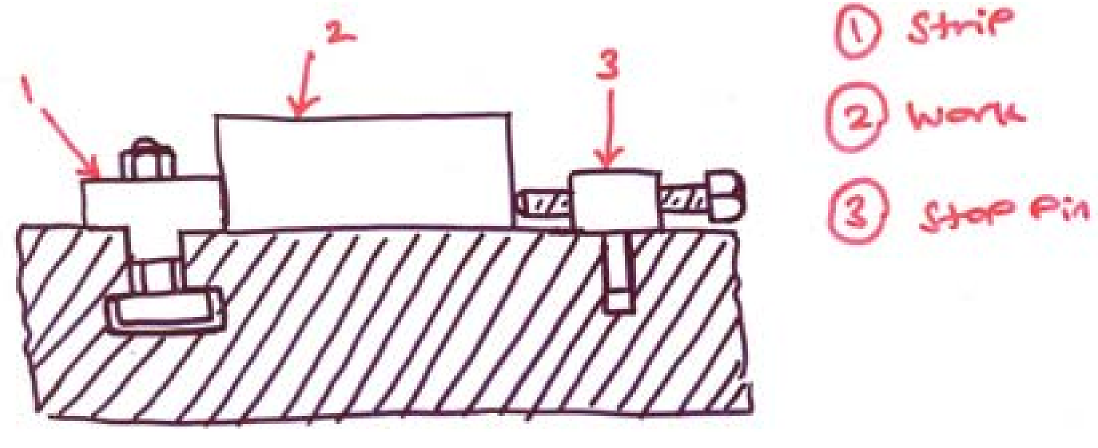
used

* A toe dog is similar in shape to that of a center punch (or) cold chisel. the head end of the toe dog is drilled slightly so that the end of the stop pin screw may fit into it.

(4')



④ Strip & stop pins:—



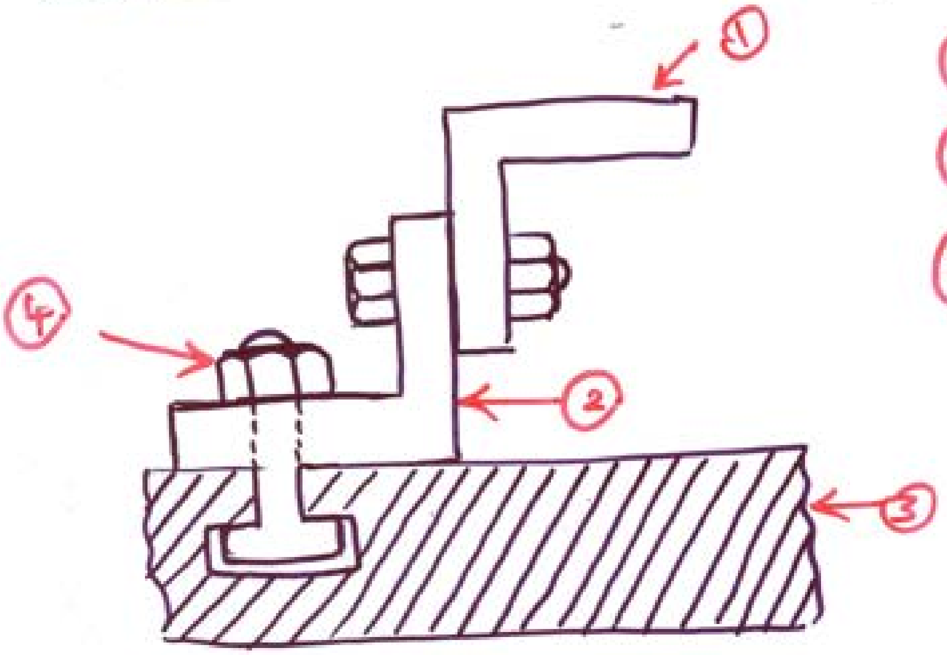
* the above figure illustrates the working of strip & stop pin for holding the work.

* work having sufficient thickness is held on the table by strip & stop pins.

* A strip is a long bar having a tongue with holes for fitting the T-bolts.

③ clamping on an Angle plate:-

① L-Shaped w/p
② Angle plate
③ Table of a Shaper
④ T-bolt & Nut



* For holding 'L' Shaped work piece, angle plates are used.

* Angle plates are made of cast Iron & has two plane surface bent at angle of 90°. One of the sides of the Angle plate is fitted on the table of the Shaper. Another Side of the angle plate hold the 'L' Shaped work piece by bolt & nut.

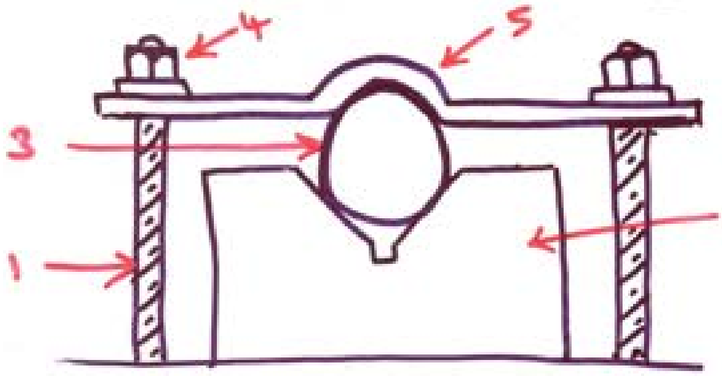
④ V-Block:-

* For holding round rods, V-blocks are used.

* Work may be Supported on two V-block at two ends & is clamped to the table by T-bolts & clamps.

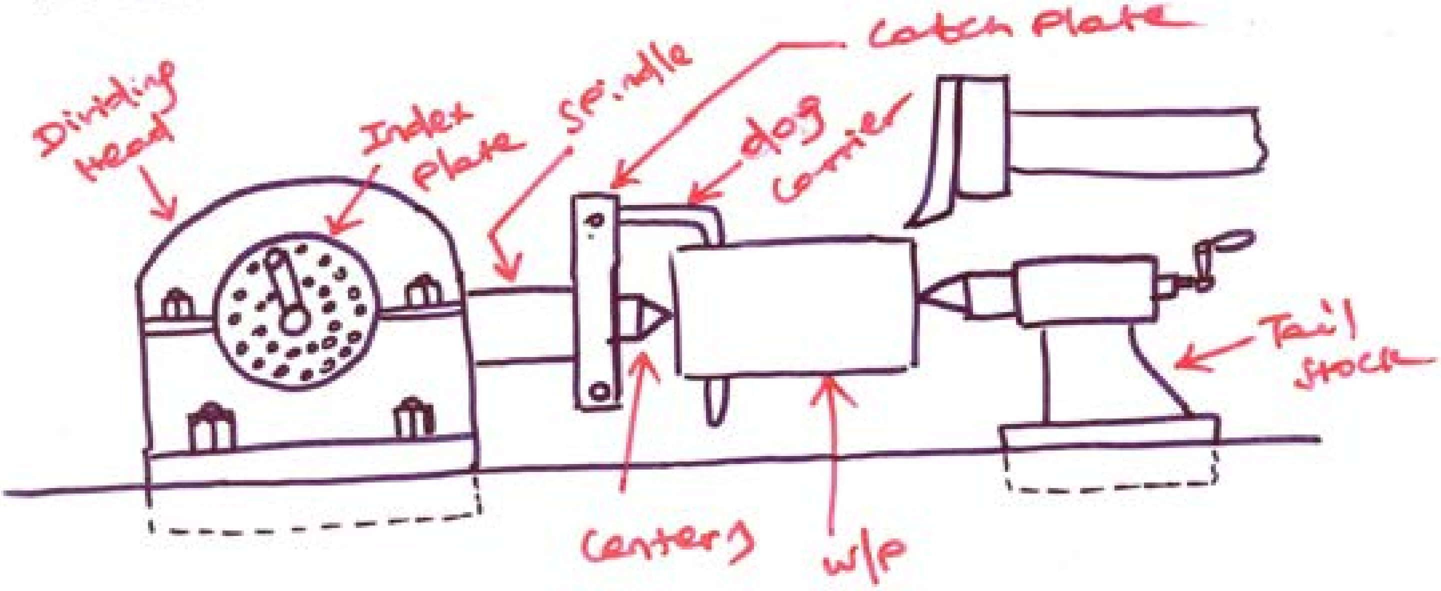
* The tool may be made to reciprocate between the two clamps for cutting grooves (or) keyways.

43



① T-bolt ② V-block
③ Work ④ Nut
⑤ Clamp

⑤ Clamping the workpiece b/w the centers:-



* This is a special attachment for cutting equally spaced grooves (or) splines on the periphery of the round work. It may be also used for cutting gears.

* After cutting each groove on the cylindrical workpiece, the crank of the index plate will be

rotated along a selected hole circle to a predetermined amount so that the periphery of the w/p is equally divided.

44

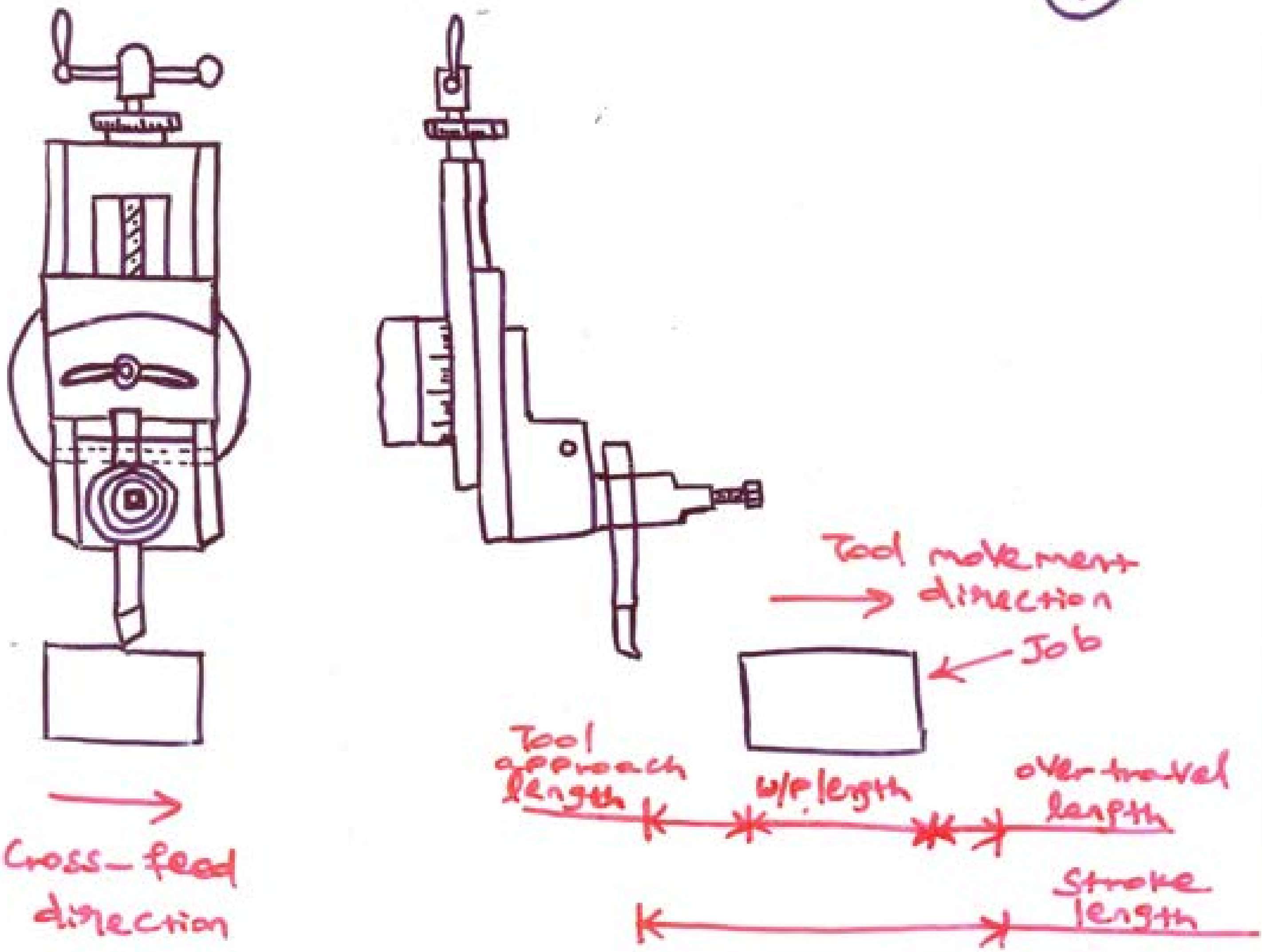
* Here, the work is held b/w the centers by means of catch plate & Dog carrier.

Shaper opreations:-

A Shaper is a Versatile machine tool primarily designed to generate a flat Surface by a Single point cutting tool. But it may also be used to perform many other operations. The different opreations which a shaper can perform are as follows

1. Machining horizontal Surface
2. Machining vertical Surface
3. Machining angular Surface
4. Cutting slots, Grooves & keyways.
5. Machining irregular Surface
6. Machining Splines (or) Cutting gears.

① Machining a horizontal Surface:-



* The above figure illustrates machining horizontal flat surface on the w/p.

* Procedure for making horizontal flat Surface on the w/p

① After the work is properly held on the table, a planning tool is set in the tool post with minimum overhung.

② The table is raised till there is a Clearance of 25mm to 30mm between the tool & the workpiece.

③ the length & the position of the stroke are then adjusted.

a) the length of stroke should be nearly 20mm longer than the work.

b) The position of the stroke is so adjusted that the tool begins to move from a distance of 12 to 15mm before the beginning of the cut and continues to move 5 to 8mm after the end of cut.

④ Proper cutting speed & feed is then adjusted.

⑤ then the depth of cut is given by rotating the down feed screw.

⑥ Then the shaper is switched on. The cross feed is given at the end of return stroke.

⑦ Both roughing & finishing cuts are performed to complete the job.

Note:-

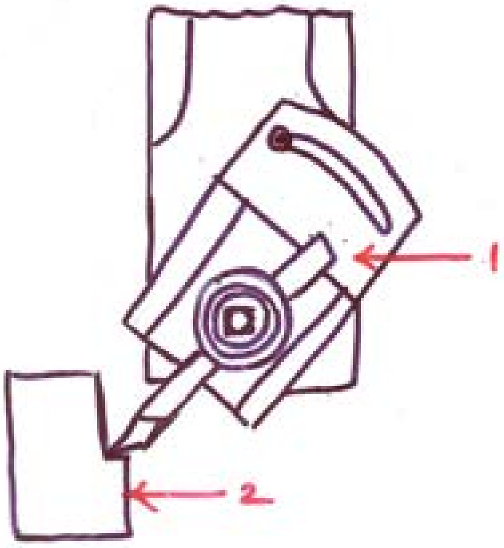
① For rough cuts, cutting speed is decreased while feed & depth of cut is increased.

$\left.\begin{array}{c}\text{depth of cut}\\ \text{for rough work}\end{array}\right\} \Rightarrow$ 1.5 to 3mm

② For finishing cut, cutting speed is increased while the feed & depth of cut is decreased.

$$\left.\begin{array}{l}\text{Depth of cut}\\ \text{for finishing cut}\end{array}\right\} = 0.075\text{mm to } 0.2\text{ mm.}$$

② Machining a Vertical flat Surface & Vertical angular Surface:-



① Apron ② work

Fig: 1

① Apron ② work

A → Swivelling Angle of Vertical Slide

Fig: 2

Steps in the machining of Vertical Surface:-

① Fig:① illustrates the machining of Vertical flat Surface.

② The work is mounted on the vice (or) directly on the table.

(3) The Surface to be machined is carefully aligned with the axis of the ram.

(4) A side cutting tool is set on the tool post and then position, length of stroke is adjusted.

(5) the vertical slide is set exactly on zero angle position.

(6) The apron is swivelled in a direction away from the surface being cut. This is necessary to enable to move upward and away from the work during the return stroke. This also prevents the side of the tool from dragging on the planed vertical surface during the return stroke.

(7) then the machine is switched on. At the end of return stroke, down feed of about 0.25mm is given by rotating the down feed screw.

(8) Both roughing & finishing cuts are performed to complete the job.

Steps involved in machining of Angular Vertical Surface:-

① Fig:2 illustrates the machining of Angular vertical surface.

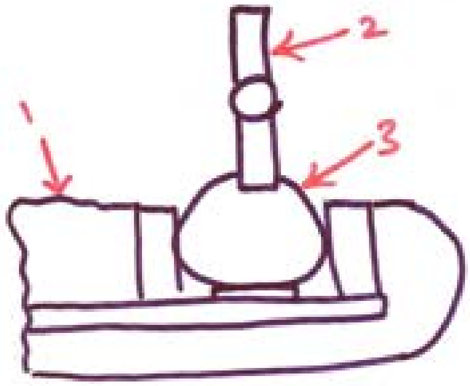
② The work is set on the table and the vertical slide of the tool head is swivelled to the required angle either towards right (or) left from the vertical position.

③ Then the apron is swivelled away from the work.

④ Then the machine is switched on. The Angular down feed is given at the end of the return stroke by rotating the down feed screw.

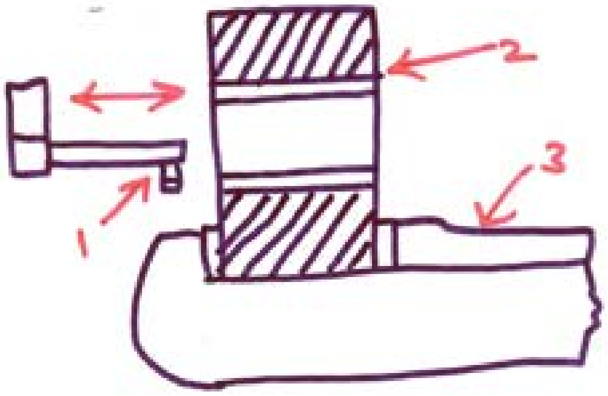
⑤ Angular machining can also be machined in a universal shaper (or) by using universal vice without swivelling the tool head.

3 cutting slots & keyways:-



1 vise 2 Tool 3 work

Cutting External keyway



1 Tool bit 2 work 3 vise.

Cutting Internal keyway

Cutting External keyway

1. For cutting slots (or) keyways, a square nose tool similar to the parting tool is selected.

2. External keyway are cut on a shaft by first drilling a hole at the blind end of the keyway. The diameter of the holes should be 0.5 to 0.8 mm oversize than the width of the keyway and the depth should be about 1.5mm larger than the depth of the keyway. This is necessary for the clearance of the tool at the end of the return stroke

③ The length & position of the Stroke is carefully adjusted so that the stroke will terminate exactly at the clearance hole. The speed is reduced while cutting a keyway. (51)

Cutting a Internal keyway:—

① Internal keyways are cut by holding the tool on a special tool holder so that the tool post will not hit against the work at the end of the stroke.

② Clapper block is locked in the clapper box to prevent the tool from lifting during the return stroke.

③ Lubrication is necessary on the work to prevent the cutting edge of the tool from wear due to dragging.

Machining Irregular Surface:—

① A Shaper can also produce a contoured surface (i.e) a convex (or) concave (or) a combination of any of the above surfaces.

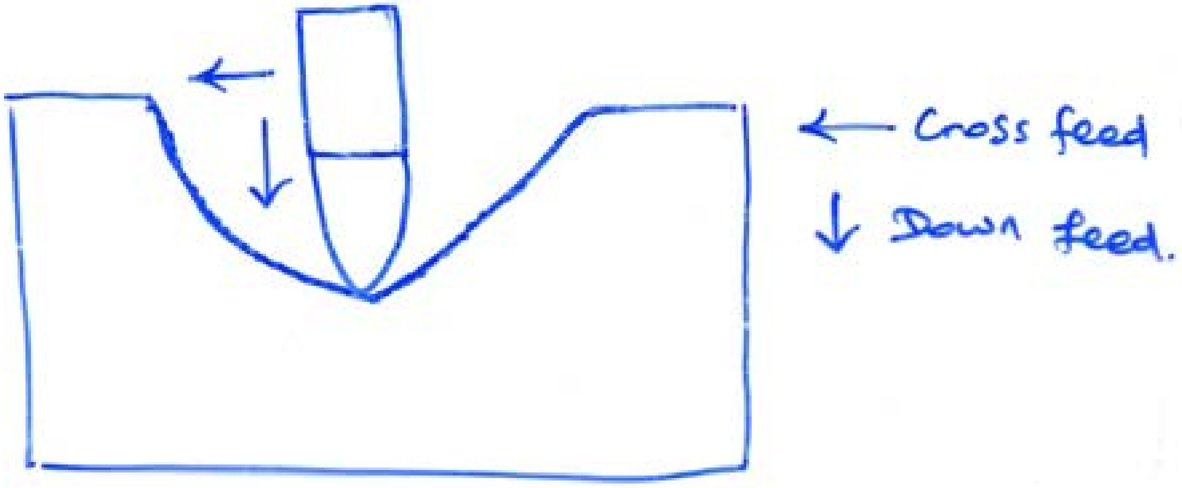
(2) To produce a small contoured surface, a forming tool is used.

(52)

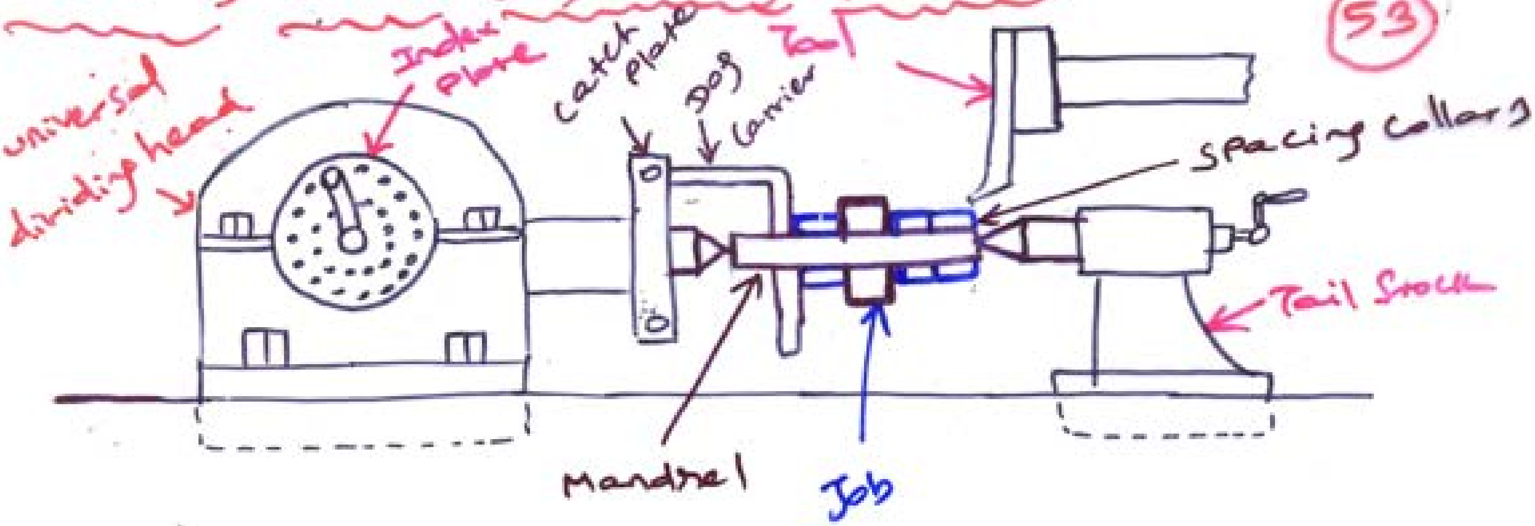
(3) If the curve is sufficiently large, power cross-feed in conjuction with manual down feed is so adjusted that the tool will trace the required contour.

(4) If the contour has too many ups & down, both of the feeds are operated by hand. A round nose tool is selected for machining irregular surface.

(5) For shallow cut, the apron may be set vertical. But if the curve is quite sharp, the apron is swivelled towards right (or) left away from the surface to be cut. The following figure shows machining of concave surface using a round nose tool.



Machining splines (or) Cutting Gears:-



* Here, After each tooth is cut, the crank of the index plate is moved to a pre-determined amount along a selected hole circle so that the workpiece will be equally divided.
* Here, a formed tool is used for cutting gears (or) splines on the shaft.

SHAPER SPECIFICATIONS:-

1. Maximum length of stroke
2. Maximum cross-wise movement of the table
3. Maximum vertical adjustment of the table
4. Type of driving mechanism
5. Power of the motor
6. Speed & feed available
7. Type of Shaper - plain (or) Universal
8. Floor space required.
9. Total weight of the Shaper
10. Ratio of cutting stroke time to return stroke time

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

$$m \sim 2:1 \text{ to } 3:2$$

Machining time calculation in a Shaper:—

(54)

Formulae's used:—

① Cutting Speed (V):—

ⓐ Time taken for cutting stroke $= \dfrac{\text{Length of cutting stroke } (L) \text{ in mm}}{\text{Cutting Speed } (V) \times 1000 \text{ in mm/min}}$

$$= \frac{L}{1000 \times V}$$

ⓑ The ratio b/w return to cutting time $m = \dfrac{\text{Return time}}{\text{Cutting time}}$

$\therefore$ Return time $= m \times$ Cutting time

$$= \frac{m \times L}{1000 \times V}$$

ⓒ $\therefore$ Time taken to complete one double stroke $= \dfrac{L}{1000 \times V}\left[1 + m\right]$

ⓓ Number of double stroke (or) Number of revolution of bull gear per minute $n = \dfrac{1}{\dfrac{L}{1000 \times V}(1+m)}$

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

(e) $\boxed{\therefore \text{Cutting Speed, } v = \frac{nL(1+m)}{1000}}$ m/minute

55

where

n → number of double Stroke/min

L → Length of the Stroke

m → Ratio of Return time to Cutting time

(2) Machining time:—

Let,

L = Length of the Stroke in mm

B = Breadth of the work in mm

S = feed in mm/double Stroke

m = the Ratio of return to Cutting time

v = Cutting Speed in m/min

a) Time taken to complete one double Stroke $= \frac{L}{1000 \times v}[1+m]$

b) Total number of double Stroke $= \frac{B}{S}$

(c) Total machining time $= \frac{L \times B}{1000 \times v}[1+m]$

Problems:—

(1) In a Shaper work, the length of the Stroke is 200mm, number of double strokes per minute is 30 and the ratio of return time to cutting time is 2:3. Find the cutting speed. (5b)

Given:—

$L = 200mm; \quad n = 30; \quad m = 2/3; \quad v = ?$

Soln:—

$$n = \frac{1}{\frac{L}{1000 \times v}(1+m)} = \frac{1000 \times v}{L(1+m)}$$

$$\therefore v = \frac{nL(1+m)}{1000} = \frac{30 \times 200(1+2/3)}{1000}$$

$$\boxed{v = 10\ m/min}$$

---

(2) Find the time required for taking a complete cut on a plate 600 x 900mm if the cutting speed is 9 m/min. The return time to cutting time ratio is 1:4 and the feed is 3mm. The clearance at each end is 75mm.

Given:-

$v = 9$ m/min; $m = \frac{1}{4}$; feed, $s = 3$ mm; 57

clearance at each end $\Big\} = 75$ mm;

Find:- Machining time required for 600×900 mm w/p ?

Soln:-

In a Shaper, a Stroke length of more than 900mm is not ordinarily available. So the work is placed on the table to take a cut of 600mm plus the clearances.

$\therefore$ stroke length, $L = 600 + 75 + 75$

$$\boxed{L = 750 \text{ mm}}$$

no of double stroke required $\Big\} = \frac{900}{3} = 300$

$\therefore$ Total machining time $\Big\} \Rightarrow \frac{L}{1000 \times v}(1+m) \times 300$

$$= \frac{750}{1000 \times 9}\left(1 + \frac{1}{4}\right) \times 300$$

$$\boxed{\text{Machining time for Single Pass} = 31.25 \text{ minutes.}}$$

Drilling Machine

It is the machine used to drill a hole on the work piece quickly at a low cost.

Here the hole is generated by the rotating edge of the cutting tool known as drill bit which exerts large force on the work clamped on the table

Types of drilling machine:-

1. Portable drilling machine
2. Sensitive drilling machine
3. Upright drilling machine
4. Radial drilling machine → plain / Semi-universal / universal
5. Gang drilling machine
6. Multiple spindle drilling machine
7. Automatic drilling machine
8. Deep hole drilling machine

① Portable drilling machine:—

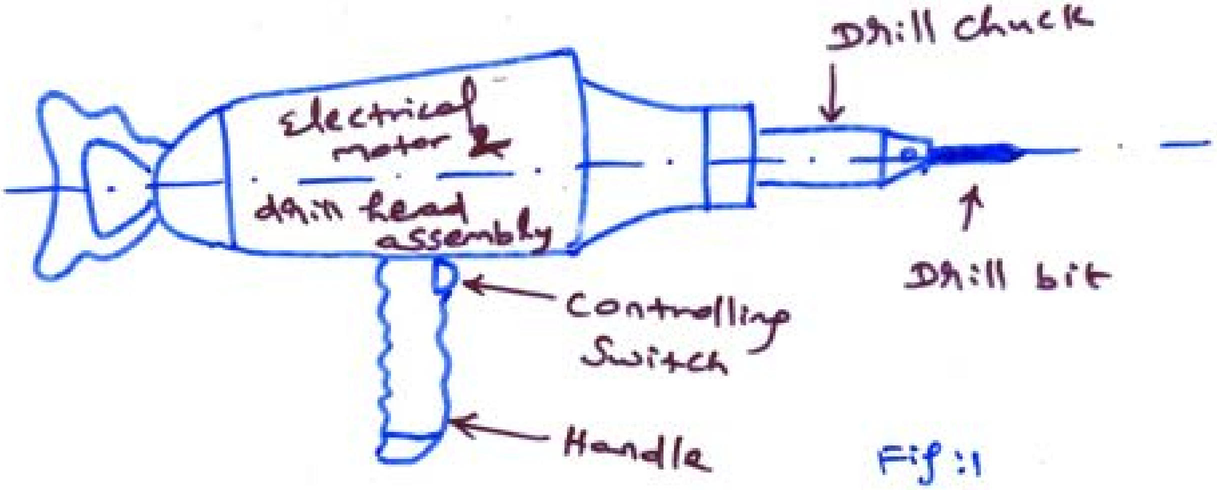


Fig:1

* It is Small & compact in size & can be taken to anywhere & the holes can be produced on the workpiece in any position.

* the maximum size of the drill that it can acconodate is not more than 12 to 18mm.

② Sensitive drilling machine:—

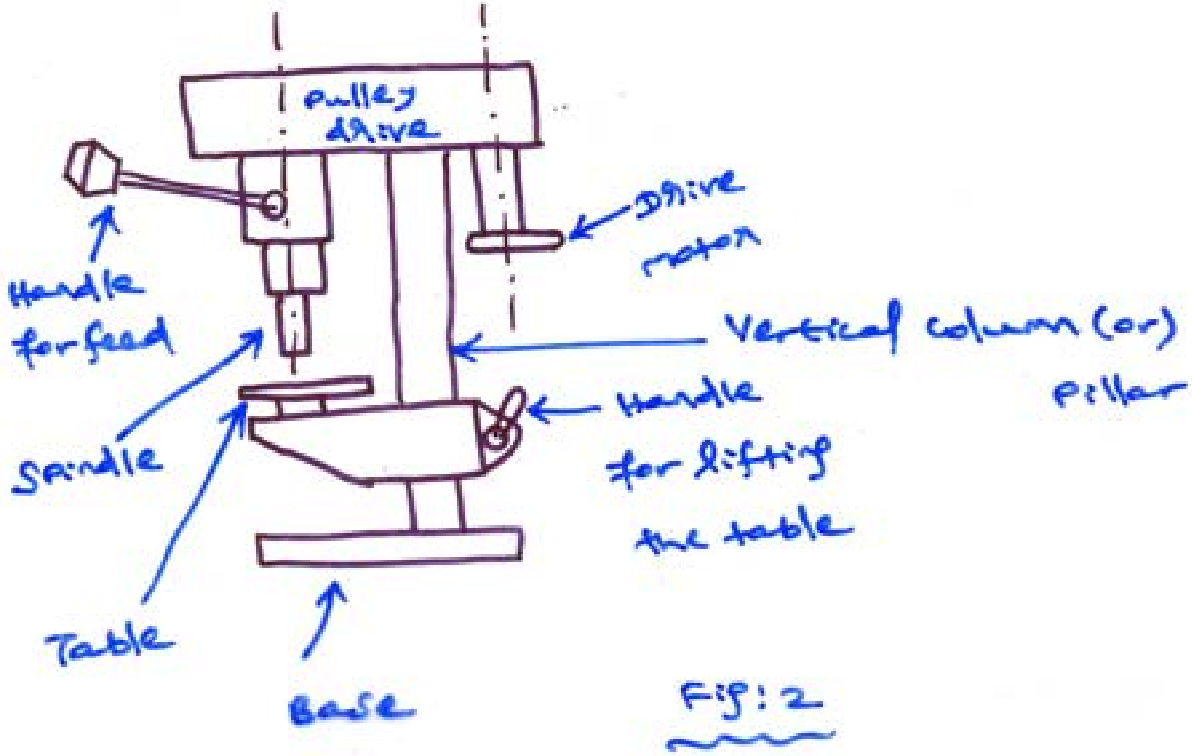


Fig:2

* used to produce small holes of dia from 1.5 to 15.5mm at high speed in light jobs.

* The base of the machine may be mounted on a bench (or) on the floor.

* It consists of

    1. horizontal table mounted on a pillar
    2. Vertical spindle for holding the drill bit.
    3. Vertical column (or) pillar
    4. Base
    5. Mechanisms for driving the vertical spindle

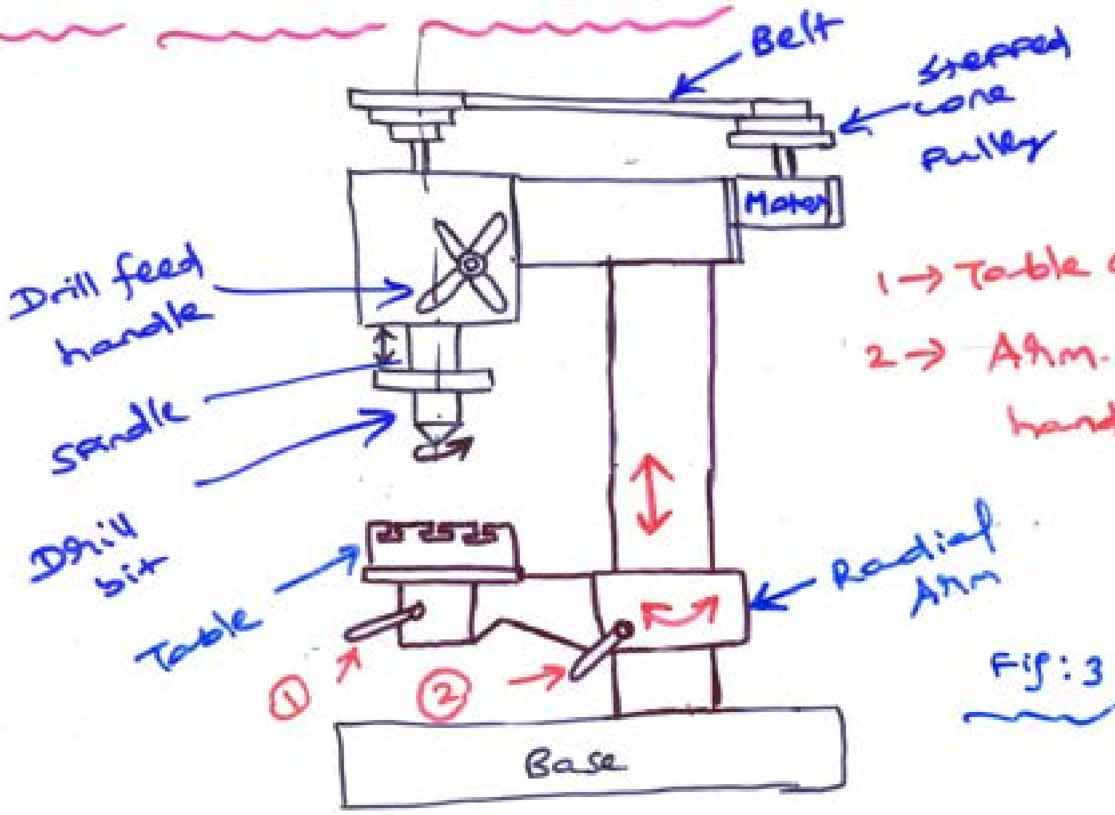
* The various speed for the spindle is achieved by means of Belt & pulley drive.

* Here, there is no automatic feed. Feed is given manually.

* Hand feed permits the operator to feel (or) sense the progress of the drill into the work so that if the drill becomes worn-out (or) jams into the work, the pressure on the drill may be released immediately to prevent it from breaking.

* As the operator senses the cutting action at any instants it is called "Sensitive drilling machine".

Upright (or) Pillar Drilling M/c:-



1 → Table clamp handle

2 → Arm. Elevating clamp handle

Fig: 3

Radial Drilling Machine:-

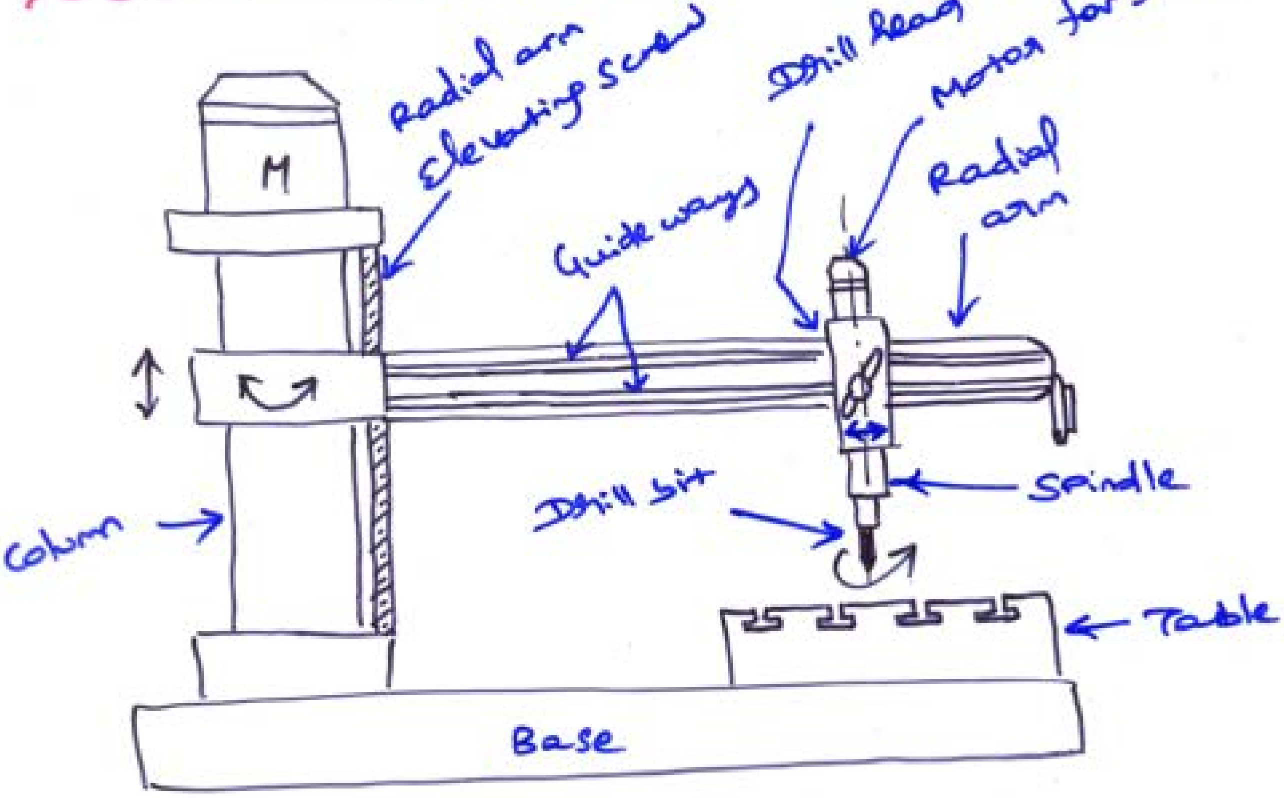


Fig: 4

③ Upright drilling machine [See fig: 3]

* Designed for handling medium sized jobs

* In construction, it is similar to sensitive drilling machine but it is larger & heavier than sensitive drilling machine & is supplied with power feed arrangement.

* A large number of spindle speeds & feeds are available in this machine for drilling different types of work.

* The table of the machine have different types of adjustments.

* It is mounted on the floor.

* It consists of the following parts

① Base ② Vertical column

③ Horizontal Arm & the table

④ spindle ⑤ driving mechanisms for spindle

① Base:—

It is a supporting member made up of cast Iron. It has T-slots on its top surface for mounting the large workpieces directly on it.

② Column:—

* Vertical member mounted on the base & carries the Radial arm & table, spindle & its driving mechanisms.

* It may be of Round type (or) Box type.
(Pillar)

③ Radial arm & table:—

* The table is mounted on a Radial arm which is mounted on a vertical column.

* It has following movements

① The movement of the Radial arm in a vertical direction along the column

② The swivelling movement of the Radial arm around the column upto 180°

③ The swivelling of the table about its own center upto 360°.

* By means of the above movements, the workpiece can be drilled in various positions.

④ Spindle:—

* used to hold the drill bit of various sizes

* The various speed for the spindle is achieved by step cone pulley & belt drive.

* The following feeds are available for Spindle

① Sensitive hand feed

② Quick traverse hand feed (64)

③ Auto-feed.

④ Radial drilling machine:- [See: Fig.4]

* This drilling machine is intended for medium to large & heavy workpieces.

* This machine consists of the following parts

① Base ② Column ③ Radial Arm

④ Drill head ⑤ Spindle mechanisms.

① Base:-

* It is a large rectangular casting

* It supports the vertical column & Table.

* The top surface of the base is accurately machined with T-Slots to mount the large size workpieces on it.

② Column:-

* This machine consists of heavy, round, vertical column mounted on a large base.

* The column supports a radial arm which can be raised (or) lowered over the column & swung around

column.

* An elevating screw is provided on the side of the column to move the radial arm up & down by a motor.

(3) Radial arm:-

65

* It is mounted on a vertical column
* It can be moved up & down over the column and can be swung around the column
* It supports the drill head.
* It has horizontal flat guide ways to move the drill head along a radial arm.
* By means of the above movements for a radial arm, the workpiece can be machined at different positions.

(4) Drill head:-

* It is mounted on a radial arm.
* It has a mechanism for driving the spindle & moving the drill head along the guide ways of the radial arm.
* the drill head has a spindle which carries the drill bit.

⑤ Mechanisms for driving the spindle:—

* The different speeds for a spindle is achieved through a gear box.

* The following feed for the spindle is available

  1. Sensitive hand feed
  2. Quick traverse feed
  3. Automatic feed.

Radial drilling machine may be classified with respect to the movement of radial arm & tool head.

1. Plain type
2. Semi-universal type
3. Universal type.

① Plain type:—

The following movements are available for a plain type radial drilling machine

* Up & down movement of the radial arm over the column.
* Swivelling movement of the radial arm around the column
* Movement of the tool head along the radial arm.

2) Semi-universal type:-

* In this type of radial drilling machine, in addition to the above three movements of the plain type radial drilling machine, the drill head can be swung about a horizontal axis perpendicular to the arm.

* The fourth movement of the drill head permits drilling hole at an angle to the horizontal plane other than the normal position.

3) Universal type:-

* In a universal machine, in addition to the above four movements, the arm holding the drill head may be rotated on a horizontal axis.

* All these five movements in a universal machine enables it to drill a work piece at any angle.

⑤ Gang drilling machine:-

* when a number of single spindle drilling machine columns are placed side by side on a common work table which is mounted on a common base, the machine is known as "Gang drilling machine".

* Four to six spindles may be mounted side by side.

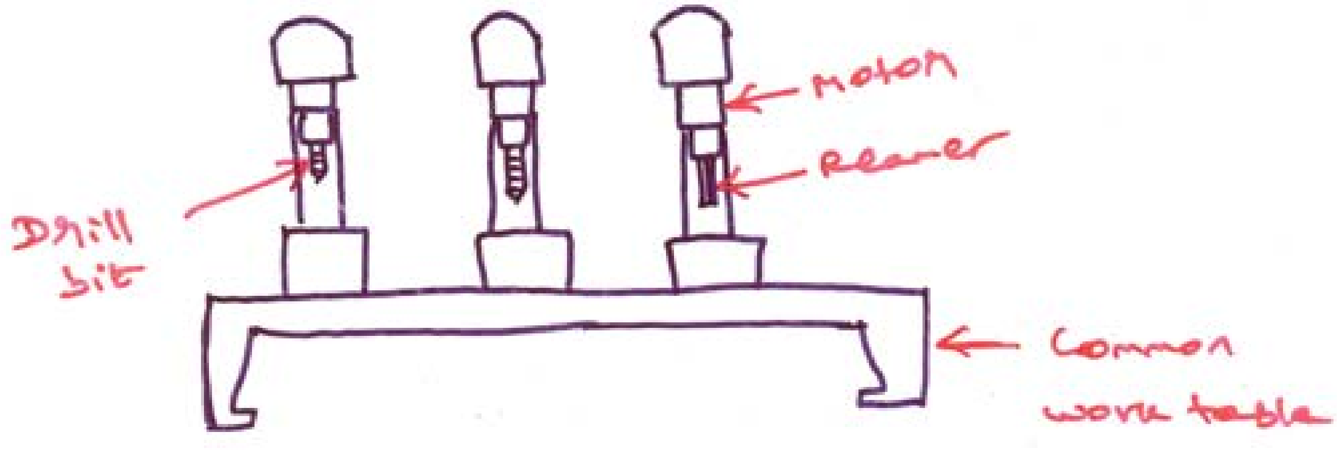
* In some machines, the drill spindles are permanently placed & in others the position of the column may be adjusted so that the space between the spindle may be varied. (68)

* The speed & feed of the spindles are controlled independently.

* This type of machine is specially adapted for production work.

* A series of operations may be performed on the work by simply shifting the work from one position to the other on the work table.

* Each spindle may be set up properly with different tools for different operations.



(6) Multiple spindle drilling machine:-

* This is used to drill a number of holes in a piece of work simultaneously & to reproduce

the same pattern of holes in a number of identical pieces in a mass production work. 69

* It has several spindle driven by a single motor & all the spindles holding drills are fed into the work simultaneously.

* Feeding motion is usually obtained by raising the table (or) lowering the tool head.

* The center distance b/w the spindles can be varied based on the different operations required.

7 Automatic drilling machine :-

* It can perform a series of machining operations at successive units and transfer the work from one unit to the other unit automatically.

* Once the work is loaded at first machine, the work will move from one machine to the other where different operations are done and the finished work comes out from the last unit without any manual handling.

* It is used purely for production purposes.

Deep hole drilling:-

* Deep hole drilling is the opreation of making a deep holes in a long shaft of L/D ratio from 6-30. such as rifle barrels, crank shaft etc.

* Deep hole drilling machines is opreated at high speed & low feed.

* Sufficient quantity of lubricant is pumped to the cutting points for removal of chips and cooling the cutting edges of the drill.

* A long shaft is usually supported at several points to prevent any deflection.

* Here, the work is usually rotated while the drill is fed into the work. This helps in feeding the drill in a straight path.

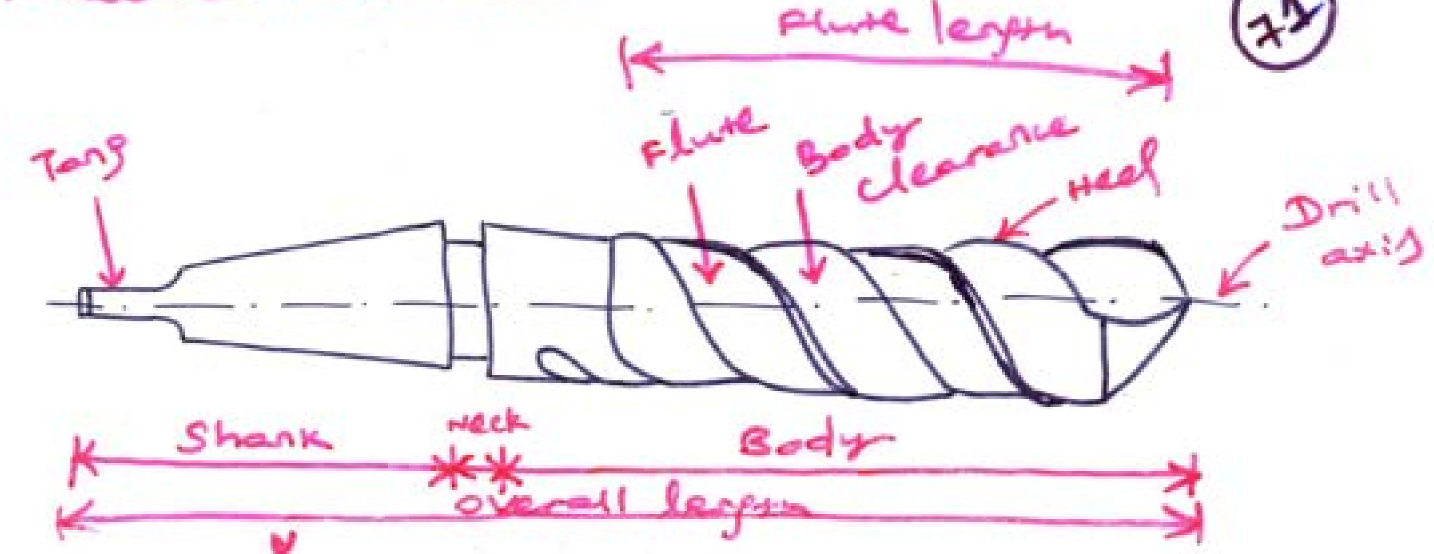
* In some machine, both the drill & the work are rotated for accurate location.

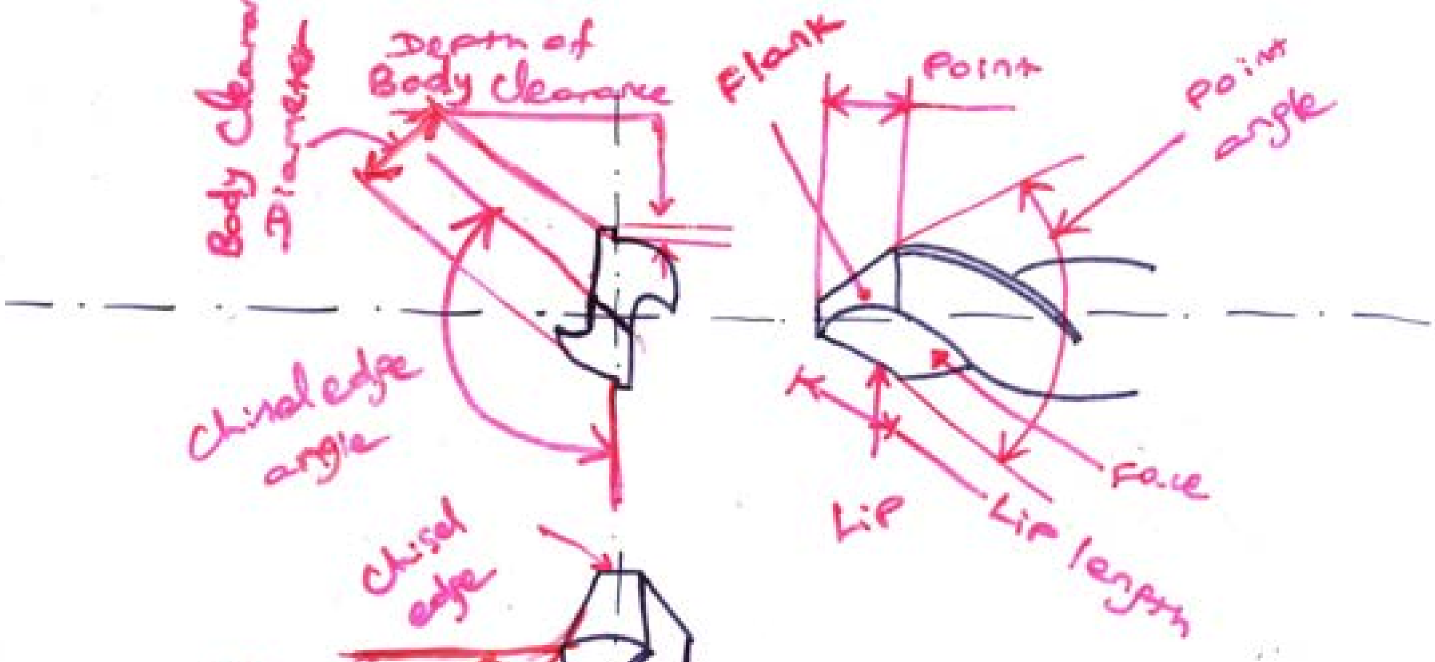
* The machine may be horizontal (or) vertical type.

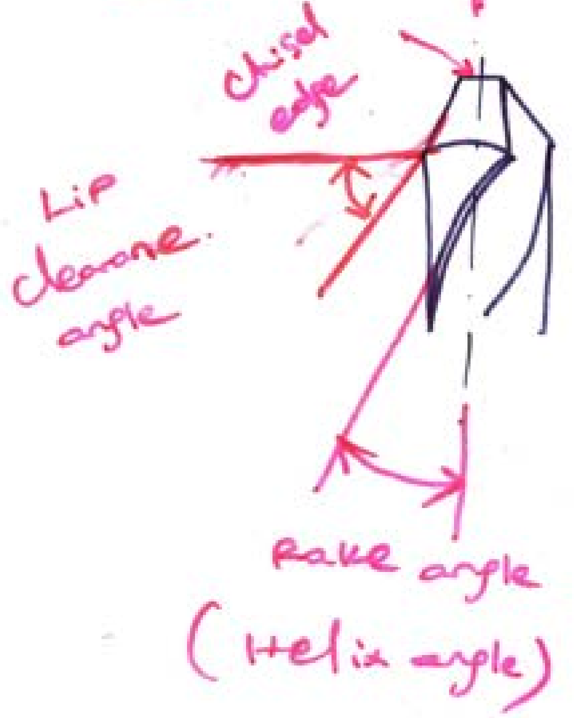
* In some machines, step feed is applied. The drill is withdrawn automatically each time when it penetrates into the work equal to its diameter. This process permits the chip to clear out from the work

Drill Tool [Twist Drill] Nomenclature :-

71







1. Point angle – 118°
2. Helix (or) Rake angle 30°–45°
3. Chisel edge angle – 120°–135°
4. Lip clearance angle – 12°

① **Axis:-** the longitudinal center line of the drill.

(72)

② **Body:-**

Portion of the drill extending from its extreme point to the commencement of the neck

③ **Shank:-**

Portion of the drill by which it is held into the spindle & driven. The most common types of shank are the taper shank & the straight shank.

④ **Tang:-**

The flattened end of the taper shank intended into a drift slot in the spindle. The tang ensures positive drive of the drill from the drill spindle

⑤ **Neck:-**

The diametrically undercut portion between the body and the shank of the drill. Diameter and other particulars of the drill are engraved at the neck

⑥ **Flutes:-**

It denotes the grooves in the body of the drill the function of the flutes are

1. To form the cutting edges of the drill
2. To allow the Chips to escape
3. To cause the chips to curl
4. To permit the cutting fluid to reach the cutting edge

(7) Flank:-

It denotes the Surface on the drill point.

(8) Face:-

the Portion of the flute Surface in the point of the drill. the chip impinges on the face as it is cut from the work.

(9) Lip (cutting edge):-

The edge formed by the intersection of flank & face.

(10) Chisel edge:-

the edge formed by the intersection of the flanks. It is sometimes called as "Dead Center". A great amount of axial thrust is required to cut a hole by a chisel edge.

(11) Body clearance:-

The portion of the body surface which is reduced in diameter to provide diametral clearance.

(12) Heel:-

The edge formed by the intersection of flute surface & the body clearance

(13) Lands:-

The cylindrically ground surface on the edges of the drill flutes.

(14) Heel:-

The edge formed by the intersection of body clearance & the flute.

(15) Point:-

The sharpened end of the drill, which is shaped to produce lips, faces, flanks and chisel edge

(16) Drill angles:-

Following are the drill angles which are ground on a twist drill for efficient removal of metal.

1. Chisel edge angle
2. Helix angle (or) Rake angle
3. Point angle
4. Lip Clearance angle

Chisel edge angle:-

The obtuse angle included b/w the chisel edge & the lip as viewed from the end of the drill. It varies from 120° to 135°.

Helix angle (or) Rake angle:-

* It is the angle formed by the leading edge of the land with the axis of the drill.

* The usual angle of the rake = 30° − 45°

Point angle:-

* It is the angle included between the two lips. The usual point angle is 118°, but for harder Steel alloys, the angle increases.

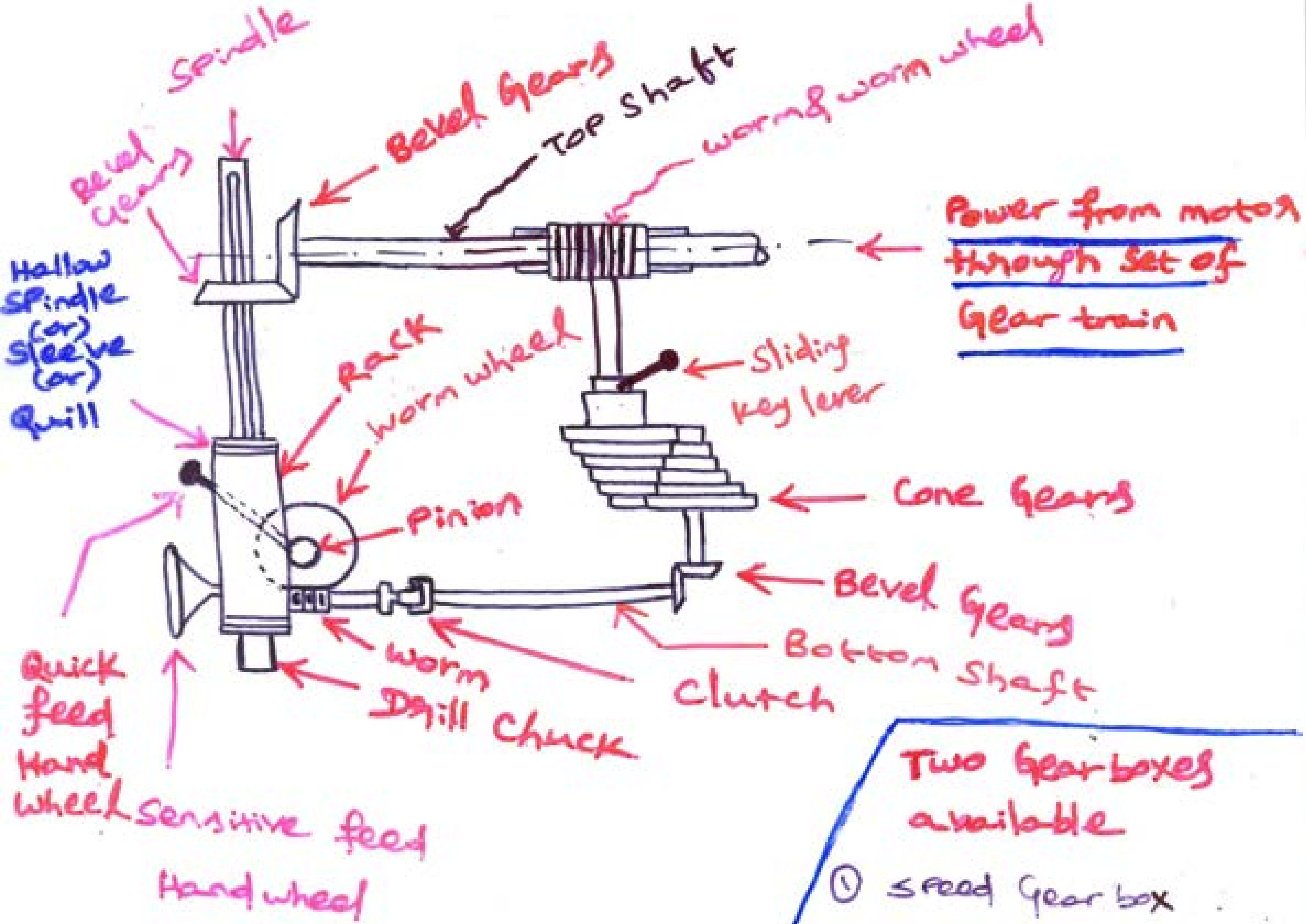
Lip clearance angle:-

The angle formed by the lip and a plane at right angle to the drill axis. The clearance angle is 12° in most cases.

Specification of Drilling machine:-

1. Maximum Size of the drill in mm that the machine can hold.
2. Table Size in Square meter.
3. Maximum spindle travel.
4. Number of spindle speeds & range of speeds in R.P.M
5. Number of auto Spindle feed & feed range in mm/rev
6. Morse taper number of the drill Spindle taper portion
7. Power input of the machine in H.P
8. Floor Space required in Square meter
9. Net weight of the machine in Tonne

Quill mechanism (or) feed mechanism :-



Two Gear boxes available

① speed Gear box

② feed Gear box

there are 3 types of feed which can be obtained in drilling machine

① Automatic feed for spindle

② Sensitive Hand feed

③ Quick traverse Hand feed.

① Auto feed for Spindle:—

* The automatic feed is applied while drilling large diameter holes as the cutting pressure required is sufficiently great.

* The power from the top shaft is transferred to the six speed feed box through worm & worm wheel.

* In the feed box,

① Six Gears mounted on the worm gear shaft are constantly in mesh with another six gears mounted on the driven shaft.

② Gears on the driven shaft are all keyed to the shaft & rotate with it.

③ Gears on the worm gear shaft are all free to rotate, but may be keyed to the shaft by a sliding key only by one gear at a time.

* When the sliding key is in the first gear, the motion is transmitted to the driven shaft from the first gear. Other gears on the worm shaft revolve freely with their mating gear on the driven shaft.

* thus six different speed for the driven shaft are obtained by sliding the key to engage with six different gears on the worm gear shaft.

79

* the motion from the driven shaft is transmitted to the bottom shaft through a set of bevel gears.

* The worm at the end of the bottom shaft receives the power by a clutch on the bottom shaft.

* when the worm at the end of the bottom shaft rotates, the worm wheel meshing with that also rotates. At the end of this worm wheel shafts, there is a pinion. this pinion meshes with the rack which is bolted to the quill (or) spindle.

* The rotation of the pinion causes the quill to move up & down giving spindle feed.

② Sensitive Hand feed:—

* when the sensitive hand feed is applied, the clutch is disconnected.

* the sensitive hand feed wheel is attached to the rear end of the bottom shaft.

* So, the rotation of the sensitive hand wheel will cause the worm & worm gear to rotate and a slow but a sensitive feed is obtained.

3 Quick traverse hand feed:-

* Quick traverse hand feed is obtained by rotating Quick traverse hand wheel which is attached with pinion

* One turn of the quick traverse hand wheel will cause the pinion to rotate through one complete revolution giving quick hand feed movement of the spindle

* Clutch should be disengaged when giving the quick traverse hand feed.

Methods of mounting (or) holding the drill bit into the spindle (or) Tool holding devices

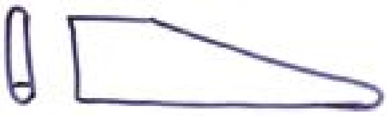
8

1. By directly fitting in the spindle
2. By using sleeve
3. By using socket
4. By using Drill Chuck.

① Directly holding the tool:-

Fig:1

Drift



③ Socket

② Sleeve:-

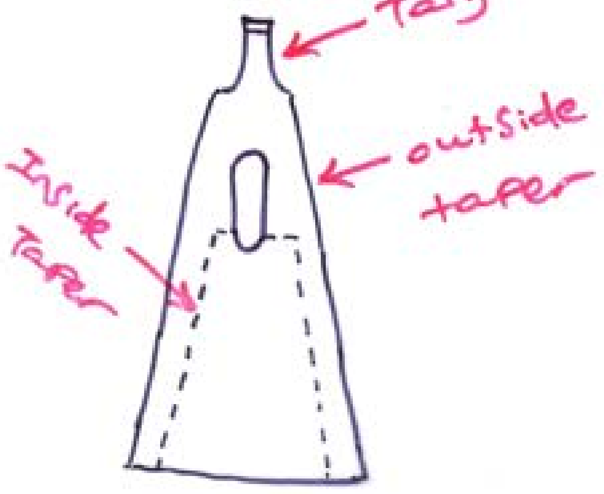


Fig:2

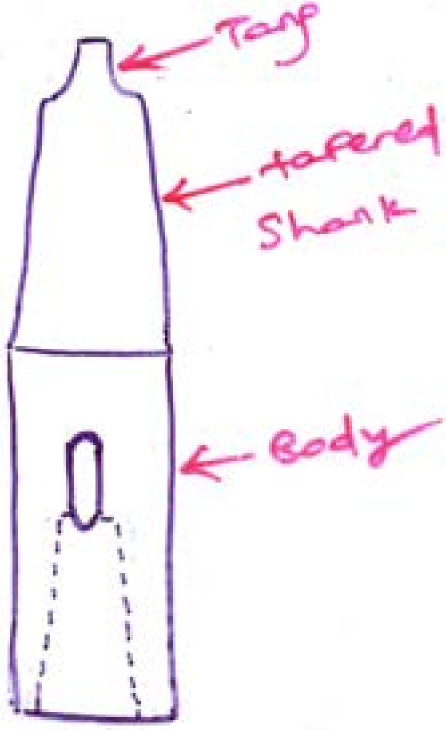


Fig:3

Drill Chuck:-

82

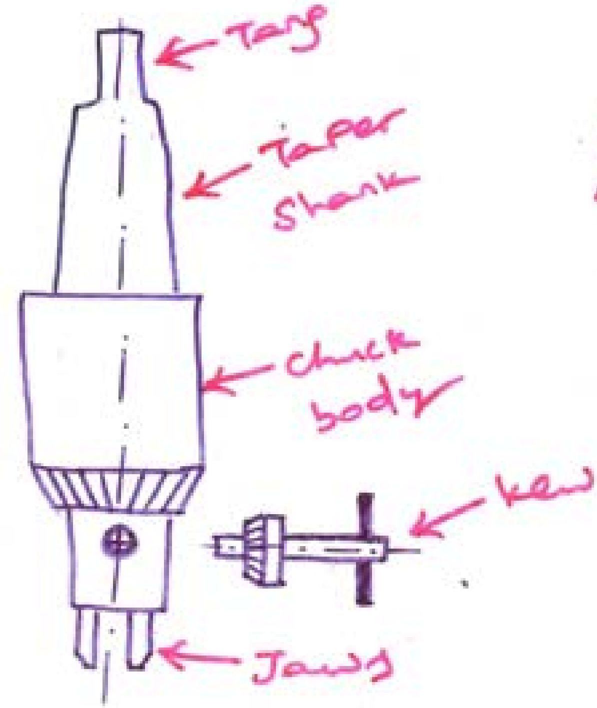
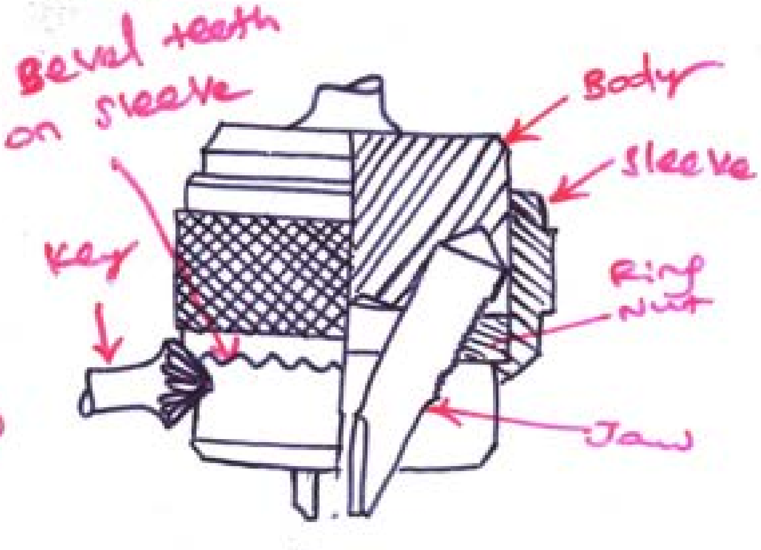


Fig: 4



Directly holding the tool:- [See fig:1]

* All general purpose drilling machine have the spindle bored out to a standard taper to receive the taper shank of the tool.
* The taper used in the drill spindle is usually Morse standard taper which is approximately 1:20.
* while fitting the tool, the shank is forced into the tapered hole & the tool is gripped by friction.
* The tool may be rotated with the spindle by friction between the tapered surface of the spindle

and Shank of the tool; but to ensure a positive drive, the tang (or) tongue of the tool fits into a slot at the end of the taper hole of the spindle.

* The tool may be removed by pressing a tapered wedge known as drift into the slotted hole of the spindle.

83

* It can be noted that the spindle should not be affected while the drift is pushed to take out the drill bit. Usually a mallet should be used in stead of a hammer.

Sleeve :- [Fig: 2]

* The drill spindle is suitable for holding only one size of the shank.

* If the taper shank of the tool is smaller than the taper in the spindle hole, a taper sleeve is used.

* The outside taper of the sleeve conforms to the drill spindle taper and the inside taper of the sleeve holds the shank of the smaller size tools.

* The sleeve has a flattend end (or) tang which fits into the slot of the spindle.

* The tang of the tool fits into the slot provided at the end of the taper hole of the sleeve.

* The tool from the sleeve & the sleeve from the spindle may be removed by forcing the drift into the corresponding slot by a mallet.

* Different sizes of tool shanks may be held in the spindle by using different sizes of sleeve

* The taper on the outer surface of the sleeve doesn't change but the inner taper surface varies with different sizes of the tool shanks.

3 Socket :- [See fig: 3]

* when the tapered tool shank is larger than the spindle taper, drill sockets are used to hold the tool. Drill sockets are much longer in size than the drill sleeves.

* The tapered shank of the socket conforms to taper hole of drill spindle & fits into it.

* The body of the socket has a taper hole larger than the drill spindle taper into which the taper shank of any tool may be fitted.

④ Three jaw self centering chuck:— [See fig: 4]

* this type of chuck is particularly adaped for holding the tools having straight shanks. (85)

* three slots are cut 120° apart in the chuck body which houses three jaws having threads cut at the back that meshes with a ring nut.

* the ring nut is attached to the sleeve.

* Bevel teeth are cut all round the sleeve body

* the sleeve may be rotated by rotating a key having bevel teeth cut on its face which meshes with the bevel teeth on the sleeve

* The rotation of the sleeve causes the ring nut to rotate in a fixed position and all the three jaws will move simultaneously by the same amount. and hold the drill bit having straight shanks.

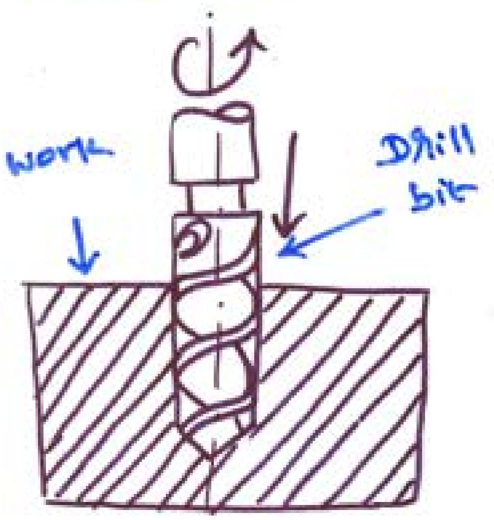
Drilling Machine operations:-

(1) Drilling (2) Reaming (3) Boring (4) Counter Boring

(5) Counter Sinking (6) Spot facing (7) Tapping

(8) Lapping (9) Grinding (10) Trepanning.

Drilling:-



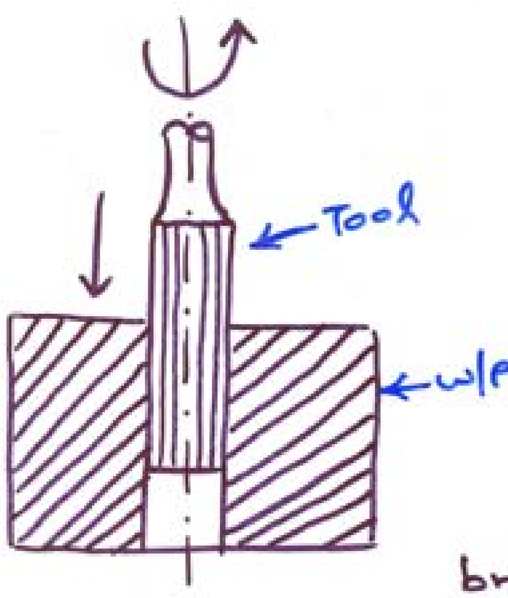
* Drilling is the opreation of producing a cylindrical hole by feeding the rotating drill into the workpiece

* Before drilling, the center of the hole is located on the w/p by drawing two lines at right angle to each other & the center punch is used to produce an indentation at the center. The drill point is pressed at this center point to produce the required hole.

* Drilling does not produce an accurate hole in a work piece and the hole so generated by drilling becomes rough and the hole is always slightly over size than the drill used due to the vibration of the spindle & drill.

* A 12 mm drill may produce a hole which 0.125 mm oversize
  A 22 mm drill may produce a hole which 0.5 mm oversize

Reaming:—



* It is the opreation of sizing & finishing a hole which has been already drilled. the tool used for this opreation is called as "Reamer" (or) "Reaming tool".

* In order to finish a hole & to bring it to the accurate size, the hole is drilled slightly undersize

* The spindle speed is made half of the speed for drilling opreation & auto feed may be given.

* Reaming tool (or) Reamer has multiple cutting edges and it doesnot originate a hole. It simply follows the path which has been already drilled and removes a very small amount of metal.

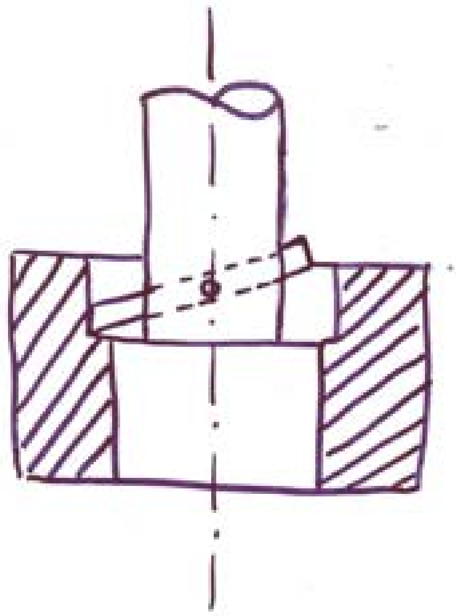
* For this reason, a reamer cannot correct a hole

* The material removed by this process is around 0.375mm.

Boring:—

* It is the opreation of enlarging the already drilled hole by an adjustable cutting tool with only one cutting

edge called "Boring tool".



* Boring opreation is performed in a drilling machine for the following reason.

① To enlarge a hole by an adjustable cutting tool with only one cutting edge. this is necessary where suitable sized drill is not available.

② To finish a hole accurately & to bring it to the required size.

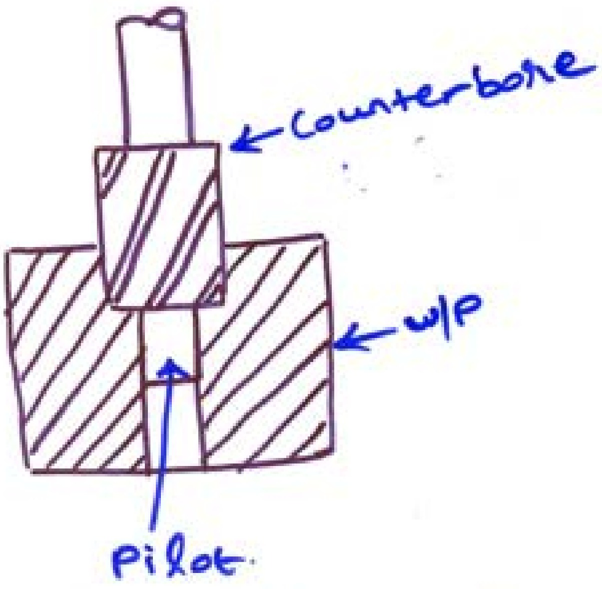
③ To machine the internal surface of a hole already produced in the casting.

④ To correct out of roundness of a hole

⑤ To correct the location of the hole as the boring tool follows an independent path with respect to the hole

* Here the cutting tool is held in a boring bar which has a taper shank to fit into the spindle socket.

Counter boring:-

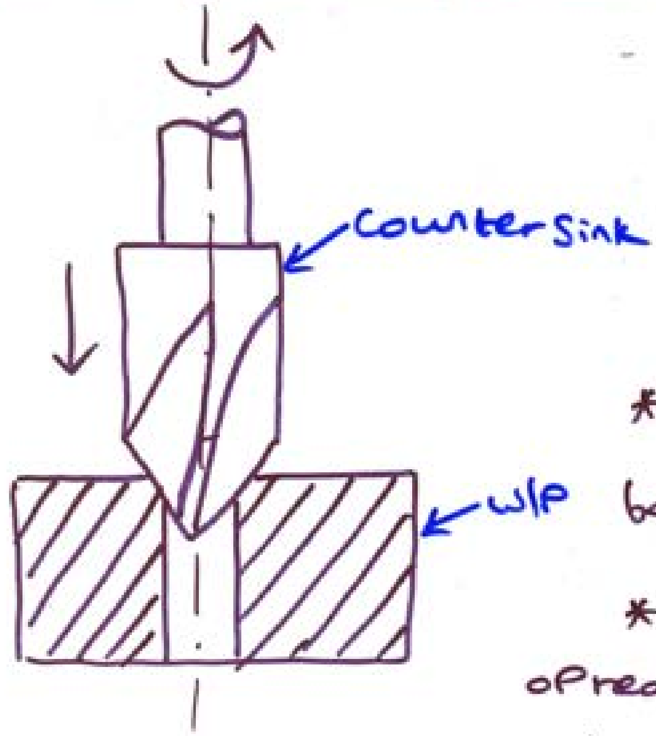


* It is the operation of enlarging the end of a hole cylindrically.
* The enlarged hole forms a square shoulder with the original hole. It is necessary in some cases to accomodate the heads of bolts, studs, pins etc.,

* the tool used for counterboring operation is called as counter bore. It is made with tapered (or) straight shank to fit into the drill spindle. The cutting edges may be straight (or) spiral teeth.
* The tool is guided by a pilot which fits into the small diameter hole having running clearance & maintains the alignment of the tool.
* these pilots may be interchanged for enlarging different sizes of holes.

Counter sinking:-

It is the operation of making cone-shaped enlargement of the end of the hole to provide recess for a flat head screw (or) counter sunk rivet.

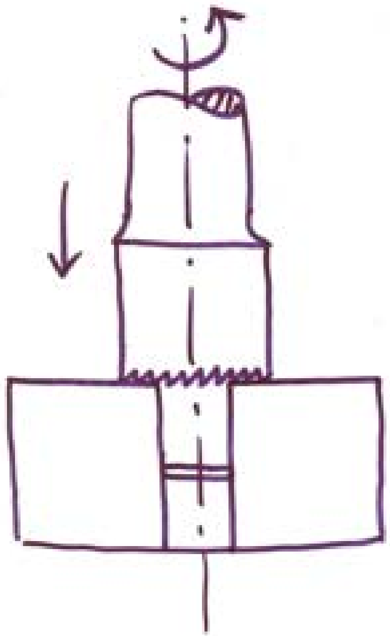


* The tool used for counter sinking opreation is called as countersink.

* Standard counter-sinks have 60°, 82° (or) 90° included angle.

* The cutting speed for this opreation is 25% less than that of drilling.

Spot facing:-

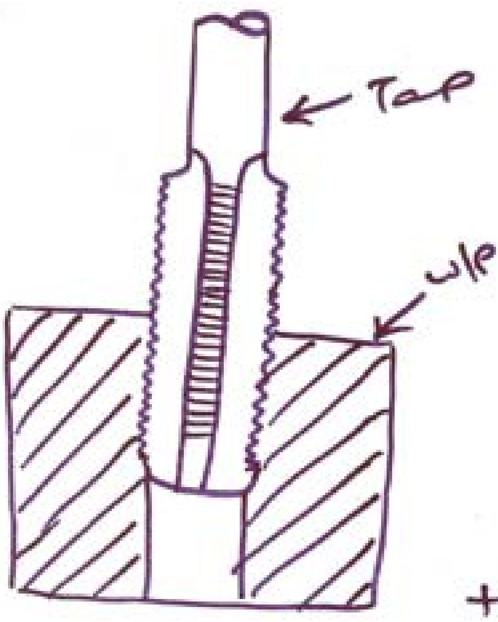


* It is the opreation of smoothing and squaring the surface around a hole for the seat for a nut (or) the head of screw.

* A counter bore (or) special spot facing tool may be employed for this purpose

Tapping:-

It is the opreation of cutting internal threads into the already drilled hole by means of a cutting tool called a "tap".



* A tap may be considered as a bolt with accurate threads cut on it.

* The threads acts as cutting edges which are hardened and ground.

* when the tap is screwed into the hole, it removes metal and cuts internal threads which will fit into the external threads of the same size

Tap drill size:-

* As the size of the thread is the outside diameter of the threads, the drilled hole must be smaller than the tap by twice the depth of the thread.

* Tap drill size may thus be derived from the following formula

$$\boxed{D = T - 2d}$$

D → Dia of drilled hole

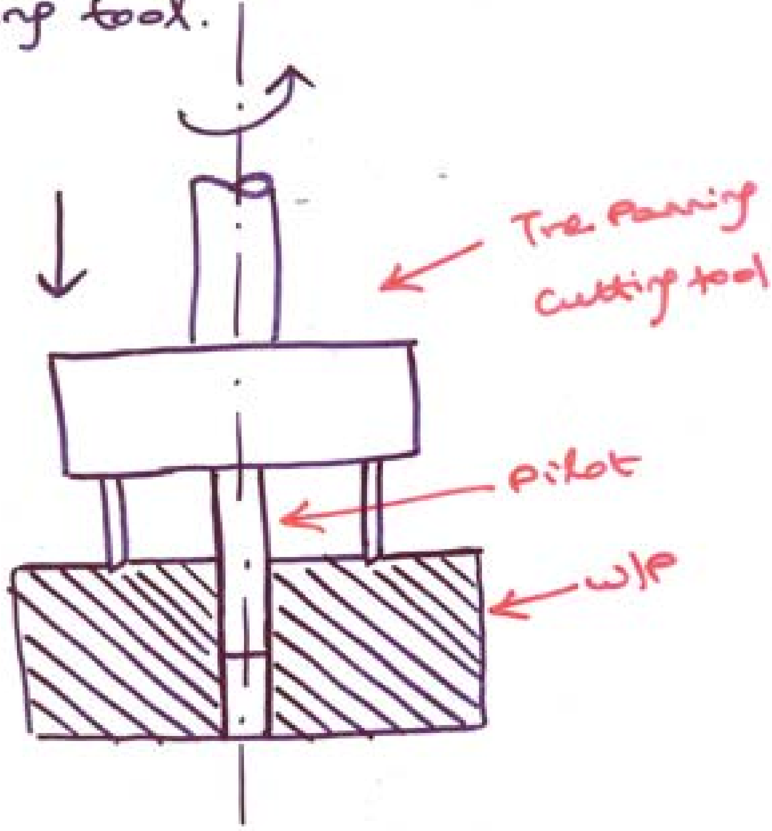
T → Dia of tap

d → depth of thread of the tap

Trepannings:-

* It is the operation of producing a hole by removing the metal along the circumference of the

hallow cutting tool.



* It is performed for producing bigger holes
* Fewer chips are removed when machining.
* Tre-panning opreation are done at higher speeds.
* this is one of the efficient method for producing a bigger hole

Feed, cutting speed, depth of cut of a drilling opreation:—

Cutting speed:—

It denotes the peripheral speed of a cutting point on the Surface of the drill in contact with work

$$\boxed{v = \frac{\pi d n}{1000} \text{ m/min}}$$

$v \rightarrow$ cutting speed

$d \rightarrow$ dia of the drill in mm

$n \rightarrow$ rpm of the drill

Feed:-

It denotes the distance the drill moves into the work at each revolution of the spindle. It is expressed in mm/rev (or) mm/min.

The feed per minute can be calculated as

$$\boxed{S_m = S_r \times n}$$

S_m = feed in mm/min

S_r = feed in mm/rev

n = r.p.m of the drill.

Depth of cut:-

Depth of cut is generally taken as one-half of drill diameter

$$\boxed{t = d/2}$$

t = depth of cut in mm.

Machining time in drilling:—

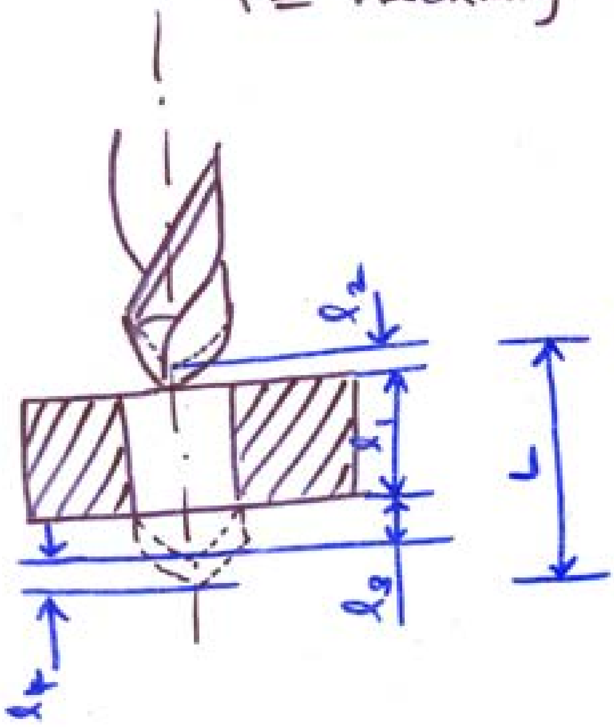
$$\boxed{T = \frac{L}{n \times S_r} \text{ min}}$$

n = r.p.m of the drill

S_r = feed of the drill in mm/rev

L = Length of travel of drill in mm

T = Machining time in min.



$$L = l_1 + l_2 + l_3 + l_4$$

where

l_1 = length of the w/p

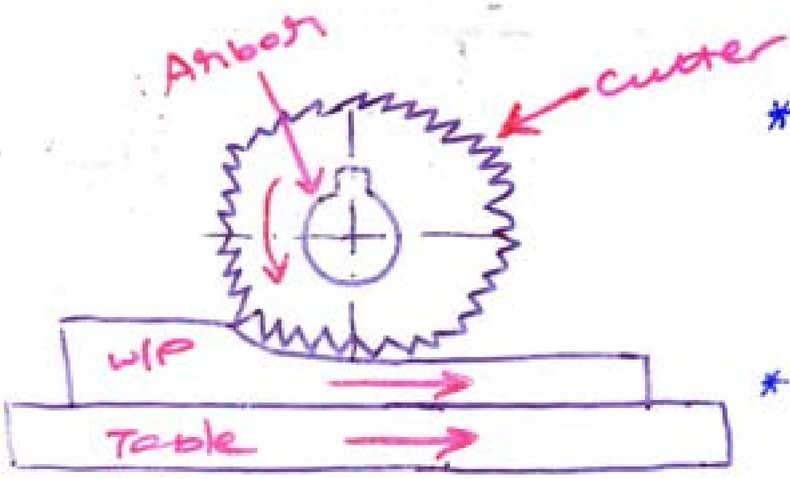
l_2 = Approach of the drill

l_3 = length of the drill point $(0.29d)$

l_4 = over travel.

MILLING

Principle of operation:-



* feed:-

  Longitudinal, cross, vertical feed.

* More life to the cutting edge.

Classification of milling m/c:-

① Column & knee type → Plain (or) Horizontal, Vertical, Ram type, Universal, Omniversal milling m/c's

② Bed-type milling m/c → (a) Simplex milling m/c, (b) Duplex milling m/c, (c) Triplex milling m/c

③ Plano type milling machine

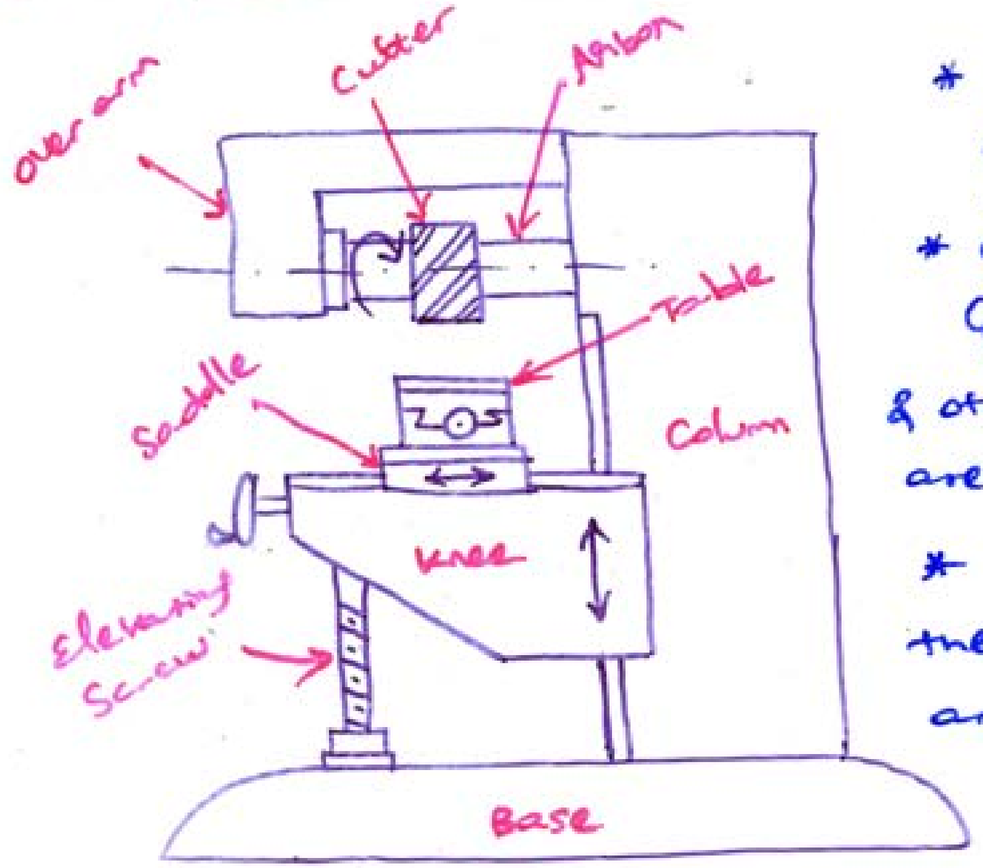
④ Special purpose milling machine

(a) Rotary table milling machine

(b) Drum milling machine

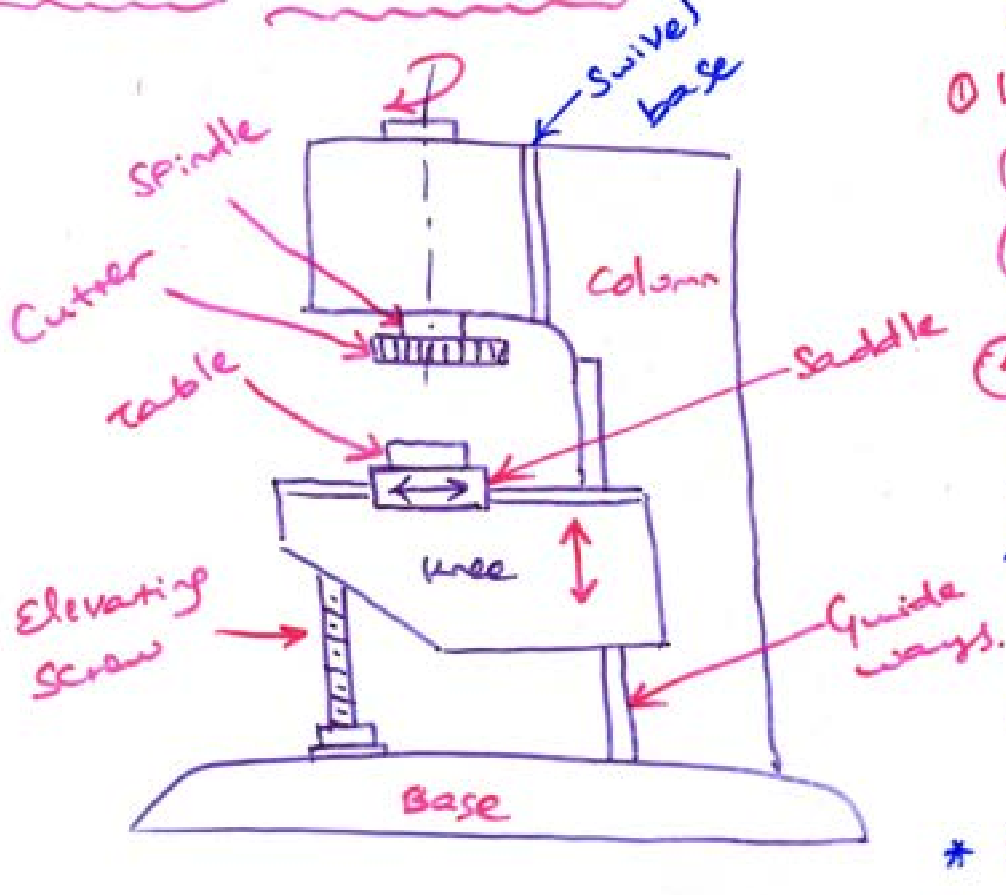
(c) Profile milling machine.

① Plain (or) Horizontal milling machine:—



* Base made of grey C.I serves as reservoir for cutting fluid.
* Column supports & Guides the knee. Motor & other driving mechanisms are housed in it.
* Overarm supports the outer end of the arbor to prevent it from bending.

② Vertical milling machine:—



Common feeds

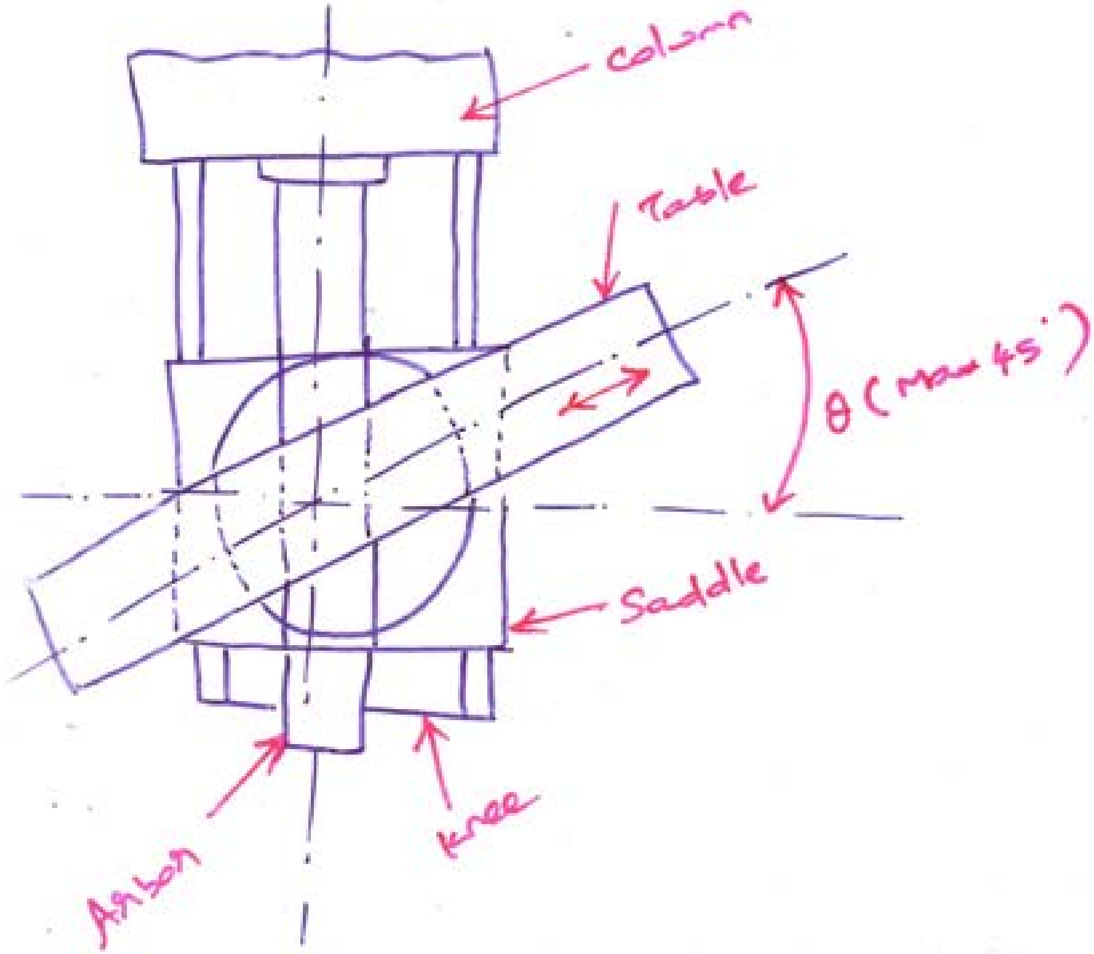
① Longitudinal by table
② cross by saddle
③ Vertical by knee
④ Rotation of spindle

with this movement, the spindle head can be swivelled at an angle.

* Mainly used for end milling and face milling operations.

{Power feed — Saddle & Table}

Universal Milling Machine:-



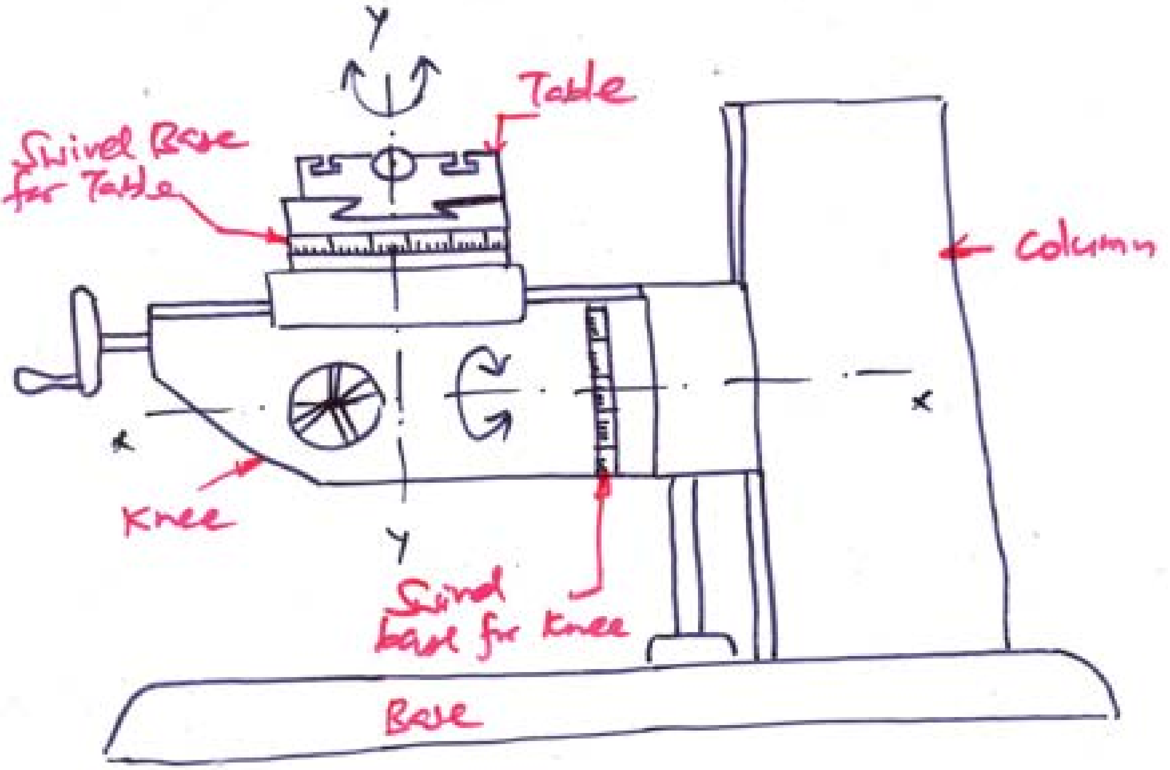
Comparison b/w Plain & Universal milling machine:-

① Table movements

② Plain milling m/c → Spur Gears, Splined Shafts, making a round rod into Square, Pentagon, Hexagonal Shaped block by using dividing head, Horizontal milling, Side face milling etc.,

Universal milling m/c → with all the oprns done on Plain milling m/c, Helical gears, Spiral bevel gears, twist drills, Reamers, milling cutters, all types of milling, drilling, Shaping oprns

Omniversal Milling m/c:-



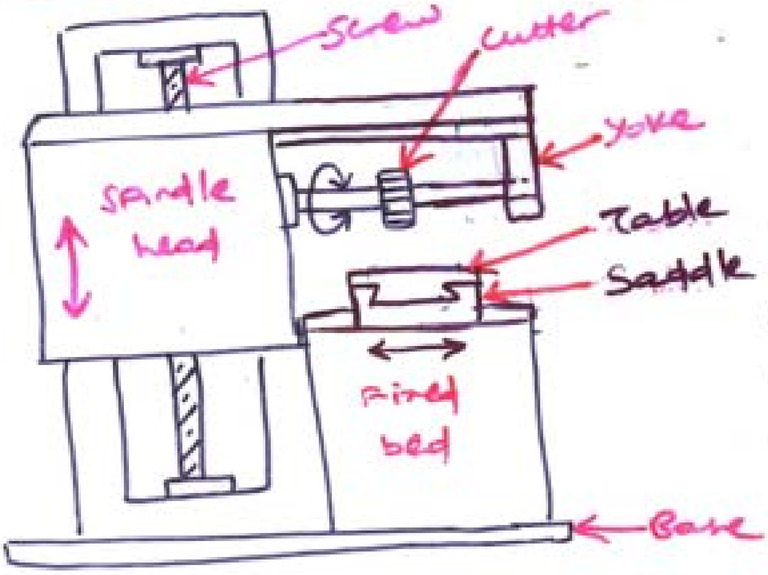
Applications:-

Taper spiral Grooves in reamers & Bevel Gears, Twist Drill, Spur & Helical Gears, milling cutters & All types of milling, drilling & shaping oprns.

Bed type Milling m/c:-

* comparatively large, heavy in construction
* Vertical motion imparted to spindle instead of Table
* Dep on no of spindle, simplex, Duplex Triplex bed Type milling m/c.

1. Simplex (Fixed bed type) Milling m/c:-

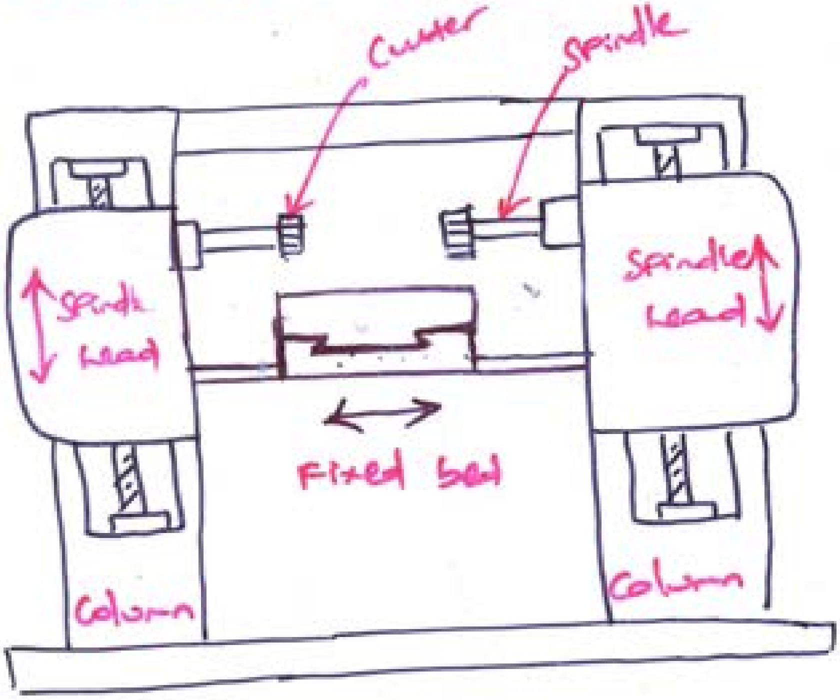


Since the bed is fixed, it can not move up, down [illegible].

The spindle head carries the spindle, which can be moved up & down along the column ways.

Table → cross-wise movement
longitudinal movement

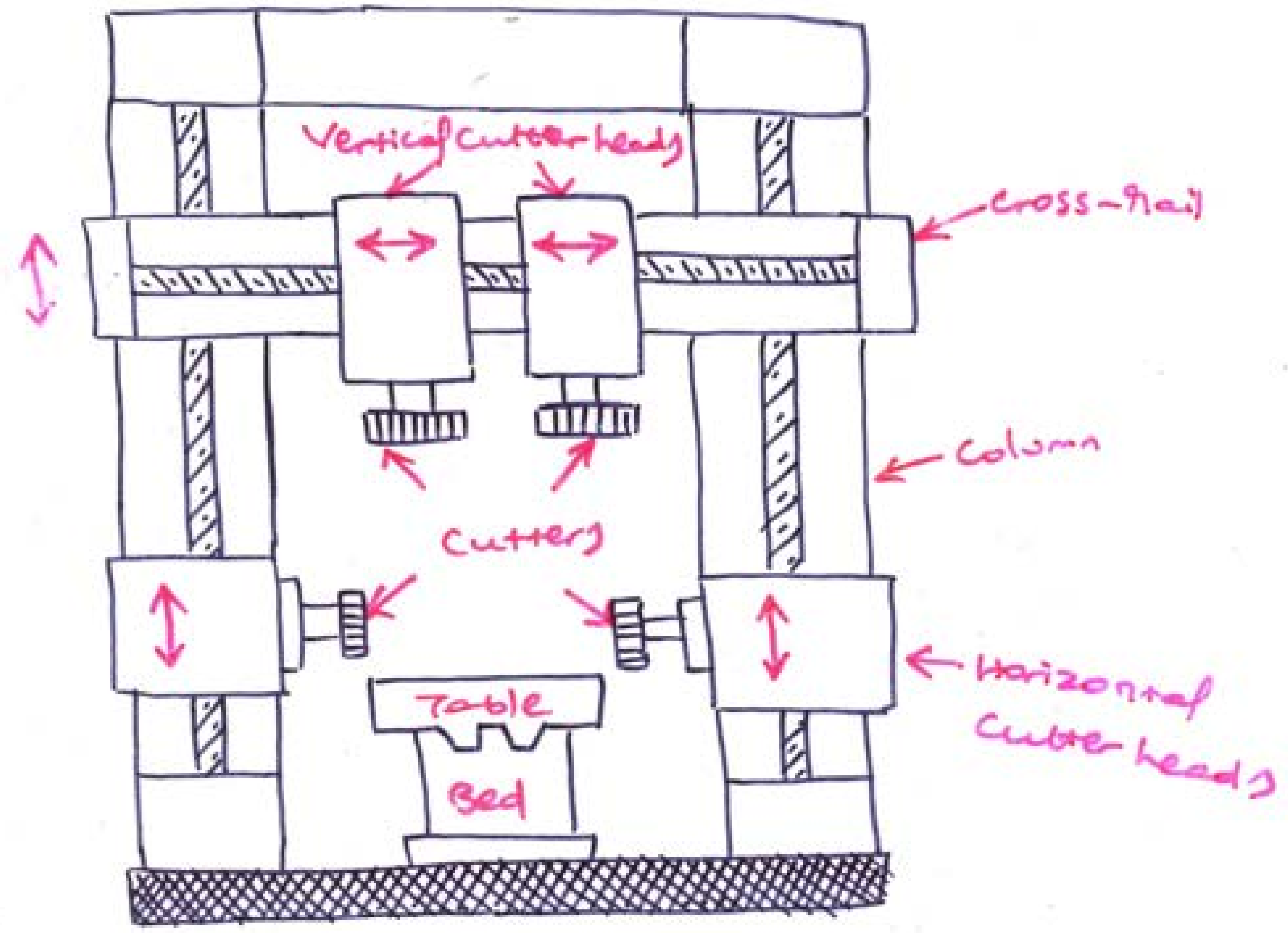
2. Duplex head fixed bed type milling m/c:-



3. Triplex head fixed bed type milling machine:-

three spindles - Two Horizontal
one vertical.

Plano-type milling machine:-



The w/p can be machined in 4 different ways.

1. By moving the table & rotating the cutters in position
2. By keeping the table stationary & feeding the cutters
3. By moving the table & milling head simultaneously
4. By keeping the table stationary and moving the cross-rail downwards & also the side cutters up & down.

Work Holding Devices:-

⑦

(1) T-bolt & Clamps (2) Angle plates (3) V-blocks
(4) Machine vice (5) Dividing head (6) Universal vice
(7) Fixtures.

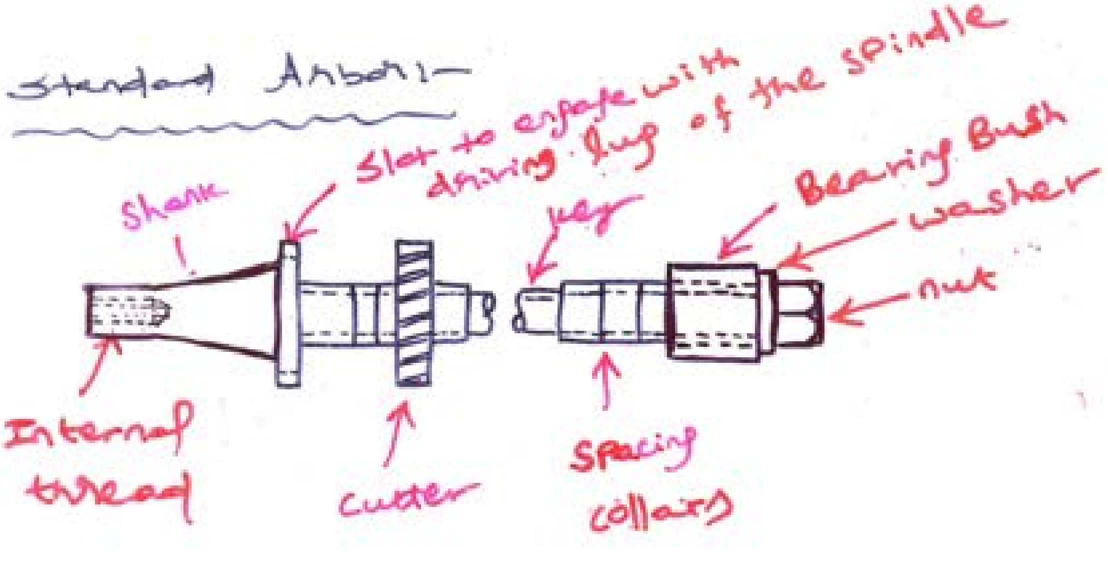
Tool-holding Devices:-

(1) Arbors (2) Adaptors (3) Collets

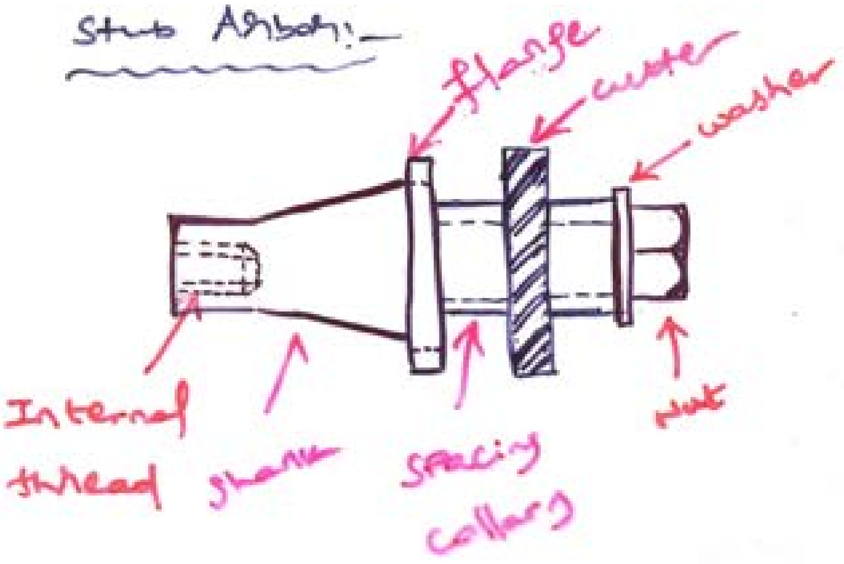
Arbors:-

(i) Standard Arbor (ii) Stub Arbor

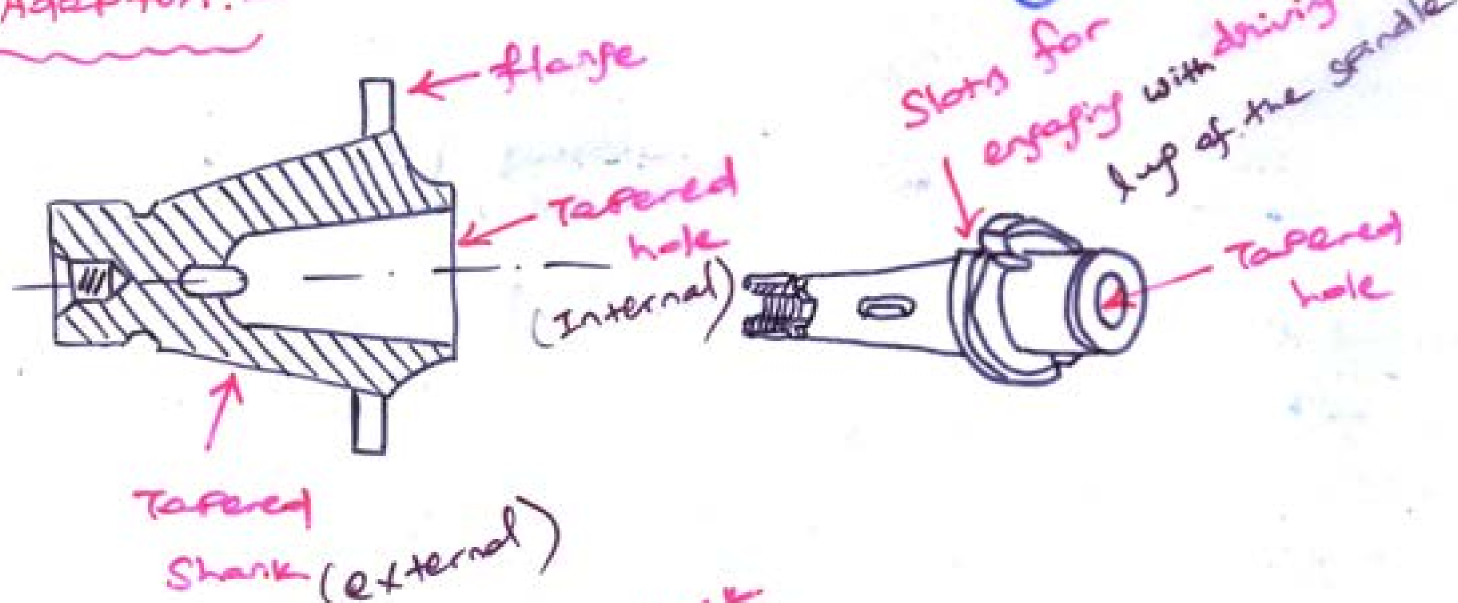
Standard Arbor:-



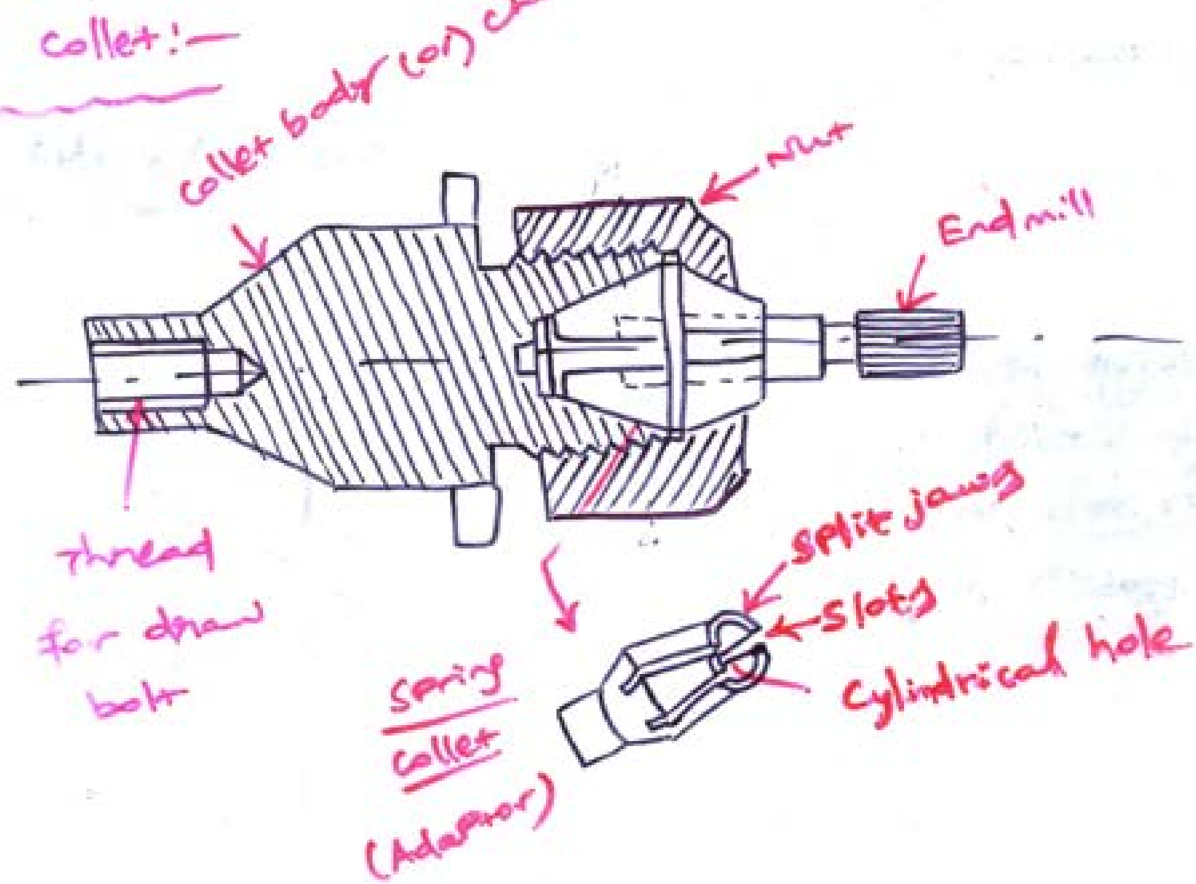
Stub Arbor:-



Adaptor:-



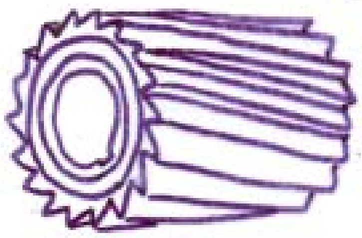
Spring collet:-



1. Collet chuck –
   1. tapered shank to fit into the tapered hole of the spindle nose.
   2. Internal threaded hole on back side
   3. Internal tapered hole on its front
   4. External threaded portion for nut.

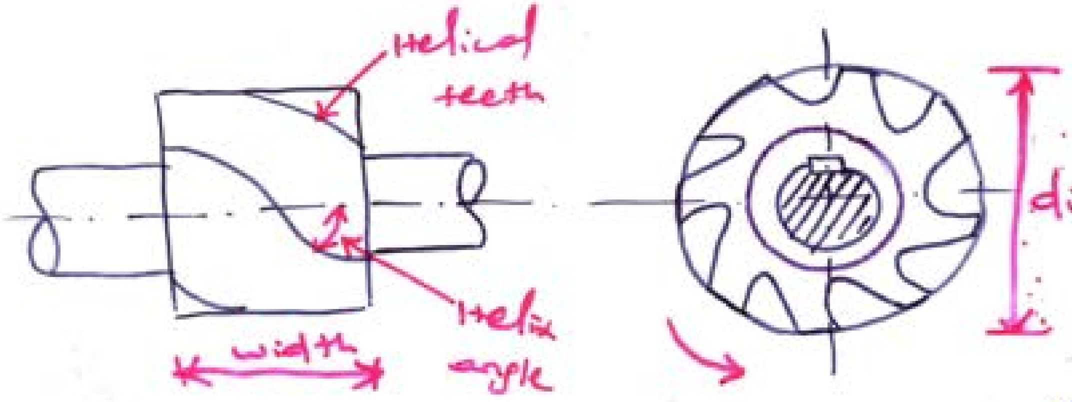
Milling cutters:-

(a)



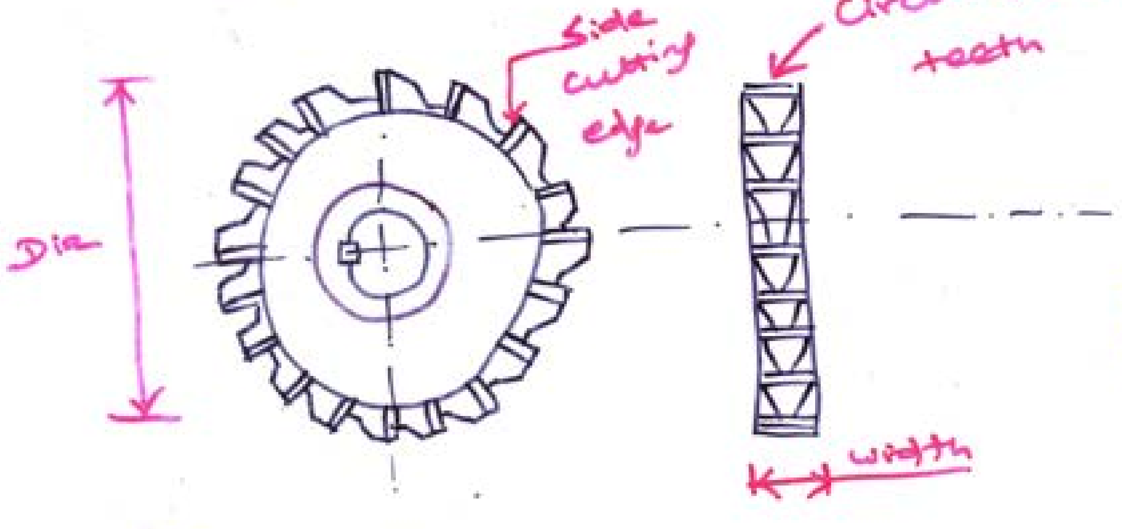
Straight teeth plain milling cutter

① Plain milling cutter:-



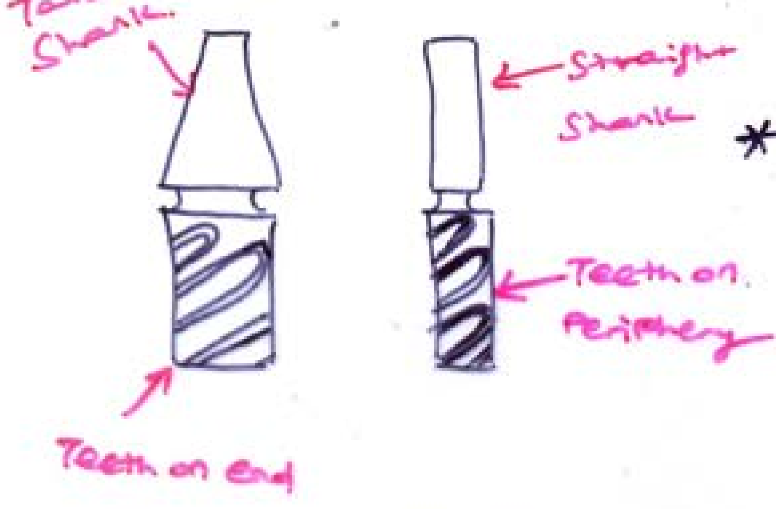
To produce flat surface ∥ to the axis of rotation of cutter

② Side milling cutter:-



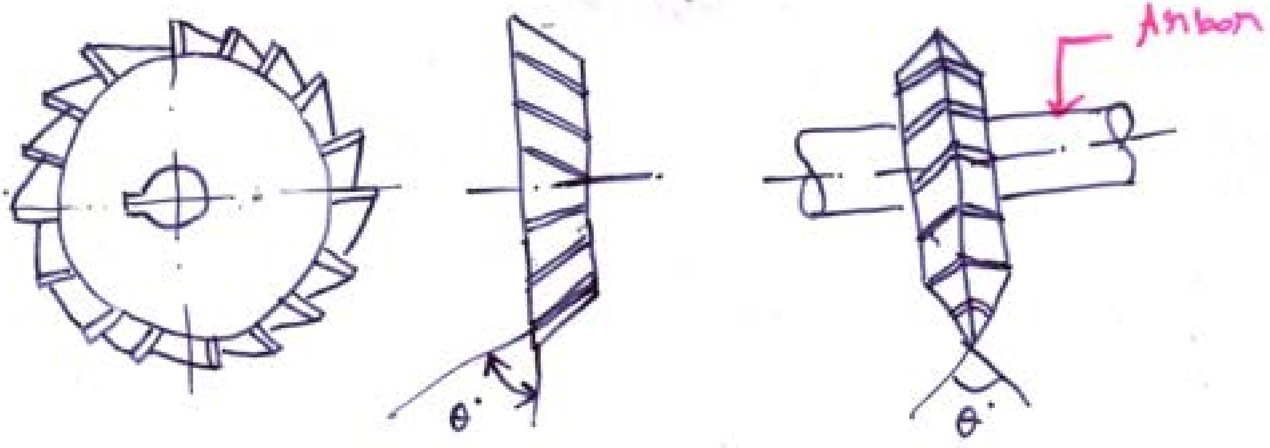
To produce flat surface on side of w/p & slots.

③ End milling cutters:-



* Periphery teeth may be Helical (or) straight
* Used to produce slots, profile milling & to produce accurate holes.

④ Angle Milling cutter:-

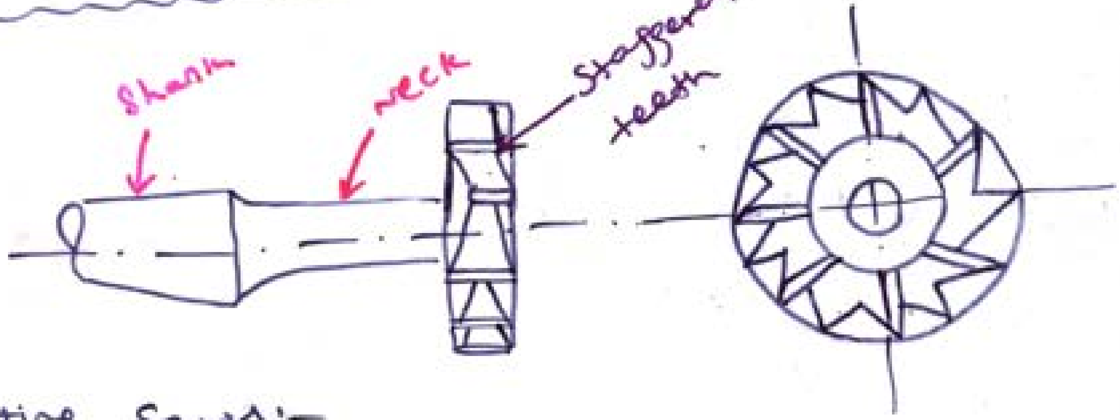


single angle milling cutter

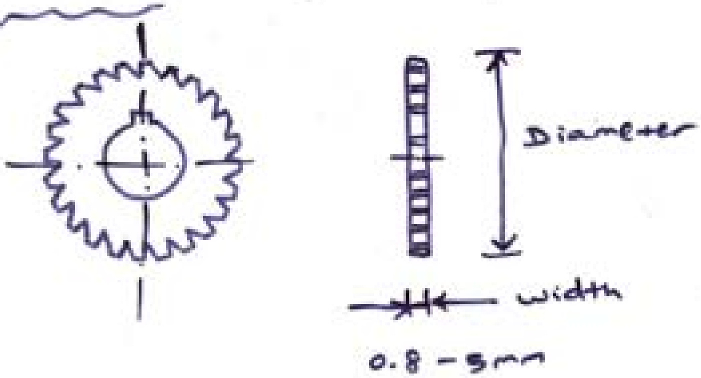
V-Grooves, Dove-tail Slots

Double Angle milling cutter

⑤ T-Slot milling cutter:-



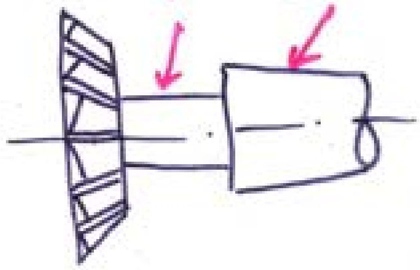
⑥ Slitting Saws:-



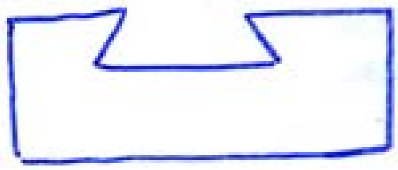
Deep Slots, Parting off the w/p.

Dove-tail Slot milling cutter:-

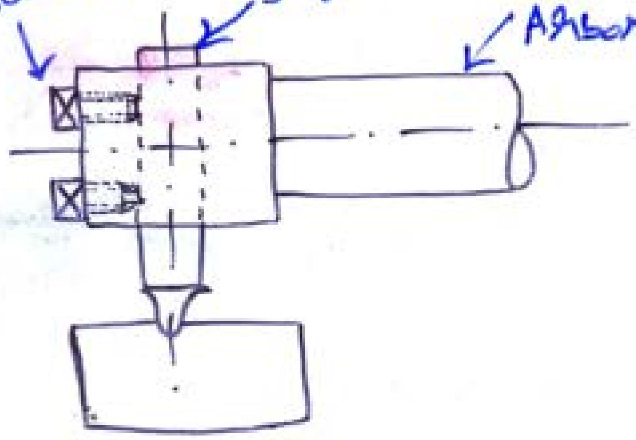
(11)



for making Dove-tail Slots



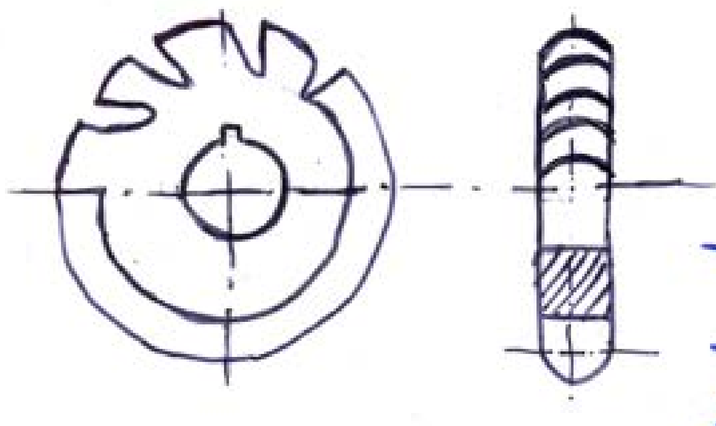
Fly-cutters:-



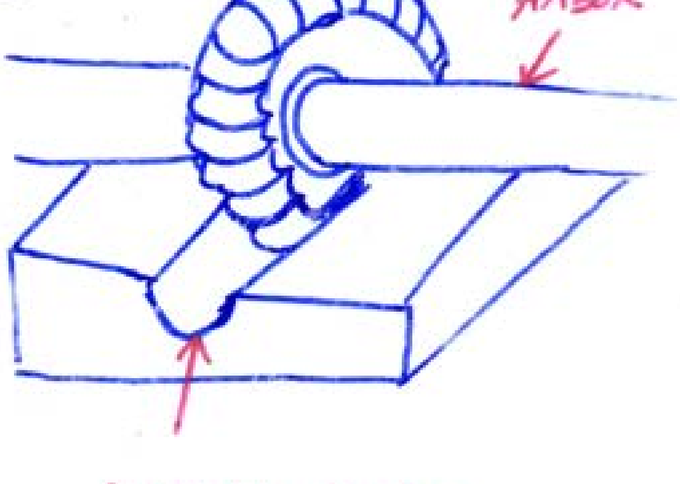
* The cutting edge may be formed to reproduce contoured Surface.
* considered as emergency tool when the Standard tool is not available.

Formed cutters:-

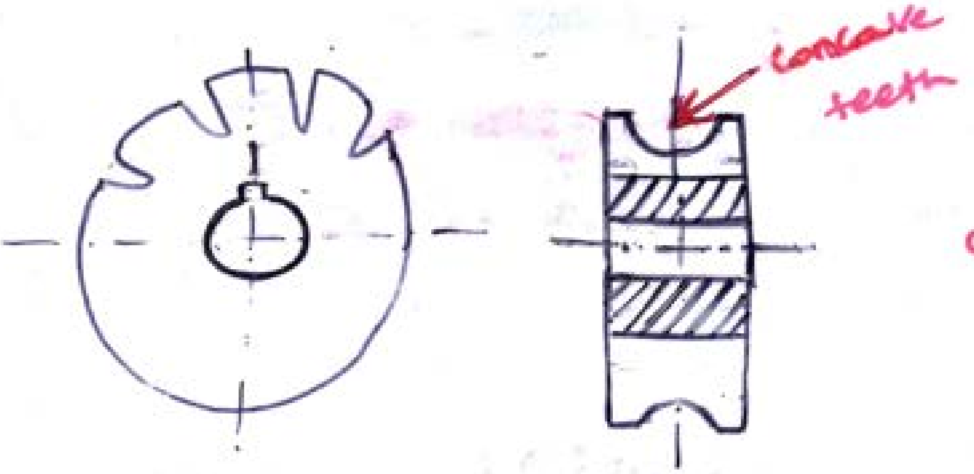
(1)

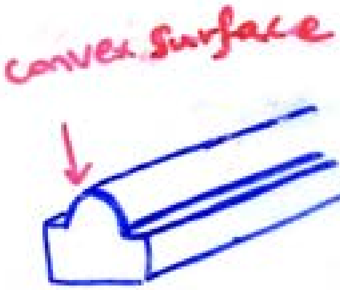


Convex milling Cutter

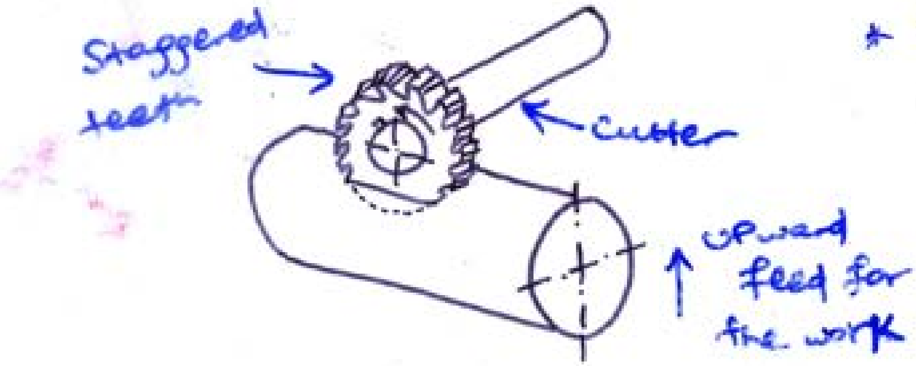


② concave cutter





Woodruff Key Slot milling cutter:—

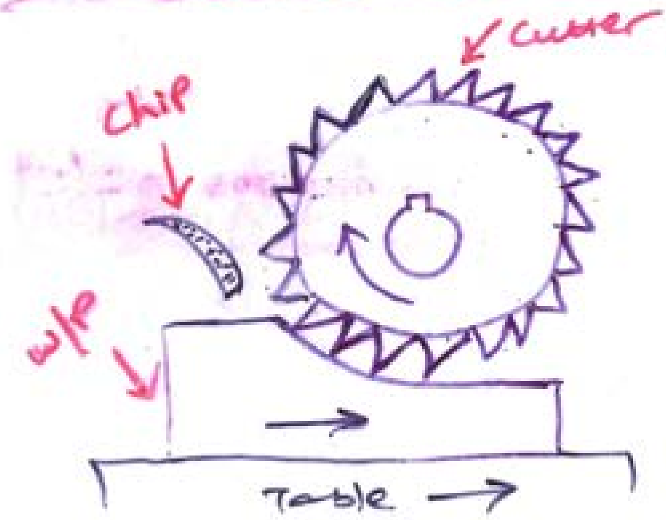


* It has Staggered (or) Straight teeth.
* Sides of the cutter ground convex
* used to cut woodruff key slot.

MILLING PROCESS

1. Peripheral milling – Milled surface $\parallel^{r}$ to the axis of rotation of cutter
    - (a) Up milling (or) Conventional milling
    - (b) Down milling (or) Climb milling
2. Face milling – Milled surface $\perp^{r}$ to the axis of rotation of cutter

Up milling (or) Conventional milling:–



* Direction of rotation
* Chip thickness
* cutting force direction
* preferred over climb milling in the case of backlash.
* chip welding & heat dissipation

Advantage:–

* Backlash eliminator is not required
* Safer operation

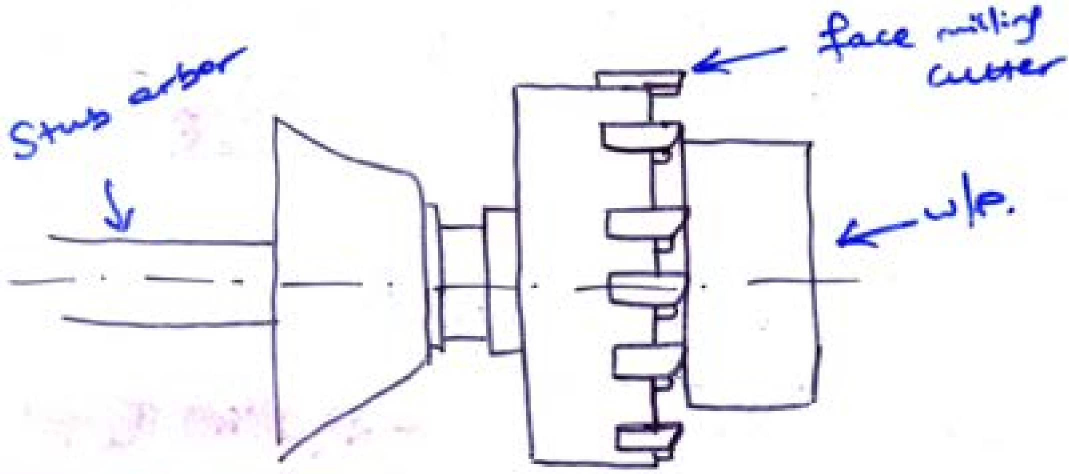
Down (or) Climb milling:–

Advantage:–

* cutter with high rake angle.
* cutter wear is less

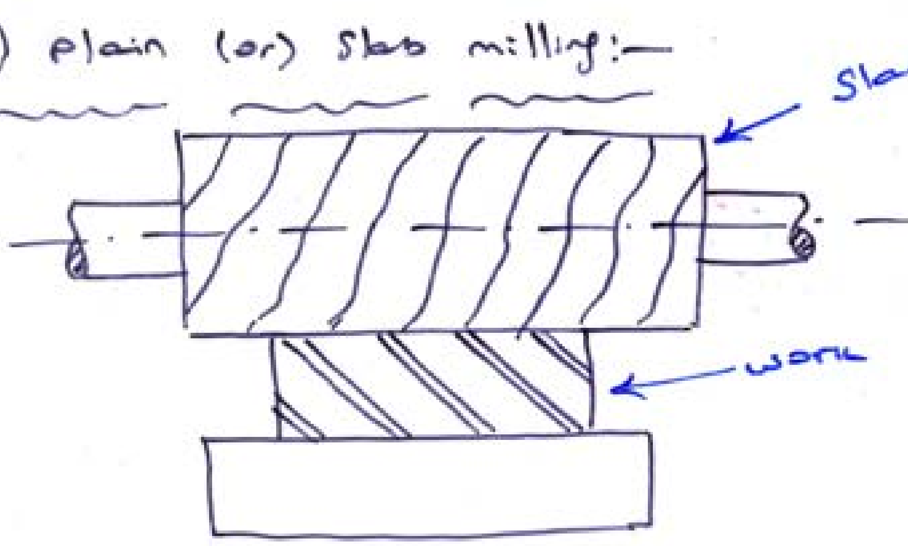
Face milling:-

* Machined surface ⊥r to the axis of rotation of cutter.
* Generally face milling cutter used for finishing.



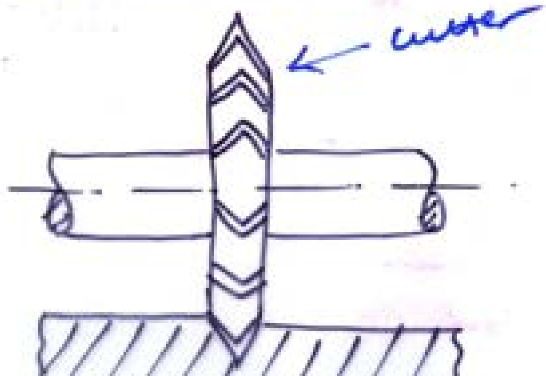
Milling operations:-

① plain (or) slab milling:-



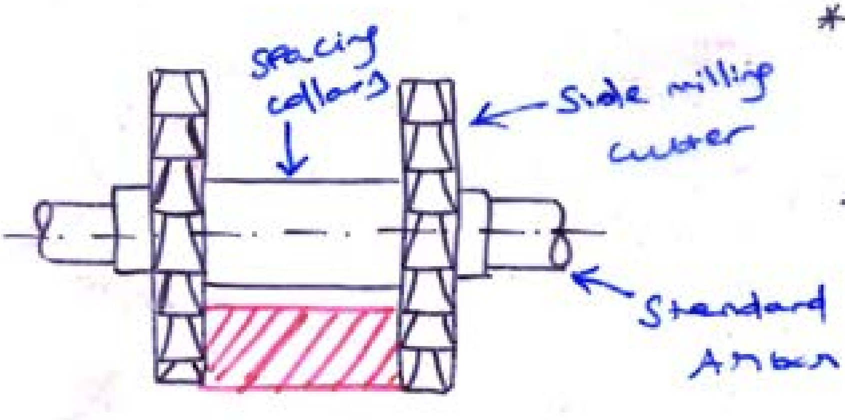
Producing flat horizontal surface ||el to the axis of the cutter.

② Angular (or) Bevel milling:-

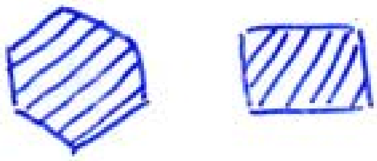


operation of machining a flat surface at an angle, other than right angle to the axis of the revolving cutter.

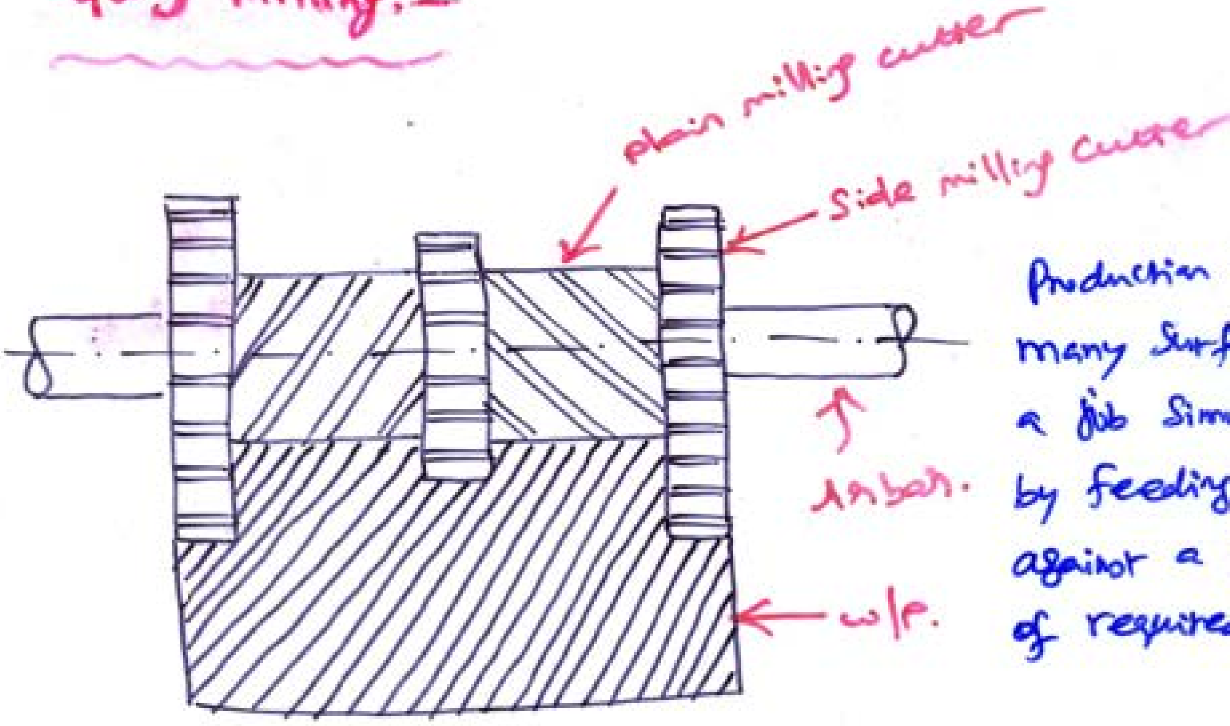
Straddle Milling:-



* Production of two Vertical flat surfaces on both sides.
* commonly used for milling square and hexagonal Surfaces.

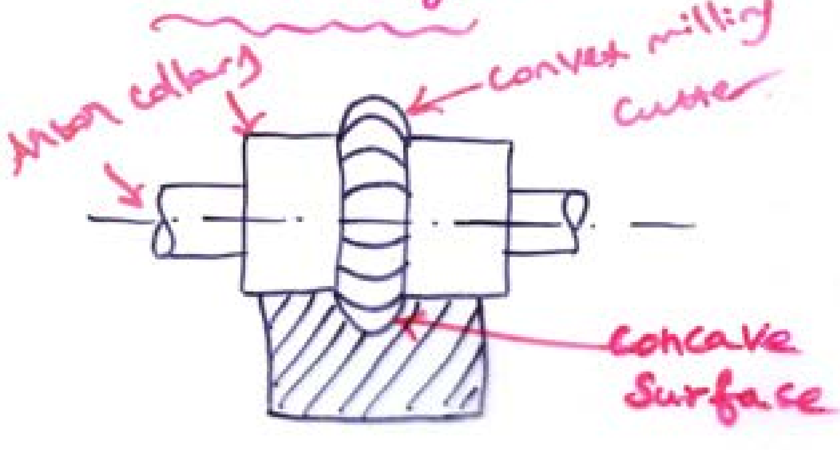


Gang Milling:-



Production of many Surfaces of a Job Simultaneously by feeding the table against a number of required Cutters

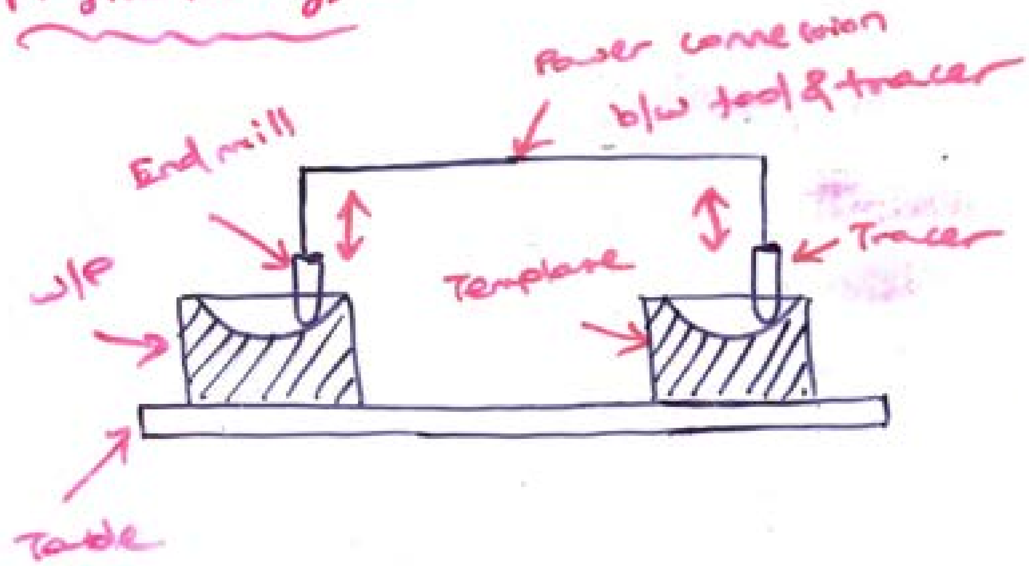
Form milling:-



* Production of Irregular Surface by using form Cutter.

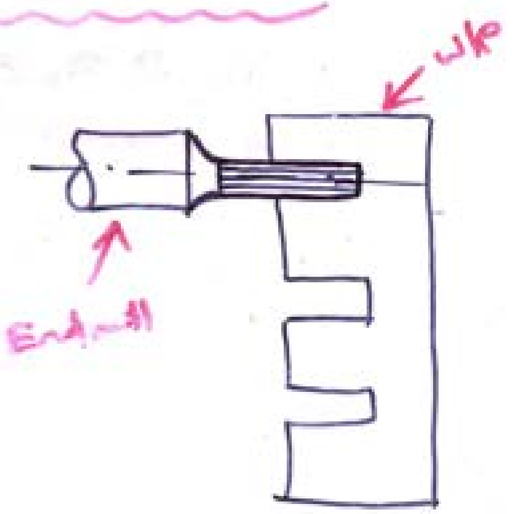
Profile milling:-

(16)



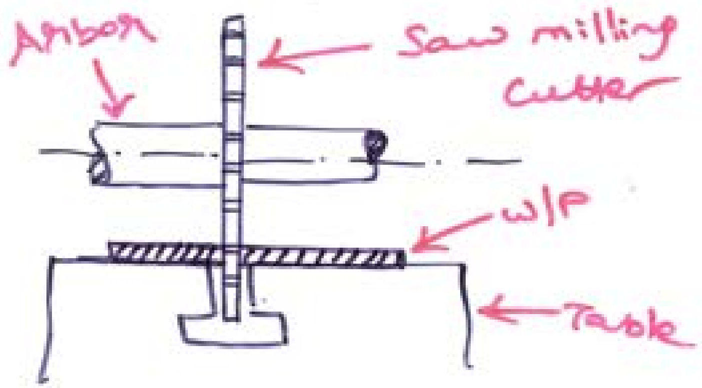
* Reproduction of outline of Template (or) Complex Shape of master die on the workpiece.

End milling:-



* used to produce flat Surface which may be vertical, horizontal (or) inclined to the table Surface
* Slots, key ways, Grooves etc

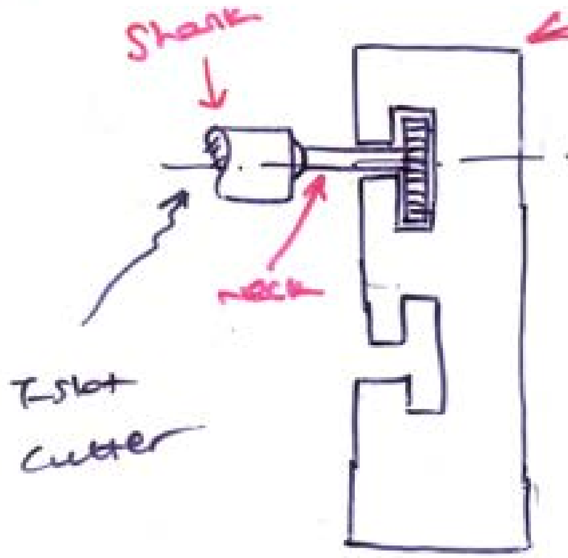
Saw milling:-



* Narrow Slots & narrow Grooves on a w/p.
* Also used for parting off the workpiece

T-slot milling:-

(17)

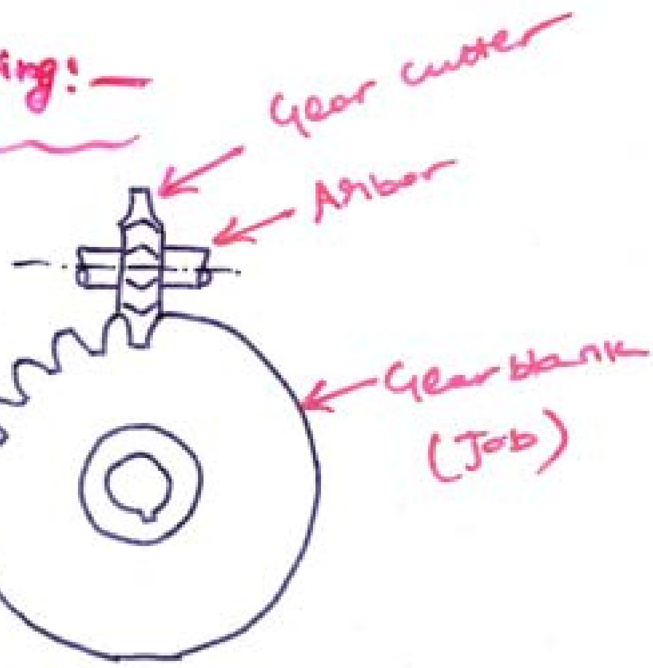


* used to produce T-slots on a w/p using T-slot cutter.

First stage :- Plain slot is made by using end milling cutter.

Second stage :- T-slot is made by using T-slot cutter.

Gear cutting:-



Gear Cutting

Machining Process:-

① Formed cutter method

ⓐ by formed disc cutter in Horizontal milling m/c

ⓑ by formed End mill in Vertical Milling m/c

ⓒ by single point form tool in Shaper (or) planer

ⓓ by Shear speed Shaping process

ⓔ by Gear broaching

ⓕ by template method.

② Gear Generation

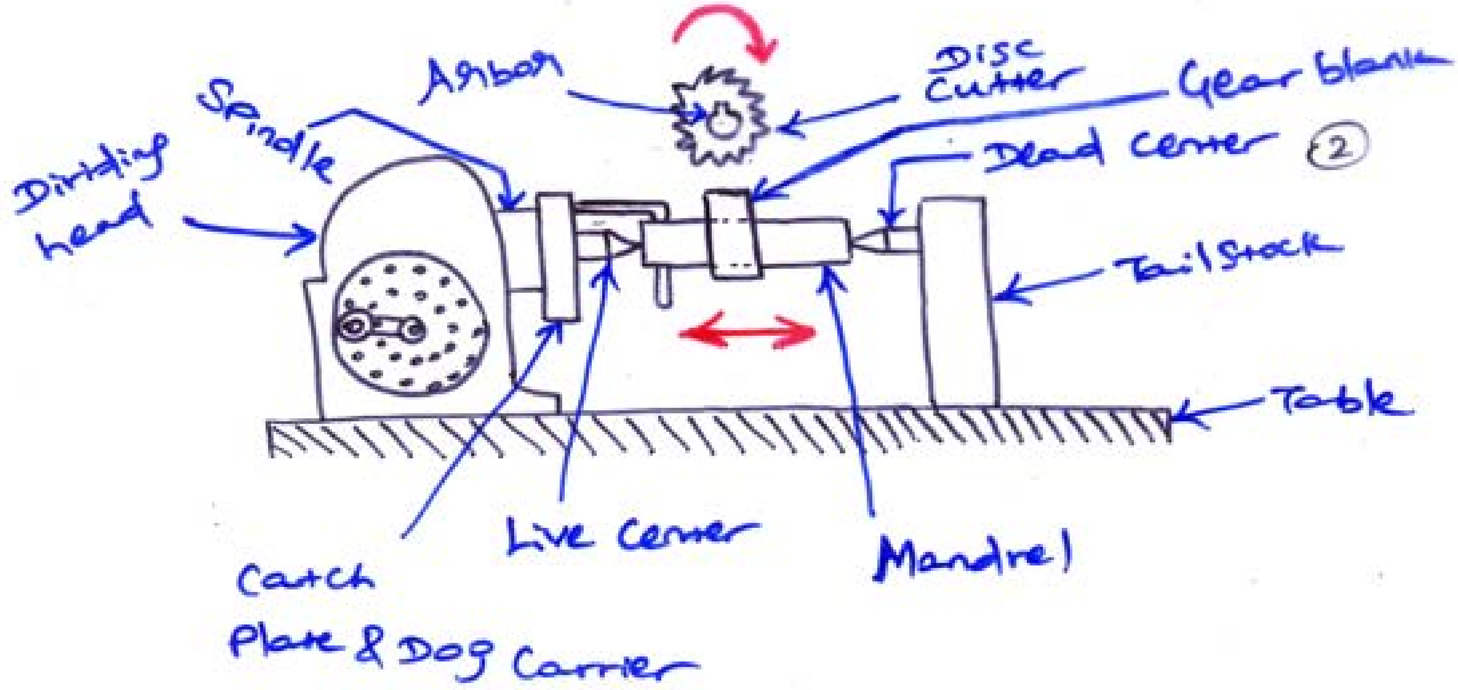
ⓐ Gear Shaping process

ⓑ Gear planing process

ⓒ Gear Hobbing process.

① Gear cutting by formed disc Cutter in Horizontal milling machine)-

① To form a spur Gear, column & knee type Horizontal milling machine is used.



② Universal Dividing head & TailStock is fit in the correct position on the table.

③ the Dividing Head Spindle has Inner bore & outer threaded portion for holding the Live center and catch plate respectively.

④ the Gear blank is fit into the mandrel which is supported in between Live & Dead center with the help of Catch plate & Dog Carrier.

⑤ Now, the formed disc Cutter of required profile is taken & is fit into arbor of the milling machine.

⑥ The cutter teeth is made to coincide with the Spindle axis by Various movement available in the milling machine

3

7. Based on the number of divisions required on the work piece, Indexing method is selected & then Indexing calculation is done so that required amount of crank movement along the hole circle of Index plate after each tooth is formed can be found out.

8. Now, the depth of cut is given by elevating the table & the workpiece is fed against the rotating cutter.

9. After one tooth is formed, it will be indexed to the next position by rotating the Indexing crank upto the required number of holes in the selected hole circle of the Indexing plate.

10. In this way, spur gear is formed in the column & knee type Horizontal milling machine.

Spur gear calculation:-

(a) Blank diameter $= m(z+2)$
(b) Tooth Depth $= 2.25m$
(c) Cutter pitch $= 3m$.

Indexing:-

$$\left.\begin{array}{l}\text{no of holes}\\ \text{to be moved}\\ \text{by crank}\end{array}\right\} = \frac{40}{Z}$$

Dividing Head & Indexing:— (4)

- Indexing is the process of dividing the periphery of a job into equal number of divisions.

- For doing this indexing, An attachment called dividing Head is used with the milling m/c. There are 3 types of dividing Head

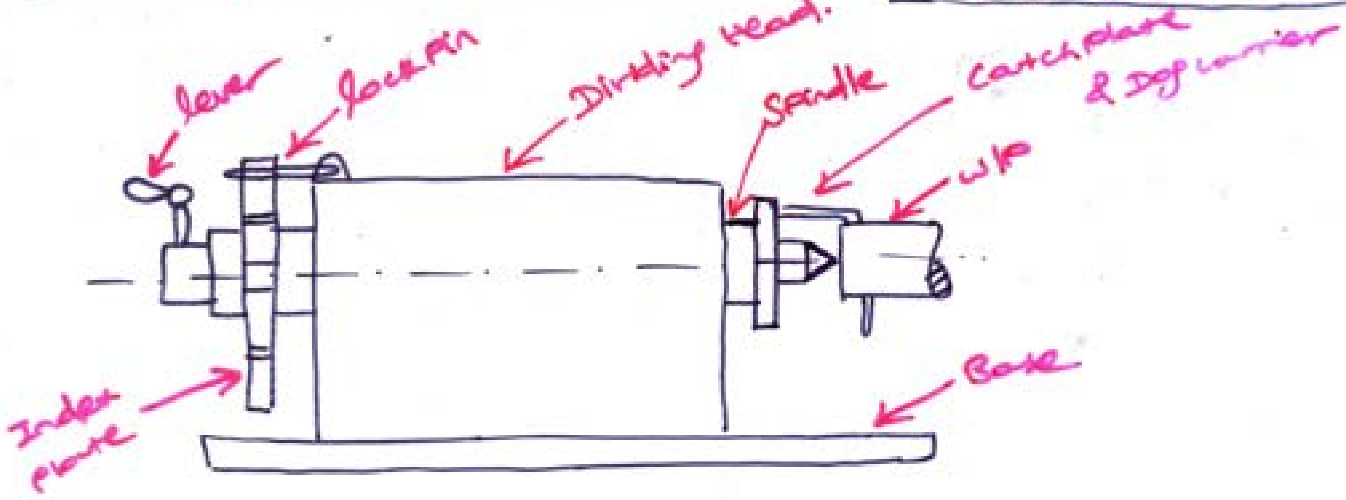
① Plain (or) Simple dividing Head

② Universal dividing Head

③ optical dividing Head.

Plain (or) Simple dividing Head:—

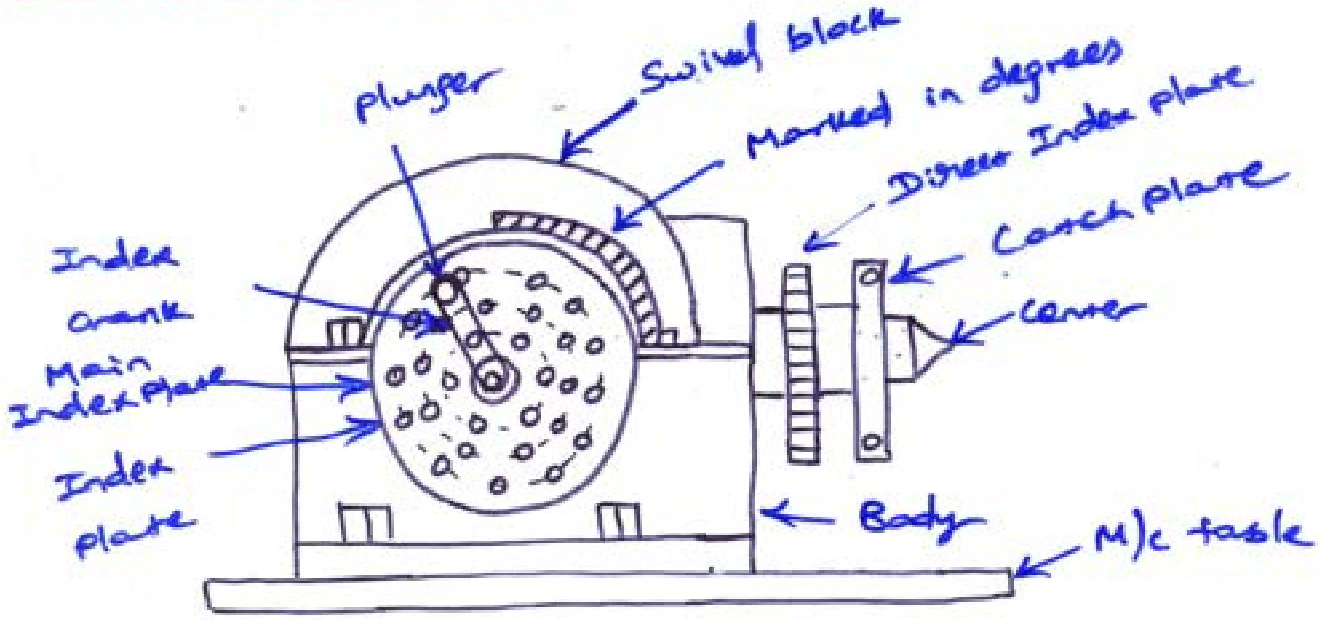
24 (or) 12 slots
2, 3, 4, 6, 12 & 24
2, 3, 4, 6, 12

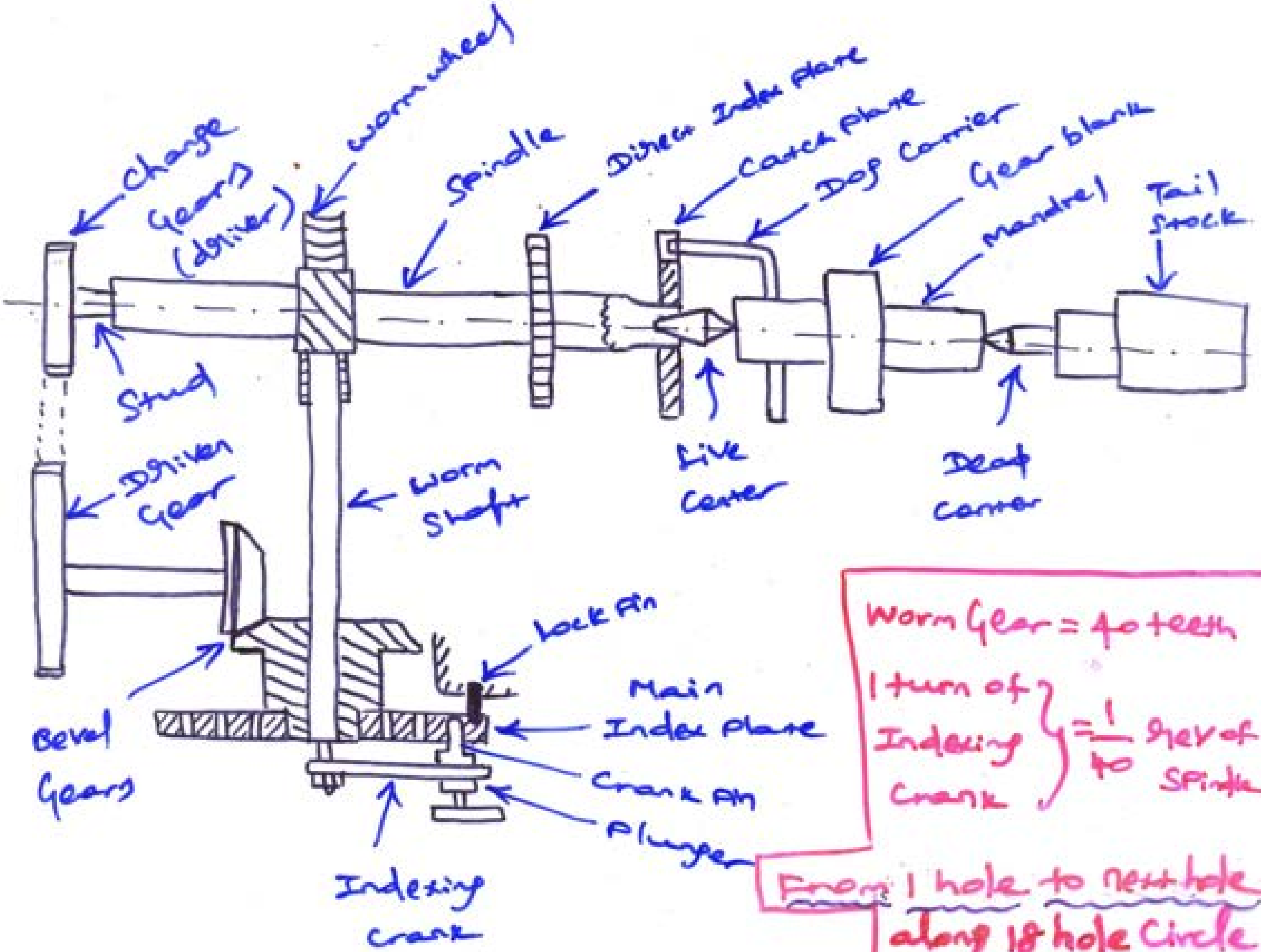


* Simplest of all dividing heads

* used for making small number of divisions on the periphery of the w/p [upto 24]

Universal Dividing Head:-





Worm Gear = 40 teeth

1 turn of Indexing Crank $= \frac{1}{40}$ Rev of Spindle

From 1 hole to next hole along 18 hole Circle

$\frac{1}{40} \times \frac{1}{18} = \frac{1}{720}$ Rev of Spindle

Uses:-

(1) To set the w/p in defined position [Horizontal, Vertical (or) Inclined] in relation to the m/c table

5° below Horizontal & 10° beyond Vertical

(2) To turn a workpiece periodically through a given angle (Indexing)

(3) To provide continuous rotary movement to the job during milling of Helical gears

(4) To hold the workpiece & support it in conjuction with the tailstock

Indexing Methods:-

(1) Direct (or) Rapid Indexing
(2) plain (or) Simple Indexing
(3) Differential Indexing
(4) Compound Indexing
(5) Angular Indexing.

(1) Direct (or) Rapid Indexing:-

* obtained by Plain (or) Universal dividing head.
* In universal dividing head, it is obtained by rotating the spindle by hand after disengaging the worm wheel & worm.
* Direct Index plate – 24 holes

factors of 24 $\longrightarrow$ (2, 3, 4, 6, 8, 12, 24)

Rules for Direct (or) Rapid Indexing:-

$$\text{No of holes to be moved} = \frac{24}{N}$$

N = No of divisions required.

example:-1 Find out Index crank movement to mill a Hexagonal bolt by direct Indexing.

$$\text{No of holes to be moved} = \frac{24}{6} = 4$$

So, After machining one side, the Index plate will have to be moved by 4 holes for 6 number of times to machining the remaining faces of the bolt.

Simple (or) Plain Indexing:-

* Suitable for divisions beyond the range of Rapid Indexing

Formula for Simple (or) Plain Indexing

$$\text{No of holes to be moved} = \frac{40}{N}$$ N - No of divisions required

B&S Index Plate:

Plate no:1 - 15, 16, 17, 18, 19, 20

Plate no:2 - 21, 23, 27, 29, 31, 33

Plate no: 3 - 37, 39, 41, 43, 47, 49

[Brown (B) & Sharp (S) M/g Company]

example:-2 Set the dividing head to mill 30 teeth on a spur wheel blank by simple Indexing.

Index crank movement $= \frac{40}{30} = 1\frac{1}{3} = 1\frac{1}{3} \times \frac{7}{7}$ (8)

$= 1\frac{7}{21}$

Thus for Indexing, one complete turn and 7 holes in 21 hole circle of Index plate will have to be moved by the Index crank

Compound Indexing!—

* Called compound Indexing due to the two separate movements of the Index crank in two different hole circle of one Index plate

* First movement — is as plain Indexing

  Second movement — ① Crank pin engaged with Index plate
  ② Lock pin at rear side of the plate is disengaged.
  ③ Then Index plate with the crank pin is rotated either forward (or) backward along the selected hole circle to the required amount

* The effective movement will be the summation of the two movement.

example for compound Indexing

(1) first movement :— 3 holes on a 15 holes circle by crank pin

Second movement :— 4 holes on a 12 hole circle forward. by Index plate

So, effective movement by worm $\left.\right\}$ $\frac{3}{15} + \frac{4}{12} = \frac{8}{15}$ revolution

Since 40 turn of worm rotates the spindle by 1 rev,

$\frac{8}{15}$ rev of worm rotes the spindle by $\frac{8}{15} \times \frac{1}{40} = \frac{1}{75}$ rev

$\therefore$ 75 divisions can be produced on workpiece

(2) first movement :— 6 holes on 18 hole circle by crank

Second movement :— 4 holes on 16 hole circle backward by Index plate

So, effective movement by worm $\left.\right\}$ $\frac{6}{18} - \frac{4}{16} = \frac{1}{12}$ rev

$\therefore$ Spindle movement $= \frac{1}{12} \times \frac{1}{40} = \frac{1}{480}$ rev

$\therefore$ 480 divisions can be produced on the w/p.

Rules for Compound Indexing :—

$$\frac{40}{N} = \frac{n_1}{N_1} \pm \frac{n_2}{N_2}$$

where,

(10)

N = number of divisions required

N_1, N_2 = The hole circle used by the crank pin & lock pin respectively

n_1 = no of hole spaces moved by the crank pin in N_1 hole circle

n_2 = no of hole spaces moved by the lock pin in N_2 hole circle

Procedure for Indexing:-

For example, Take a indexing for 69 division by Compound Indexing.

① Resolve into factors the number of divisions required

$$\boxed{69 = 23 \times 3}$$

$$\begin{array}{r|l} 3 & 69 \\ \hline 23 & 23 \\ \hline & 1 \end{array}$$

② Choose at radom hole circle

$\boxed{23 \ \& \ 33}$

Plate no:2 ⇒ 21, 23, 27, 29, 31, 33

③ Subtract the hole number of one circle from other

$$\boxed{33 - 23 = 10}$$

④ Factor the difference

$10 = 2 \times 5$

$$\begin{array}{r|l} 2 & 10 \\ 5 & 5 \\ & 1 \end{array}$$

10.1

⑤ Factor the no of turns required by the worm to make one revolution of the spindle

$40 = 2 \times 2 \times 2 \times 5$

$$\begin{array}{r|l} 2 & 40 \\ 2 & 20 \\ 2 & 10 \\ 5 & 5 \\ & 1 \end{array}$$

⑥ Factor the selected hole circle

$33 = 3 \times 11$

$23 = 23 \times 1$

⑦ Place the factors for no of divisions required & the factors for difference of hole circles above the horizontal line and place the factors for no of turns required by worm for 1 rev of spindle & hole circles selected below the horizontal line.

$$\frac{\begin{array}{l} 69 = \cancel{23} \times \cancel{3} \\ 10 = \cancel{2} \times \cancel{5} \end{array}}{\begin{array}{l} 40 = \cancel{2} \times 2 \times 2 \times \cancel{5} \\ 23 = \cancel{23} \times 1 \\ 33 = \cancel{3} \times 11 \end{array}}$$

⑧ Cancel out the common terms If the all the factors above the horizontal line is cancelled, the chosen hole circles are correct. otherwise choose another Set of hole Circles. In this case, the choosen hole circles [23 & 33] are correct as all the factors above the horizontal line are cancelled. (10.2)

(9) The no of hole spaces to be moved by the pin along the hole circle is found out by multiplying all the remaining factors below the horizontal line

$$\boxed{2 \times 2 \times 1 \times 11 = 44}$$

(10)

$$\therefore \frac{40}{69} = \frac{44}{23} - \frac{44}{33} = 1\frac{21}{23} - 1\frac{11}{33}$$

$$\boxed{\frac{40}{69} = \frac{21}{23} - \frac{11}{33}}$$

therefore, After each tooth is cut

(1) First, move the crank of Index plate by 21 holes along 23 hole circle

(2) Second, move the lock pin on the back side of the index plate by 11 holes along 33 hole circle in a reverse direction.

(4) Differential Indexing :-

1. The movement of Index Crank Similar to the Simple Indexing
2. The Simultaneous movement of the Index plate, when the crank is turned.

Change gears :- 24, 28, 32, 40, 44, 48, 56, 64, 72, 86, 100.

with this Change gears & B&S Index plate, It is Possible to Index any number ~~from~~ upto 382.

Special gears :- 46, 47, 52, 58, 68, 70, 76, 84

with this gears, index = 383 to 1008

Example :- 4

Index 83 divisions.

First of all, Check it can be indexed by plain indexing (or) not.

$$\text{Index crank movement} = \frac{40}{N} = \frac{40}{83}$$

As 83 hole circle is not available So, Differential Indexing ~~should~~ can be used.

Rule for Differential Indexing :-

1. $$\text{Gear ratio} = \frac{\text{Driving gear on spindle}}{\text{Driven gears on bevel Gear shaft}} = \frac{(A-N)\times 40}{A}$$

where,

A = the selected number which can be indexed by plain Indexing & the number should be nearer to the no of divisions required.

N = Number of divisions to be indexed.

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

Then the index crank will have to be moved by an amount for N number of times for complete division of work.

If (A−N) is positive, the index plate must rotate in the same direction.

If (A−N) is negative, the index plate must rotate in a opposite direction to the crank

To achieve these conditions, the number of idle gears used depends on the following factors

(a) gear train is simple & (A−N) is positive, only one idle gear is used.

(b) gear train is compound & (A−N) is positive, no idle gear is used.

(c) gear train is simple & (A−N) is negative, two idle gears are used.

(d) gear train is compound and (A−N) is negative, only one idle gears are used.

Example: 4

Index for 83 divisions.

First of all, findout whether the number can be indexed by plain indexing (or) not.

$$\text{Index crank movement} = \frac{40}{N} = \frac{40}{83}$$

Since, there is no 83 hole circle, the number cannot be indexed by plain Indexing. Therefore it is the case of differential Indexing.

Plate no: 3 37, 39, 41, 43, 47, 49

Assume A as 86

① Gear ratio $= (A-N) \times \frac{40}{A} = (86-83) \times \frac{40}{86}$

$$= 3 \times \frac{40}{86} = \frac{72}{24} \times \frac{40}{86}$$

② Therefore, Drivers $= 72, 40$

Driven $= 24, 86.$

③ Index crank movement $= \frac{40}{86} = \frac{20}{43}$

For complete Indexing, the index crank will have to be moved by 20 holes in 43 hole circle for 83 times.

④ As $(A-N)$ is positive & the gear ratio is compound, no idle gear is required.

Angular Indexing:-

40 Rev. of the Crank = 1 Rev of the Spindle

= 360°

∴ 1 Rev of the Crank $= \frac{360}{40} = 9°$

Formula:-

Index crank movement $= \frac{\text{Angular Displacement of work}}{9}$

$= \frac{\text{Ang. Dis of work in minutes}}{540}$

$= \frac{\text{Ang displa of work in sec}}{32,400}$

Example:- 5

Index an angle 19° 40'

$19° 40' = (19 \times 60) + 40$

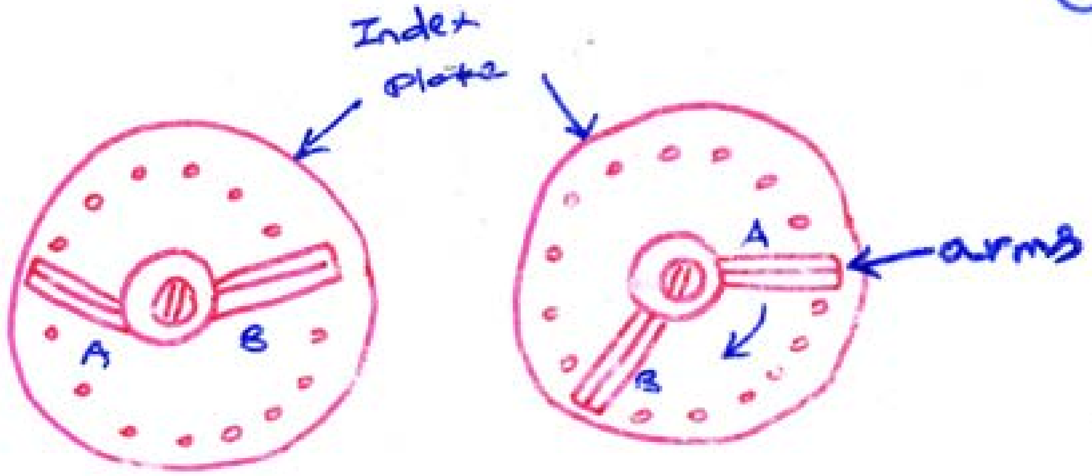
$= 1180'$

∴ Index Crank movement $= \frac{1180}{540} = 2\, 5/27$

The index crank Should be moved two Complete turns & 5 holes in 27 hole circle

Secton arms:-

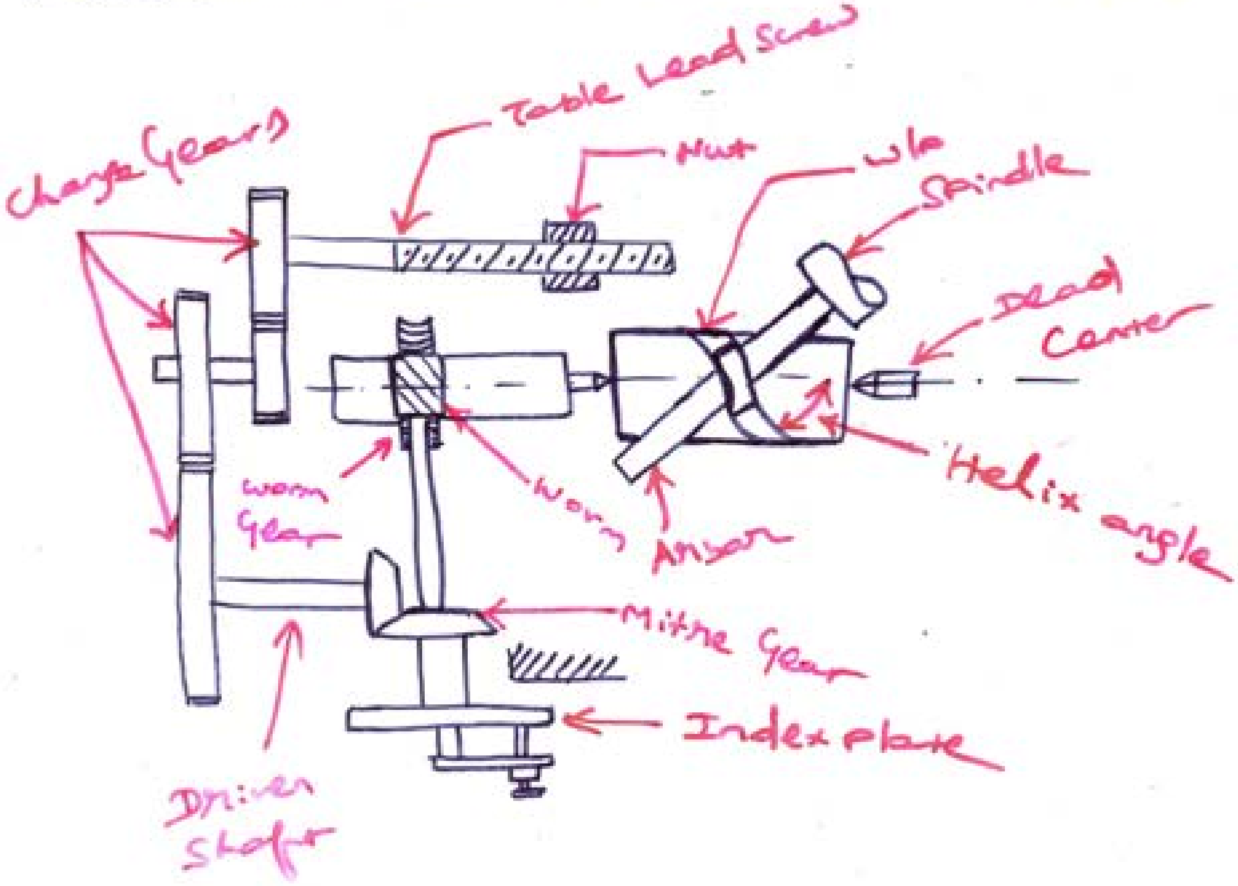
15



* Radial distance b/w the arms can be adjusted depending on the no of holes to be moved.

Helical Gear milling in universal milling machine:-

(16)



① The Gear blank is fit into the mandrel which is supported in between Live center & Dead Center of universal Dividing head & Tail Stock respectively.

② Cutter of required profile is taken & fit into the arbor of the milling m/c.

③ Then the workpiece axis is made to coincide with the cutter teeth by various movements available in the milling machine.

④ Then, the table is swivelled to the required Helix angle.

(5) * For forming a Helical teeth, the Gear blank shall be (17)
rotated to a distance of lead of the Helical gear when machining takes place. So, here the mitre Gear of the universal dividing head is connected to the table lead screw through a Gear train.

* So when the feed is given, continuous rotary motion is given to the workpiece from the table lead screw through a set of gear train in the universal Dividing head.

* As the gear train is connected to the table lead screw, differential Indexing cannot be done. only simple Indexing method is used for Indexing.

(6) Now, the table is fed against the rotating cutter to form a tooth on a Gear blank.

(7) After one tooth is formed, w/p is indexed to the next position by moving the indexing crank to the required number of holes in the selected hole circle and the next tooth is formed.

(8) In this way, Helical is formed by universal milling machine.

Helical Gear milling calculation:-

(18)

(a) Blank Diameter $= m\left(\frac{z}{\cos\beta} + 2\right)$

(b) Tooth depth $= 2.25m$

(c) Cutter pitch $= 6m$

(d) Index crank movement $= \frac{40}{N}$

(e) Lead of the Helical Gear, $T_2 = \frac{\pi D}{\tan\beta}$

(f) Gear ratio $= \frac{\text{Driver}}{\text{Driven}} = \frac{40 T_1}{T_2}$

(g) Cutter teeth, $z' = \frac{z}{\cos^2\beta}$

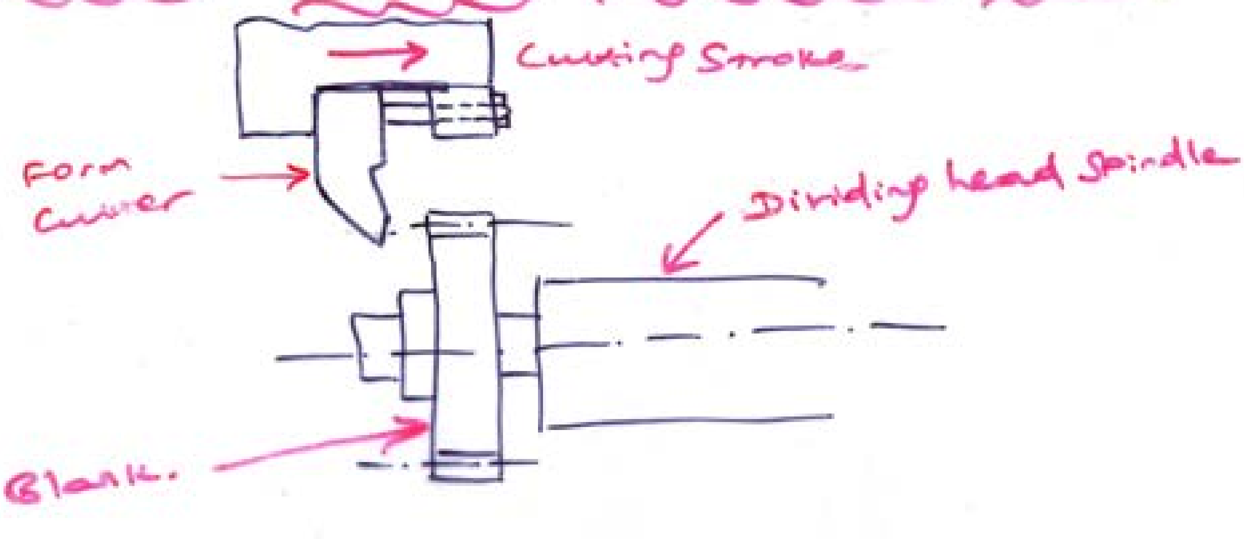
$\beta \rightarrow$ Helix angle
$m \rightarrow$ Module of Gear
$z \rightarrow$ No of teeth
$N \rightarrow$ No of divisions

$T_1 \rightarrow$ Pitch of lead screw
$T_2 \rightarrow$ lead of Helical Gear.

Cutter No	No of teeth to be cut
No. 1	from 135
No. 2	55 - 134
No. 3	35 - 54
No. 4	26 - 34
No. 5	21 - 25
No. 6	17 - 20
No. 7	14 - 16
No. 8	12 - 13

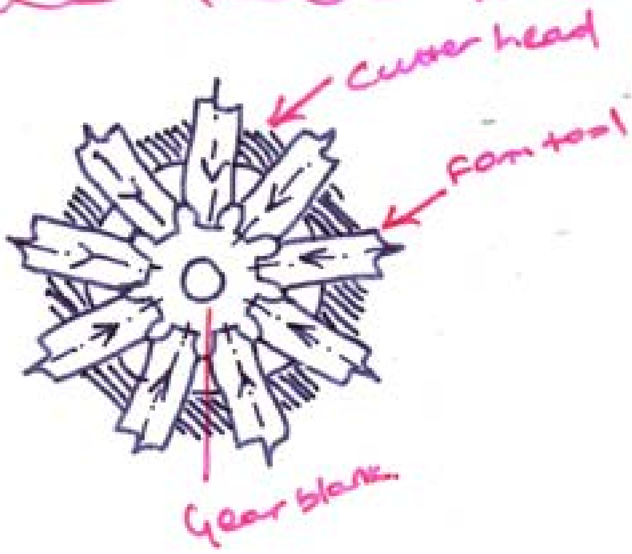
Based on z' value, the Cutter Should be chosen from the table

Gear cutting by Single Point formed Tool:-



Gear cutting by Shear Speed Process:-

(1a)

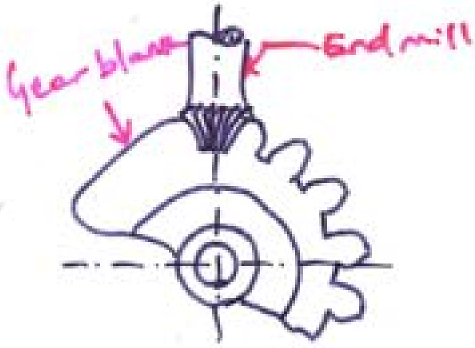


* All teeth are formed simultaneously by a ring of formed blades
* Work - Ram & Reciprocates
* Radial Adjustment for the cutter for
  1. relief during retraction
  2. Depth of cut.

uses:-

Spur Gears, Splined Shaft, Ratchet wheels etc.,

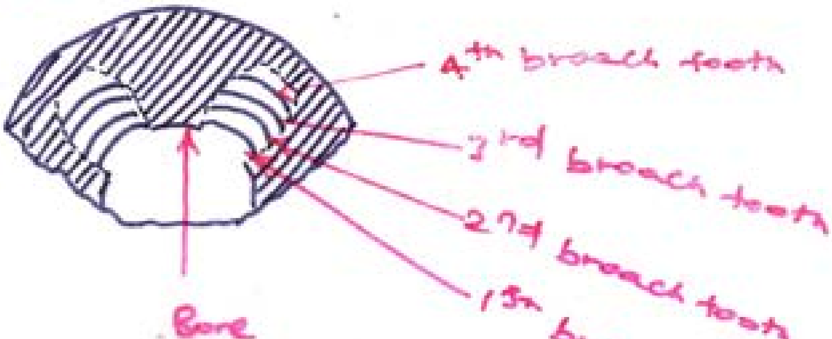
Gear milling using a formed End mill:-



* Done in VMC
* Simple Indexing is used

  Crank movement = $\frac{40}{Z}$
* Spur, helical, herring bone Gears.

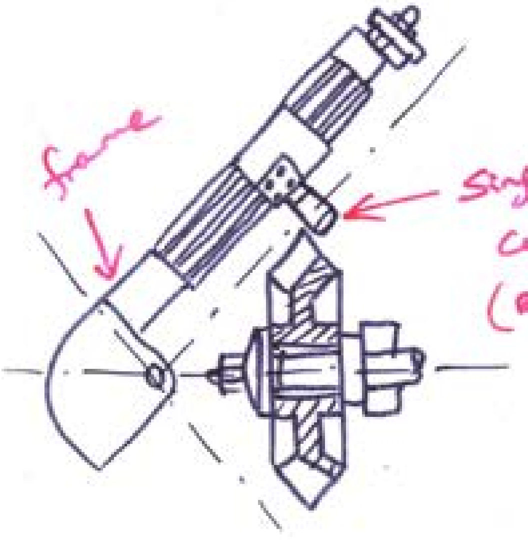
Gear broaching:-



Concentric circle denotes depth of cut per tooth.

Template Method!—

(20)



① It involves the production of a gear tooth profile by a single point cutting tool which is reciprocated & made to follow a guided path by a template whose profile corresponds to the shape of the gear tooth being cut.

Gear Generating method!—

① Gear shaping process
② Gear planning process
③ Gear Hobbing process

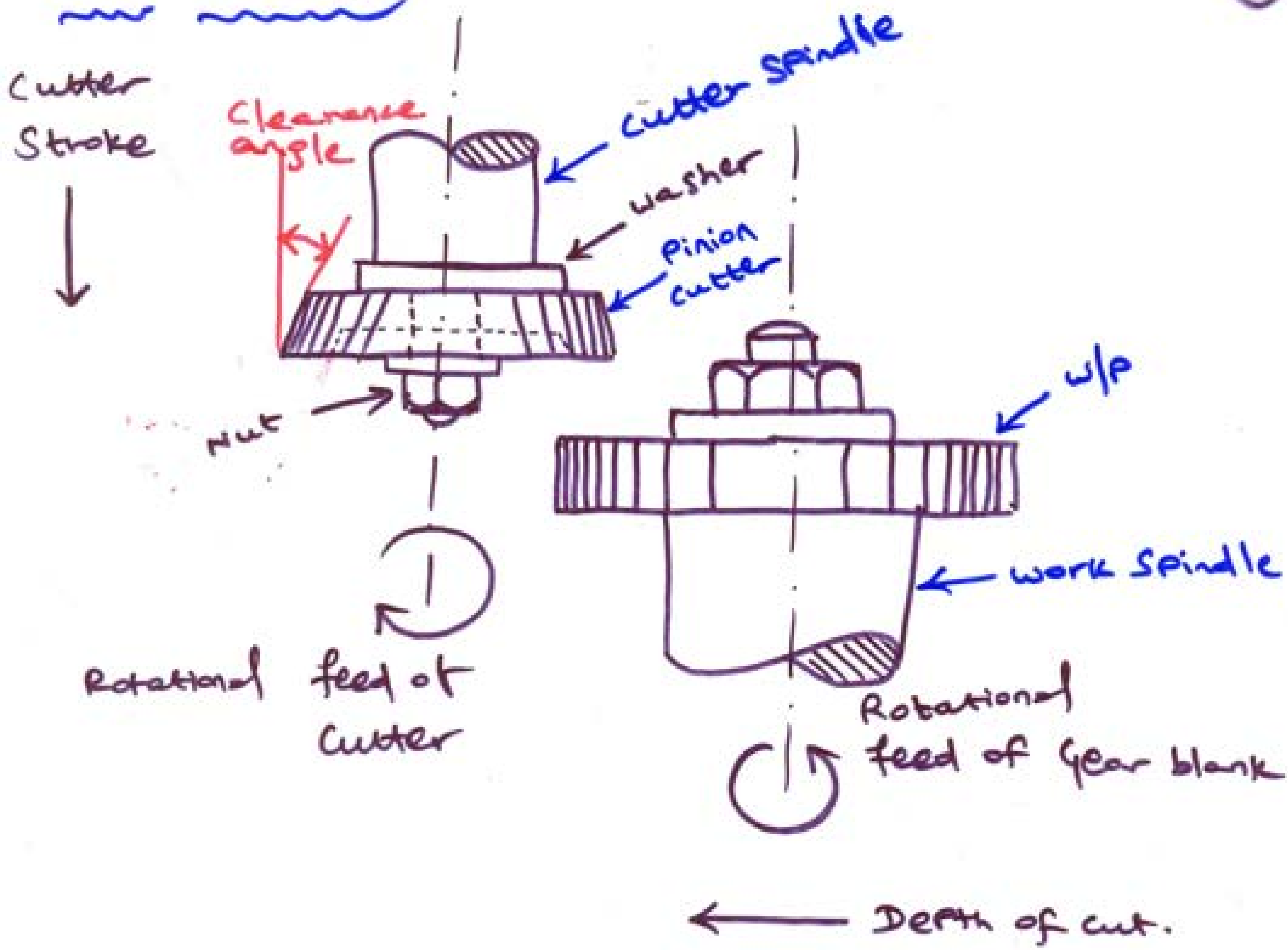
Principle:—

Gear Generation is based on the fact that any two involute Gears will mesh together.

* Here one Gear is made as a cutter & the other is made as a Gear blank.
* Cutter is made to reciprocate over the entire width of the Gear blank.
* Cutter is fed towards w/p (or) Vice-versa to give a depth of cut.

Due to the relative rolling motion b/w the cutter & blank, Gear teeth are generated on the Gear blank.

Gear shaping



* Gear shaping is one of Gear Generation method used for cutting cylindrical external Gears.

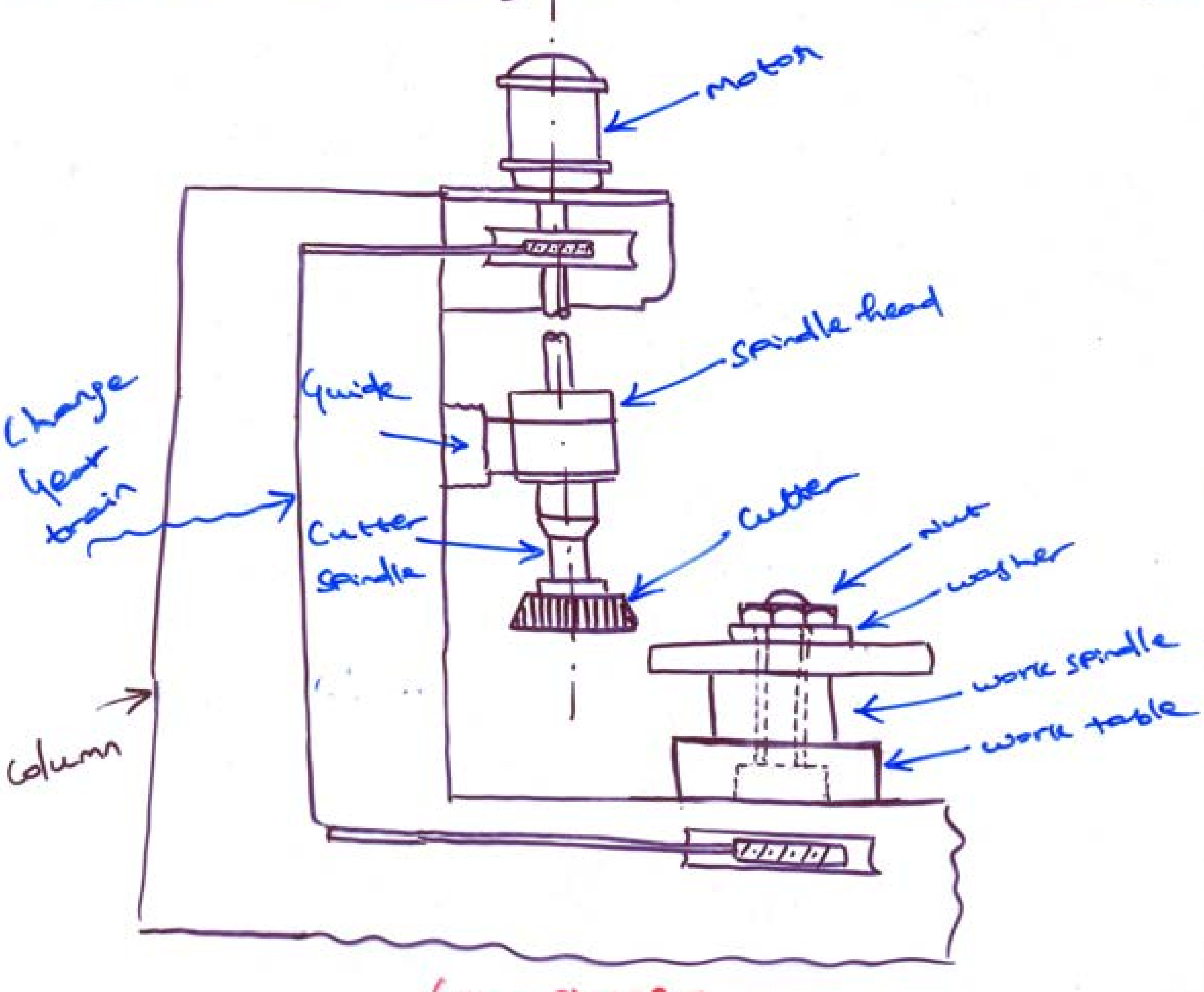
* Gear shaping is done on a special type of machine called Gear Shaper.

* Here, pinion type cutter with proper Rake & clearance angle is used.

* Here, the axis of rotation of cutter & Gear blank are parallel.

* Here the cutter & work spindle are separately connected with gear trains. It gives correct relative speed of rotation to these two spindles. So, the rotation of the cutter generates the tooth profile on the gear blank. The rolling movement is continued until all the teeth on the blank are cut.

* During return stroke, the work is relieved from the cutter by a suitable mechanism. It is done to avoid the rubbing of cutter over the cut surface



Gear shaper

The various movements available in the gear shaper are given below.

1. Rotational feed for cutter & workpiece
2. Radial feed of the cutter towards the blank for giving the depth of cut.
3. Reciprocating motion of the cutter along the width of the gear blank
4. Withdrawal motion of the blank away from the cutter during return stroke

Application:-

1. Gear shaping is used for generating both internal & external spur gear.
2. Helical gear can also be generated using special attachment.

Advantages:-

1. Both internal & external gears can be generated.
2. Various sizes of gears can be generated using a single cutter.

③ For the Same modules a single type cutter can be used irrespective of the number of teeth in the gear.

④ As the cutting action is continuous, the rate of production will be high.

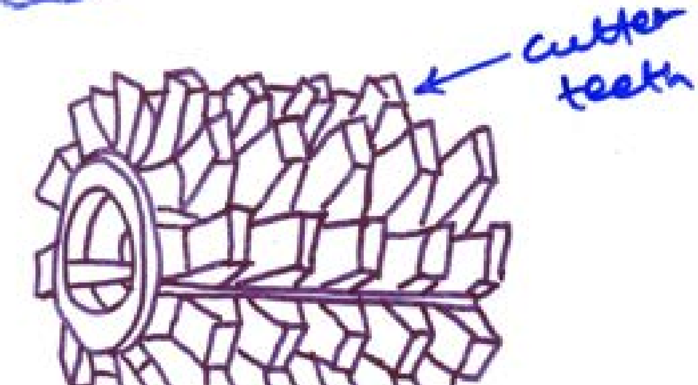
24

Limitations:-

① Worm Gear can not be produced.

② There is no cutting in the return stroke of the cutter. So, there is a need to make return stroke faster than the cutting stroke

Gear Hobbing:-



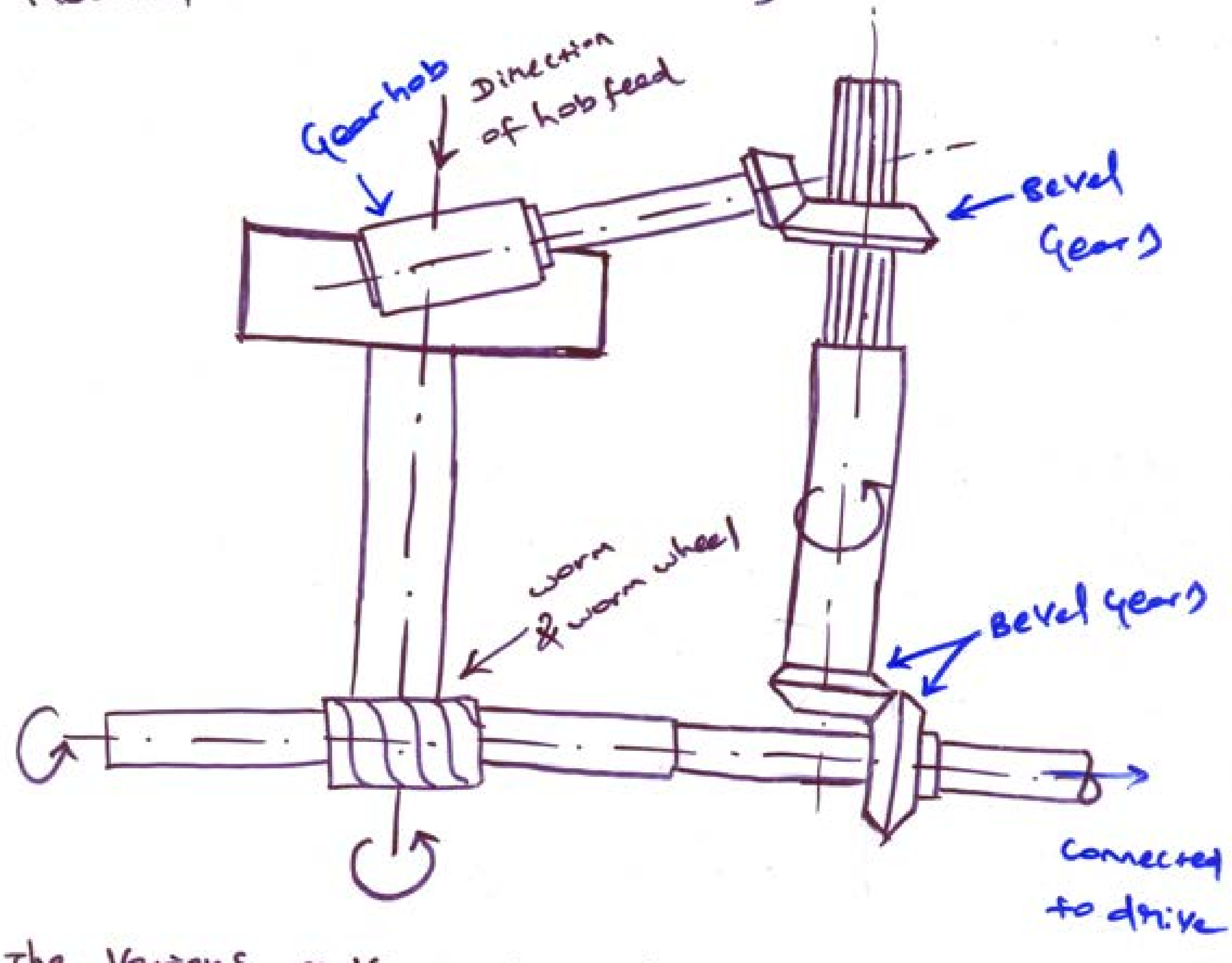
Hob

* the process of generating a gear by means of a multi point rotating cutter called "hob" is known as hobbing.
* this hob looks like a worm & this process looks like a meshing of worm & worm wheel.

* The hob may be either single threaded (or) multi threaded part.

(25)

* In gear hobbing opreation, the gear blank is mounted on a vertical arbor but the hob is mounted on a horizontal rotating arbor.



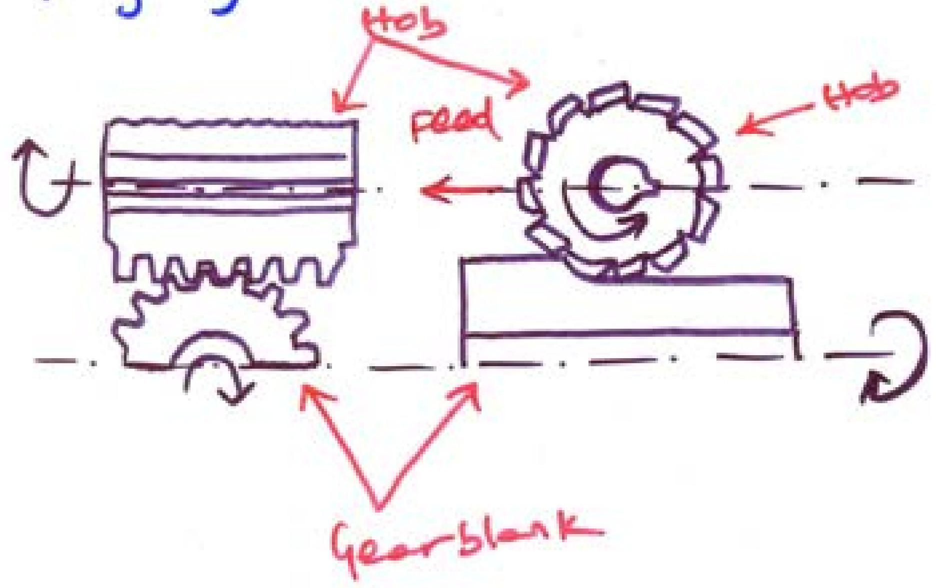
The various movements available in a Gear hobbing machine

1. Rotation of the hob [cutter]
2. Vertical feed of the hob over the width of the Gear blank

③ Rotational feed of the work piece

④ Radial feed of Gear blank towards the Cutter for giving the depth of cut.

(26)



* In gear hobbing, the axis of rotation of cutter (hob) & the work piece are perpendicular to each other.

* Here both Gear blank & hub are made to rotate like worm & worm Gear in mesh.

For cutting a helical Gear, the axis of hub is inclined to 'α'.

Lead angle, $\alpha = \theta + (90^\circ - \alpha_1)$ – Helix of the hub & Gear blank are different.

$\alpha = \theta - (90^\circ - \alpha_1)$ – Helix of hob & Gear blank are same

where,

$\theta \rightarrow$ Helix angle of the Gear to be formed.

$\alpha_1 \rightarrow$ Helix angle of the hub.

Applications:-

Hobbing is used for generating spur, helical & worm gears. (27)

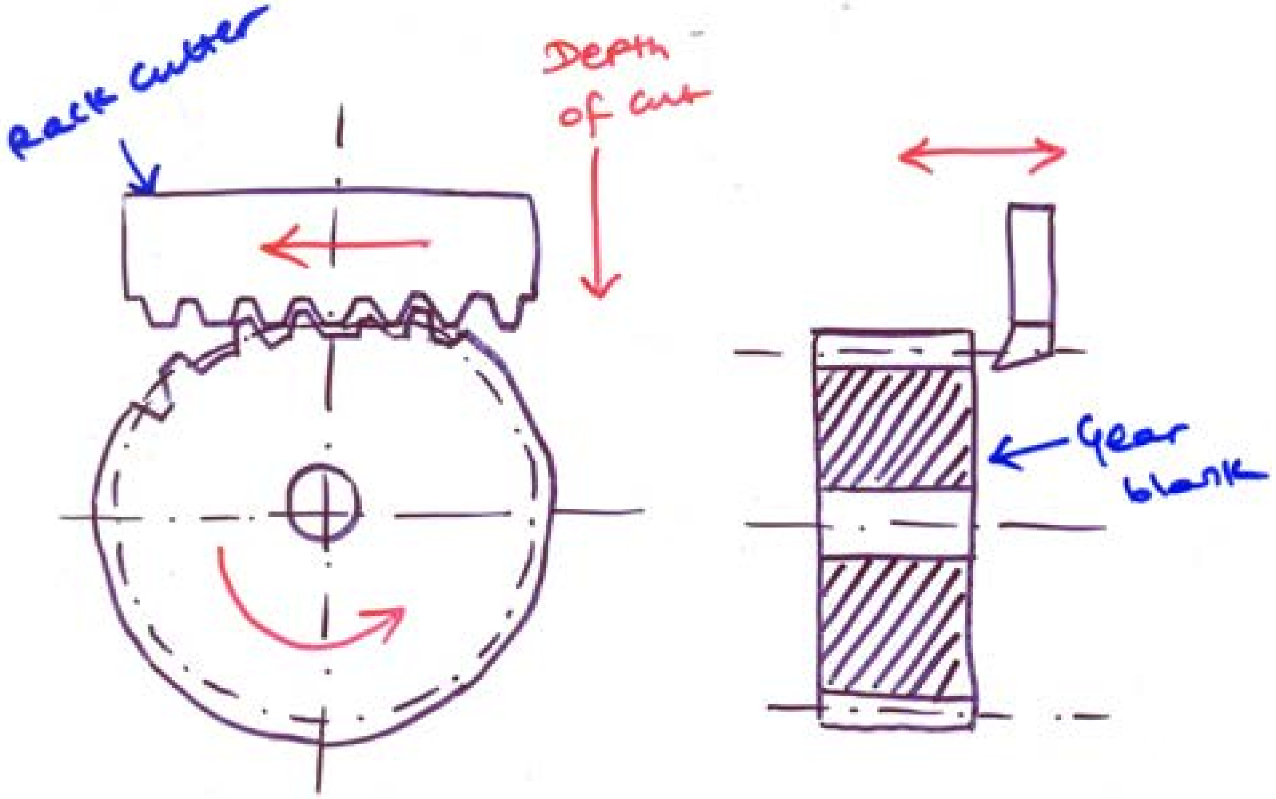
Advantages:-

1. Using a single hob, any number of teeth of the same module can be produced.
2. Spur & helical gear can be produced using the same hob.
3. Several gear blanks mounted on the same arbor can be processed simultaneously. So it can be used for mass production.

Limitations:-

1. Internal gear cannot be produced
2. Rolling motion b/w the gear blank & hob is not continuous. So, production time is more

Gear planning:-



* It is done on the Gear planner.
* Here rack cutter is used for making the teeth on the Gear blank
* Here, the cutter & the Gear blank's movements are like a movement of a rack & pinion in mesh.
* Here the Gear blank is fitted on the table. The relative rolling movement between the Gear blank & cutter is obtained by the Gear train.
* Here, after the Rack has cut 2 (or) 3 teeth on the Gear blanks it will be brought to the starting position & the operation is repeated till

all the teeth are cut on the Gear blank

* In the Gear planning process, the following movements will be given.

① Radial feed of the cutter towards the Gear blank for giving the depth of cut.

② Relative rotational feed & horizontal feed of the Gear blank & cutter respectively

③ Reciprocating movement of cutter over the width of the Gear blank.

④ The withdrawal of cutter from the Gear blank during the return Stroke. It will avoid the rubbing action of cutter during the return Stroke

Application:-

used to cut external Spur & helical Gear.

Advantage:-

① the Same module gear of any number of teeth can be cut using a single rack cutter.

② Used to cut external Spur & helical gear.

Limitation:- ① Internal & worm gear cannot be produced.

② Due to stepping back of cutter, production time is high.

Gear Finishing process:—

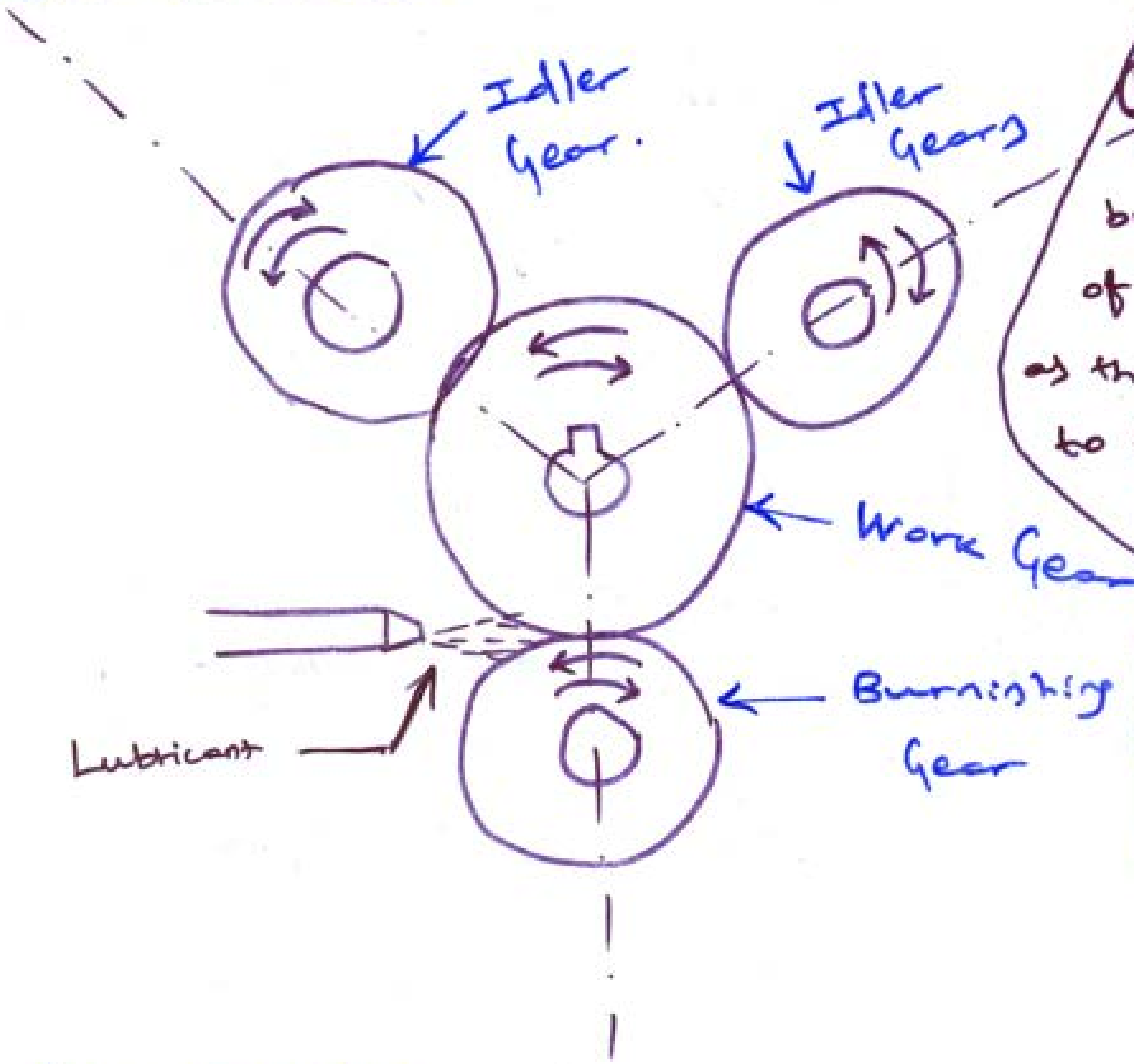
* Gears manufactured by different machining process will have rough surfaces.
* The machined gears may have errors in tooth profiles, concentricity, Helix angles.
* For quiet & Smooth opreation of Gears, these errors & rough surfaces should be removed. Gear finishing opreations are done for this purpose.

1. Burnishing – for Gears not hardened.
2. Shaving – for Gears not hardened.

③ Grinding – for Gears which are hardened.

④ Lapping – for Gears which are hardened

Gear Burnishing:–



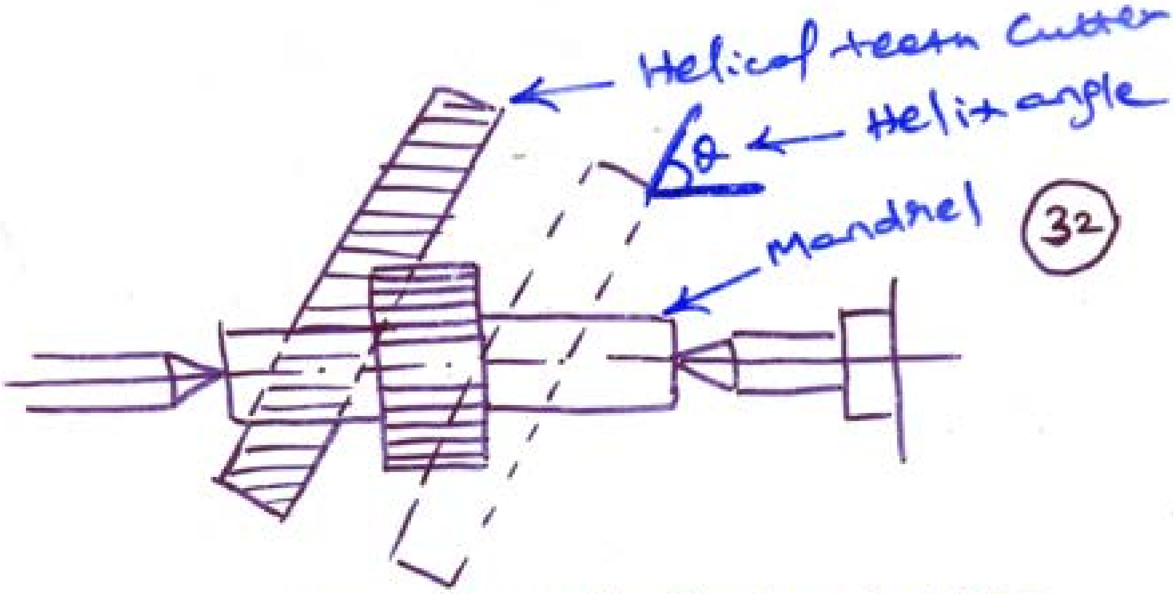
① Here, the Hardened burnishing gear of same module as that of the Gear to be finished is made to mesh with the Gear to be finished.

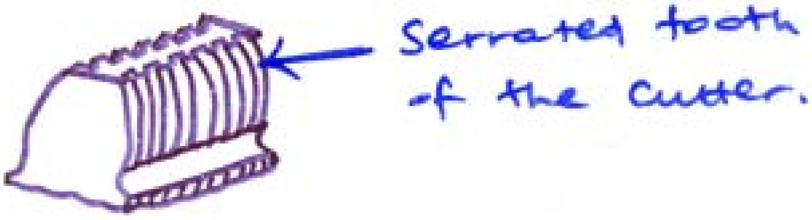
② The idler gears are used to support & give the pressure

Gear Shaving:–

* Most common method of Gear finishing.

* In this method, a very hard gear shaving cutter is used to remove fine chips from the Gear teeth.





* Rotation of the work Gear
* Rotation of Cutter & Reciprocation of Shaving cutter.

Gear Grinding:-

It is used for finishing of gears after hardening. Gear Grinding is done for

1) Increasing the accuracy & Surface finish of the gears
2) Removing the distortion after heat treatment.

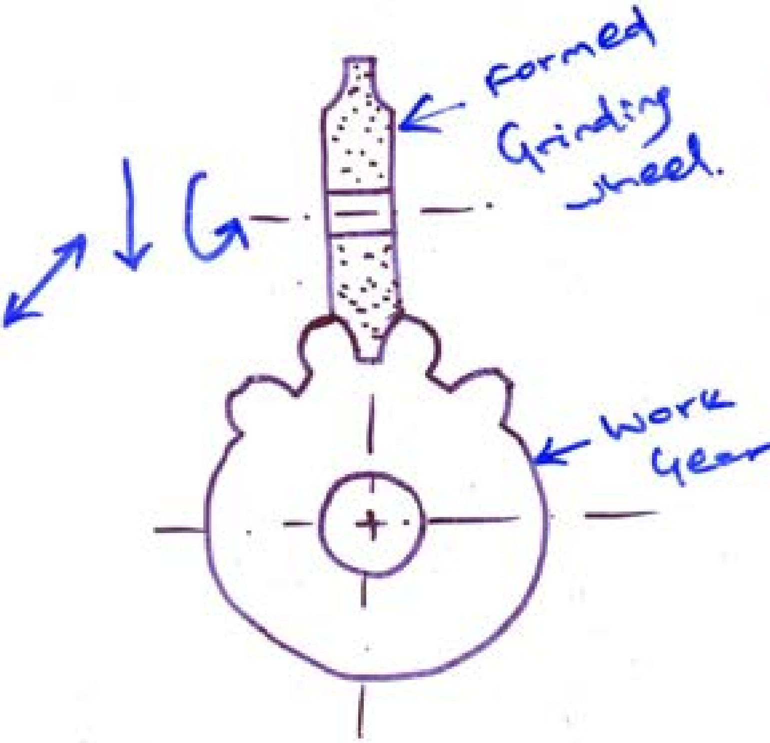
There are two methods of Gear Grinding.

They are

a) Formed wheel Grinding

b) Generation Gear Grinding.

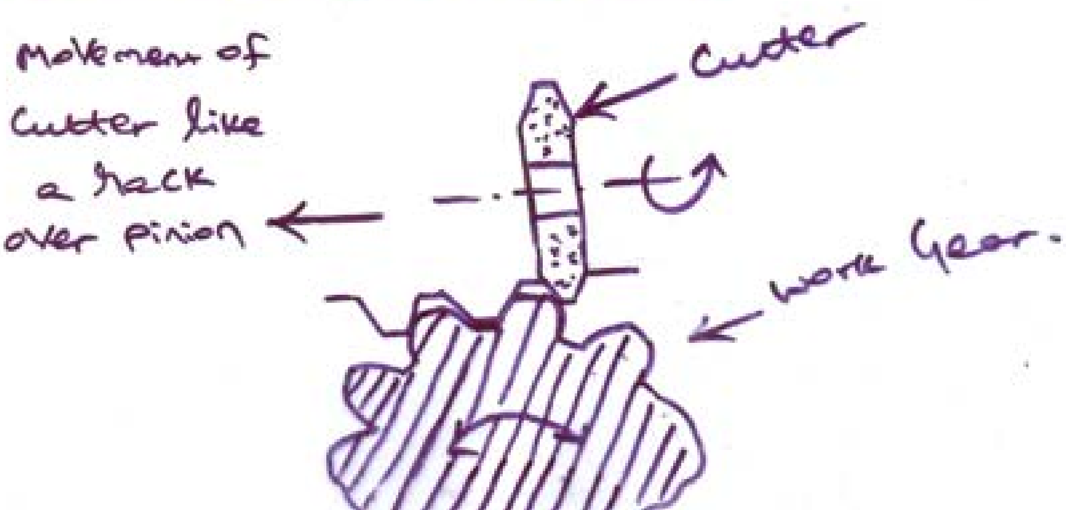
Formed wheel Grinding:—



* Three passes required for completing one tooth space.

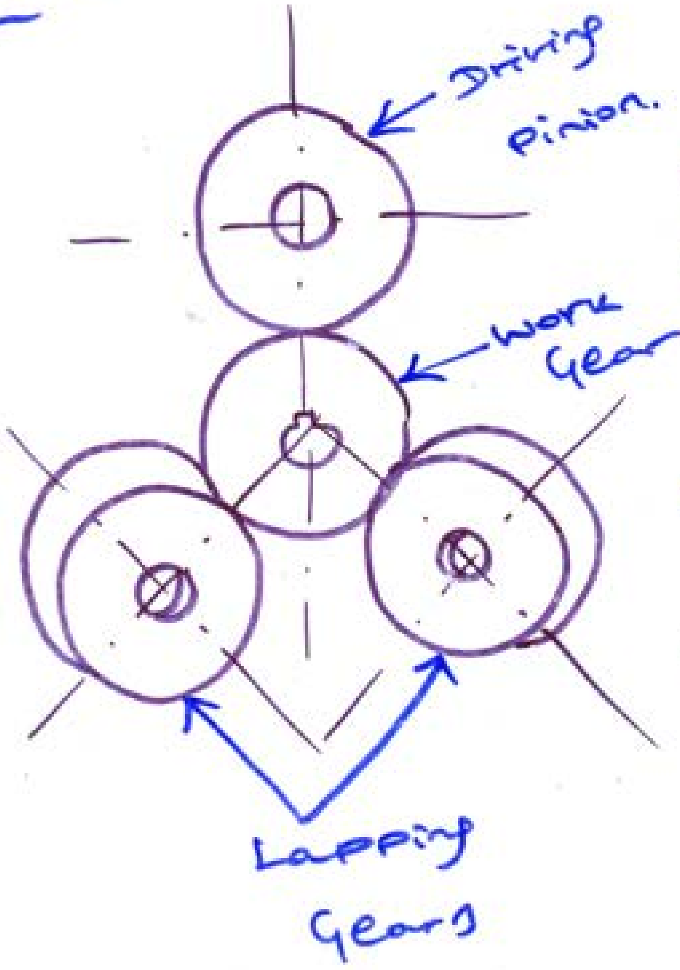
* Coolant should be used while finishing.

Generation Gear Grinding:—



* The work gear is rolled along an imaginary rack.
* Rolling is done in both the direction to grind both sides of the tooth.
* The Grinding wheel rotates & reciprocates along the length of the tooth.

Gear lapping:-



* The Surface of the Lapping Gears are inclined at about 4° to the work Gear Surface whereas the Surface of the Driving pinion is ‖ to the work Gear Surface

* When the gears rotate, a lapping compound is applied b/w them. The compound is a mixture of a very fine abrasive powder and Kerosene.

Unit – IV

ABRASIVE PROCESS AND BROACHING

1

Introduction :–

* Grinding * self-sharpening of Grinding wheel.
* for what purpose ?

Types of Abrasives :–

(i) natural Abrasives (ii) Artificial Abrasives.

(i) natural Abrasives	(ii) Artificial Abrasives
(a) Sand Stone	(a) Aluminium oxide
(b) Emery	(b) Silicon Carbide
(c) Corundum	(c) Artificial Diamond
(d) Diamond	(d) Boron Carbide

Types of bond :–

* Adhesive Substance used to hold abrasive Grains together to form the Grinding wheel
* Sufficiently Strong to withstand Stresses
* Their Choice – Grinding speed, Cutting pressure, heat formation etc

* Types :–

(i) Organic →
- Metalic
- Vitrified (V)
- Silicate (S)

(ii) Non-organic →
- Resinoid (B)
- Rubber (R)
- Shellac (E)
- oxy-Chloride (O)

Grit (or) Grain Size:-

- Indicates the Size of the abrasive grains (or) Size of the cutting teeth of the Grinding wheel.
- Denoted by the number indicating the number of meshes per linear inch (25.4mm) of the Sieve through which the grains can pass through.
- Larger is the grit number, Smaller will be the Grit (or) Grain size (or) vice-versa.

Grinding opreation	Grit (or) Grain Size						
Coarse	10	12	14	16	20	24	
medium	30	36	46	54	60		
Fine	80	100	120	150	180		
very fine	220	240	280	320	400	500	600

Grade:-

- Grade (or) Hardness indicates the strength with which the bonding material holds the abrasive grains in the grinding wheel.

Soft	A	B	C	D	E	F	G	H		
Medium	I	J	K	L	M	N	O	P		
Hard	Q	R	S	T	U	V	W	X	Y	Z

STRUCTURE OF WHEELS:-

* Denotes the spacing b/w abrasive grains (or) the desity of the wheel.

Structure	Symbol							
Dense	1	2	3	4	5	6	7	8
open	9	10	11	12	13	14	15	or more

Designation of Grinding wheel:-

Sequence ⇒ Prefix Abrasive Grain Grade Structure Bond Suffix

W A 36 K 5 R 17

W → Mfr's abrasive type Symbol (optional)

A → A – Al_2O_3, C – SiC, D – Diamond

36 →

Coarse	medium	fine
10	30	80
12	36	100
14	46	120
16	54	150
20	60	180
24		

K → Grade Scale } A–H ⇒ soft; I–P ⇒ medium; Q–Z – Hard

5 → Dense

1–8 ⇒ Dense

9–15 ⇒ open

R → V – Vertified, B – Resinoid, R – Rubber, E – Shellac, S – Silicate, O – oxychloride

17 → Mfr's Abrasive type Symbol (optional)

Selection of Grinding wheel:-

(4)

* quick stock removal, high-class surface finish, close dimensional tolerances.

* Parameters ① Constant factors
  ② Variable factors.

Constant factors:-

① Physical properties of the material to be ground
② Amount and rate of stock to be removed
③ Area of contact
④ Type of Grinding m/c.

Variable factors:-

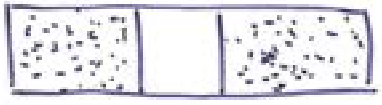
① work speed ② wheel speed
③ condition of the Grinding m/c ④ Personal factors.

Reconditioning of Grinding wheel:-

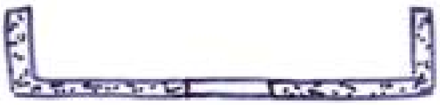
* Wear, Loading, Glazing $\Rightarrow$ wheel loses its cutting capacity

* Avoid $\Rightarrow$ Dressing, Truing.

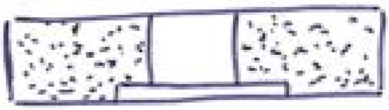
SHAPES OF GRINDING WHEEL:-



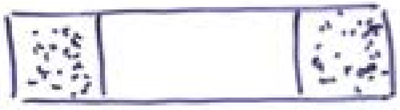
(i) Straight



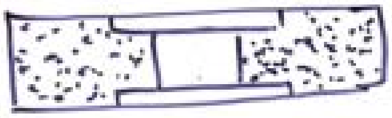
(ii) Straight cup



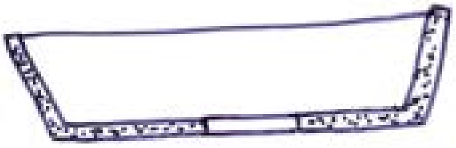
(iii) Recessed on one side



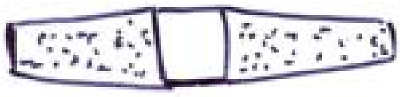
(iv) Cylindrical



(v) Recessed on both sides

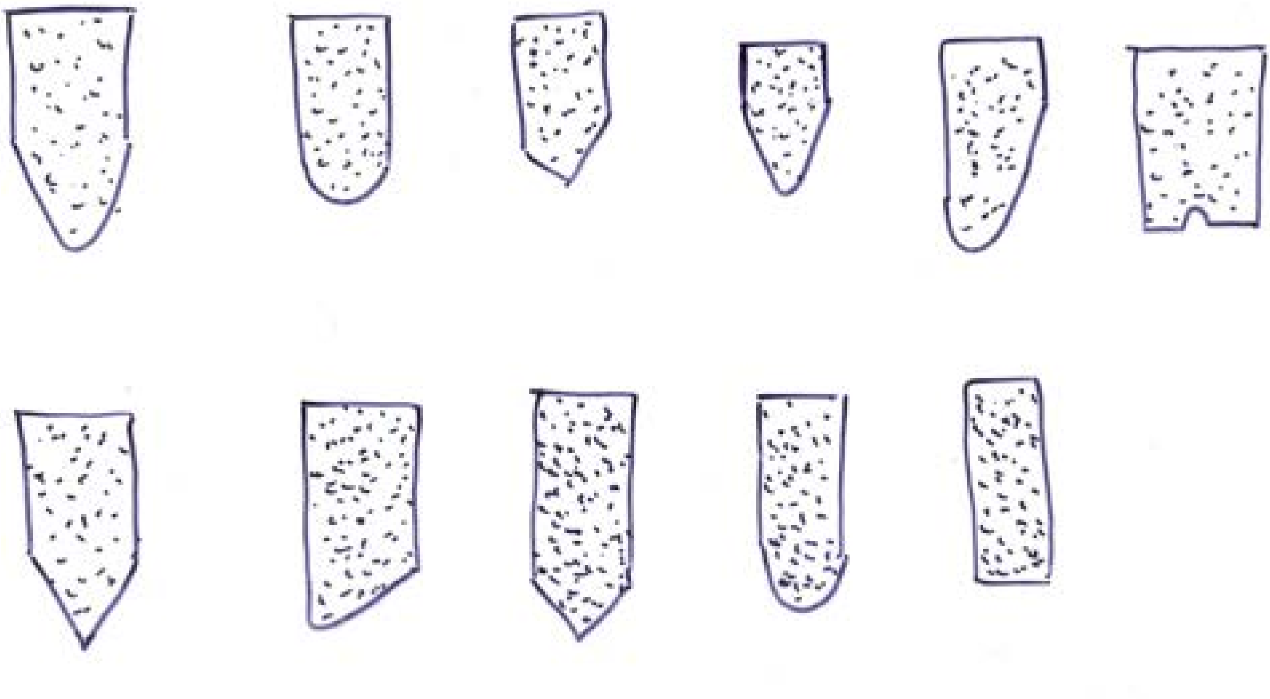


(vi) Flaring cup



(vii) Tapered

(viii) SAUCER type.



Standard Shapes of Straight Grinding wheel faces

Glazing of Grinding wheel:-

* Glazing of wheel is a condition inwhich the face of the grinding wheel takes a glass-like appearance. that is, the cutting points of the abrasives have become dull & Smooth due to its wear.

* Continued work with the glazed wheel increases the Smoothness of the face & decreases its cutting Capacity.

* Glazing of wheel takes place when a wheel
  1. is too hard
  2. revolves at high speed.

* Remedy for Glazing is to
  1. to give optimum speed for the wheel.
  2. to use a softer wheel.

Loading of the Grinding wheel:-

* The Grinding wheel may become loaded if the particles of the metal being ground will adhere to the face of the grinding wheel & fill up the pores of the wheel face. the adhered particles of the metal being ground with the face of the

Grinding wheel prevent the Grinding wheel to cut freely.

* Loading may be caused by

(7)

① Grinding softer material

② using a wheel of too hard a bond and running it too slowly

③ taking cuts that are too deep & by not using the right cutting fluid.

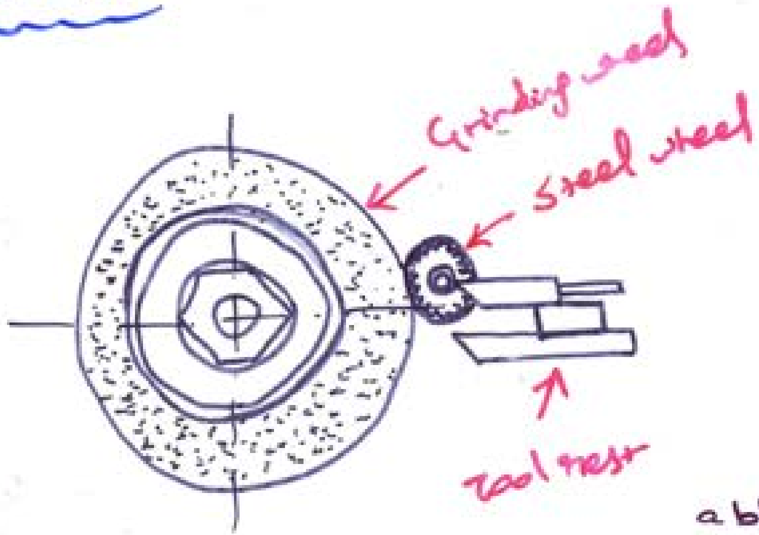
* The remedy for loading is

① to run the wheel at optimum speed

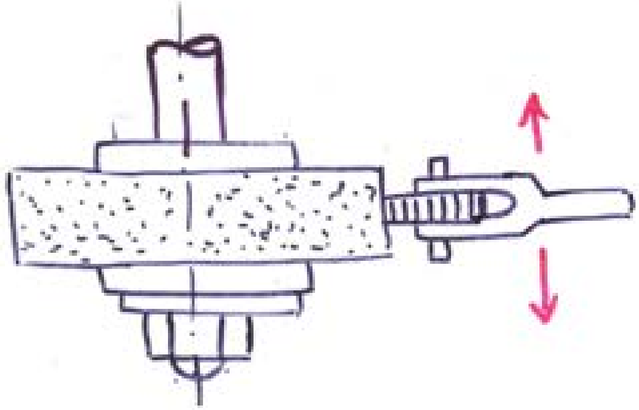
② to use a softer wheel.

Dressing:-

8



* Process of removing loaded particles and breaking away the glazed surface by the tool called "dresser" so that the abrasive particles are again presented to the work surface.



Tool types

a) Star dressing Tool

b) Round abrasive stick

c) Diamond dressing tool.

Truing:-

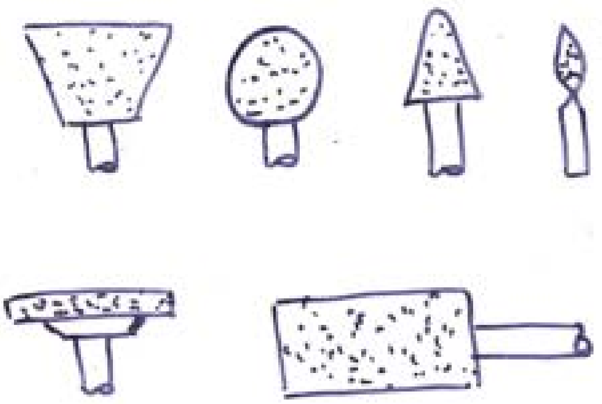
* Truing is the process of changing the shape of the grinding wheel as it becomes worn from an original shape owing to the breaking away of the abrasive particles & bond.

* This is done to make the wheel rotate true with its axis.

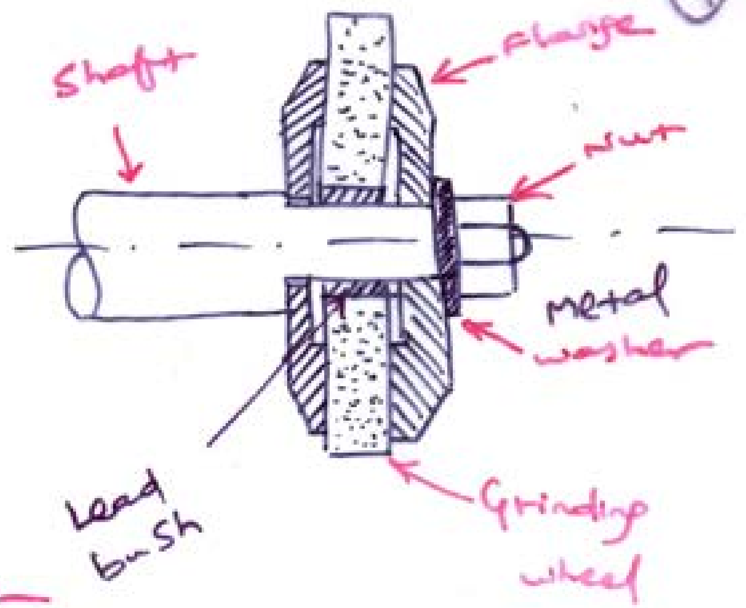
* It is also done to change the face contour for form grinding.

* Truing & dressing are done with the same tools but not for the same purpose.

Mounted wheels & Points:-



Mounting of Grinding wheel:-



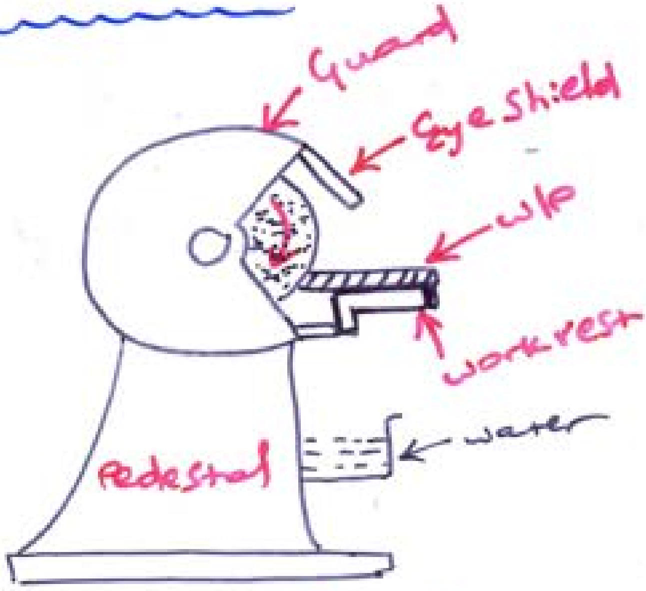
Types of Grinding Machines:-

1. Type of operation —— (a) Tool (b) cut-off Grinder
2. Quality of finish —— (a) Rough (b) Precision Grinder
3. Type of Surface —— (a) cylindrical (b) Internal (c) Surface (d) Special purpose etc.,

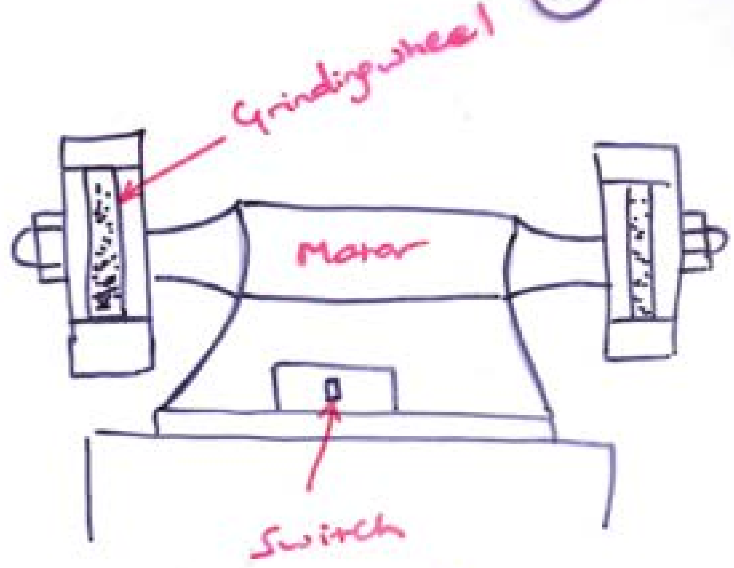
Rough Grinders
- (1) floor stand Grinders
- (2) Bench grinders
- (3) Portable grinders
- (4) Abrasive belt grinder
- (5) Swing frame Grinder

Precision Grinder
- Cylindrical Grinder
  - Center type Plain
  - Center type Universal
  - Centerless
- Internal Grinder
  - Chucking type
  - Planetary type
  - Centerless
- Surface Grinder
- Tool Surface and Special Grinder

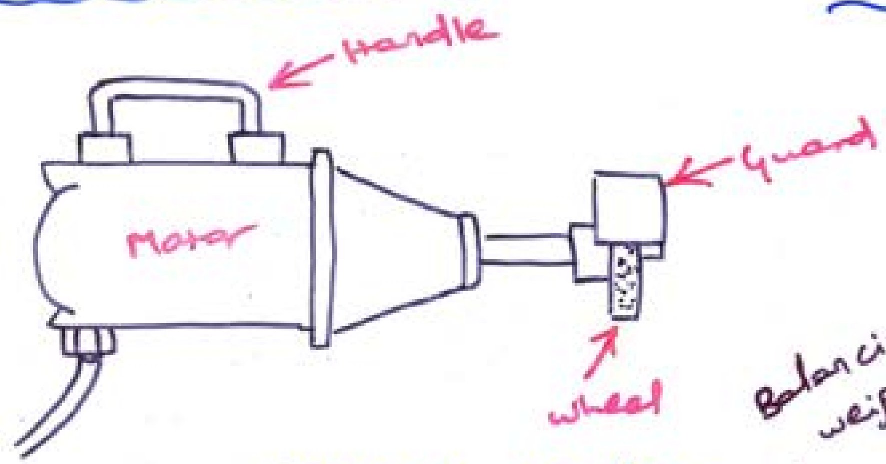
Rough Grinders:-



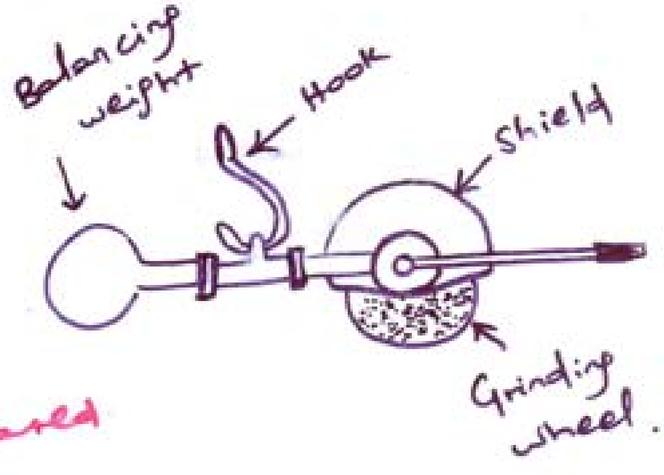
Floor Stand Grinder



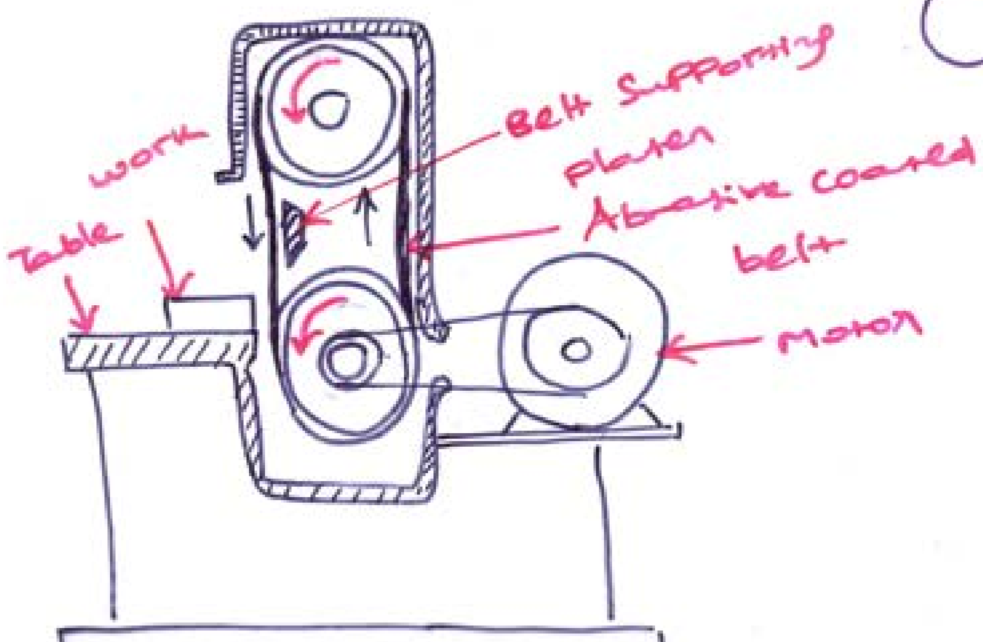
Bench Grinder



Portable Grinder



Swing frame Grinder



Abrasive Belt Grinder

Precision Grinders:-

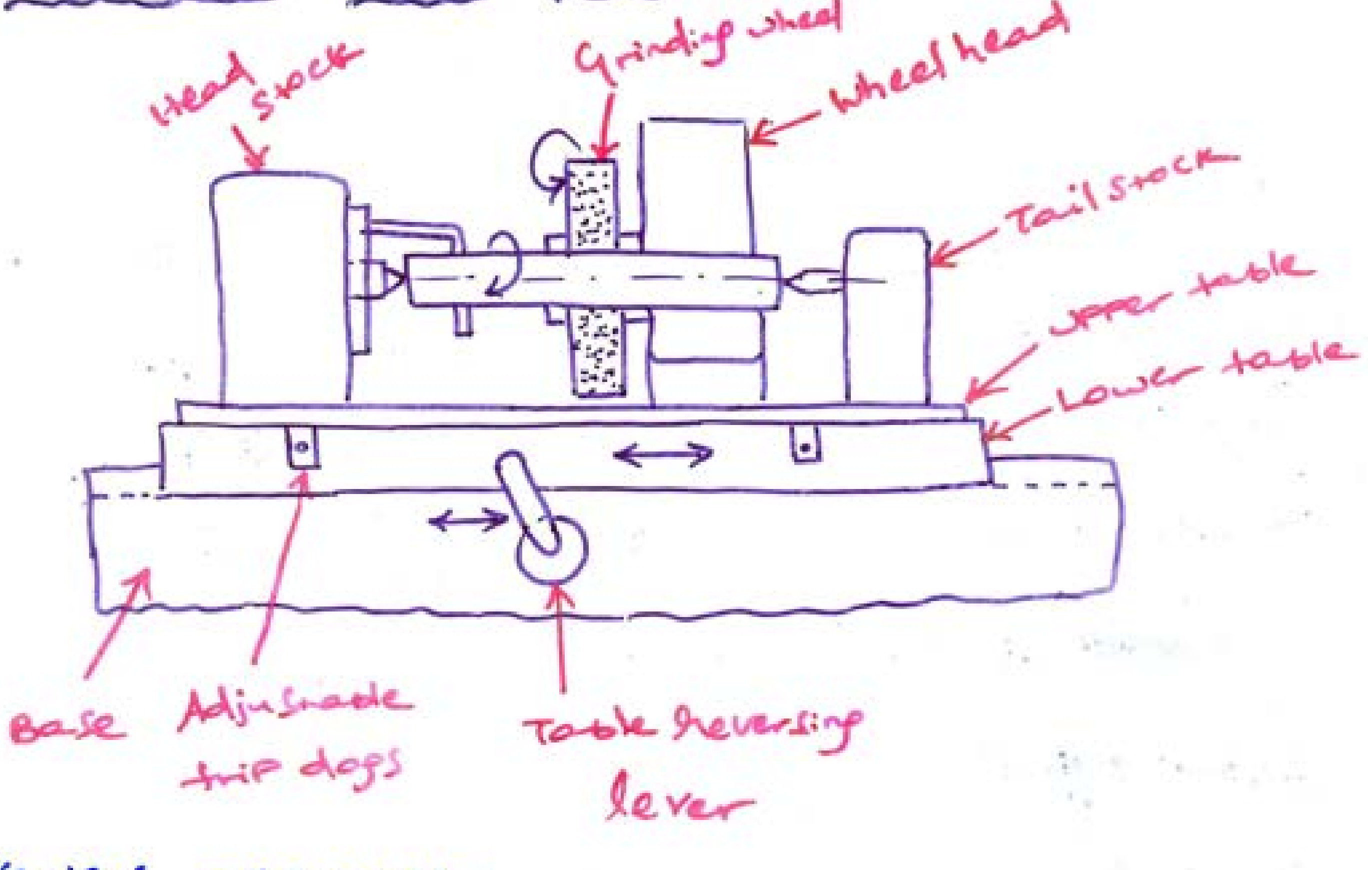
11

Cylindrical Grinders:-

(i) Center type Grinders → Plain type / Universal type

(ii) Centerless type Grinders.

Center type plain Cylindrical Grinder



Various movements:-

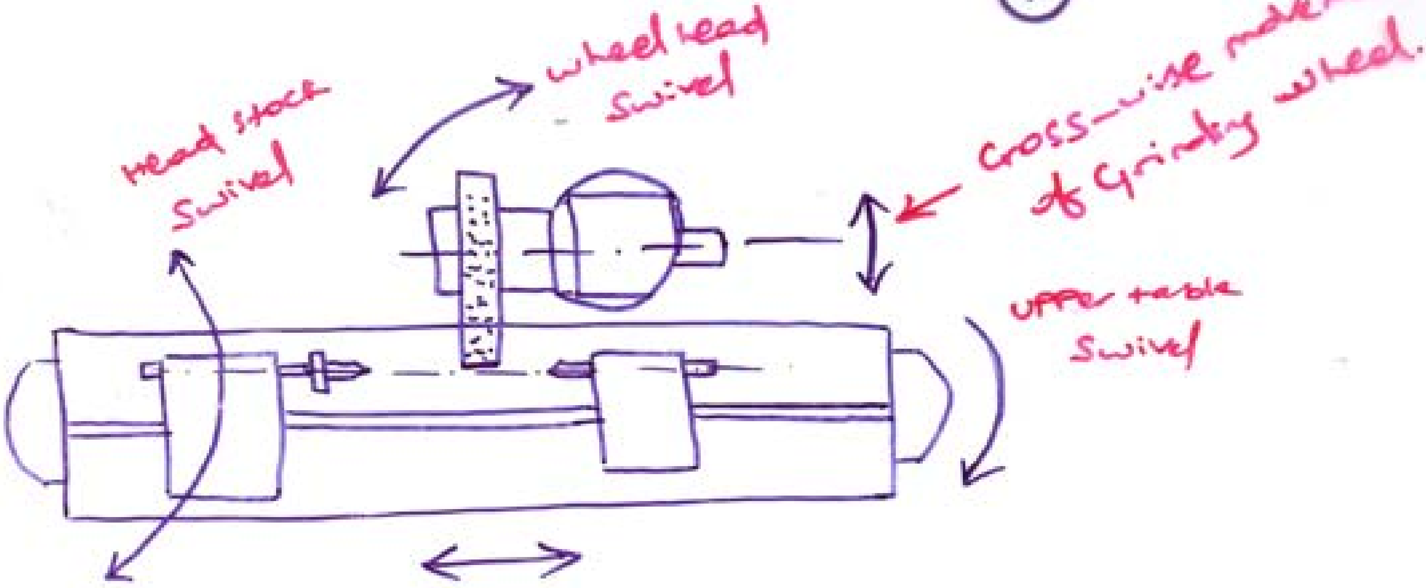
① Longitudinal feed along guideways of the bed. [Hand (or) Power]

② Swivelling of upper table on the lower table [10° on either side]

③ Rotation of a Grinding wheel.

④ The Movement of the wheel perpendicular to the Table ways. by hand (or) Power. [cross feed]

center type universal Grinder:-



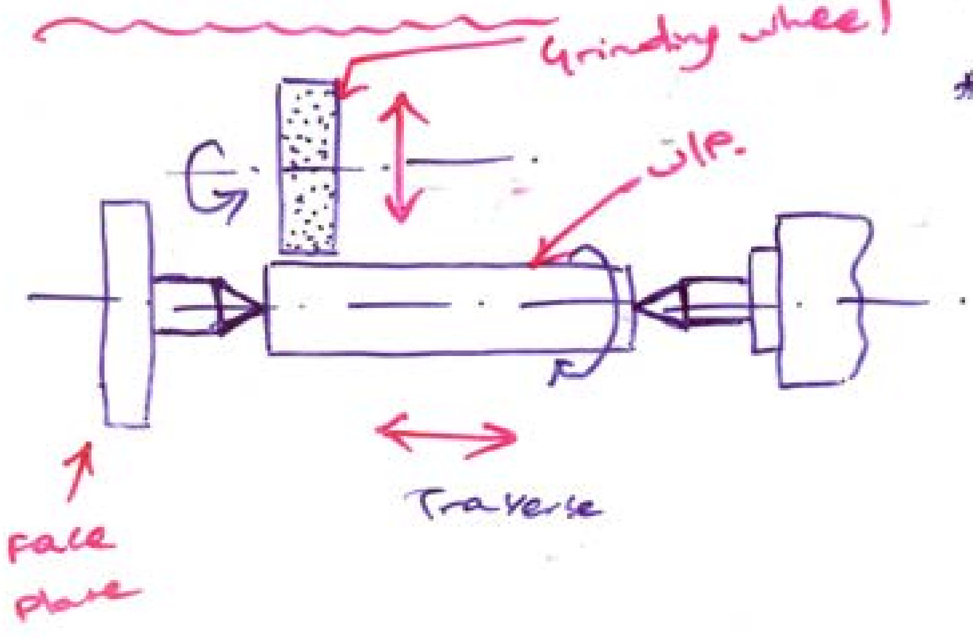
* Revolution of workpiece b/w the centers
* Swivelling of Head stock in a horizontal plane
* Swivelling of upper table in a horizontal plane
* Swivelling of wheel head in a horizontal plane.
* Longitudinal feed of the table
* Cross-wise movement of the grinding wheel.

Types of operations in cylindrical Grinding machine:-

(i) Traverse Grinding

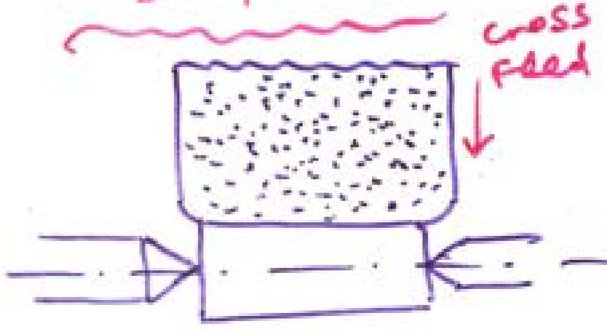
(ii) plunge Grinding

Traverse Grinding:-

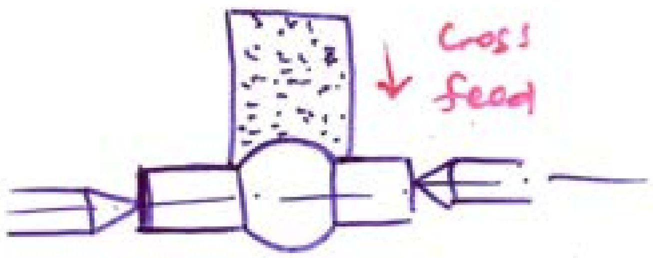


* used when the job length is more than the width of the grinding wheel.

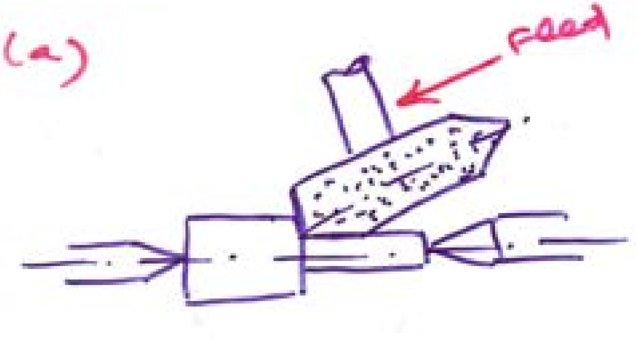
Plunge Grinding:-



(a)

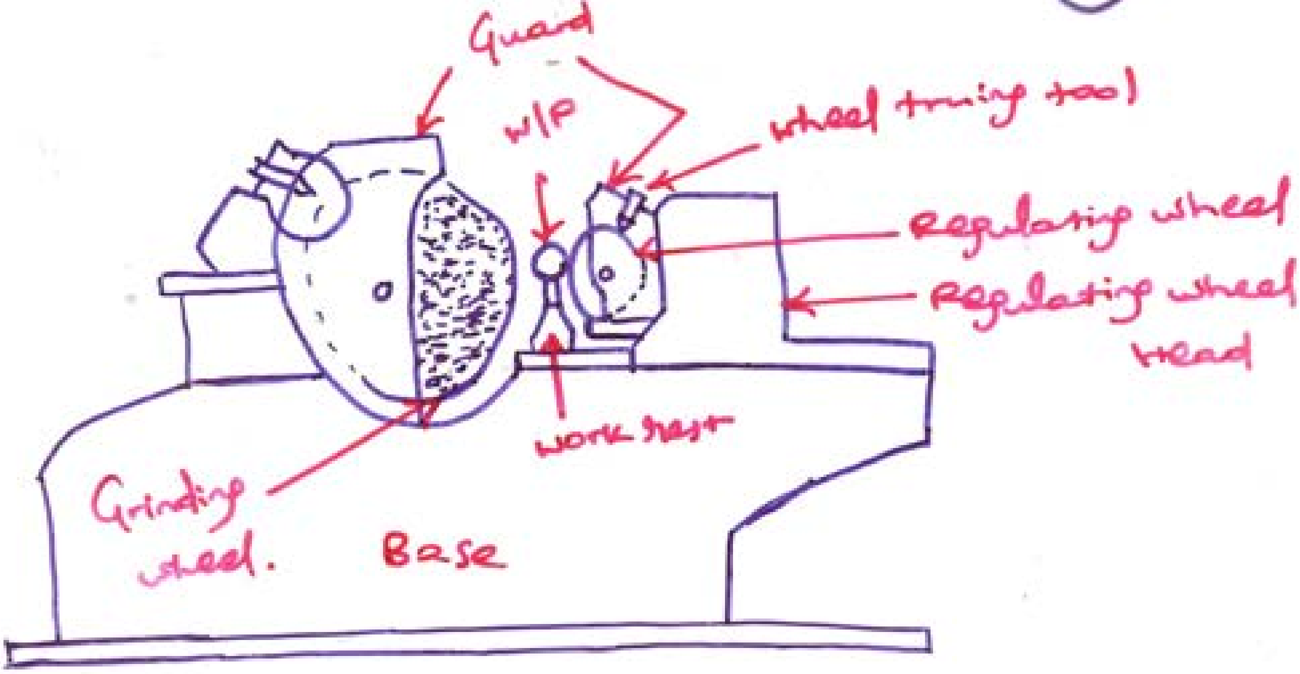


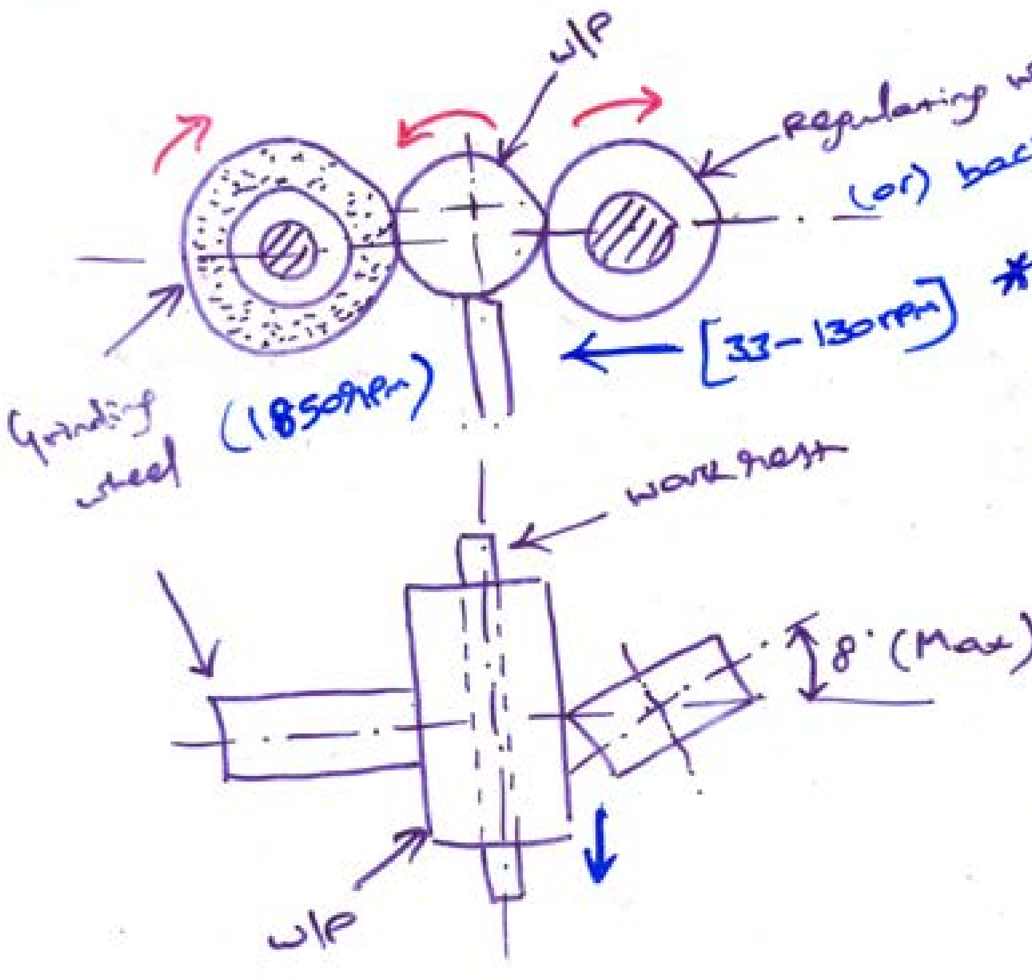
(b)



(c)

Centerless Grinders:-





* Method of Grinding exterior Cylindrical, taper & formed surfaces on the w/p that are not held & rotated on centers.

* The regulating wheel is inclined at an angle of 8° to permit the axial movement of the workpiece in between the wheels.

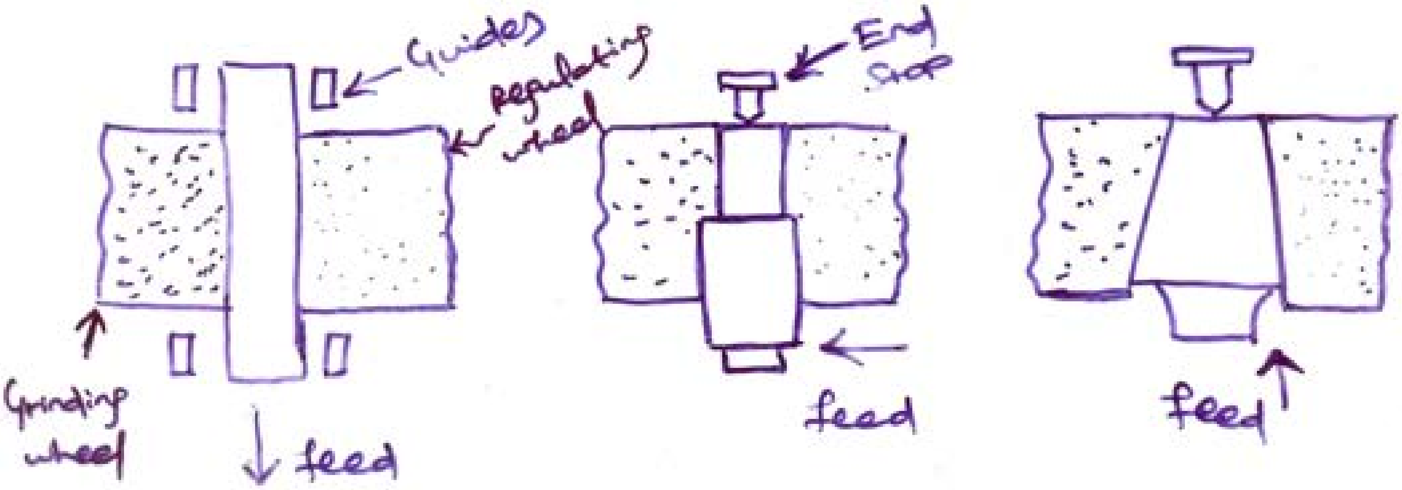
$\boxed{S = \pi d n \sin\alpha}$

Methods of Centerless Grinding:-

(15)

(i) through feed.

(ii) In-feed (iii) End feed.



(a) through feed (b) In feed (c) End feed Grinding

(a) through feed:
* long shaft, bars, roller pins etc.
* for 1 pass, 0.2mm

(b) In feed:
* similar to plunge grinding
* used for Grinding formed surfaces

(c) End feed Grinding:
* used for Grinding tapered Surfaces.

* End stop prevents the axial move of the w/p.

Internal Grinders:-

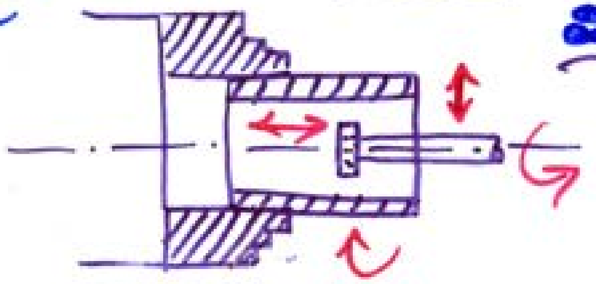
* used to finish straight, tapered (or) formed holes.
  (a) Chucking type
  (b) planetary type
  (c) centerless type

Chucking type Internal Grinders:—

(16)

According to General construction, there are **2** types.

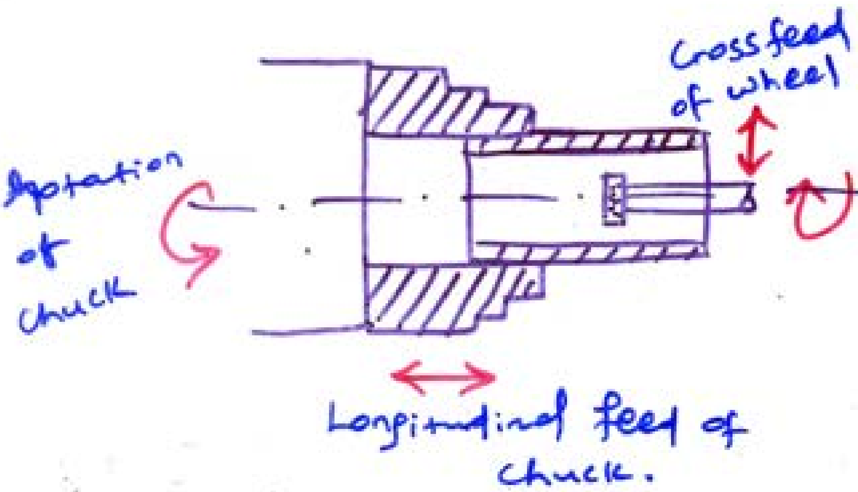
Type-I



w/p → rotation

Grinding wheel } ⇒ Rotation, longitudinal feed, cross-feed.

Type-II



w/p → rotation, longitudinal feed.

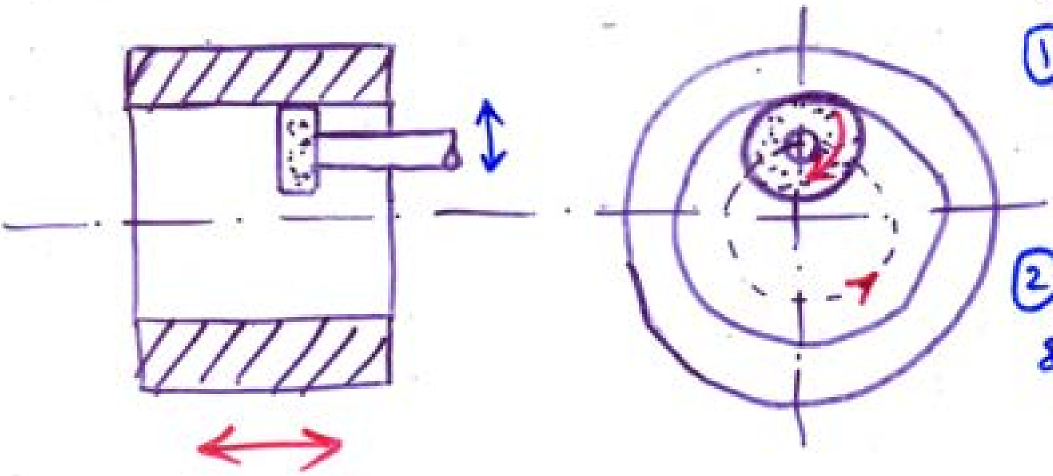
Grinding wheel } ⇒ Rotation, cross-feed.

Plain Internal Grinder :— The workhead can be swivelled to 45°.

Universal Internal Grinder:—

* work head — Cross feed can be given & It is swivelled upto 90°.

Planetary type Internal Grinders :-



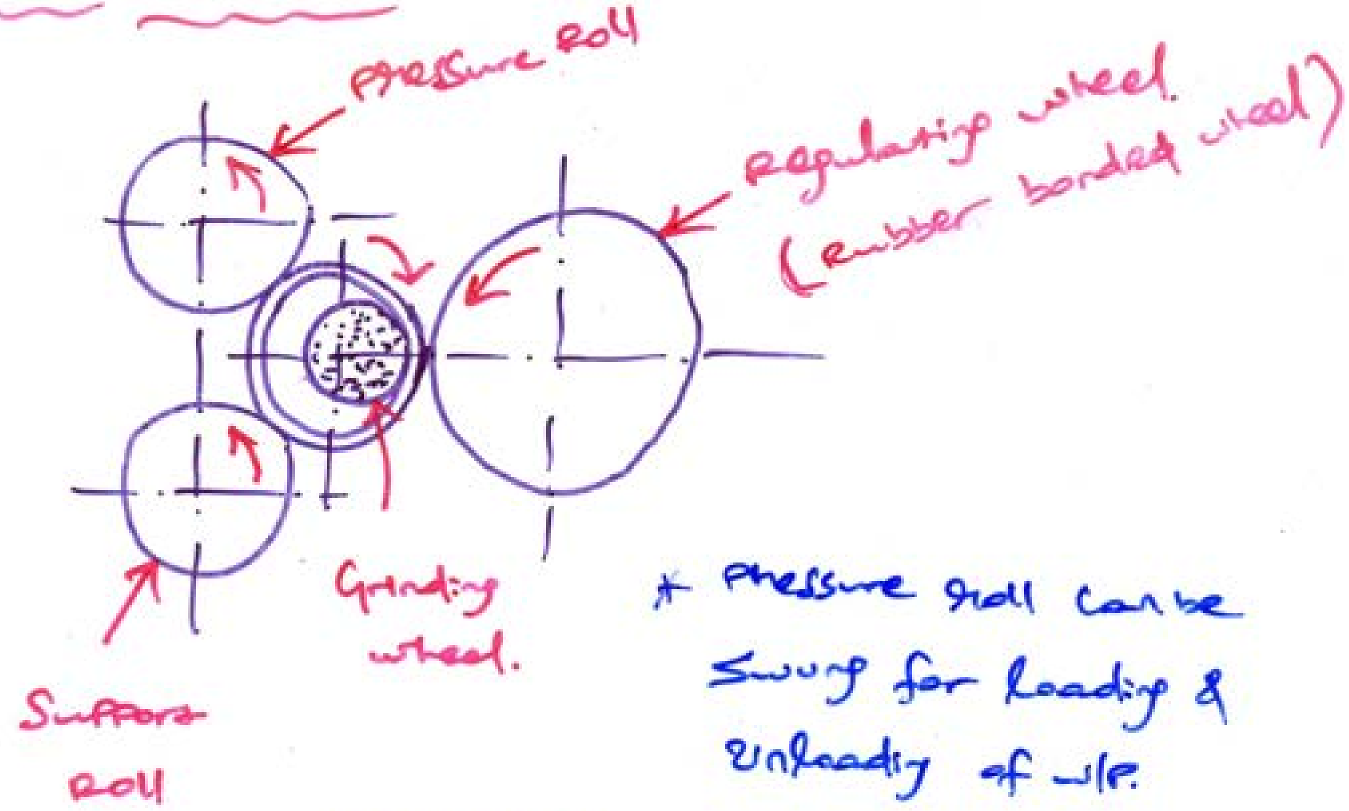
Movements

(1) Longitudinal (or) Reciprocating movement of Chuck.

(2) Planetary motion & cross-feed of wheel.

used for finishing holes for heavy Jobs that can't be conveniently rotated by a Chuck

Centerless Grinding:-



* Pressure roll can be swung for loading & unloading of w/p.

Movements

Grinding wheel ⇒ Reciprocation, rotation, cross-feed.

Advantages of centerless Grinding:-

(18)

① As the w/p is supported throughout its entire length when grinding takesplace, there is no tendency for chatter (or) deflection of the work.

② Small, fragile (or) slender workpieces can be ground easily.

③ The process is continuous & is adopted for production work

④ Less metal is removed due to floating condition.

⑤ no center holes, no chucking (or) mounting of work on mandrels (or) other work holding devices are required.

Disadvantages:-

① In hollow workpiece, there is no certainty that the outside diameter will be concentric with the inside diameter.

② work having multiple diameter is not easily handled.

Surface Grinding M/c :-

(19)

* Used to produce & finish flat & plane Surfaces
* Angular & formed Surfaces
* Heavy workpieces — fixture
  Small workpiece — Magnetic chuck

Types

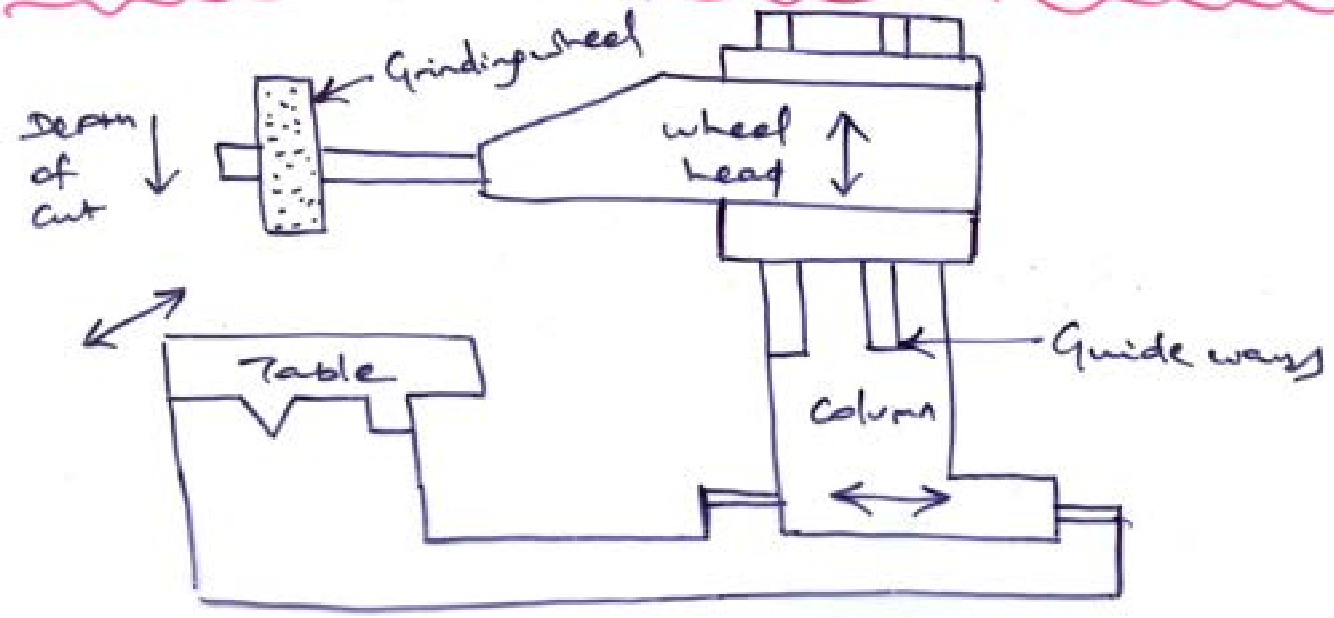
(1) Horizontal Spindle Reciprocating table Surface Grinder

(2) Horizontal Spindle Rotary table Surface Grinder

(3) Vertical Spindle Reciprocating table Surface Grinder

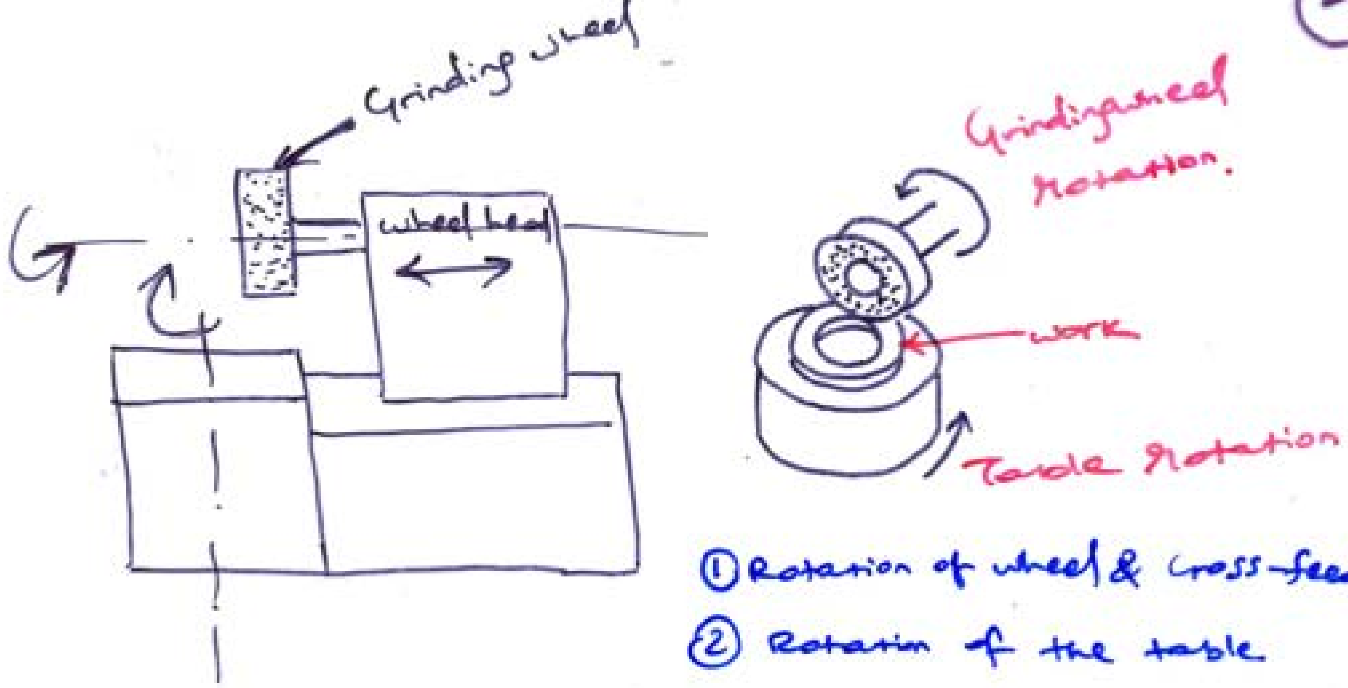
(4) Vertical Spindle Rotary table Surface Grinder

(1) Horizontal Spindle Reciprocating table Surface Grinder



Horizontal Spindle Rotary table Surface Grinder:-

(20)



① Rotation of wheel & cross-feed of wheel

② Rotation of the table

Vertical Spindle Reciprocating table Surface Grinder

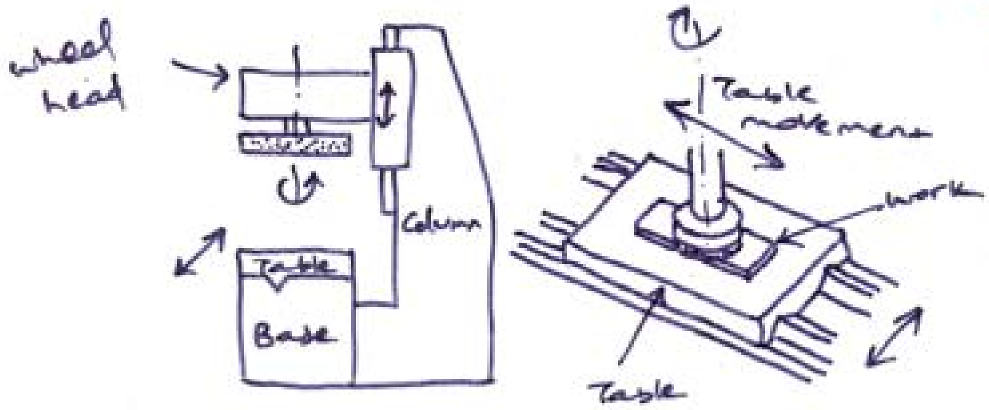
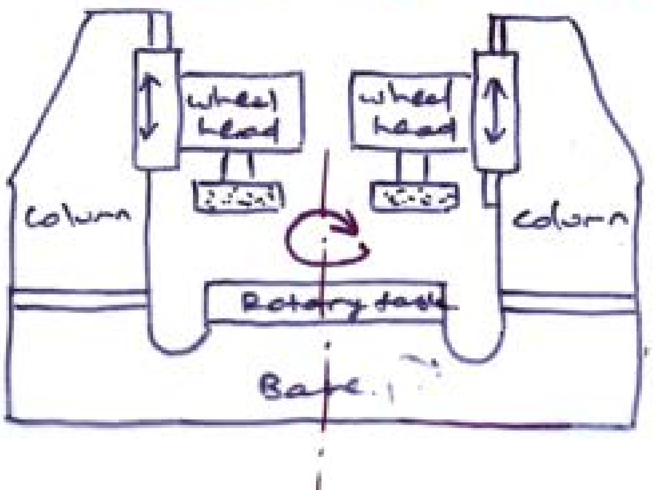


Table – C. feed & L. feed

wheel – Vertical & rotation.

Vertical Spindle Rotary Table Surface Grinder:-



used to Grind large quantity of workpieces.

Unit - V

①

CNC machine Tool & Part Programming

Introduction:-

* Numerical control has been developed out of need for higher productivity, lower cost & more precise manufacturing.

* In NC System, operation instructions are inputted to the machine as a code of numbers, letters, & special characters. These coded instructions are then automatically carried out in the machine tool in a pre-determined sequence with pre set speed, feed, etc., without human intervention.

NC & its Components:-

* NC stands for numerical control. It is a technique of automatically operating the machine tool based on a code of letters, numbers and special characters.

* The complete set of coded instructions responsible for executing an operation is called a:

"Part program".

* This part program is translated into electric signals to drive various motors to opreate the m/c to carry out the required opreations.

* NC system containing the following components.

1. Program of Instructions
2. Tape punch
3. Tape reader
4. Machine controller
5. NC machine.

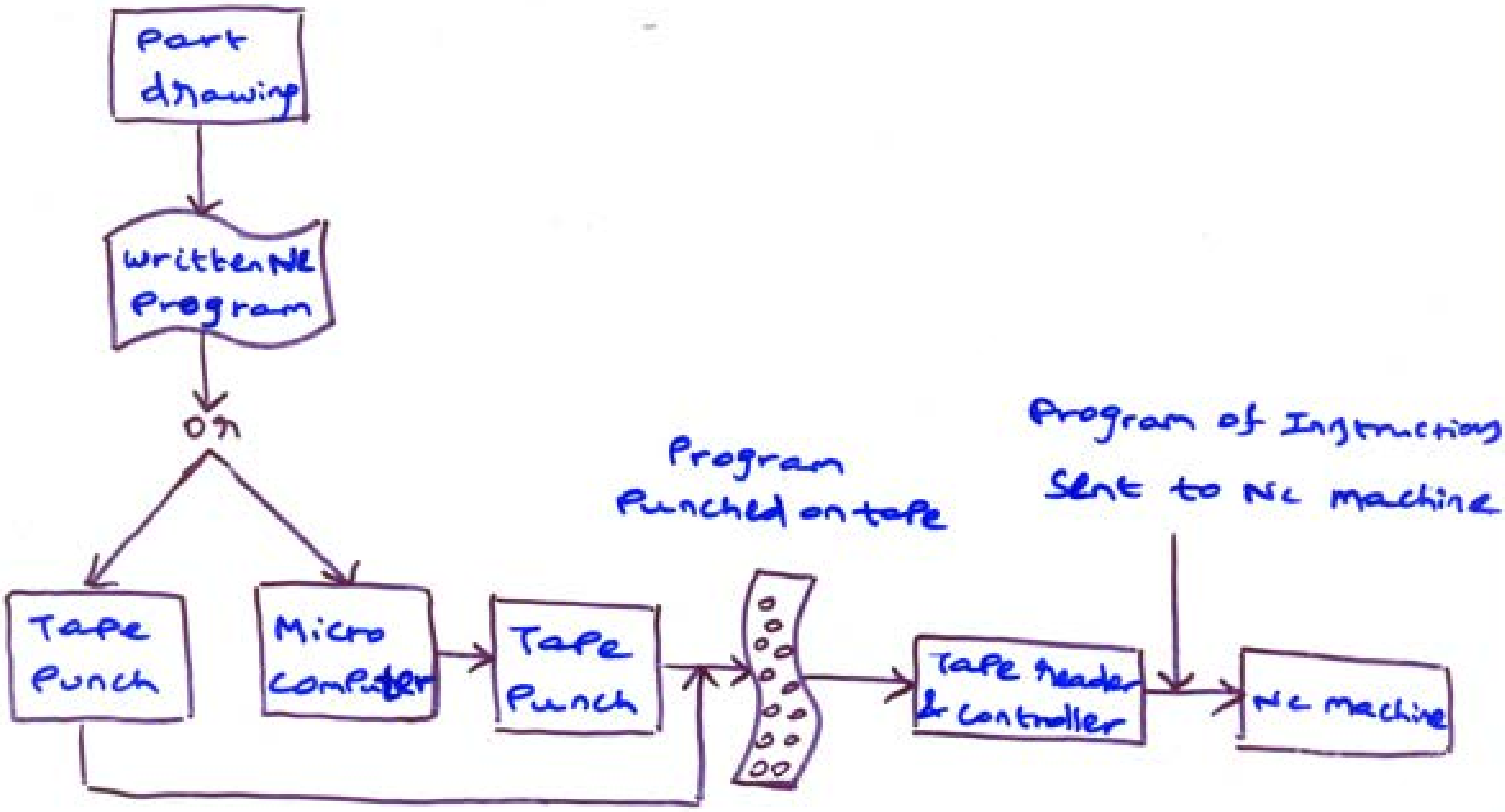
1. Program of Instructions:-

* The program of instructions, often called part program is the detailed set of instructions for the NC machine to produce a component. the part program is a mixture of Alphabetic codes, numeric data & special characters.

* This part program is punched in a input media (usually paper tape) in a specified format

* This input is read by a tape reader which converts the instructions on a paper tape into the electric signals & transfers it to the machine controller to opreate the machine slides & to generate specific

Surfaces on the job.



2 Tape Punch:-

* Usually it is a paper tape of 1" width. Paper mylar, Aluminium mylar (or) plastics are also used as a tape materials.

* Punching machine (flexo writers) of various types are used to key in program of instructions to tapes.

* Presently tapes are prepared by micro-computers by keying in the information from the manuscript.

1. once the entire program has been entered, it is checked & corrected if needed.

2. Then the computer activates the tape punching unit to produce the tape.

③ the computer can also generate print-out through its printer. ④

③ Tape reader:-

A tape reader reads the hole pattern on the tape & converts the pattern to a corresponding electric signal.

④ Machine Controller:-

* Controller receives the electric signals from the tape reader & causes NC machine to respond.

* It contains a decoder/encoder, an interpolater and facilities to execute auxillary functions which are machine dependent.

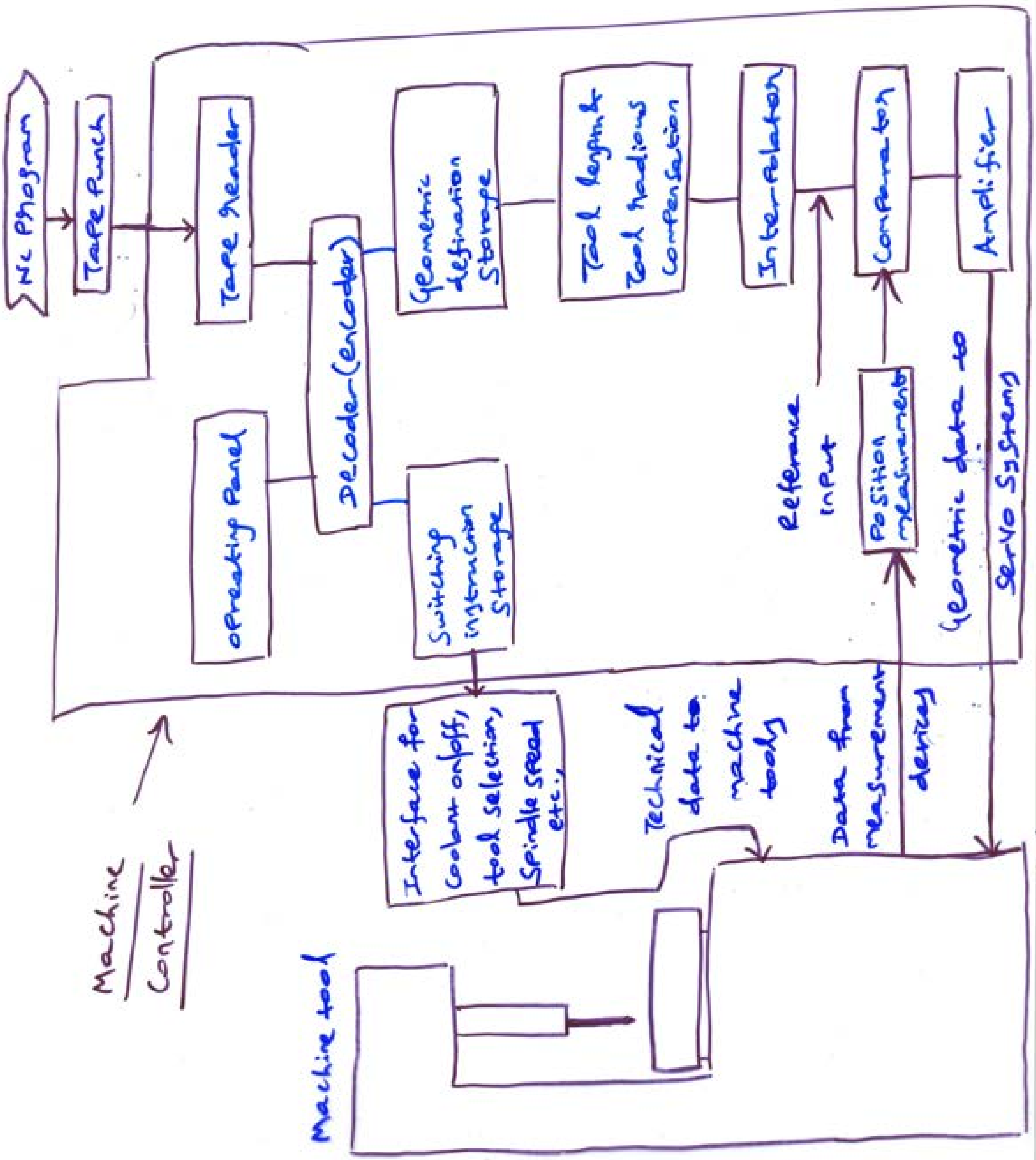
* The decoder/encoder receives data & stores them in two separate memory locations. one for the part geometry data & the other for process data

Process data includes switching functions for adjusting feed rates, spindle speeds, tool changes, cutting fluid applications etc., Geometric data consists information about tool motion, tool length, tool radious, tool compens-ation etc.,

* The function of interpolater is to break down the curves (or) simple line into small individual increments

for each controlled motion of the machine tool. (5)

* Controller also interfaces drive motors, transducers and other control functions of the machine tool.



(5) NC machine:-

* NC machine responds to the electric signals from the controller. Accordingly the machine executes various slide motions & spindle rotations to manufacture a part.

* Transducers are fitted to feed back

(1) The R.P.M of the spindle (2) the amount of cut on the job.

* NC machine tools range from single spindle drilling machine to complex machines having multiple motions, tool changers, high capacity tool magazines and multi axis control.

Position & Motion Control in NC machine:-

In NC machine, there is a servo control unit which is a group of electrical, mechanical, hydraulic & pnematic devices used to control the slide position of NC machine tool. The servo control unit is classified into two types.

a) Open loop System

b) Closed loop System.

open loop System:-

* It involves feeding of tape, intrepretation of information by a tape reader, storing the data in the buffer storage.

* After storing, it is converted into electrical signal and the electrical signal is sent to the machine control unit [MCU]

* The control unit is connected to servo control unit which controls the slide movement.

* In open loop system, there is no feed back devices to ensure whether the obtained slide movement is same as desired (or) not and if not, what error is present.

Closed loop System:-

* It is similar to open loop System, but it carries an additional feed back device

* A feed back device is nothing but an transducer

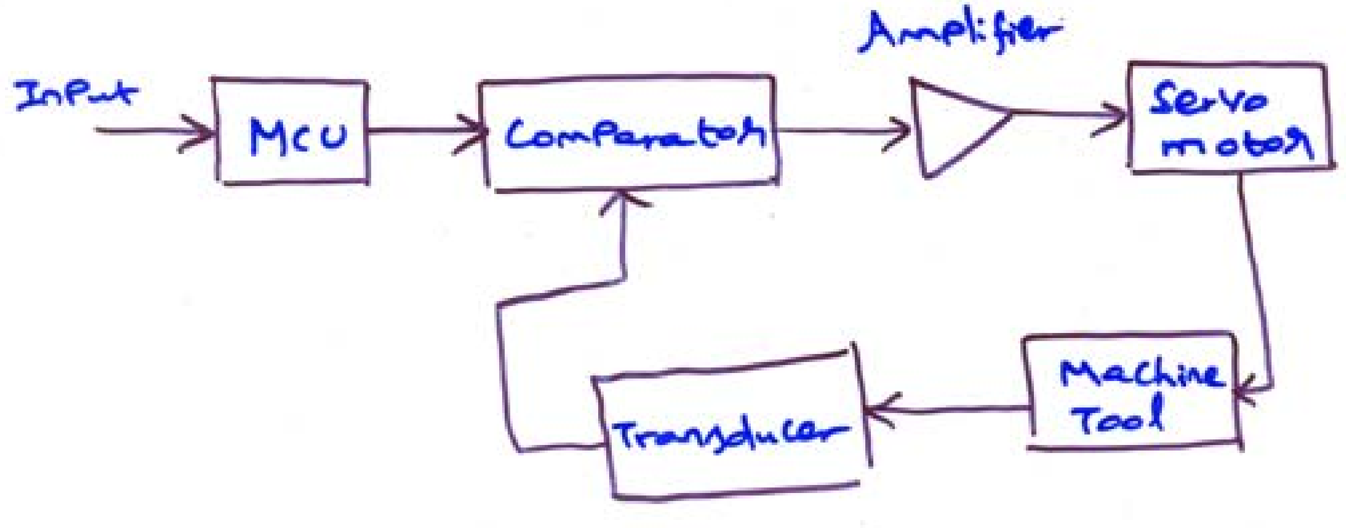
accompanied by a Comparator.

* As this is similar to open loop system, the motion is same upto the servo control.

* The transducer fed back the obtained slide displacement to the Comparator.

* The Comparator compares the obtained slide displacement with the applied slide motion and error if any, is fed back to the servo Control unit through an amplifier

* The Servo Control unit sends correct commands to servo motor & the cycle continues.



* The transducer used in the closed loop system is classified into two types.

a) Analog transducer

b) Digital transducer

a) Analog transducer:-

* It produces a variable electrical voltage which varies with the rotational speed of the shaft.

* This voltage can be easily measured and converted into the linear distances moved by the slides.

b) Digital transducer:-

* It converts the rotary motion of machine screw into the countable electric pulses.

* The number of electric pulses indicates the linear distance moved by the table corresponding to the lead screw rotation.

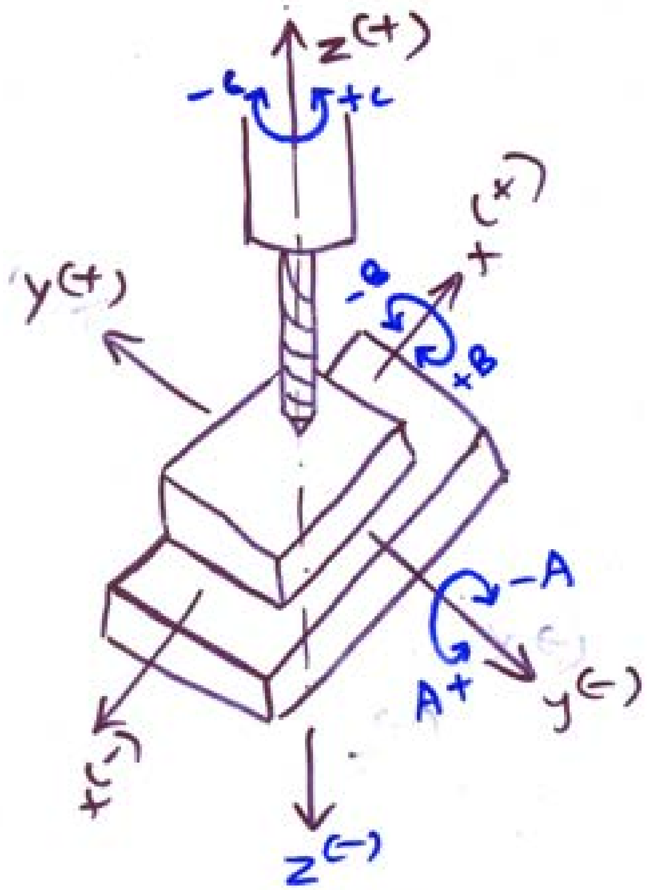
NC machine - Axis of motion:-

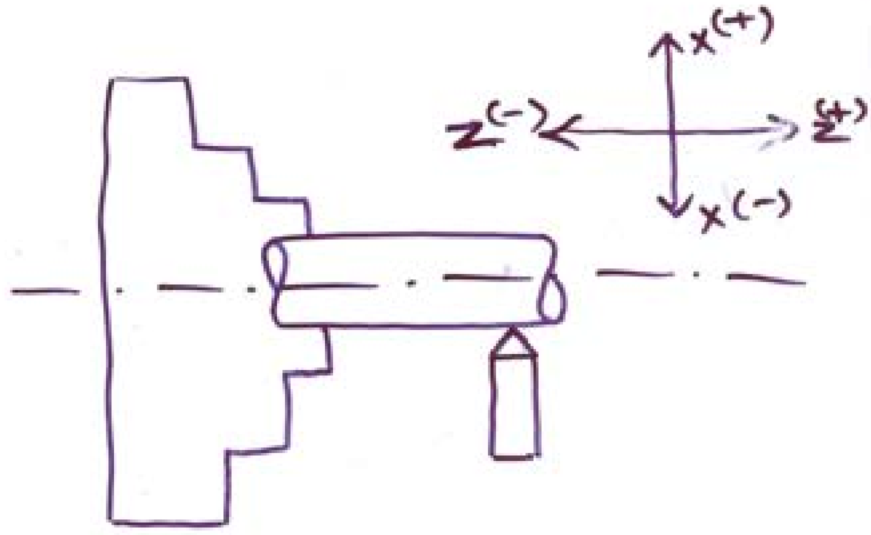
* The location of NC tool at any point in the NC machine is mentioned by Cartesian Co-ordinate System. It consists of x, y, z axes which are mutually perpendicular to each other.

* The NC machine axis of movement follows the right hand rule

* The axes [x, y, z] of motion in NC machine are specified as follows. 10

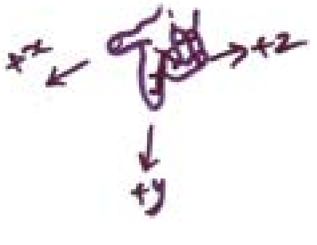
1. z-axis is always spindle axis
2. x-axis is always horizontal axis & parallel to the surface of the work
3. y-axis is perpendicular to both x-axis & z-axis.





CNC milling

CNC turning



* Rotary motion

1. Rotation about an axis parallel to x-axis is "A."

② Rotation about an axis Parallel to Y axis is "B"

③ Rotation about an axis Parallel to Z axis is "C"

* U, V, W axes are Parallel to X, Y & Z axes respectively. ⑪

Classification of NC System:—

① According to the motion Control of tools in NC System.

① Point-to-Point NC System
② Straight cut NC System
③ Contouring NC System

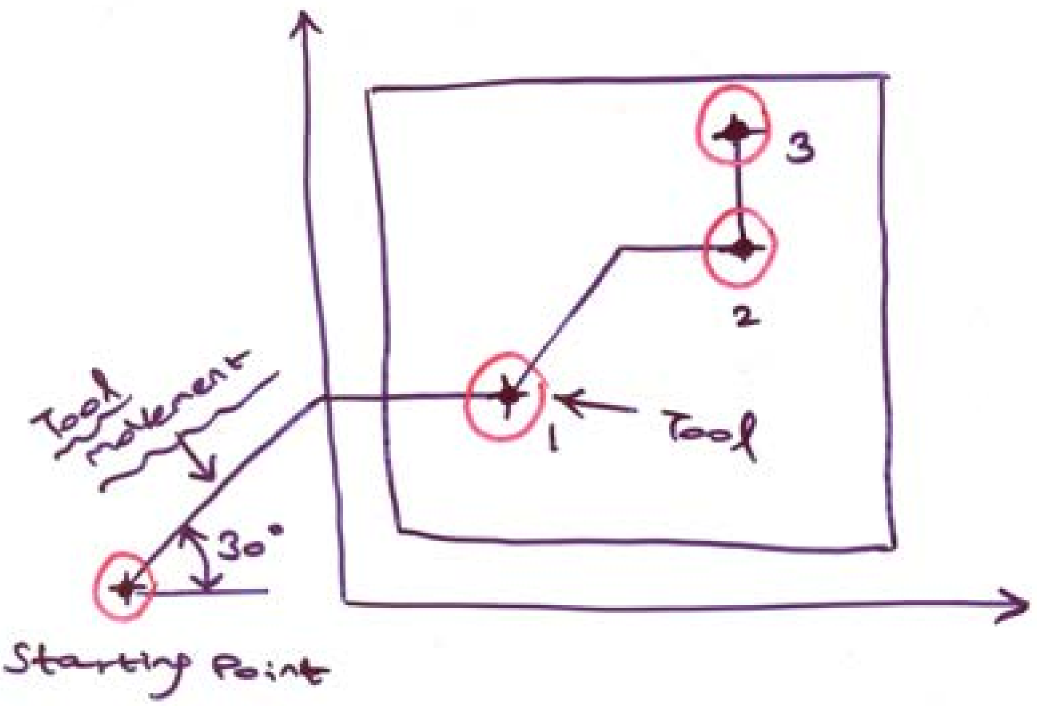
① Point to Point (PTP) System:—

* It refers to the motion of tool at a faster rate to a Point followed by a manufacturing opreation at that Point. The drilling opreation is one of the opreation inwhich PTP System is adopted

* As Shown in figure, the tool is moved from a Starting Point to Point 1 & drilling opern

is done at point 1. Then the tool is moved to point 2 & point 3 followed by the drilling operation at the corresponding points.

(12)



example

NC drilling machine

(2) Straight line (or) Straight cut System:-

* In straight cut NC, the tool moves parallel to one of the major axis at a desired rate suitable for machining.

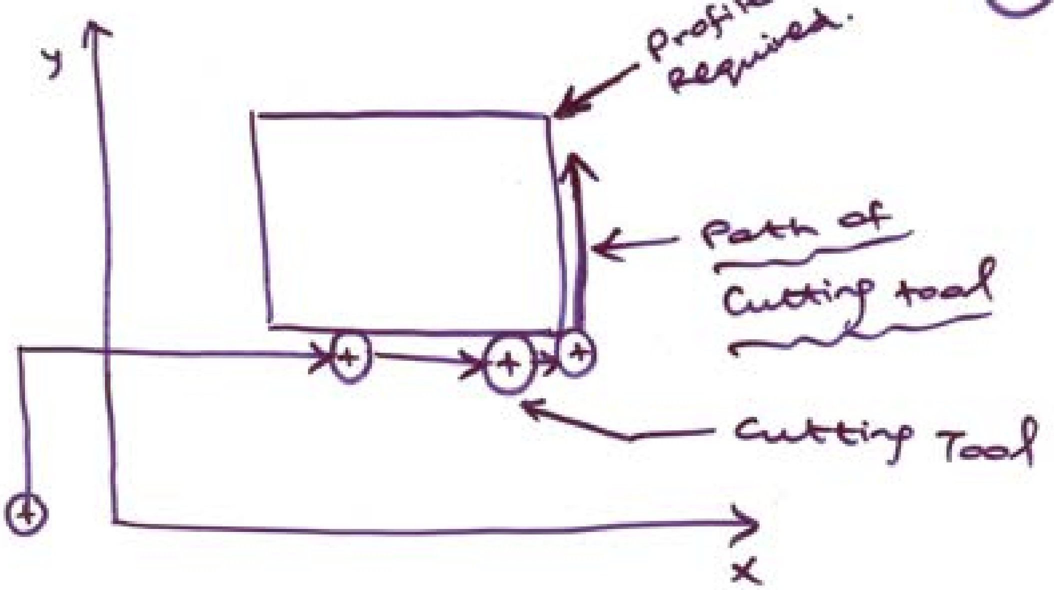
* It is not possible to combine the axis of motion. Hence the tool motion is only along X-axis, Y-axis, Z-axis. Due to this, Angular cuts can not be produced in this type of machine.

* example :-

Milling workpieces of rectangular profile

* Any NC machine tool capable of Straight Cut movement can perform Point-to-Point opreation also.

(13)



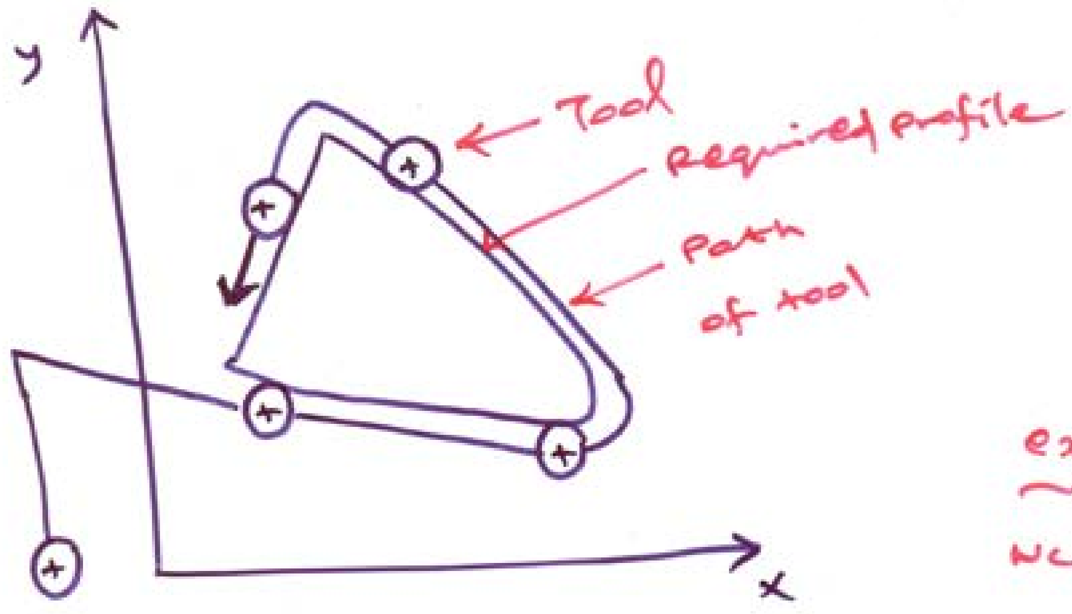
(3) Contouring (or) Continuous NC System:-

* In this type of system, there is a relative motion between the tool & the workpiece.

* All the axis of motion might move simultaneously. Due to this, different curves & Profiles can be cut.

* Actually it is a combination of PTP & Straight Cut NC System.

* An NC machine tool capable of doing Contouring type NC opreation is capable of doing PTP & Straight Cut NC opreation also.



example

NC milling machine

Contour control in NC

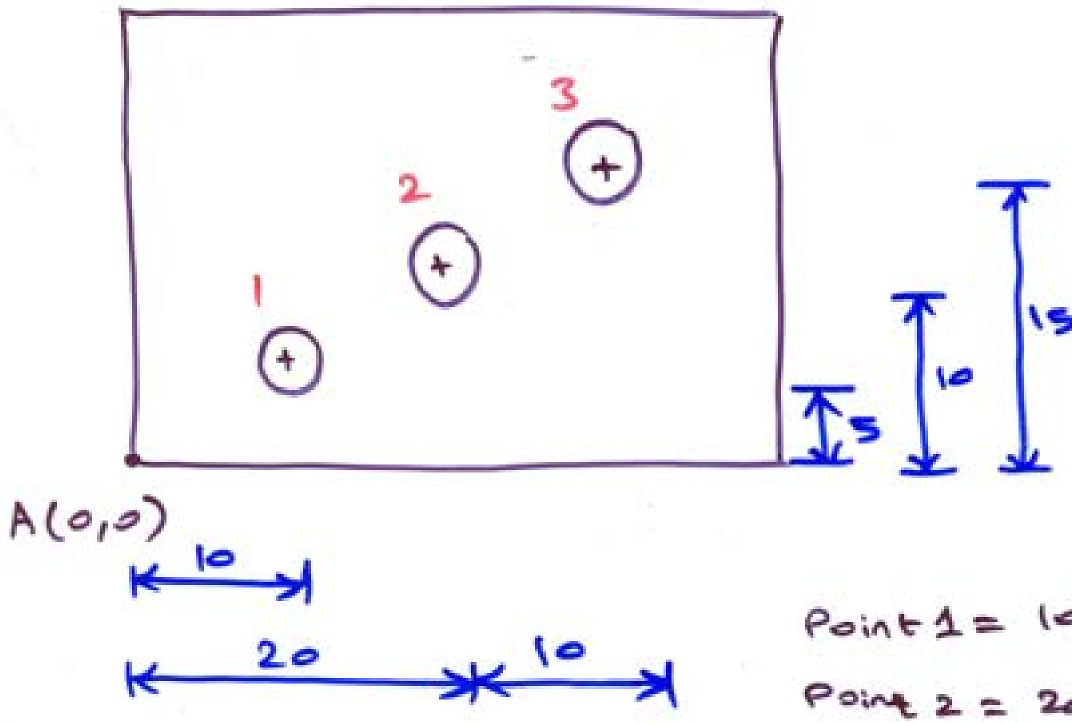
(2) According to the tool Positioning (or) modes of programming

a) Absolute Programming NC System
b) Incremental Programming NC System

a) Absolute System:-

* In this System, all the Position of the tool are measured from the same Zero Point.

* The following figure shows the Positions of the tool measured from the Same Zero Point "A".



Point 1 = 10, 5

Point 2 = 20, 10

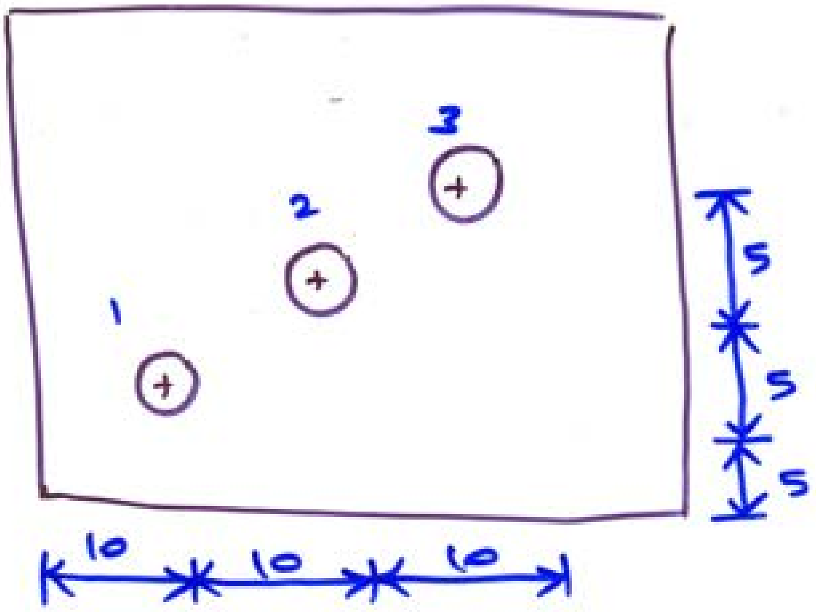
Point 3 = 30, 15

b) Incremental System:-

* An incremental system is one in which the reference point to the next instruction is the end point of the preceding operation.

* The main disadvantage of this system is that if an error occurs into the dimension of any location, all the location marked after that will carry the same error.

* The following figure shows the various position of tool as per the incremental system.



Point 1 = 10, 5 ; Point 2 = 10, 5 ; Point 3 = 10, 5

3. According to the Servo Control System, the NC System is classified into 2 types

a) open loop control System
b) Closed loop Control System

} It was previously discussed.

Zero Point:—

there are two types of zero point in NC machine.

1. Machine zero point
2. workpiece zero point → fixed zero point / → floating zero point.

① Machine Zero Point:-

* It is called as Reference Point. It is set by the manufacturer when the machine is manufactured. It is far away from the Spindle axis.

* Usually after all the opreation is completed on the workpiece, the tool is brought back to the Reference Point.

* Tool changing also is done after the tool is sent to the Reference Point.

② Workpiece Zero Point:-

* It is the origin Point from where all the Position of the tool & work table is defined. It is of two types

ⓐ Fixed Zero point

ⓑ Floating Zero Point.

ⓐ fixed zero point:-

Here, the origin is always Predefined and fixed. Usually it is at lower left corner of the work table.

ⓑ Floating zero point:-

In modern NC machines, Floating zero

concept is provided, which allows the opreator to define this origin. (18)

* In this type of machine, the operator can set the origin where ever he want it to be set on the workpiece.

* The setting of zero point is done manually by the opreator, by bringing x, y & z axes to the point at which the origin is to be defined and by pressing the zero button at that point.

NC Part Programming:—

* It is a set of instructions which instruct the machine tool about processing steps to be performed to manufacture a component.

* There are three types of programming techniques

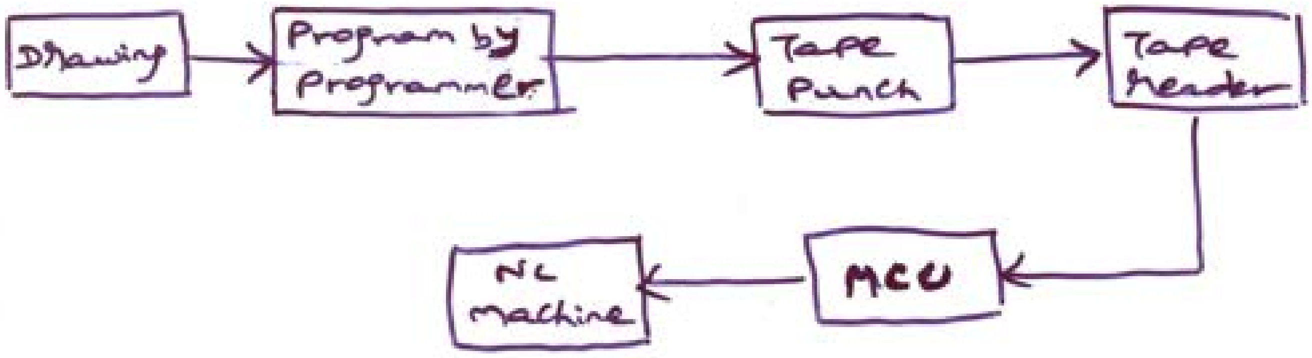
1. Manual part programming
2. Computer-Assisted Part Programming
3. Manual data input.

Manual Part Programming:—

* In manual part programming, the NC part program required for a particular part is written

by a programmer. This part program is typed with a flexo writer where typing causes the typed paper and punched tape to be prepared simultaneously.

19



* In a part program, each line of instruction is called as a block which is composed of one word (or) an arrangement of words. Blocks may vary in length [(ie) they contain variable number of words]. The block ends with end of block (eob) character. Each word is represented by an adress followed by a number.

(ex):— N30 G90 G01 X30.0 Z−20.0 F150 ;

N30	G90	G01	X30.0	Z−20.0	F150	;
word	word	word	word	word	word	eob character

① Sequence number (N code) :—

This code (or) address is used to indicate the block number. The NC part program contains number of blocks. Each block is identified by the block number. N30 ⇒ Block number: 30.

② Preparatory function (G-codes):—

* This codes inform the controller what type of action is to be carried out.

* In general, G-codes is typed at the beginning of block after N-code so that it can set the control for a particular mode when acting on the other words in the same block (or) all other subsequent block.

* G-codes may be modal (or) non-modal.

* For modal type, G-codes specification will remain in effect for all subsequent block unless replaced by another G-code.

* X, Y, Z codes are used to indicate howmuch distance the tool (or) work table along a particular direction.

③ Feed rate (F-code):—

This code is used to indicate the feed rate by which the tool (or) worktable should move along a particular direction. It is in terms of inch/minute (or) mm/minute.

④ Spindle speed (S-code):—

This code is used for the rotation of the spindle at a particular rpm.

⑤ Tool number (T-code):—

It is used to indicate which tool is to be used for a particular operation.

⑥ Miscellaneous codes (M-codes):—

* this codes are used to control the parts of the NC machine

Ex:— N20 M03 S1500; (spindle motor on at a speed of 1500 r.p.m)

N30 M09; (Coolant motor off)

N40 M06 T02; (Tool should be changed by tool number: 2)

Computer Aided Part Programming:—

* In computer aided part programming, much of the computational work needed in manual programming is performed by the computer processor

* In this programming type, the programmer prepares the set of instructions in the high level computer language which is converted to machine tool level program with the help of processor.

Manual Data input:-

* It is a procedure inwhich the part programmer directly keys in the program into the MCU of the machine tool.

* Most of the modern CNC machine is having this facility.

* This facility helps the programmer to change any existing program before the machining opreations.

Advantages of NC System:-

* High Productivity due to less setup & lead time
* Less Scrap

  As human errors are eliminated, Accurate components are produced. Hence Scrap is reduced.

* High quality due to high accuracy.

* Flexibility in design

  In NC System, complicated profile can be easily produced at a faster rate.

* Reduction in inventory
* Safety to the opreator

Disadvantages of NC System:-

* High initial cost
* High maintanenence cost
* Skilled opreator

for part programming, well trained and highly skilled opreator is required.

* The program can not be stored & edited. So for every program, tape punch should be used.

CNC System:-

* CNC stands for "Computer Numerical Control". CNC machine is having computer attached with machine control unit [MCU].
* Here, the part programming can be fed into the computer memory by pendrive (or) tape punch.
* The part programmes which was fed into the CNC machine can be stored & edited.
* The part programmes is converted into electrical signals by the microprocessor in the computer.

The features of the CNC machine:-

① Cathode ray tube which is capable of simulating cutting parameters and show the positions of machine table & the cutting tool before the part is actually loaded on the machine tool. Actual cutting position may also be shown when the part is being machined. The entire program also can be listed on the screen.

(2) Provisions of absolute & incremental programming which are incorporated by G90 (absolute) and G91 [incremental] codes.

(3) Provision of inch (or) metric data input through G70 (inch) and G71 (metric) address.

(4) Availability of manual data input (MDI) to incorporate changes in part programming as & when necessary & lot of programmes can be stored and can be used as & when necessary.

(5) Point-to-point system straight cut system & contouring path system are available in one CNC machine.

(6) Cutter diameter & length compensations are incorporated.

(7) Provision of high volume program & data storage area for future storage & use are incorporated with hard disks.

(8) Use of canned (or) fixed cycle programming to reduce complexity in programming is allowed. 25

(9) Incorporation of provision of subroutine/sub programming and macros.

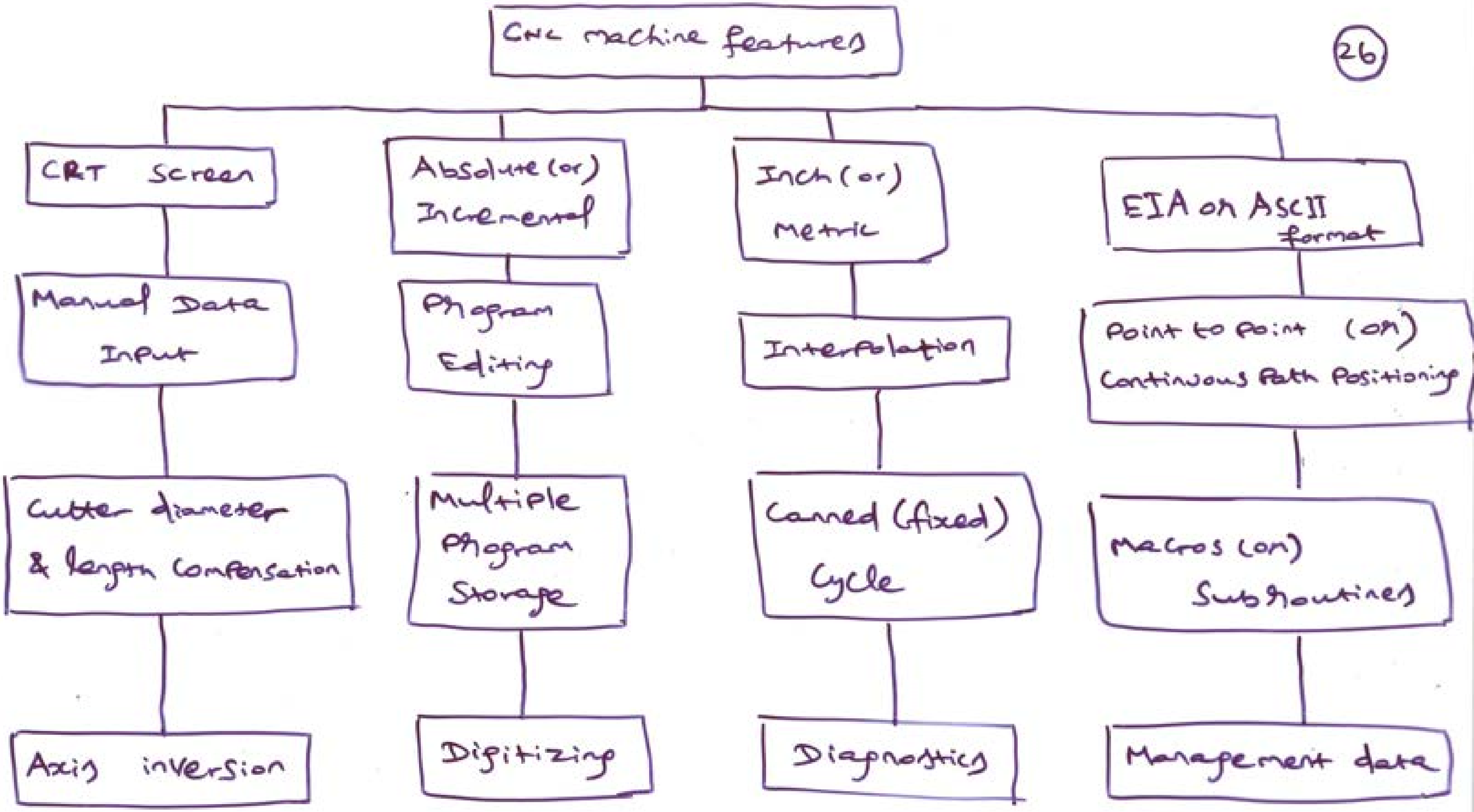
(10) Capability to create axes inversion (mirror image) to produce right (or) left hand part from the same program.

(11) Digitizing to make part programming directly from the existing part.

(12) A program can be loaded in MCU & hence, dependency on tape reader is eliminated.

(13) Adaptable to both EIA or ASCII tape formats

(14) Advanced interpolation methods like helical and cubic make it more versatile. In previous available NC systems only linear, circular and parabolic interpolators are available


CNC machine features
CRT Screen
Manual Data Input
Cutter diameter & length Compensation
Axis inversion
Absolute (or) Incremental
Program Editing
Multiple Program Storage
Digitizing
Inch (or) Metric
Interpolation
Canned (fixed) Cycle
Diagnostics
EIA or ASCII format
Point to Point (or) Continuous Path Positioning
Macros (or) Subroutines
Management data

Direct Numerical control:—

* It is a manufacturing System in which a number of machines are controlled by a Central computer through a direct connection of telecommunication lines.

* Instead of using a tape reader as in NC machines, the part program is transmitted to the machine directly from the computer memory. One Computer can control more than 200 Separate machines.

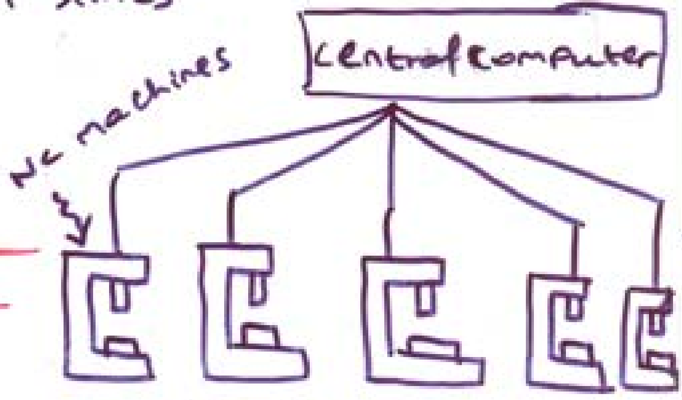
* The Direct Numeric control System consists of four components

① Control computer

② Bulk memory which Stores NC Part Program

③ Tele communication lines

④ Machine tools



Advantages of CNC machines:—

Advantages of CNC machines are Similar to NC machine. Some additional advantages due to additional feature in CNC machine are as follows.

① Program Storage

As computer is available, multiple programs can be stored in the machine.

② Reliability of System :-

As the data is directly entered with the help of computer, no need to use punched tape. It improves the reliability of the system.

③ Online Part Programming :-

The part program can be done online with editing if required.

④ Metric (or) inches :-

The program can be written in metric (or) inches through G70 (inch) & G71 (metric) address.

⑤ Interpolations :-

In NC System, there is interpolation for straight and circular path, but in CNC, helical, parabolic and cubic interpolation also can be made.

⑥ Tool Compensation :-

For the purpose of tool offset & tool wear, (29)
cutter diameter & its length compensation are available in CNC machine

(7) Axis inversion:-

Capability to create axes inversion (mirror image) to produce right (or) left hand part from the same program.

(8) canned (fixed) cycle:-

Use of canned (fixed) cycle programming to reduce complexity in programming is available.

Disadvantages of CNC machine:-

1. High initial cost
2. High maintanence cost.
3. Costly control system.
4. Need of skilled opreator for part programming.
5. whole machine will stop functioning if there is any problem in computer
6. Costly software

Advantages of DNC System:—

1. Control of more than one NC machine
2. Elimination of Punched tape & tape reader
3. Convenient Storage of NC Part Programs in computer files.
4. Greater computational capability & flexibility
5. The data for tools & cutters can be centrally maintained & updated.
6. The data related to manufacturing can be effectively collected & hence inventory can be better controlled

Disadvantages of DNC System:—

1. If the central goes down, all machines becomes inactive
2. Initial cost is too high.

Comparison between NC, CNC & DNC System:- (31)

Sl.no	Parameters	NC	CNC	DNC
1.	Flexibility	Less	High	High
2.	Tape editing	Not possible	Possible	Possible
3.	Productivity	Less	High	Highest
4.	Number of programs stored	only one at a time	multiple programs	multiple programs
5.	Number of operations done at a time	one	one	multiple
6.	Initial cost	Low	High	Highest

Various NC machines:-

1. Single spindle drilling machine
2. Lathe
3. Milling machine
4. Turning centers
5. Machining centers
   - (a) Horizontal machining center
   - (b) Vertical machining center

(1) Single spindle drilling machine :-

* It is the most simple NC machine.
* It is programmed on three axes.

X-axis → Table movement to the right (or) left of the column

Y-axis → Table movement towards (or) away from the column

Z-axis → Up & down movement of the spindle.

(2) Lathe :-

* Machine tool to manufacture & machine round parts
* It is programmed on two axes.

X-axis → Cross motion of the cutting tool.

Z-axis → Carriage travel towards (or) away from the Head Stock

(3) Milling machine :-

* Accepted → one of the most versatile machine tools used in the manufacturing industry.

* It is programmed on three axes.

X-axis → Table movement left (or) right of the column

Y axis → Table movement towards (or) away from the column.

Z axis → up & down movement of the spindle

4 Turning Centers:—

* Turning centers are more accurate and productive than engine lathe & has a higher spindle rate.

* It is having Turret tool head which can accomdate 12 Tools and as such can produce wide range of components without tool change.

* It is programmed for two axes.

X-axis → Cross wise movement of the Tool

Z-axis → Movement of the Tool towards (or) away from the Head stock.

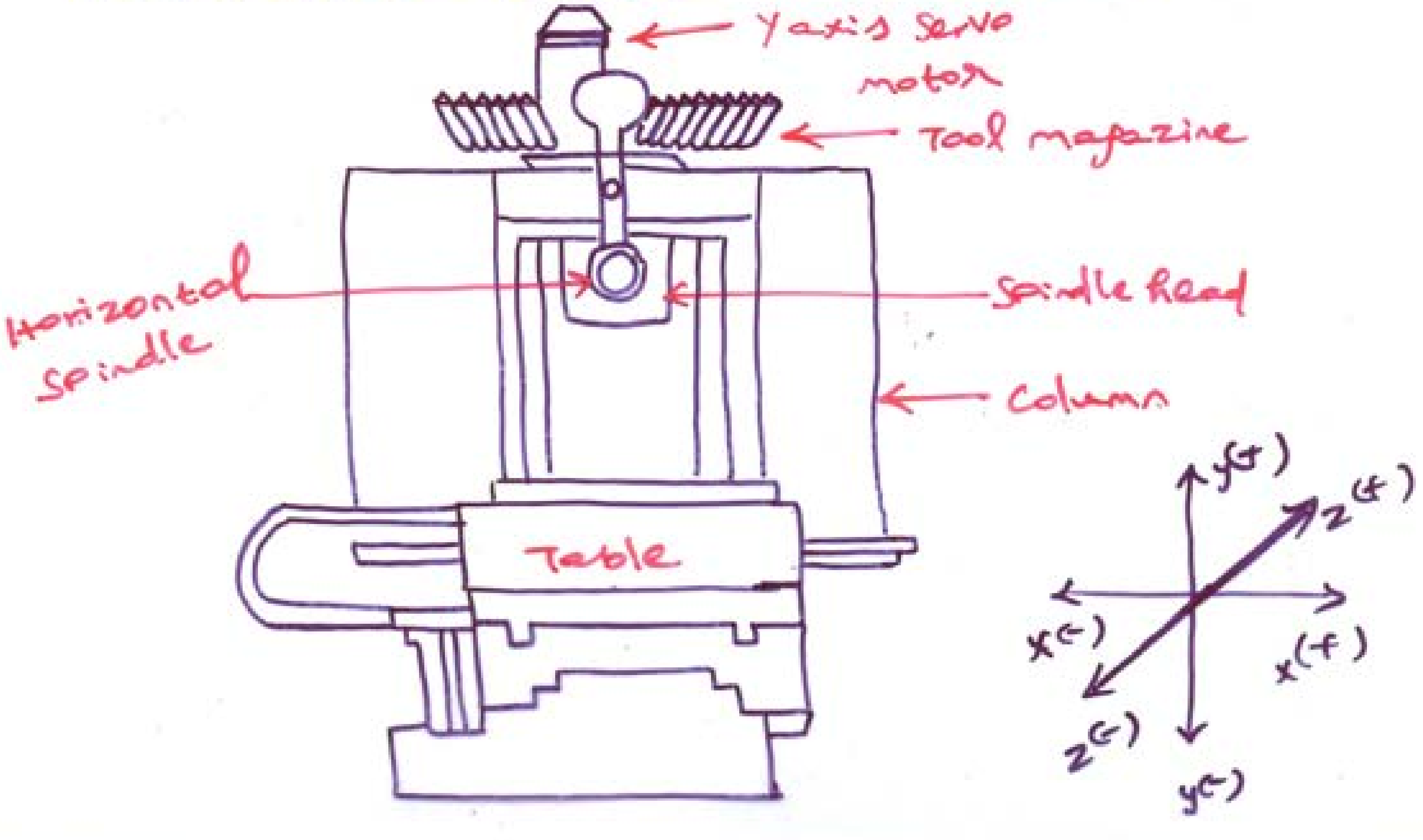
(5) NC machining centers:—

Machining centers are of two types

(1) Horizontal machining center (HMC)
(2) Vertical machining center (VMC)

Machining center are capable of a variety of machining opreations. For this reason, a variety of tools are required in a machining center. thus a machining center must have Automatic tool changer (ATC), Automatic pallet changer apart from other parts.

(1) Horizontal machining centers (HMC):—



* Horizontal machining center (HMC) has horizontal spindle head. Here the following movements are there

(35)

① X axis for table movement

② Y axis is for vertical movement of the spindle

③ Z axis is for horizontal movement of the spindle perpendicular to X & Y axis.

* In this machining center, tool magazine which holds lot of tools is there and Automatic tool changer is there. After completing one operation, Automatic tool changer remove the old tool from the spindle head & fix the new tool into the spindle head.

* Horizontal machining center may be fixed column type (or) travelling column type.

* Fixed column type carries a removable table called as pallet

* After machining, a pallet with a component is removed & another pallet which was already loaded with the workpiece is then clamped

on the pallet receiver & machined.

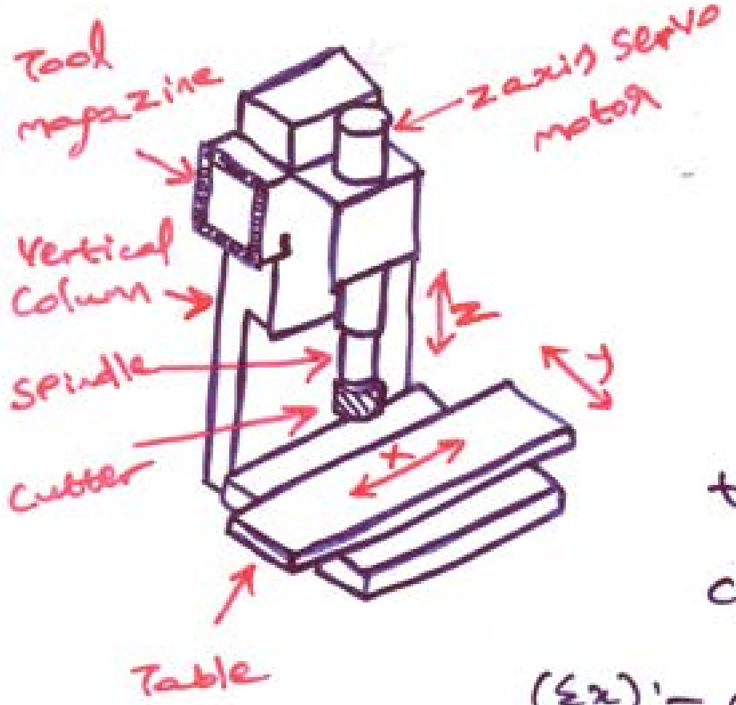
* In travelling column type machining center, there are two tables. After machining, components are on one table and column travels to the second table which carries a fresh workpiece for machining.

* Components which is to be machined is unloaded from the first table & a fresh workpiece is loaded on it.

* Machine comes under this category
    1. CNC Horizontal milling machine
    2. CNC Grinder. etc.,

Vertical machining center:- [VMC]

* these machines are generally single spindle machine.
* Vertical machining centers has
    1. x-axis control for the table movement left (or) right
    2. y-axis control for the table movement towards (or) away from the column
    3. z-axis control for the vertical movement of the spindle



* It carries a vertical spindle head which can slide along a vertical guide ways provided on the column. (37)

* It has tool magazine, tool transfer arm, Automatic Pallet changer etc.,

(Ex):- CNC vertical milling machine, CNC drilling machine etc.,

Advantages of machining center:-

1. Machining centers have high metal removal rate capabilities
2. Machining centers are highly versatile and increases productivity.
3. It is more flexible and economical than conventional machines.
4. It is mainly used for mass production.

Automatic Tool Changer (ATC):-

In order to reduce the time for changing over of tools, machining centers are provided with automatic tool changing facility. It is done by

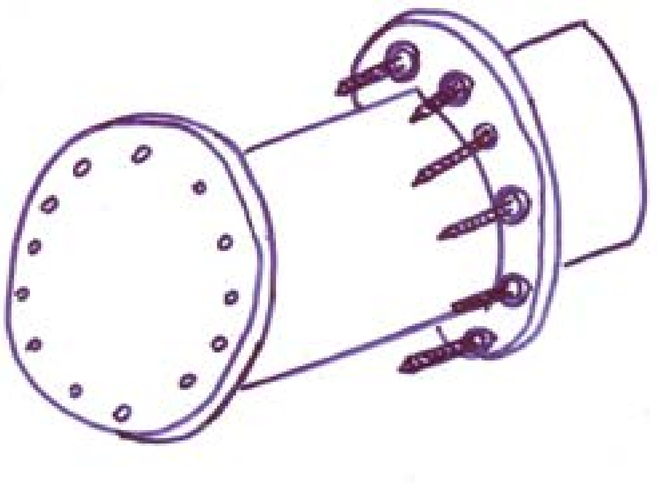
Automatic tool changer (ATC) unit of the machining center. The ATC consists of the following two units

(1) Tool magazine (2) Tool transfer arm

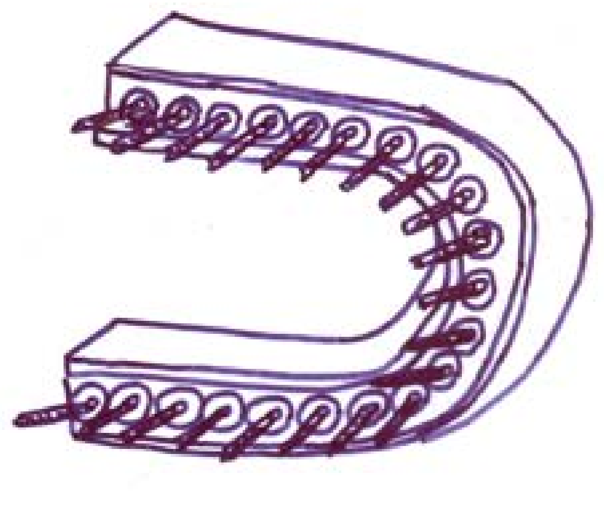
Tool magazine:-

It is a tool storing unit, attached to the machining center. It contains number of pockets (from 10 to 400) to hold number of tools. Each tool is inserted into the pockets and numbered. To select the required tool, in the part program these numbers are denoted.

The tool pockets are arranged in drum (or) chain. Based on that, it is classified as "drum type tool magazine" and "chain type tool magazine".

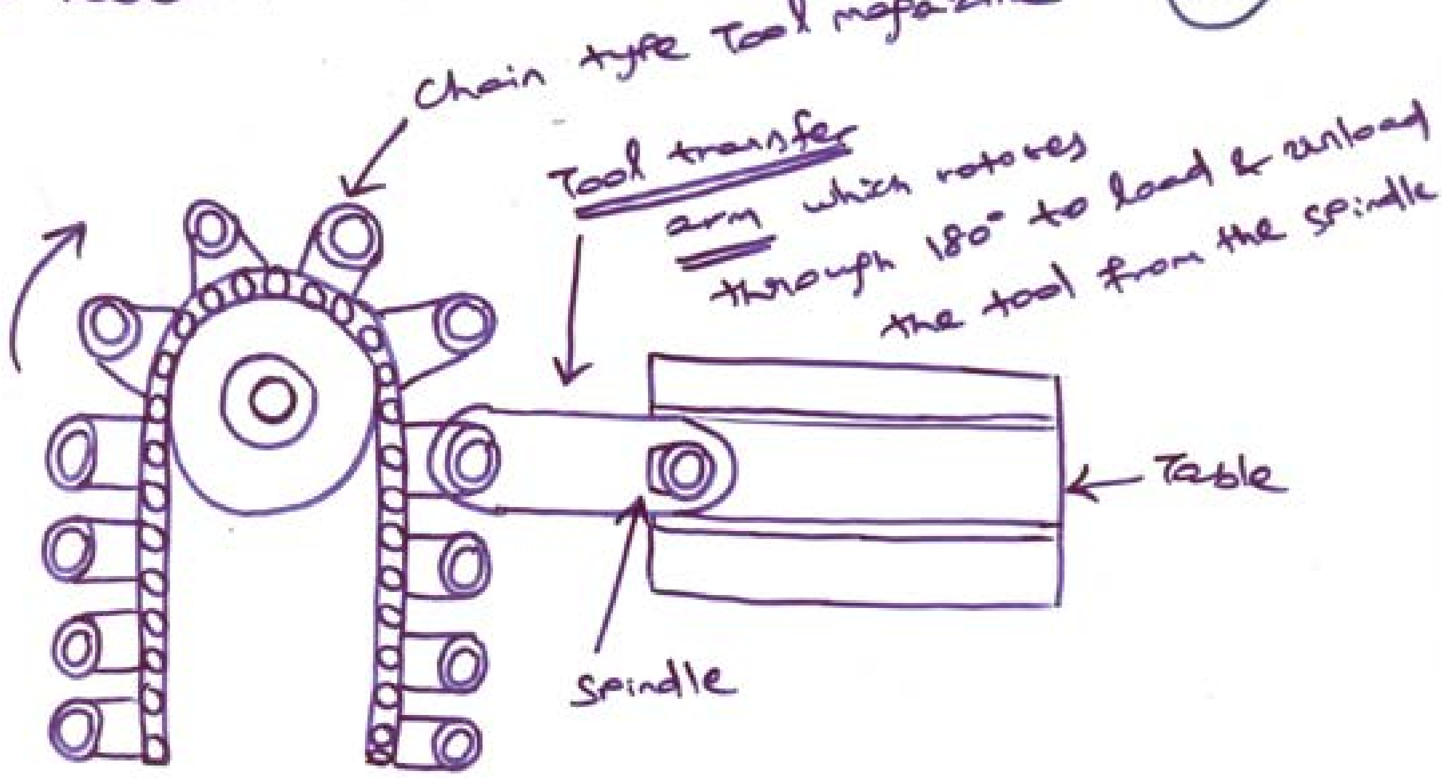


a) Drum type tool magazine



b) Chain type tool magazine

Tool transfer Arm:-



* Tool transfer arm is used to pick the tool from the tool magazine & the spindle and fix the new tool to the spindle & fix the old tool to the tool magazine by rotating through 180°.

* As per the part program instruction, after one operation is over, the magazine indexes to the next tool position. then one end of the tool transfer arm picks the required tool for next operation from the tool magazine and the another end of the arm picks the old tool from the spindle. Then it will rotate through 180° & fix the new tool to the spindle & old tool to the tool magazine

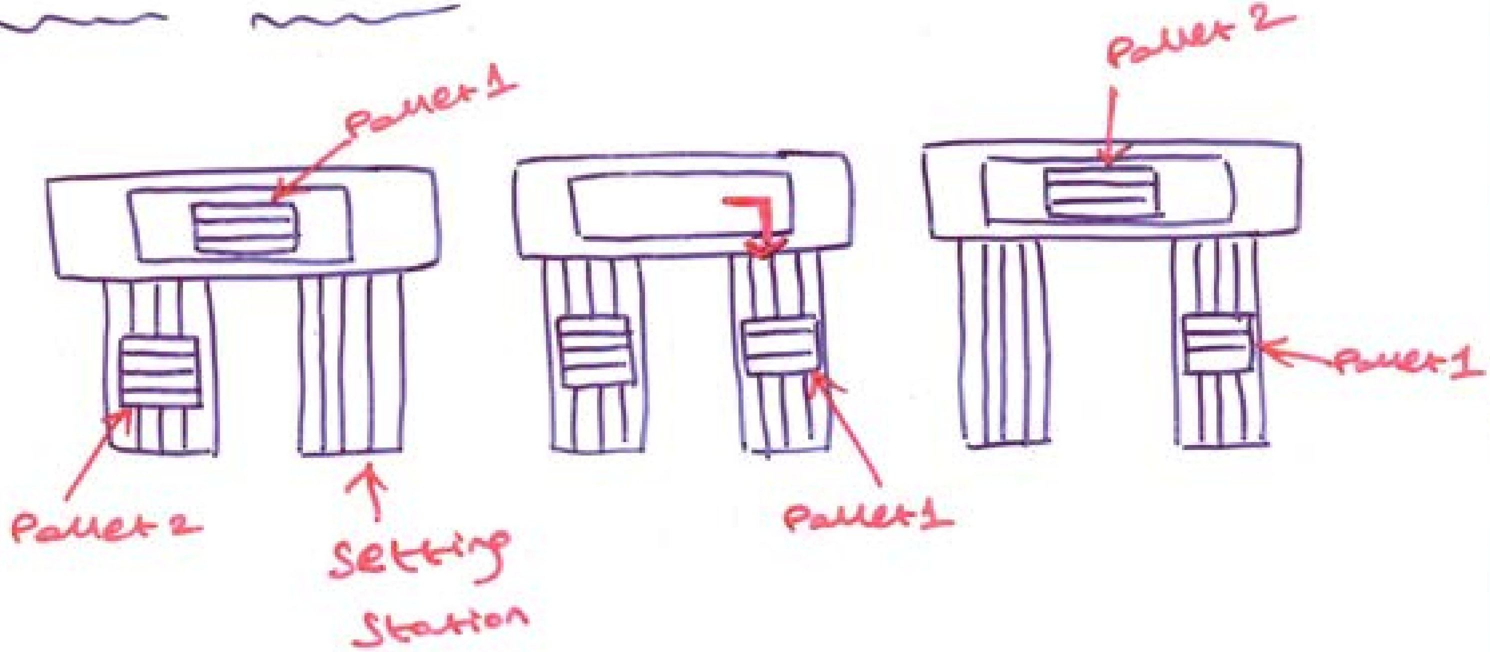
Automatic Pallet Changer:—

* Machine downtime because of loading-unloading, clamping-releasing of the workpiece can be minimized with the help of automatic workpiece loader/unloader System.

* In this System, the workpieces are mounted on the pallet and the pallets are moved around the machine in a logical manner. This System is called as "Pallet Changing System".

* According to the logical movement of the pallet, the System can be linear (or) rotary.

(i) Linear motion:—



(a) (b) (c)

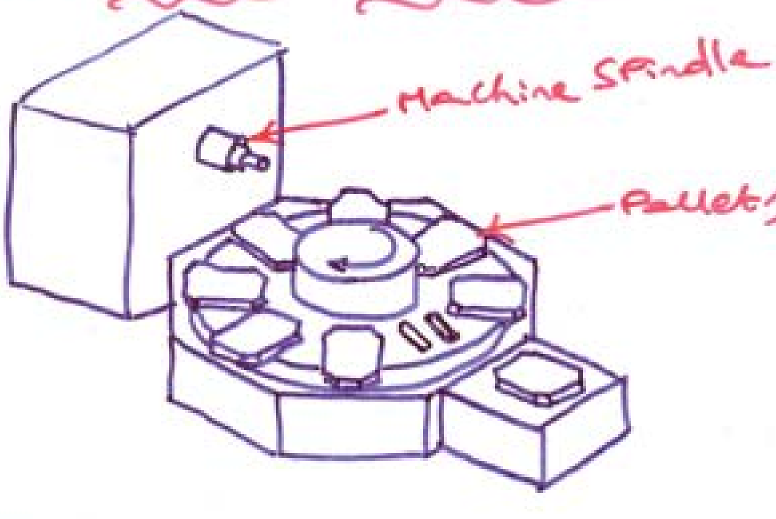
* In the fig (a), the workpiece on the left side track is waiting for completion of machining opreation of earlier workpiece.

(41)

* In the fig (b), after completion of earlier workpiece it moves onto the unloading table and the next component is ready to move onto the machining table.

* In the fig (c), the next component moves onto the machining table and this process continuous.

Rotary Pallet Changer System:—



* In this pallet changer system, the number of pallets are clamped on the rotating table.

* For the machining purpose of the workpieces, the table is moved in rotary motion with the help of indexing mechanism.

Canned cycle (or) Multiple cycle (or) fixed cycle:-

A canned cycle is a combination of tool movements that performs any one particular machining function such as drilling, boring, turning etc., this cycle automatically generates multiple movements from a single block. (42)

Ex:- Multiple turning clcyle & finishing cycle

G71 U_ R_ ;
G71 P_ Q_ U_ W_ F_ ;
G70 P_ Q_ F_ ;

where,

U → Depth of cut for each pass along x-axis
R → Retraction amount in x axis.
P → Starting block; Q – Ending block;
U → Stock in x axis
W → Stock in y axis
F → feed rate

G71 → Multiple Rough turning clcle
G70 → Multiple finish turning clcle

Subroutine (or) Sub Programe:-

when a component has repetitive pattern machining at different places, in stead of writting the same program each & every time, the Program for the repetitive opreation is written & stored in a separate file & is called whenever needed. This program is called as "Sub-programme".

Syntax for Sub Program

M98 0011234;

M99;

where,

M98 - Sub Program call;

001 - No of repeats;

1234 - Sub Program number;

M99 - Sub program End.

Interpolation:-

It is the method of specifying the Path to be generated for machining. There are the following types of Interpolation.

(1) Linear Interpolation (2) Circular Interpolation

(3) Helical Interpolation (4) Cubic Interpolation.

⑤ Parabolic Interpolation.

① Linear Interpolation :-

* It is used when a straight line path is to be generated on the workpiece of NC machine.

* The linear path can be Horizontal, Vertical (or) Inclined.

* The G-code — G01 is used for linear interpolation & four parameters [x, y, z & f] are required to specify G01 code.

Syntax for G01 :-

G01 X_Y_Z_F_;

where

x = Value for x co-ordinate

y = Value for y co-ordinate

z = Value for z co-ordinate

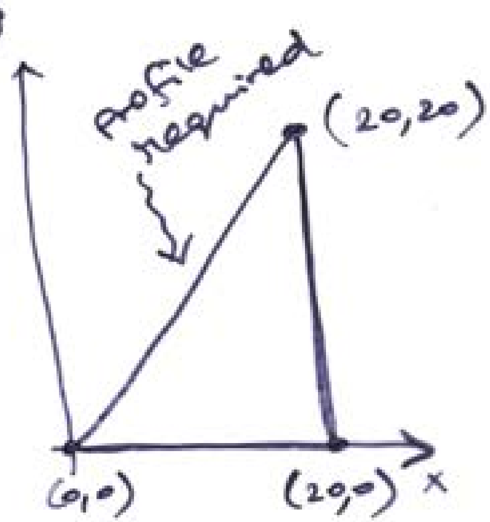
F = feed rate

example :-

G01 X20.0 Y0.0 Z-1.0 F1.5;

X20.0 Y20.0 Z-1.0;

X0.0 Y0.0 Z-1.0;



2) Circular Interpolation:-

* The movement of the tool along a circular path is called as "Circular Interpolation".

* Five parameters are required for specifying the G-code used for circular interpolation [G03 (or) G02]

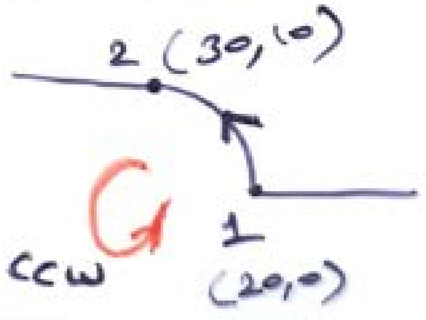
* G03 code is used for making the circular path in counter clockwise direction and G02 code is used for making the circular interpolation in a clockwise direction

* Syntax for G03/G02 code:-

G02/G03 X_ Y_ Z_ R_ F_ ;

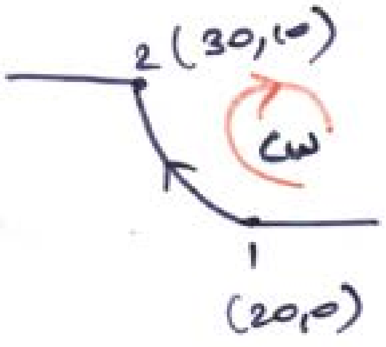
where
x, y, z → co-ordinate values;
R → Radious of the Arc;
F → feed rate in mm/rev (or) mm/minute

example:-

1)


G03 X30.0 Y10.0 Z-1.0 R10.0 F1.5;

(2)



G02 X30.0 Y10.0 Z-1 R10.0 F1.5;

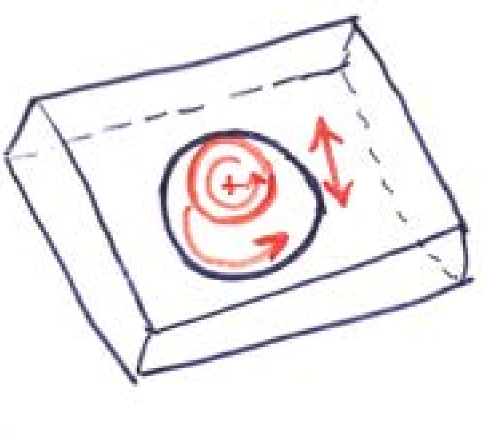
(3) Helical Interpolation:-

* The movement of the cutting along a Helical path is called as "Helical Interpolation".

* To make this interpolation happens the cutting tool is moved simultaneously along 3 axis X, Y & Z.

* It is mainly used for making Aero parts.

* By means of Helical interpolations any bigger dia hole can be made with smaller diameter tool with less torque & less power.



Producing a bigger hole with a smaller dia tool

Slide movement in CNC machines:-

Precise positioning and repeatability of machine tool slides are the major functional requirements of CNC machines. A plain slide way will give stick-slip. To prevent this, there are different slide way systems are used. These have low wear, negligible stick slip, good damping capability, low co-eff of friction properties.

Requirement of a good slide way system:-

1. Low co-eff of friction
2. Low rate of wear
3. Must have good damping capacity
4. Must give a smooth drive.
5. There must be no stick-slip.

In CNC machines, the following slideways type may be used.

a) Plastic inserted slide ways

b) Recirculating ball screw & nut type slide ways.

a) Plastic inserted slide ways

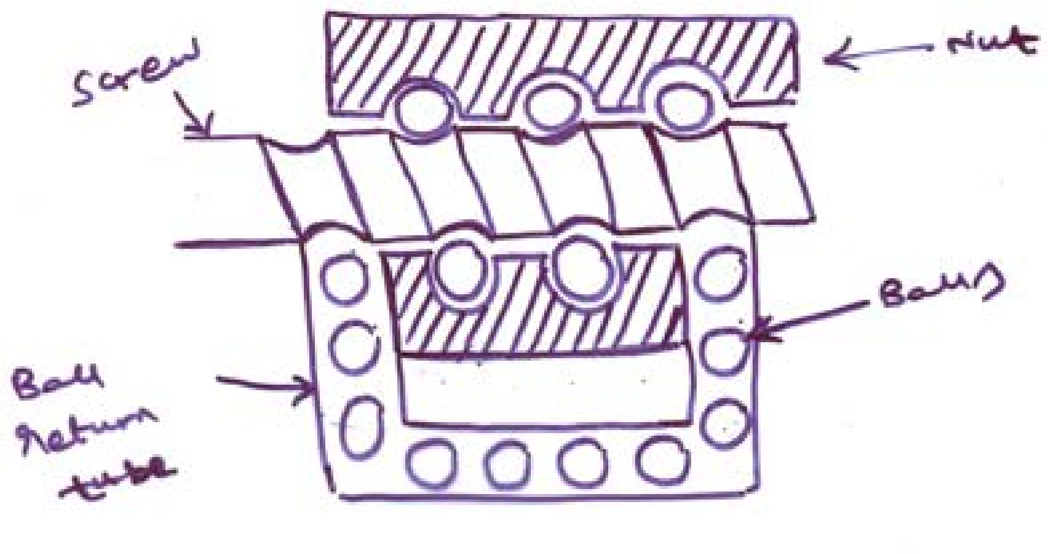
In this slide ways, plastic inserts are bonded to the underside of the sliding members. The inserts can be thermo plastic (or) thermo setting types. For these coated slide ways, the friction is less.

Advantages:-

1. Has less co-eff of friction.
2. Increase in strength
3. Good wearing quality
4. Has self lubricating properties.
5. Worn out inserts may be replaced

b) Recirculating Ball Screws & nut type Slide ways

* Recirculating Ball screw & nut arrangement are used to transmit motion to the slides.



* It consists of screw thread which acts as ball race to hold steel balls. The balls are surrounded by a nut. 49

* The balls rolling in the grooves exit from the trailing end of the nut and are picked up by return tube inserted from outside and are recirculated into the leading end of the nut.

* The rotation of the ball between the screw & the nut moves the slide attached to the nut.

* Here the sliding contact between the screw & nut is changed into the point contact due to the steel ball. So, the friction & hence wear is reduced.

Advantages:-

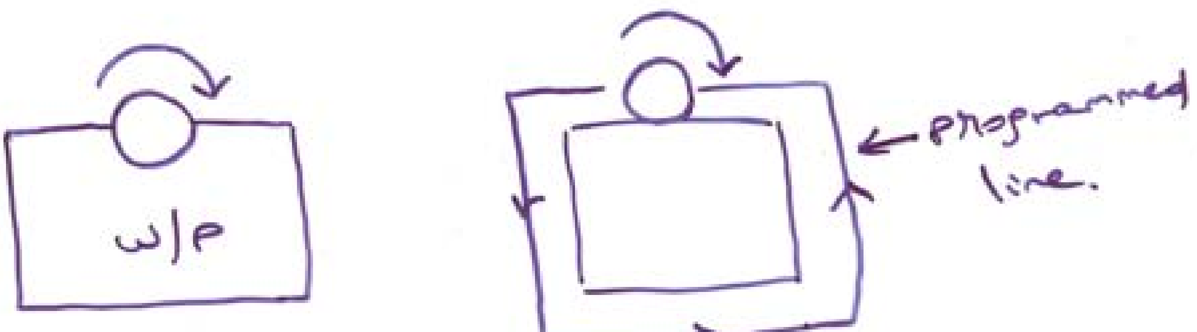
1. Very low co-eff of friction
2. Higher transmission efficiency
3. No stick-slip.
4. Backlash between the parts are eliminated by preloading the assembly.
5. Reduced friction & hence wear.

Tool offset (or) compensation:—

* the word offset refers to the allowance made by the CNC machine for the radious & length of the tool to cut the job. Programming on a CNC machine is always done according to the center point of the cutter. If the offset is not set, the tool will follow a wrong path.

* Since the diameter & length of a tool may vary, an offset value needs to be set so that the tool can be moved to the correct position for the cutting required.

* For example, if you were using a 10mm cutter the workpiece would be reduced by 5mm on each edge (or) 10mm overall. To overcome this the, the radious of cutter is given as a compensation.



no offset

offset is given
— cutter is moved on the programmed line

Tool length offset (or) Tool length compensation :-

Normally, part programmes are prepared for a particular tool length the variation if any in a tool is given in the form of tool length offset and it is the difference between the programmed length and actual length of the tool. The same will be taken into account by the machine during operation.

example:-

51

G43 H1 Z10;

* H1 refers to tool length compensation of tool one, which has a value of -3.000 in the offset table.
* A minus value increases the tool length whereas the positive value decreases the tool length.

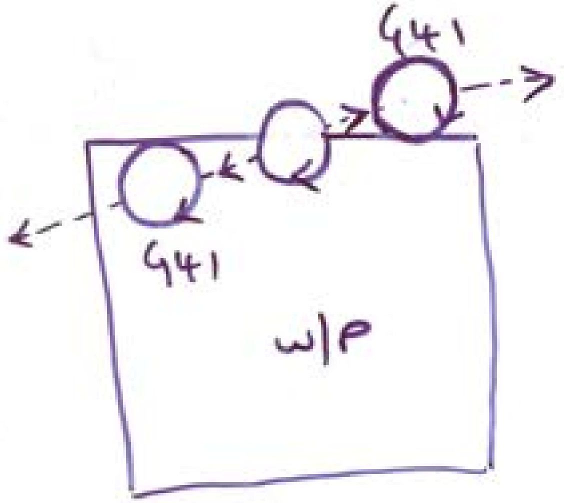
Tool radious (or) diameter compensation:-

Here, the tool radious (or) diameter is given as a compensation value

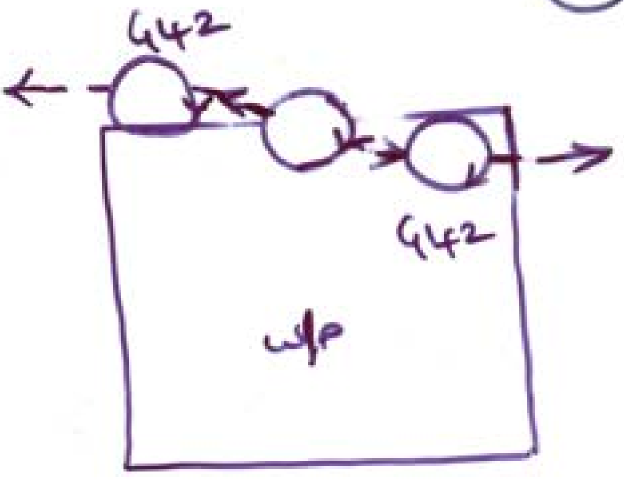
G41 – Cutter radious compensation to the left

G42 – Cutter radious compensation to the right

G40 – cancels the cutter radious compensation.

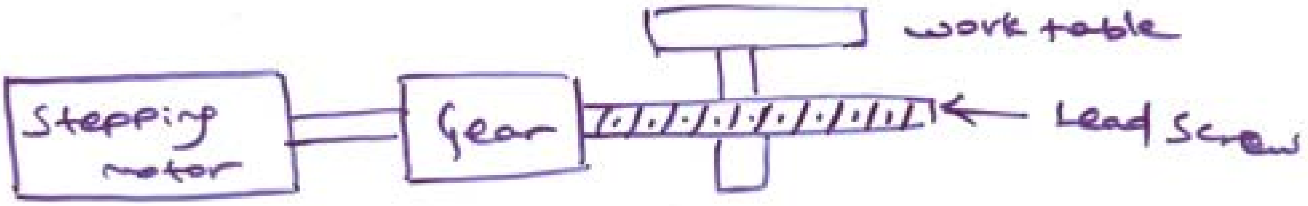


Codes for left cutter compensation



Codes for right cutter compensation

Formulaes used for calculating the slide movement :–



open loop System

Each Pulse makes the Stepping motor to rotate at an angle (or) a fraction of revolution

① Step angle, $\alpha = \frac{360}{N}$; $N \rightarrow$ No of Pulses required for 1 revolution of Stepping motor.

Let $n \to$ Total no of pulses received by stepping motor

(53)

(2) ∴ Total angle made by the stepping motor $\Big\}$ $A \Rightarrow n \times \left(\frac{360}{N}\right)$

(3) no of revolution $= \frac{n}{N}$ $\left[\begin{array}{c}\text{for 1:1 gear ratio} \\ \text{between lead screw} \\ \text{\& motor shaft}\end{array}\right]$

(4) If the pitch of the lead screw is P (in/rev), then the distance made by the worktable (x) axially

$$x = P\left(\frac{n}{N}\right)$$

(5) Pulse frequency (f) in pulses/sec is given by

$$f = \frac{N \times RPM}{60}$$

(6) Speed of the worktable, $\boxed{v = P \times RPM}$ in/min

Problem :-

(1) A stepping motor has $N=150$, $P=0.2''$/rev; If $n=2250$ pulses, what is the distance travelled in x direction? what should be the pulse frequency for a travel speed of 16 in/min?

Soln:-

54

① Distance travelled by the worktable axially $\Big\}\; x = P\left(\frac{n}{N}\right)$

$$= 0.2\left(\frac{2250}{150}\right)$$

$$\boxed{x = 3''}$$

② Pulse frequency, $f = \frac{N\,(RPM)}{60}$

$$= \frac{150 \times \left[\frac{v}{P}\right]}{60} = \frac{150 \times \left(\frac{16}{0.2}\right)}{60}$$

$$= \frac{150 \times 80}{60} = 200\ Hz$$

—— x —— x —— x —— x —— x —— x —— x ——

② A stepping motor of 200 steps/revolution, is mounted on the lead screw of a drilling machine. If the pitch is 0.1"/rev & f = 2000 Hz, what is the speed of the table?

Soln:-

$N = 200$; $P = 0.1''/rev$; $f = 2000\ Hz$; $RPM = ?$

$$f = \frac{N\,(RPM)}{60}$$

$$\therefore RPM = \frac{60 \times f}{N} = \frac{60 \times 2000}{200} = 600\ rpm.$$

Problems in a closed loop System

(3) Consider a CNC worktable driven by a closed loop control system consists of a servomotor, lead screw & optical encoder. The lead screw has a pitch $p = 0.2''$ and is coupled to the motor shaft with a screw to motor gear ratio of 1:4. The encoder generates 150 pulses/revolution of the lead screw. If the no of pulses & pulse rate received by the control system are 2250 & 200 Hz respectively, calculate

a) Table speed b) motor speed in r.p.m

c) Distance travelled by the table

Soln:—

$N = 150$ pulses/rev; Pitch of lead screw $\} = p = 0.2''/\text{rev}$

$n = 2250$; $f = 200\text{ Hz}$;

(i) RPM for table & motor = ?

(ii) Distance by table axially, $x = ?$

$$f = \frac{N\,(\text{RPM})}{60}$$

$$\therefore \left.\begin{matrix}\text{RPM}\\ \text{of the table}\end{matrix}\right\} = \frac{60 \times f}{N} = \frac{60 \times 200}{150} = 80\text{ rpm.}$$

$\therefore$ RPM of the motor shaft $\Big\} = 4 \times 80$

$= 320$ rpm.

Distance by the table axially $\Big\}$ $x = \left(\frac{n}{N}\right) \times P$

$$= \left(\frac{2250}{150}\right) \times 0.2 = 3''$$

$$\boxed{x = 3''}$$

NC Part Programming

Important G-codes

(57)

① G00 – Rapid traverse :–

Syntax :– G00 X Y Z (Milling)

G00 X Z (Turning)

② G01 – Linear Interpolation :–

Syntax :– G01 X Y Z F (Milling)

G01 X Z F (Turning)

F → Feed rate

③ G28 – Go to Reference Point

Syntax :– G28 U0.0

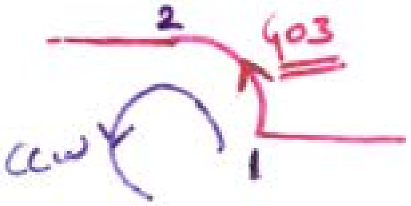
G28 W0.0

U → Go to reference (X)

W → Go to reference (Z)

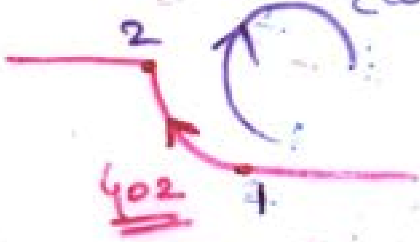
④ G03/G02 – Spindle on CCW & CW

Syntax :– G03/G02 – X Y Z R F



R → Radius

F → Feed rate



G90 – Absolute Dimensioning

G91 – Incremental Dimensioning

G20 – Imperial Units

G21 – Metric Units

G98 – Feed / minute

G99 – Feed / revolution.

M codes:–

M04 / M03 – Spindle on CCW & CW respectively

M05 – Spindle Stop.

M06 – Tool Change

M07 – Coolant on

M09 – Coolant off

M30 – Program Stop and Return

M98 – Sub Program Call

M99 – End of Subprogram

Multiple/canned cycles:-

① Turning cycle:-

Block ① G71 U _ R _ ;

Block ② G71 P _ Q _ U _ W _ F _ ;

Block ③ G70 P _ Q _ F _ ;

where,

G71 – Rough turning cycle

G70 – Finish turning cycle

Block ①

U → Depth of cut for each Pass.

R → Retraction amount at the end of each Pass.

Block ②

P → Block number for Start of opr n

Q → Block number for End of opr n

U → Stock in X axis

W → Stock in Z axis

F → Feed rate

* Feed rate for finishing cycle should be less.

Grooving cycle:-

G75 R 0.5

G75 X40 Z-10.0 P500 Q1500 F0.05

where

R → Retraction amount

P → Depth of cut - Peck amount (Microns)

Q → Movement along z direction (microns)

F → Feed in mm/rev

Drilling cycle:-

G74 R0.5

G74 Z-50.0 X0.0 Q500 F0.2

where

R → Retraction

Q → Depth of cut - Peck amount [Microns]

F → Feed in mm/rev

Single threading cycle:-

G92 X23.8 Z-50 F1.2

G92 X23.6

G92 X23.2

G92 X23.0

G92 X22.8

G92 X22.6

G92 X22.4

Thread = M24 X 1.2

where
P = Pitch value.

Depth of thread = P × 0.649

Minor dia = Major diameter − (2 × Thread depth)

Multiple facing Cycle:-

Block:1 G72 W0.25 R1.0 ;

Block:2 G72 P10 Q11 U0.0 W 0.5 F0.5;

Block:3 G70 P10 Q11 F0.25;

N10 G00 Z0.0;

N11 G01 X0.0;

Block:1

W → Depth of cut along Z axis

R → Retraction amount

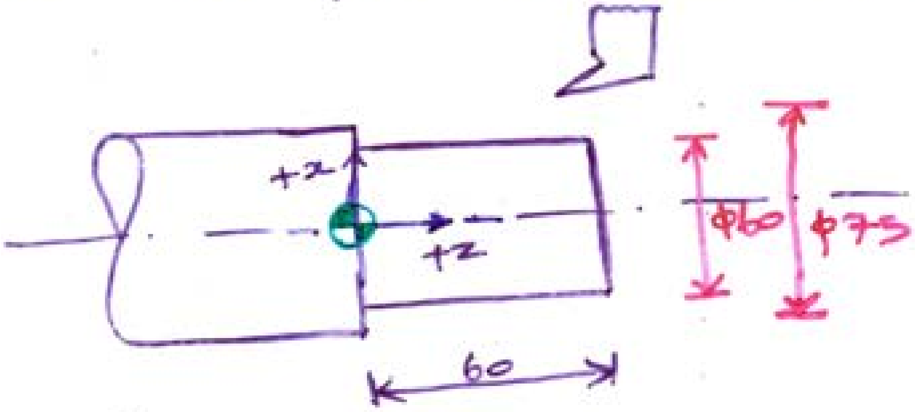
Block:2

U, W ⇒ Stock along X & Z axis respectively

G70 — finishing Cycle

CNC Part Programming

① Write a Manual Part Program to turn the component shown on a CNC Lathe from 75mm bar stock. The following Data may be assumed.



(a) there will be two rough turnings and one finish turning. The first cut is with a depth of 3mm for a length of 58 mm. The second with a depth of 3mm for a length of 59 mm and the third with a depth of 1.5mm for a full length of 60mm.

(b) the shoulder of the workpiece is also machined during each cut.

(c) the spindle speed is 400rpm and the feed rate is 0.5mm/rev.

Make a free hand sketch showing relevant points of tool position for each of the three turning oprns and then write the manual part program. State also what each line of the program does.

```
O1234                              [Program number]
[Billet X75 Z100
G28 U0.0 ;                         [Go to Reference Point]
G28 W0.0 ;
G21 G99 G90;                       [Metric Units & feed/rev & Absoute Programming]
M06 T0101 ;                        [Tool change : 01 & offset no : 1]
M04 S400 ;                         [Spindle on ccw; Speed = 400rpm]
M07 ;                              [coolant on]
G00 Z62.0 X76.0;                   [Rapid traverse]
G71 U0.25 R1.0;
G71 P100 Q101 U0.0 W0.0 F0.5;
N100 G01 X69.0;
N101 Z2.0;
```

Rough turning : 1

Multiple Rough turning cycle to a depth 3mm & length 58mm

```
G71 U0.25 R1.0;
G71 P102 Q103 U3.0 W1.0 F0.5;
G70 P102 Q103 F0.25;
N102 G01 X60.0;
N103 Z0.0;
```

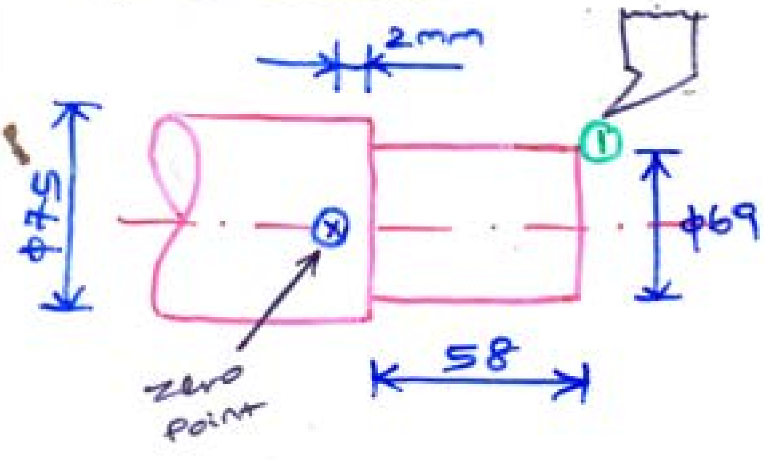
Rough turning : 2

depth :- 3mm
Length :- 59mm

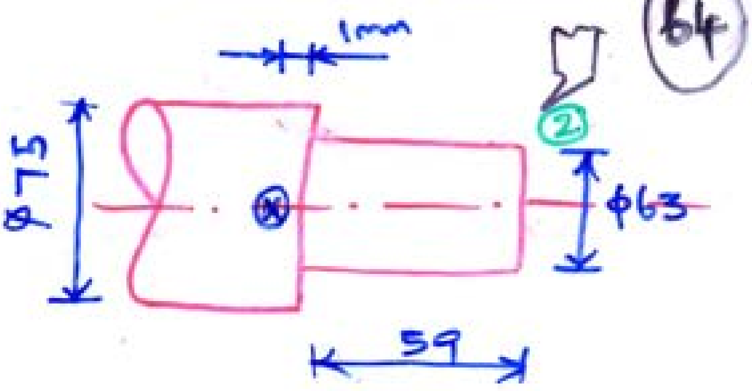
G70 → Finish turning for depth = 1.5 length = 60mm

```
M05;                               [Spindle off]
M09;                               [Coolant off]
G28 U0.0;
G28 W0.0;
M30                                [Program Stop & return]
```

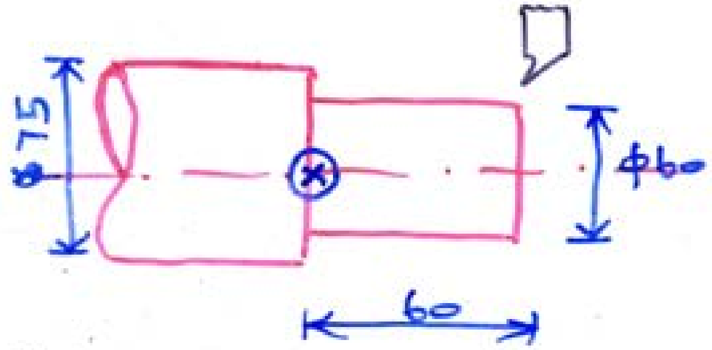
Rough turning:-1



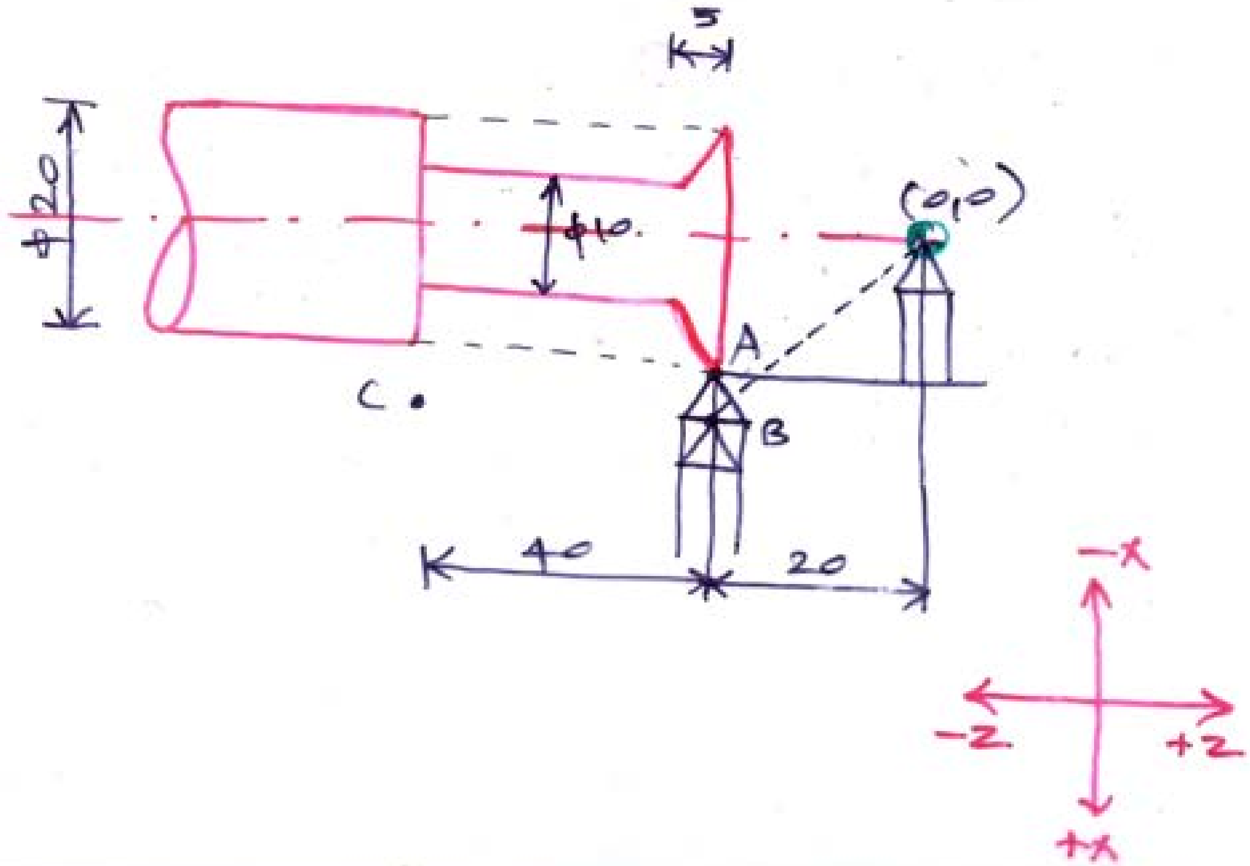
Rough turning:-2



Finish Turning

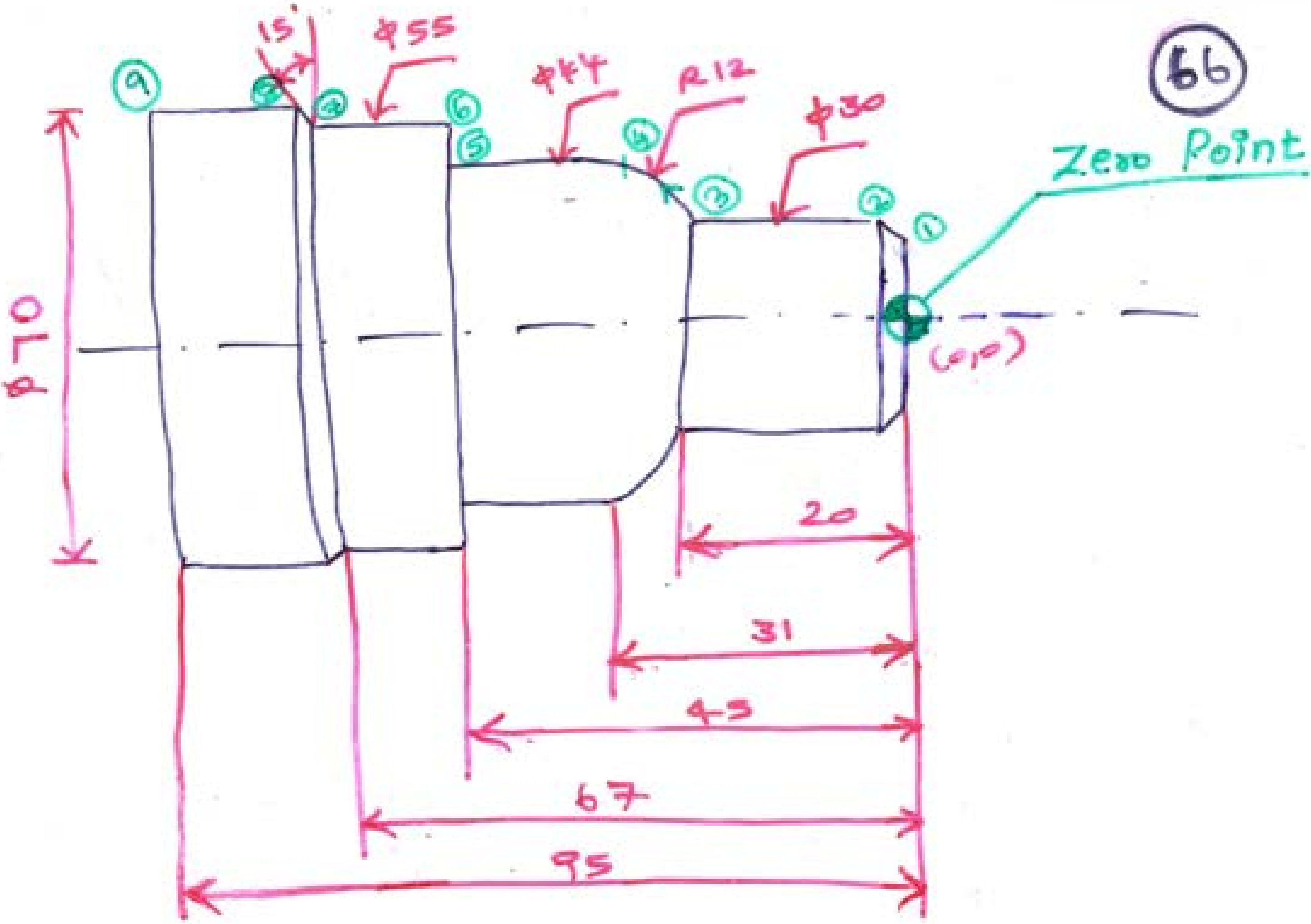


(2) Write a part program for the part shown below.



```
O1234                                   (Program number)                    65
[BILLET X20 Z80
G28 U0.0;
G28 W0.0;                               (Go to referencepoint)
G21 G99 G90;                            [Metric units & feed/rev
M06 T0101;                              (Tool change) & Absolute Programming]

M04 S1000;                              (Spindle on CCW)

M07;                                    (Coolant on)

G00 X0.0 Z0.0;                          (Rapid traverse)

     X30.0 Z-20.0;                      (Point B)
     X30.0 Z-60.0;                      (Point C)  Rough
G71 U0.25 R1.0;                         (Multiple turning cycle)

G71 P100 Q200 U0.5 W0.5 F0.5;

G70 P100 Q200 F0.25;                    (Multiple finish turning cycle)
N100 G01 X10.0;                         (Linear Interpolation)
         Z-25.0;

N200 G01 X20.0 Z-20.0;

M05;                                    (Spindle stop)
M09;                                    (Coolant off)
G28 U0.0;                               (Go to Reference Point)
G28 W0.0;

M30;                                    (Program stop & return)
```

(3) A 110mm long cylindrical rod of ϕ75mm is to be turned into a component as shown in fig using a CNC Lathe. Write a CNC program for manufacturing this component.



O1234 [BILLET X75 Z110

G28 U0.0;

G28 W0.0;

G21 G99 G90;

M06 T0101;

M04 S1000;

M07;

G00 X76.0 Z2.0;

G71 U0.5 R1.0;

G71 P100 Q200 U0.5 W0.5 F0.5;

G70 P100 Q200 F0.25;

N100 G01 X26.0;

G01 X30.0 Z-2.0;

G01 Z-20.0;

G03 X44.0 Z-31.0 R12.0;

G01 Z-45.0;

G01 X55.0;

G01 Z-67.0;

G01 X70.0 Z-69.0;

N200 G01 Z-95.0;

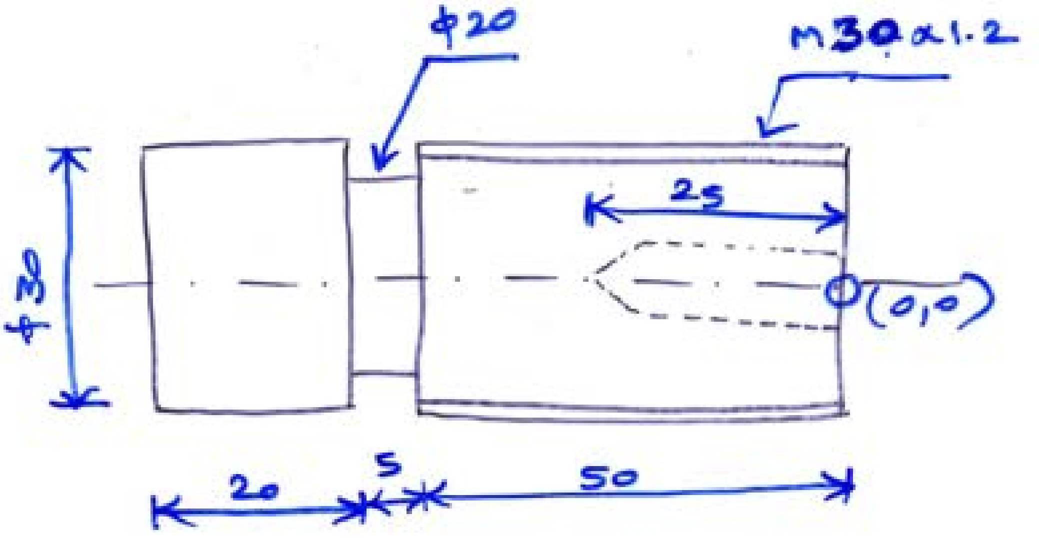
M05;

M09;

G28 U0.0;

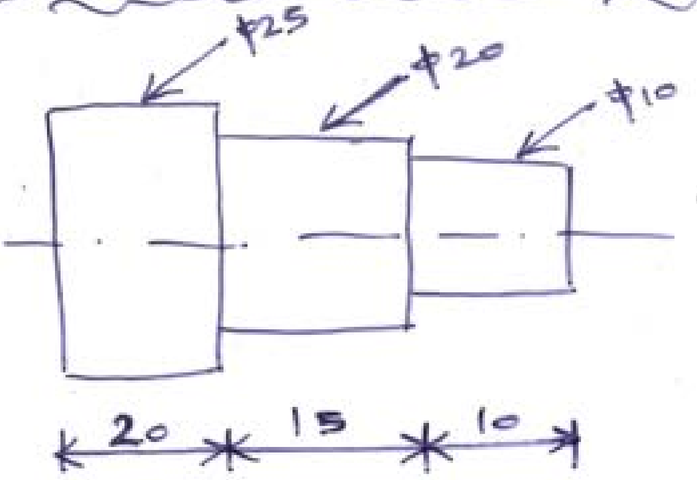
G28 W0.0;

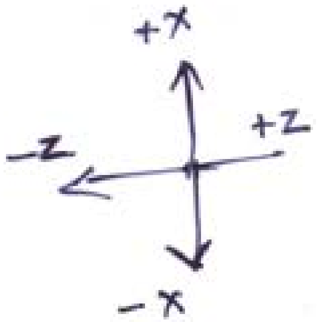
M30;




```
O1234
[BILLET X35 Z85
G21 G99 G90;
G28 U0.0;
G28 W0.0;
M06 T01; M03 S1500; M07;
G00 X35.0 Z2.0;
G71 U0.25 R1.0;
G71 P10 Q11 U0.25 W0.25 F0.5;
G70 P10 Q11 F0.25;
N10 G01 X30.0;
N11 G01 Z-75.0;
G28 U0.0;
G28 W0.0;
M06 T02;
G00 X31.0 Z2.0;
G00 Z-52.0;
G75 R0.5;
G75 X20.0 Z-55.0 P500 Q1500 F0.05;
G28 U0.0;
G28 W0.0;
M06 T03;
G00 X31.0 Z2.0;
G92 X29.5 Z-50.0 F1.2;
G92 X29.0;
G92 X28.5;
G92 X28.44;
G28 U0.0;
G28 W0.0;
M06 T03;
G00 X0.0 Z2.0;
G74 R0.5;
G74 Z-25.0 Q500 F0.2;
G28 U0.0; G28 W0.0;
M05; M09; M30;
```

Multiple turning & multiple facing cycle






```
O1234
[BILLET X30 Z60
G21 G99 G90;
G28 U0.0;
G28 W0.0;
M06 T0101;
M03 S1000;
M07;
G00 X30.0 Z2.0;
G71 U0.25 R1.0;
G71 P100 Q200 U0.25 W0.0 F0.5;
G70 P100 Q200 F0.25;
N100 G01 X10.0;
         Z-10.0;
         X20.0;
         Z-25.0;
         X25.0;
N200     Z-45.0;
G72 W0.25 R0.5;
G72 P101 Q111 U0.0 W0.25 F0.5;
G70 P10 Q11 F0.25;
N101 G01 Z-1.0;
N111 G01 X0.0;
G28 U0.0;
G28 W0.0;
M05;
M09;
M30;
```

NOTE: G21 Pro in mm

G99: Feed rate / rev

G71, Multiple Turn cycle.

CNC MILLING

G - codes

G94 - feed/min

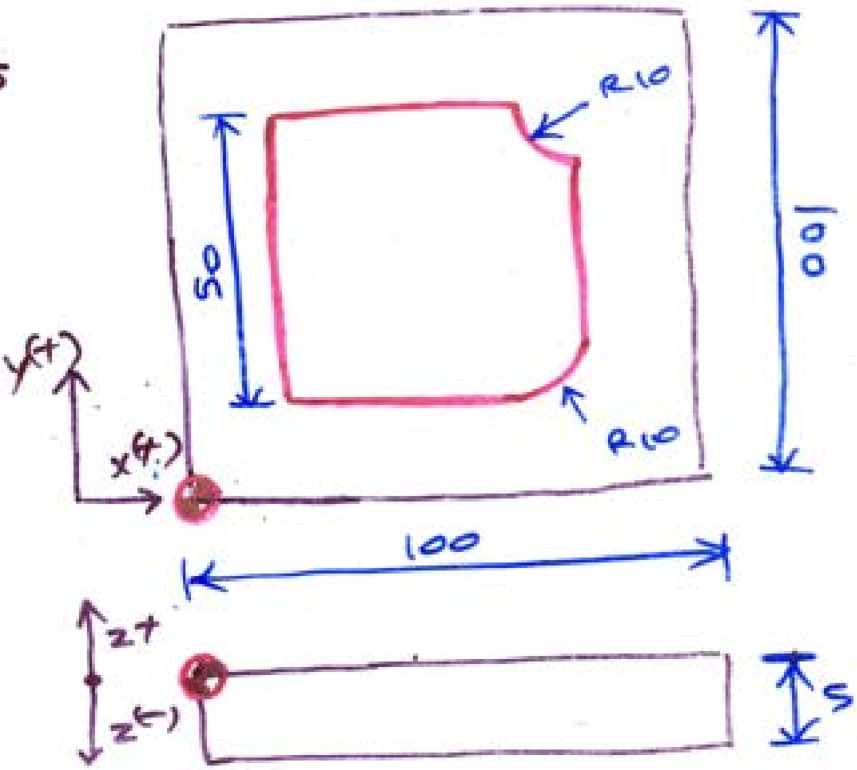
G95 - feed/rev

M - codes

M98 - Sub Program call

M99 - End of Sub Program

1. Write a CNC program for the following profile



[BILLET X100 Y100 Z5

[Tool Def T1 D5

```
G21 G94 G90 G54;
G28 W0.0;
G28 U0.0 V0.0;
M06 T1;
M03 S1000;
    G00 X25.0 Y25.0 Z5.0;
G01 Z-1 F60.0;
G01 X25.0 Y75.0;
    X65.0 Y75.0;
```

G03 X75.0 Y65.0 R10.0 F50.0;

G01 X75.0 Y35.0 F60.0;

G02 X65.0 Y25.0 R10 F50.0;

G01 X25.0 Y25.0 F60.0;

G01 Z5.0;

G28 W0.0;

G28 U0.0 V0.0;

M05;

M30;

70

G83 – Peck Drilling Cycle

Syntax:– G83 X Y Z Q R F K

X & Y : Value of x & y

Z : Drill hole bottom

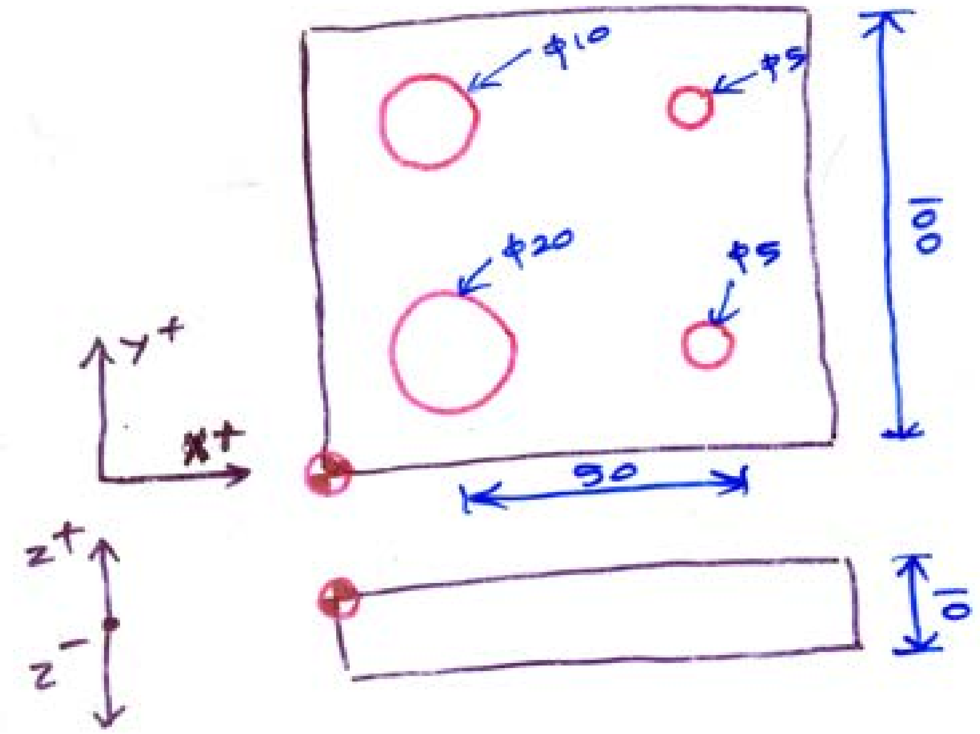
R : Retract position in Z

Q : Depth to increase on each peck

F : Cutting feed rate

K : Number of repeats.

② write a CNC Program for the following Profile



[BILLET X100 Y100 Z10

[Tool Def T1 D5

[Tool Def T2 D10

[Tool Def T3 D20

G54 – offset number

G94 – Feed/min

G90 – Absolute programming

G21 – Metric Units

```
G21 G94 G90 G54;
G28 W0.0;
G28 U0.0 V0.0;
M06 T3;
M03 S800;
M07;
    G00 X25 Y25 Z2;
G83 X25 Y25 Z-5 Q1 R0.5 F40 K1;
```

G28 W0.0;

G28 U0.0 V0.0;

M05;
G55;
M06 T02;

M03 S800;

G00 X25 Y75 Z2;

G83 X25 Y75 Z-5 Q1 R0.5 F40 K1;

G28 W0.0;

G28 U0.0 V0.0;

M05;
G56;
M06 T1;

M03 S800;

G00 X75 Y75 Z2;

G83 X75 Y75 Z-5 Q1 R0.5 F40 K1;

G00 X75 Y25 Z2;

G83 X75 Y25 Z-5 Q1 R0.5 F40 K1;

G28 W0.0;

G28 U0.0 V0.0;

M05;

M09;

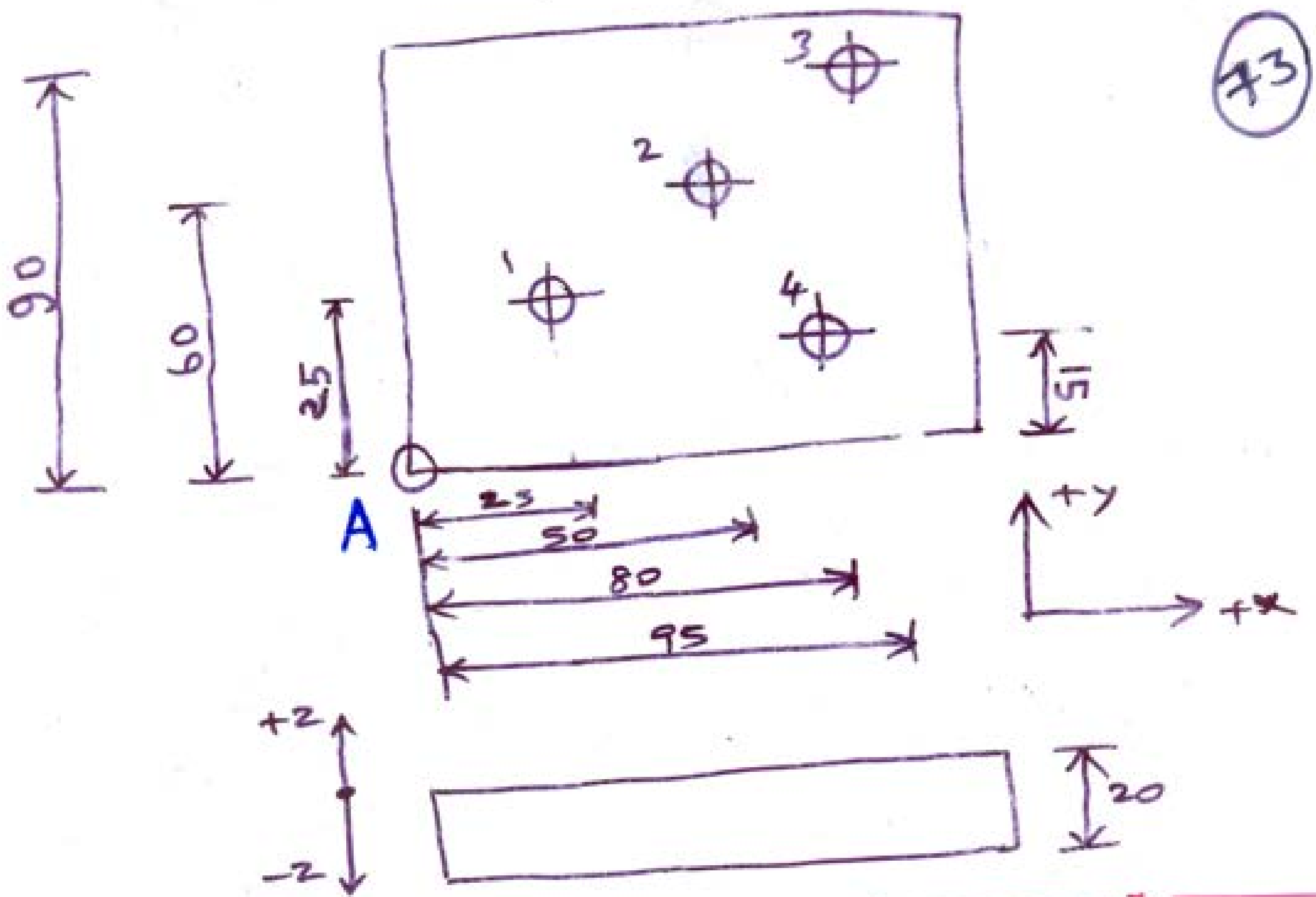
M30;

Syntax for Sub Program } M98 P0011234 { 001 = Number of repeats
1234 = 4 digit number

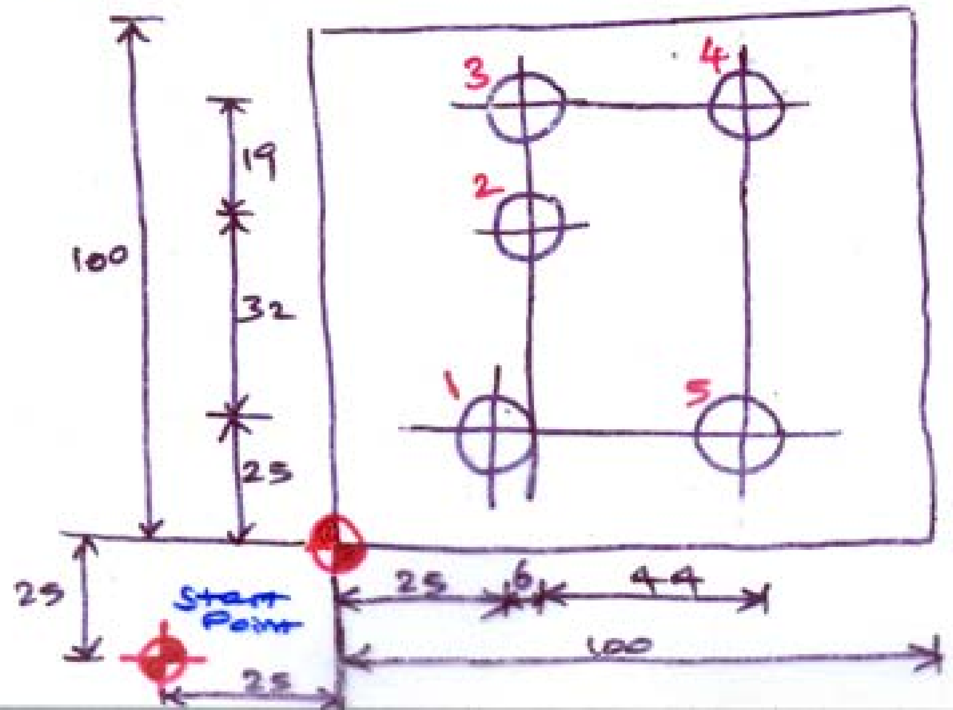
(3) Write a Part Program for drilling holes for the part shown in figure. The plate thickness is 20 mm.

NOV/DEC-2010

73

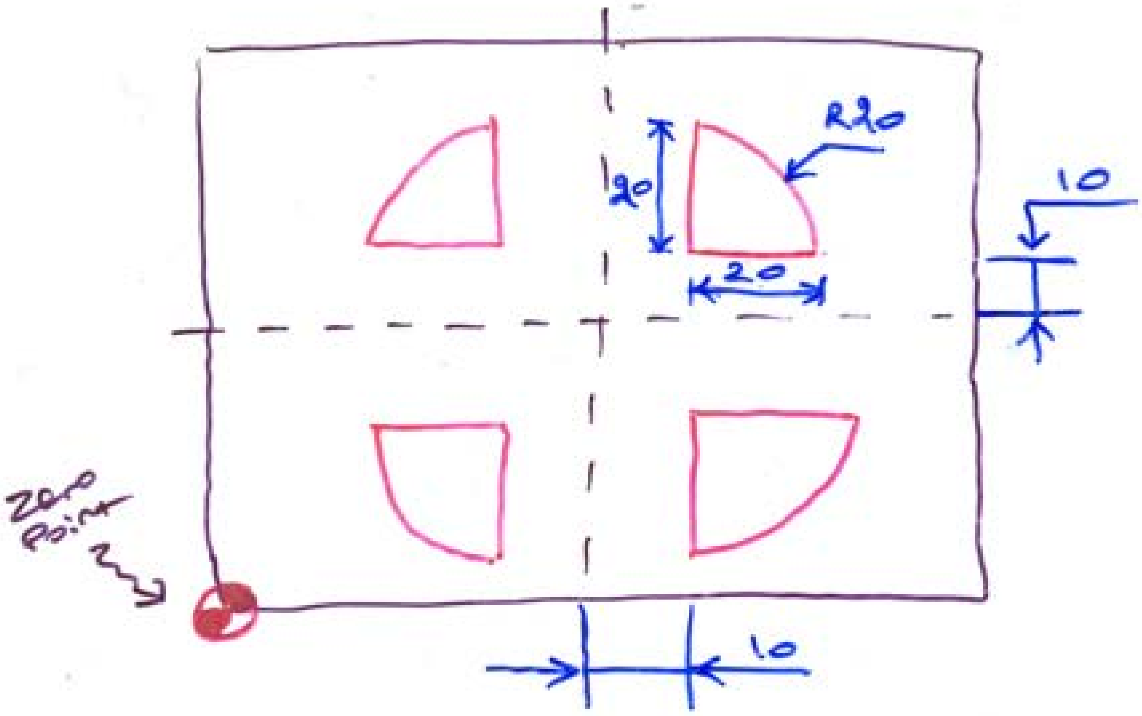


(4) Write a program to drill five hole in the five location.



Write the CNC milling program for the component shown using Mirroring command.

74



[BILLET X100 Y100 Z10

[Tool DEF T1 D5

[Edge move X-50 Y-50

```
G21 G94 G54 G90;
G28 W0.0;
G28 U0.0 V0.0;
M06 T1;
M03 S1000;
M07;
G00 X0 Y0 Z5;
M98 P0011234;
M70;
M98 P0011234;
M80;
```

Sub program 1234

```
G91;
G00 X10 Y10;
G01 Z-6 F50;
G01 X20 Y0;
G03 X-20 Y20 R20 F30;
G01 X0 Y-20;
G01 Z6;
G00 X-10 Y-10;
G90;
M99;
```

75

Mcodes for mirroring:-

M70 – X axis Mirror on

M80 – X axis Mirror off

M71 – Y axis mirror on

M81 – Y axis Mirror off

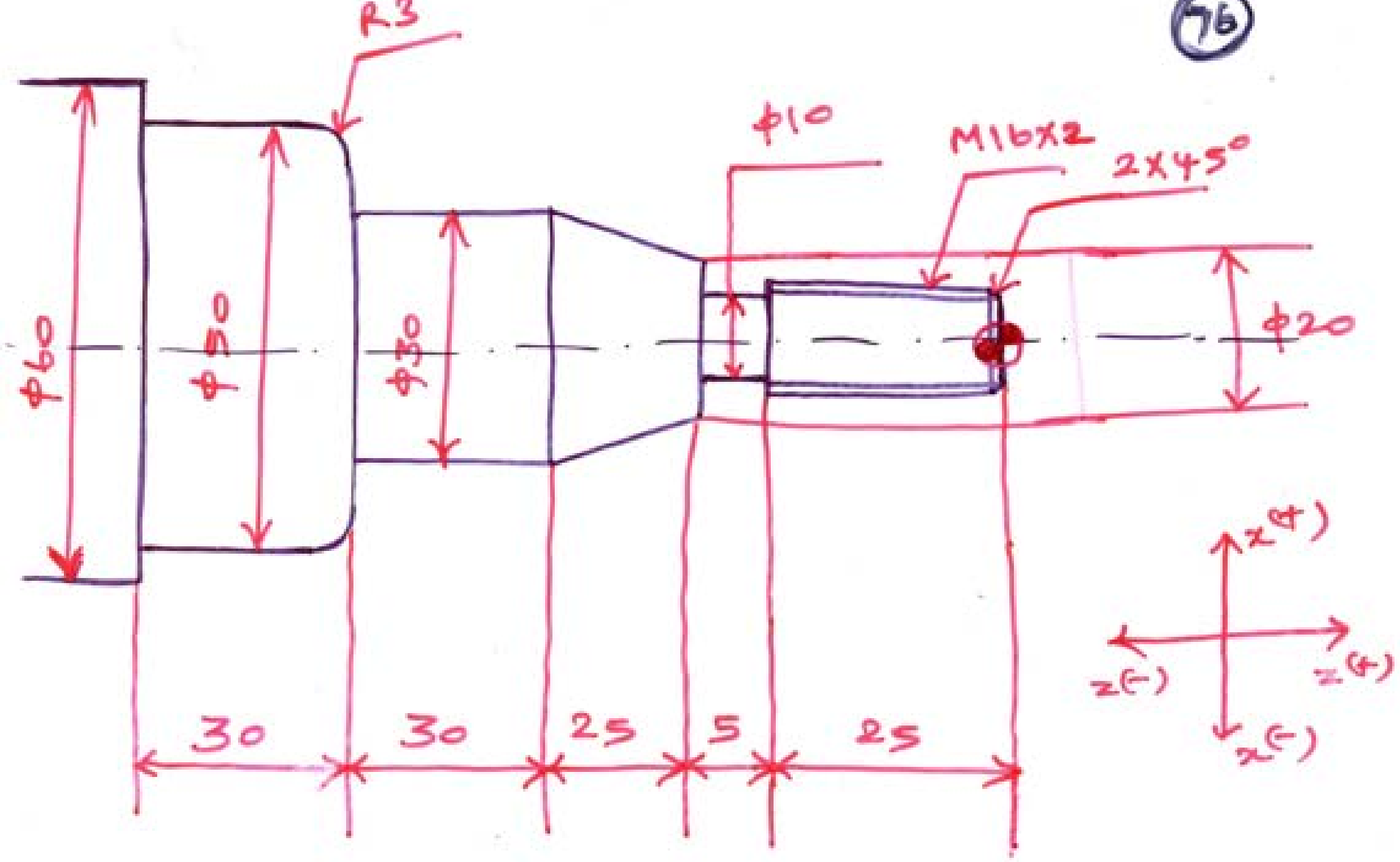
continuation of the program

```
M71;
M98 P0011234;
M81;
M70;
M71;
M98 P0011234;
M80;
M81;
G28 W0.0;
G28 U0.0 V0.0;
M05;
M09;
M30;
```

write the CNC program for the profile shown below

(Nov/Dec'2016)

(96)



O1234

[BILLET X63 Z130

G28 U0.0;

G28 W0.0;

G21 G99 G90;

M06 T0101 [Tool change to T01 - Single Point Tool]

M04 S1000;

M07;

G00 X63.0 Z2.0;

```
G71 U0.25 R1.0;
G71 P100 Q200 U0.25 W0.0 F0.5;
G70 P100 Q200 F0.25;
N100 G01 X16.0;
         Z-30.0;
         X20.0;
         X30.0 Z-55.0;
         Z-85.0;
         X44.0;
      G02 X50.0 Z-88.0 R3.0;
N200 G01 Z-115.0;
 M05;
 G28 U0.0;
 G28 W0.0;
 M06 T0202;
 G00 X17.0 Z-27.0;
 M04 S500;
 G75 R0.5;
 G75 X10.0 Z-30 P500 Q1000 F0.05;
 M05;
 G28 U0.0;
 G28 W0.0;
```

[Tool Change : T02 - Grooving Tool width = 2mm offset no: 02]

[G75 - Grooving cycle]

M06 T0303; [Tool change to T03 — threading tool]

G00 X17.0 Z2.0;

78

M04 S300;

G92 X15.6 Z-25.0 F0.2;

G92 X15.2 Z-25.0;

G92 X14.8 Z-25.0;

G92 X14.4 Z-25.0;

G92 X14.0 Z-25.0;

G92 X13.702 Z-25.0;

M05;

G28 U0.0;

G28 W0.0;

M06 T0101;

G00 X17.0 Z2.0;

M04 S1000;

G01 X12.0 Z0.0 F0.5;

X16.0 Z-2.0;

[Chamfering opreation]

M05;

M09;

G28 U0.0;

G28 W0.0;

M30;

G92 = Thread cutting cycle

Pitch value, P = 2

Depth of thread $= P \times 0.649 = 1.298$

Minor dia = Major dia − [2x depth of thread]

$= 16 - 1.298 = 13.702$

Depth of cut required $\Big\} = \dfrac{16 - 13.702}{2} = 1.149$

Depth of cut for each pass $\Big\} = 0.2$ mm

No of passes = 6

APT Programming:-

* APT means automatically programmed Tools.

* To program in APT, the workpart Geometry must first be defined. Then the tool is directed to various point location and along the surfaces of the workpart to carryout the machining opreations.

* There are four types of statements in the APT language

1. Geometry statements
2. Motion statements
3. Post processor statements
4. Auxillary statements

1. Geometry statements:-

It defines the geometric elements such as a points, lines, circles etc., of the workpart.

Syntax of Geometry statements

Symbol = keyword to define the geometry / Co-ordinates to define the geometry.

example:- P1 = POINT / 20,30,0

It defines the Co-ordinate (20, 30, 0) as a point and assign the same to "P1".

② Motion Statement:—

It describe the path of the cutting tool.

Syntax:—

Motion Command keyword / Co-ordinate Position

example:—

① GOTO/P1 ② FROM/P0

③ Post Processor Statement:—

It is used to control the opreation of the spindle, feed & other features of the machine.

Syntax:—

Post Processor keyword / related data

example:—

① SPINDLE/450 [spindle speed = 450 rpm]

② FEDRAT/1.5 [feed rate = 1.5 mm/rev]

④ Auxillary Statement:—

It is used to specify other details like cutter size, tolerance and so on.

Syntax:-

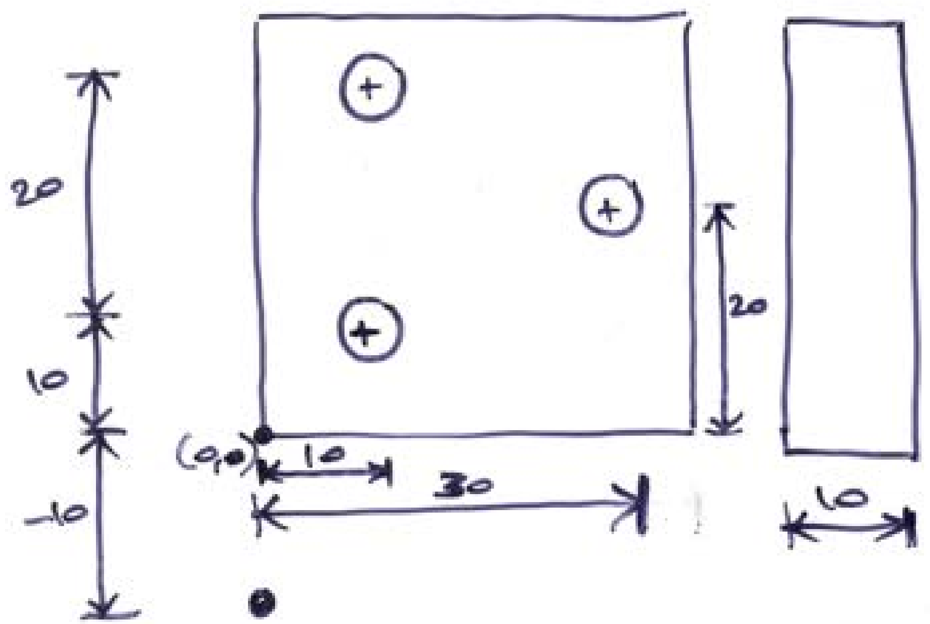
Auxillary key word | related data.

(81)

example:-

CUTTER/50 [cutter diameter = 50 mm]

(1) Write the APT program for following profile



(Geometry definations)

P0 = POINT/0, -10, 10 [Initial cutter position]

P1 = POINT/10, 30, 10

P2 = POINT/10, 10, 10

P3 = POINT/30, 20, 10

(Pre Processor (Auxillary definations) 82

SPINDLE/550 [Takes the spindle speed 550 r.p.m]

FEDRAT/1.5 [feed rate = 1.5 mm/rev]

COOLT/ON [Coolant motor on]

CUTTER/18 [18 mm dia drill bit]

[MOTION STATEMENTS]

FROM/P0 [Tools starts from P0]

GOTO/P1 [Tool go to P1]

GODLTA/0,0,-10 [Drilling takes place at P1 to a depth of -10 mm]

GODLTA/0,0,+10

GOTO/P2

GODLTA/0,0,-10 [Drilling of hole at P2]

GODLTA/0,0,+10 [Tools travels back]

GOTO/P3

GODLTA/0,0,-10

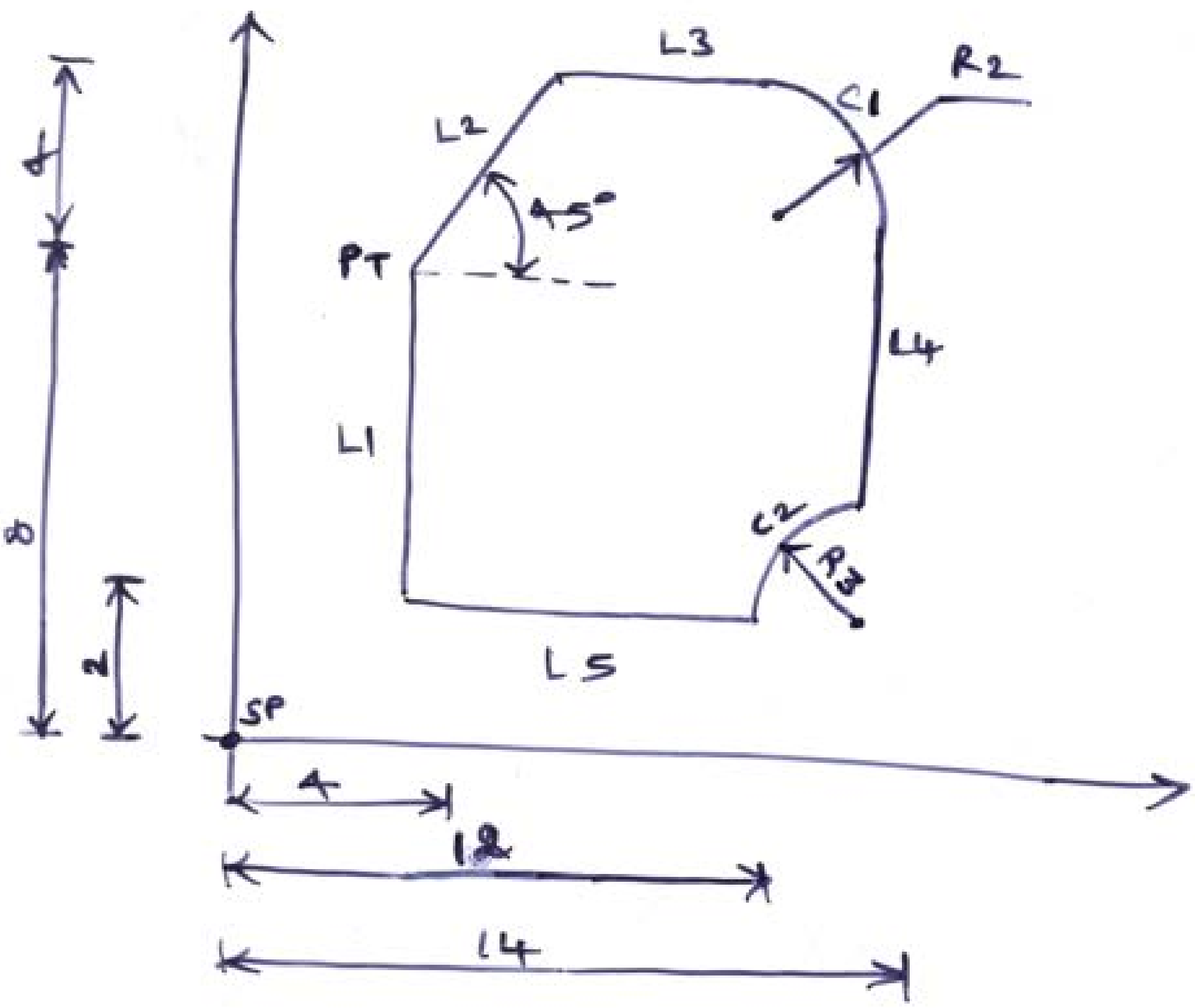
GODLTA/0,0,+10

GOTO/P0

FINI [Finishes line of the program]

② write the APT Program for the following profile

83




```
MACHINE/MILL
SP = POINT/0,0,0
L1 = LINE/4,2,0,4,8,0
PT = POINT/4,8,0
L2 = LINE/PT,ATANGL,45
L3 = LINE/8,12,0,12,12,0
L4 = LINE/14,5,0,14,10,0
L5 = LINE/4,2,0,12,2,0
C1 = CIRCLE/12,10,0,2
```

84

```
C2 =CIRCLE/14, 20, 3
INTOL/0
OUTTOL/ 0.005
CUTTER/15
SPINDLE /2000, CCW
COOLANT/ON
FEDRAT/ 20.0
FROM/SP
GO/TO, L1 [ cutter to move from current location SP
            until it is just tangent to L1 ]
TLLFT, GOLFT/L1, PAST, L2
GORGT/L2, PAST, L3
GORGT/L3, TANTO, C1
GOFWD/C1, TANTO, L4
GOFWD/L4, PAST, C2
GORGT/C2, PAST, L5
GORGT/L5, PAST, L1
GOTO/SP
COOLNT/OFF
SPINDL/OFF
FINI.
```