MACHINING LAB MANUAL

DEPARTMENT OF MECHANICAL ENGINEERING

PREPARED BY: Dr. ITISHREE ROUT
AND
Dr. AJIT KUMAR PATTANAIK

GOVERNMENT COLLEGE OF ENGINEERING
KALAHANDI, BHAWANIPATNA

MEPC3202 MACHINING LABORATORY (0-0-3)

Course Objectives:

This laboratory course provides hands-on experience in fundamental machining operations, including turning, milling, grinding, and threading. Students will analyze tool geometry (ASA/ORS systems), measure cutting forces, and evaluate the role of coolants. Emphasis is placed on practical skills for operating lathes, milling machines, and grinders to manufacture precision components.

List of Experiments:

1. A study on tool geometry in both ASA and ORS system.
2. Preparation of a threaded joint using drilling and tapping operations.
3. Perform operations like taper turning, thread cutting, knurling and groove cutting on a lathe machine.
4. Determine the cutting forces during turning of a cylindrical component in lathe machine.
5. Perform the gear cutting on milling machine.
6. Working with shaper/planer/slotting machine.
7. Working with surface and cylindrical grinding machine.
8. A study on the importance of coolant during machining.

Course Outcomes:

CO1: Remembering (Knowledge): Identify tool geometry systems (ASA, ORS) and components of machining tools (e.g., lathe, milling machine).

CO2: Understanding (Comprehension): Explain the principles of operations like taper turning, thread cutting, and gear milling.

CO3: Applying (Application): Perform machining operations (drilling, tapping, grinding) and measure cutting forces using instrumentation.

CO4: Analyzing (Analysis): Compare the effects of coolants and cutting parameters on surface finish and tool life.

CO5: Creating (Synthesis): Manufacture components by integrating multiple machining processes (e.g., turning + threading) to meet design specifications.

CONTENTS

Sl. No.	Experiment Name	Page No.
01	A study on tool geometry in both ASA and ORS system.	04
02	Preparation of a threaded joint using drilling and tapping operations.	09
03	Perform operations like taper turning, thread cutting, knurling and groove cutting on a lathe machine.	13
04	Determine the cutting forces during turning of a cylindrical component in lathe machine.	18
05	Perform the gear cutting on milling machine.	22
06	Working with shaper/planer/slotting machine.	25
07	Working with surface and cylindrical grinding machine.	36
08	A study on the importance of coolant during machining.	40

A study on tool geometry in both ASA and ORS system

Experiment No-01

Aim:

To study the various tool geometry parameters and their representation in American Standards Association (ASA) and Orthogonal Rake System (ORS) for a single point cutting tool.

Apparatus Required:

- Single Point Cutting Tool
- Tool Holder
- Protractor / Bevel Protractor
- Surface Plate
- Vernier Caliper
- Tool Maker's Microscope (optional)
- Marking Instruments

Theory:

Demonstration (expression) of tool geometry in:

• Machine Reference System

This system is also called ASA system; ASA stands for American Standards Association. Geometry of a cutting tool refers mainly to its several angles or slope of its salient working surfaces and cutting edges. Those angles are expressed w.r.t. some planes of reference. In Machine Reference System (ASA), the three planes of reference and the coordinates are chosen based on the configuration and axes of the machine tool concerned. The planes and axes used for expressing tool geometry in ASA system for turning operation are shown in Fig. 1.

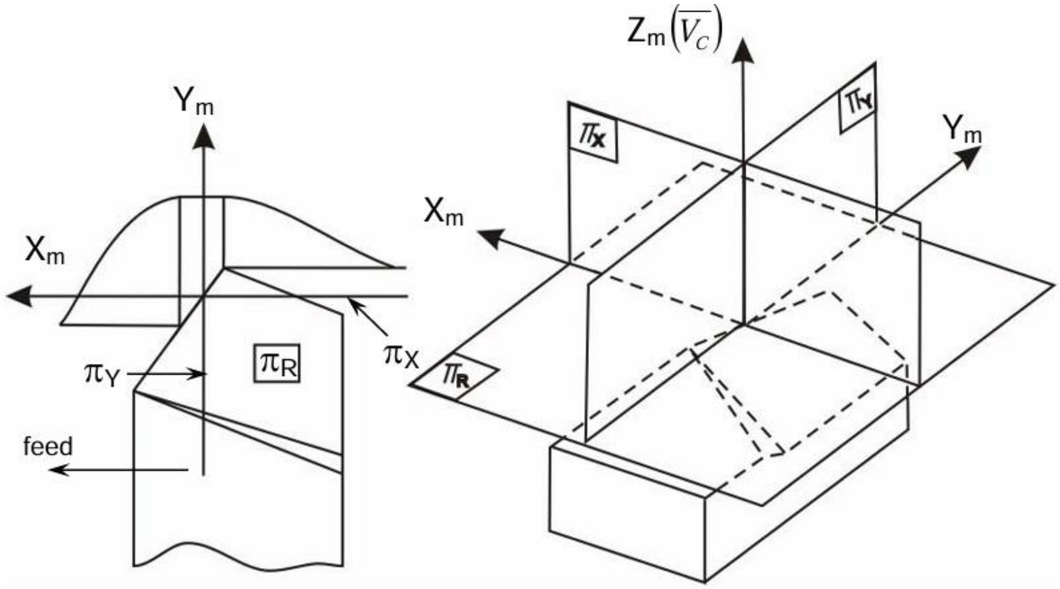


Fig. 1 Planes and axes of reference in ASA system.

The planes of reference and the coordinates used in ASA system for tool geometry is:

π_R - π_X - π_Y and X_m – Y_m - Z_m

where,

π_R = Reference plane; plane perpendicular to the velocity vector (shown in Fig. 1)

π_X = Machine longitudinal plane; plane perpendicular to π_R and taken in the direction of assumed longitudinal feed

π_Y = Machine Transverse plane; plane perpendicular to both π_R and π_X [This plane is taken in the direction of assumed cross feed]

The axes X_m, Y_m and Z_m are in the direction of longitudinal feed, cross feed and cutting velocity (vector) respectively. The main geometrical features and angles of single point tools in ASA systems and their definitions will be clear from Fig. 2.

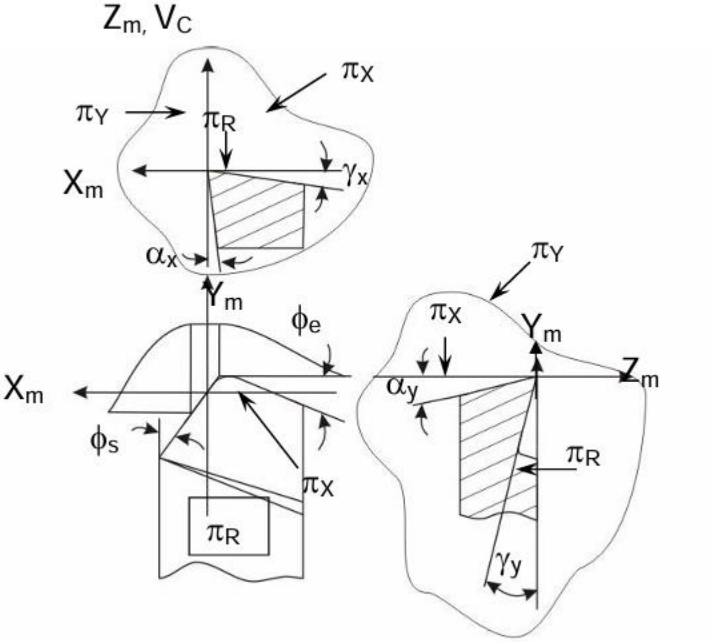


Fig. 2 Tool angles in ASA system.

Definition of:

- Rake angles: [Fig. 2] in ASA system

γ_x = side (axial rake: angle of inclination of the rake surface from the reference plane (π_R) and measured on Machine Ref. Plane, π_X.

γ_y = back rake: angle of inclination of the rake surface from the reference plane and measured on Machine Transverse plane, π_Y.

- Clearance angles: [Fig. 2]

α_x = side clearance: angle of inclination of the principal flank from the machined surface (or V_c) and measured on π_X plane.

α_y = back clearance: same as α_x but measured on π_Y plane.

- Cutting angles: [Fig. 2]

φ_s = approach angle: angle between the principal cutting edge (its projection on π_R) and π_Y and measured on π_R

φ_e = end cutting edge angle: angle between the end cutting edge (its projection on π_R) from π_X and measured on π_R

- Nose radius, r (in inch)

r = nose radius: curvature of the tool tip. It provides strengthening of the tool nose and better surface finish.

ASA Signature:

$$\gamma_x,\ \gamma_y,\ \alpha_x,\ \alpha'_y,\ \phi_e,\ \phi_s,\ r\ (\text{inch})$$

- **Tool Reference Systems**
- **Orthogonal Rake System – ORS**

This system is also known as ISO – old.

The planes of reference and the co-ordinate axes used for expressing the tool angles in ORS are:

π_R - π_C - π_O and X_o - Y_o - Z_o

which are taken in respect of the tool configuration as indicated in Fig. 3.

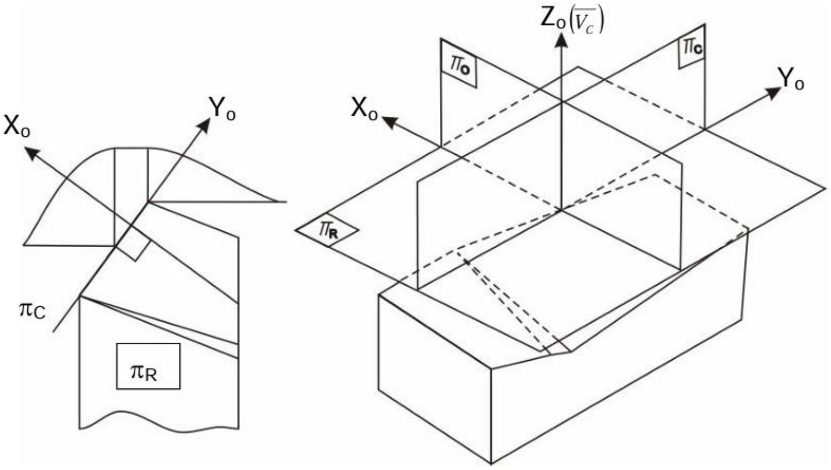


Fig. 3 Planes and axes of reference in ORS.

where,

π_R = Reference plane perpendicular to the cutting velocity vector, V_C

π_C = cutting plane; plane perpendicular to πR and taken along the principal cutting edge

π_O = Orthogonal plane; plane perpendicular to both πR and πC

and the axes.

X_o= along the line of intersection of π_R and π_O

Y_o= along the line of intersection of π_R and π_C

Z_o= along the velocity vector, i.e., normal to both X_o and Y_o axes.

The main geometrical angles used to express tool geometry in Orthogonal Rake System (ORS) and their definitions will be clear from Fig. 4.

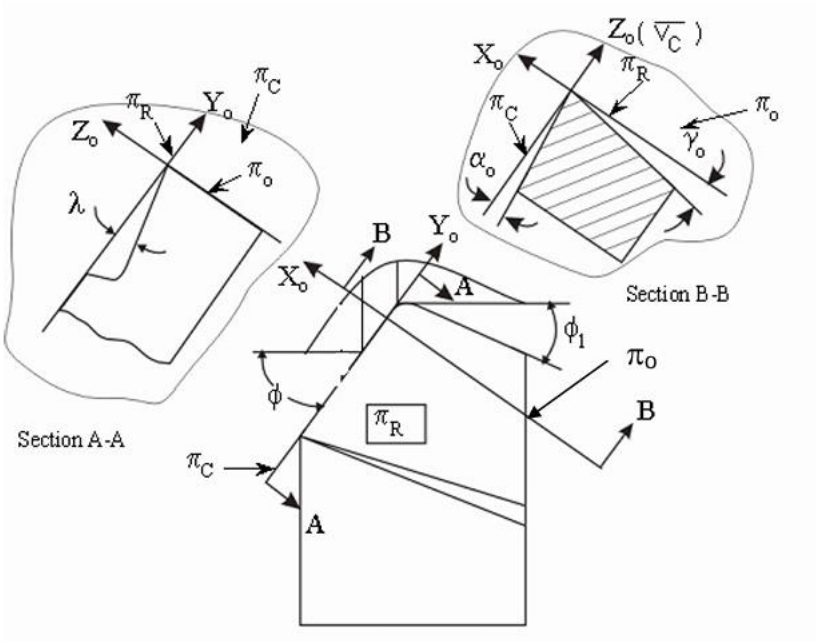


Fig. 4 Tool angles in ORS system.

Definition of –

- Rake angles [Fig. 4] in ORS

γ_o = orthogonal rake: angle of inclination of the rake surface from Reference plane, π_R and measured on the orthogonal plane, π_o

λ = inclination angle; angle between π_C from the direction of assumed longitudinal feed [π_X] and measured on π_C

- Clearance angles [Fig. 4]

αo = orthogonal clearance of the principal flank: angle of inclination of the principal flank from π_C and measured on π_o

α_o' = auxiliary orthogonal clearance: angle of inclination of the auxiliary flank from auxiliary cutting plane, π_C' and measured on auxiliary orthogonal plane, π_o' as indicated in Fig. 5.

- Cutting angles [Fig. 4]

φ = principal cutting-edge angle: angle between π_C and the direction of assumed longitudinal feed or π_X and measured on π_R

$\varphi 1$ = auxiliary cutting angle: angle between π_C' and π_X and measured on π_R

- Nose radius, r (mm)

r = radius of curvature of tool tip

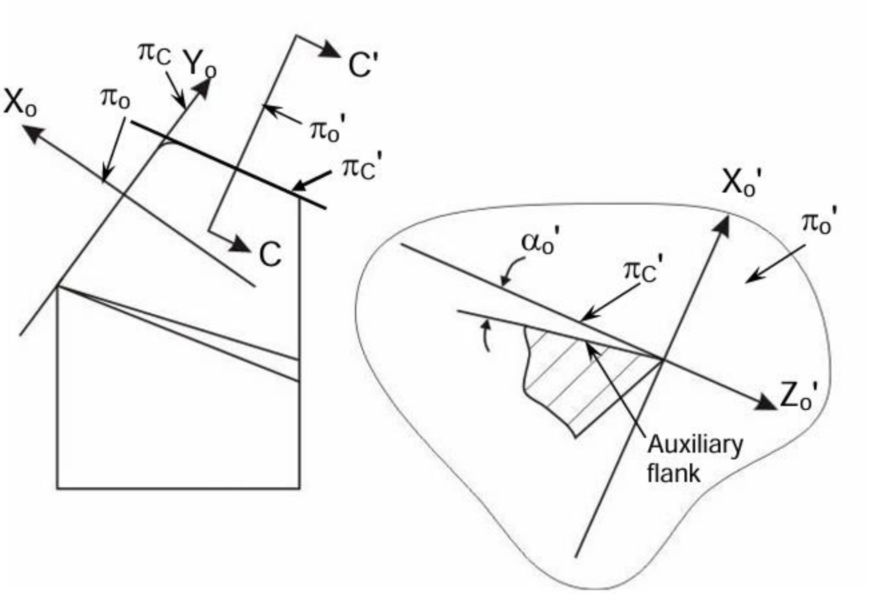


Fig. 5 Auxiliary orthogonal clearance angle.

ORS Signature:

$\lambda, \gamma_o, \alpha_o, \alpha_o', \phi_1, \phi, r$ (mm)

Procedure:

1. Observe the given single-point cutting tool carefully.
2. Identify the various surfaces — rake face, flank face, cutting edge, and nose.
3. Using a bevel protractor, measure the various angles according to ASA system.
4. Redraw the same tool geometry and convert the measured angles into the ORS system using standard relations.
5. Record the observations in a tabular form.
6. Discuss the effect of each angle on tool performance.

Result:

The geometry of the single point cutting tool was successfully studied in both ASA and ORS systems. The tool angles were measured and converted between the two systems.

Precautions:

- Handle the cutting tool carefully.
- Avoid parallax error while reading angles.
- Keep the tool and measuring instruments clean.

Preparation of a threaded joint using drilling and tapping operations.

Experiment No-02

Aim:

To drill the given work piece as required and then to perform to make, counter boring, countersinking and tapping operations.

Materials Required: mild steel specimen, coolant (oil and water mixture), lubricant oil, nut and bolt.

Machine Required: Drilling machine

Measuring Instruments: Vernier calipers

Cutting Tools:

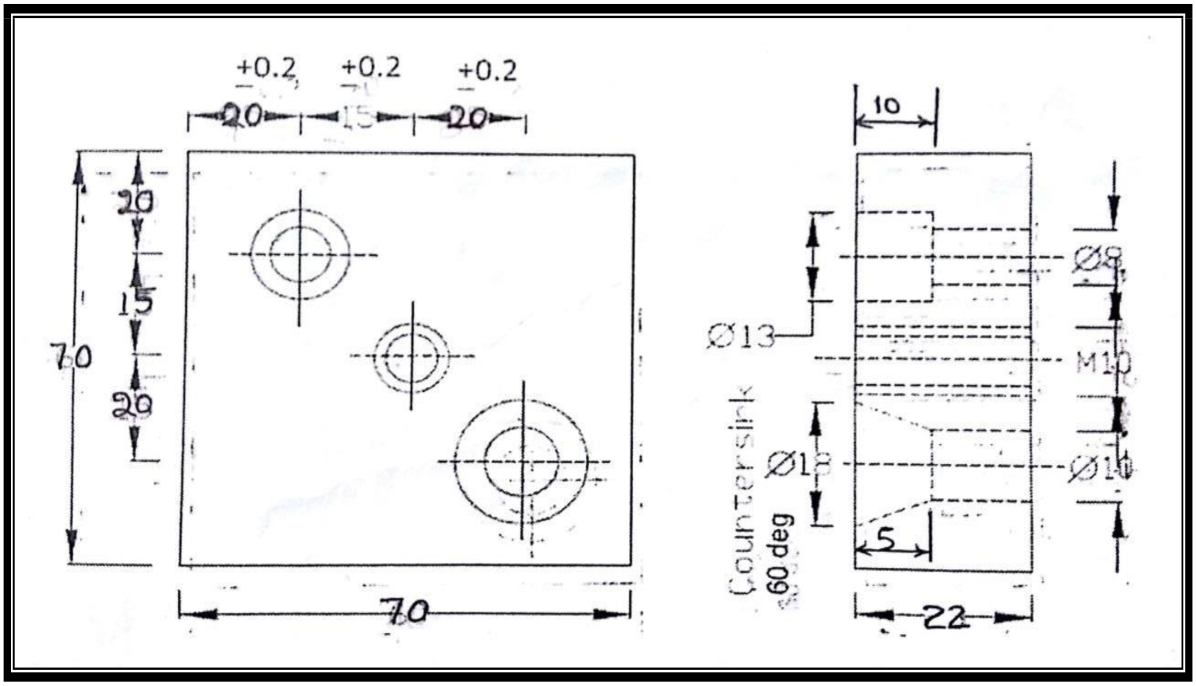
Button pattern stock, Dies, Drill bits, Hand taps, Tap wrench

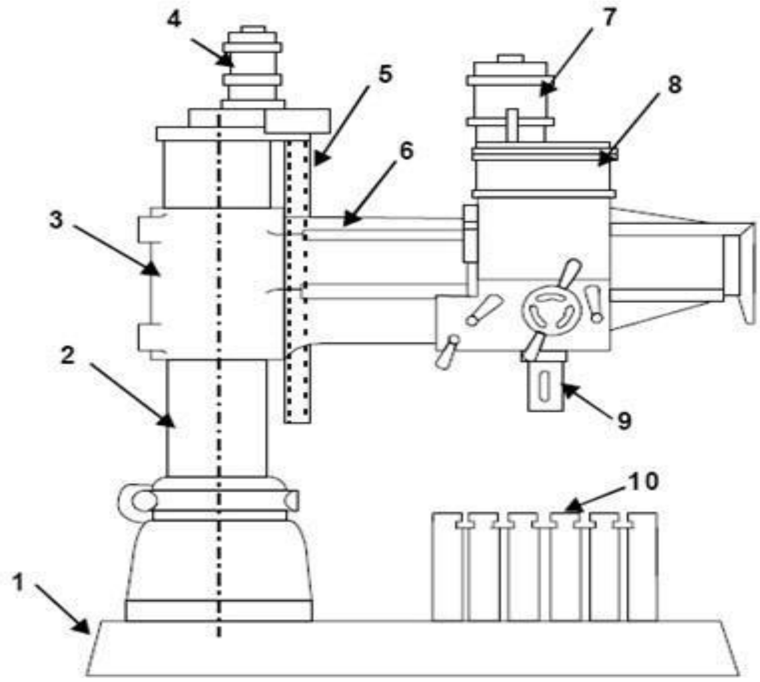
Work holding fixtures:

Bench vice, V-Block

Miscellaneous tools:

Brush, Allen Keys





PARTS OF RADIAL DRILL:

1. Base
2. Column
3. Radial arm
4. Motor for elevating screw
5. elevating screw
6. Guide ways
7. Motor for driving spindle
8. Drill head
9. Drill spindle
10. Table

Observation:

Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Drilling m/c							

Speed and Feed Data (Table B)

No.	Drilling M/c.	
	Speed	Feed
1		
2		
3		
4		
5		

Sequence of Operations:

1. Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced.
2. Note particularly the features, which control the geometrical form of the surface.
3. Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.
4. Record the obtainable speed and feed values (Table B).
5. Note down the special features *of* the speed and feed control on each machine.
6. Mark the center of hole and center punching
7. Drill bid

$D_d = d_h - p$

Where,

1. d_h - dia. of the hole,
2. D_d – dia. of drill bit,
3. p = pitch

8.Use the suitable drill size for required

tapping D=Dia. of tap

Tap Drill size = (D-1.3p) +0.2 – for metric threads

9.Chamfering of specimen

10.Use the sequential tapping as tap set 1, 2, and 3

11.Internal taping of drilled specimen

12. Filling of specimen on which external threading to be done

13. Measuring the diameter of the specimen & choosing of dies according to it

14. Dying operation (external threading) of the specimen.

Precautions:

1. Coolant must be sued while drilling
2. Lubricating oil must be used to get smooth finish while tapping.

Advantages:

1. The universal movements of the tool head permit the drill tool located at any desired position over the stationary work piece.
2. Possible to work on odd, shaped jobs and to drill larger diameter holes.
3. Accurate precision drilling is possible.

Disadvantages:

1. A skilled worker is a must.
2. Only small size holes can be drilled.

Applications:

1. Origination and / or enlargement of existing straight through or stepped holes of different diameter and depth in wide range of work materials – this is the general or common use of drilling machines
2. Making rectangular section slots by using slot drills having 3 or four flutes and 1800 cone angle
3. Boring, after drilling, for accuracy and finish or prior to reaming

Counter boring, countersinking, chamfering or combination using suitable tools.

Perform operations like taper turning, thread cutting, knurling and groove cutting on a lathe machine

Experiment No-03

Aim:

To perform taper turning, thread cutting, knurling and groove cutting operations on the given work piece.

Material Required:

Mild steel rod of 25 mm diameter and 100 mm long.

Tools Required:

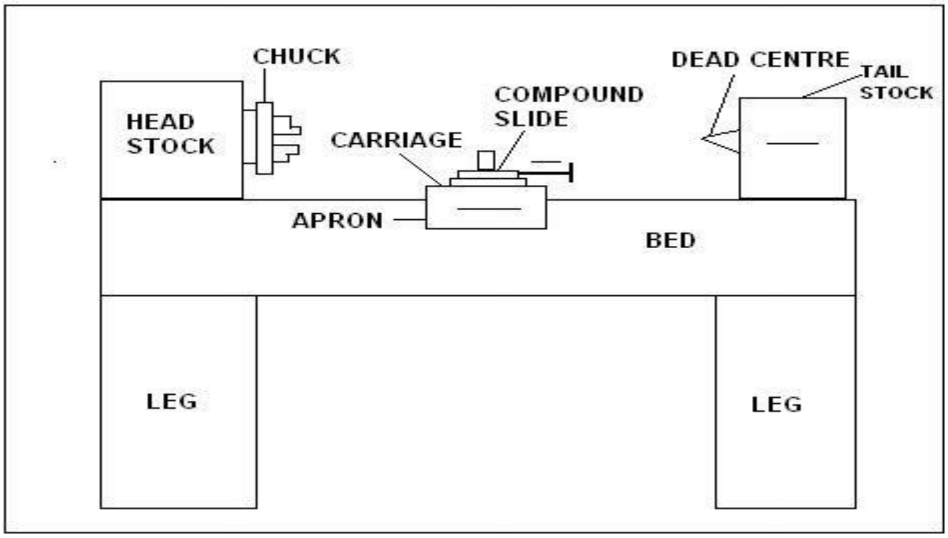
Vernier calipers, steel rule, spanner, chuck spanner, and H.S.S. single point cutting tool.

Specification of Lathe:

Length of bed 1390 mm
Width of bed 200 mm
Height of centers 165 mm
Admit between centers 700 mm
Lead screw pitch 4TPI
Power of the motor 1 h.p.

Theory:

Lathe removes undesired material from a rotating work piece in the form of chips with the help of a tool which is traversed across the work and can be fed deep in work. The tool material should be harder than the work piece and the later help securely and rigidly on the machine. The tool may be given linear motion in any direction. A lathe is used principally to produce cylindrical surfaces and plane surfaces, at right angles to the axis of rotation. It can also produce tapers and bellows etc.



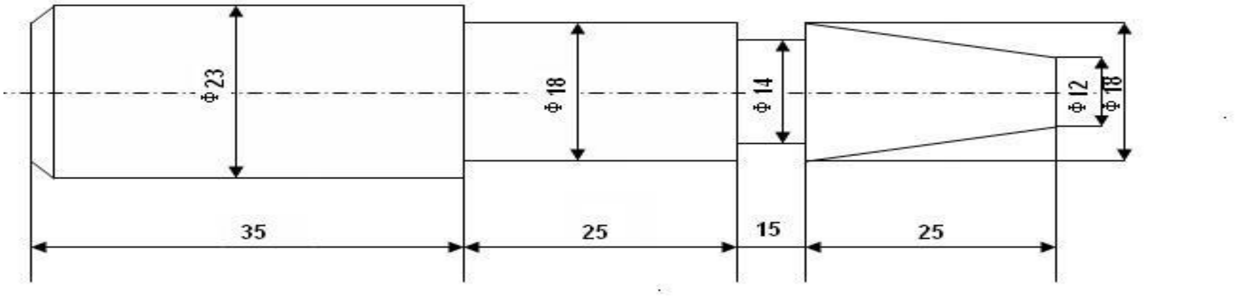
BLOCK DIAGRAM OF A LATHE

A lathe (shown in fig.) basically consists of a bed to provide support, a head stock, a cross side to traverse the tool, a tool post mounted on the cross slide. The spindle is driven by a motor through a gear box to obtain a range of speeds. The carriage moves over the bed guide ways parallel to the work piece and the cross slide provides the transverse motion. A feed shaft and lead screw are also provided to power the carriage and for cutting the threads respectively.

Sequence of Operations:

- Centering
- Facing
- Plain turning
- Chamfering
- Step turning
- Grooving
- Taper turning

Procedure for Step Turning and Taper turning:

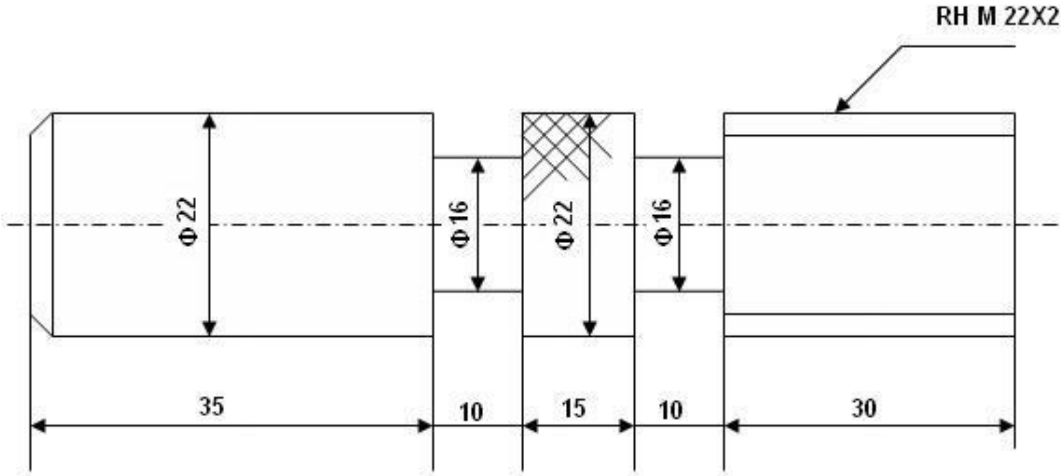


STEP TURNING AND TAPER TURNING

ALL DIMENSIONS ARE IN MM

- The work piece is fixed in a 3-jaw chuck with sufficient overhang.
- Adjust the machine to run the job to a required cutting speed.
- Fix the cutting tool in the tool post and centering operation is performed so that the axis of the job coincides with the lathe axis.
- Give the feed and depth of cut to the cutting tool
- Facing operation is performed from the center of the job towards outwards or from the circumference towards the center.
- Plain turning operation is performed until the diameter of the work piece reduces to 23 mm.
- Check the dimensions by using vernier calipers.
- Then chamfering is done on the 23mm diameter surface.
- Reverse the work piece in the chuck and facing operation is performed to reduce the length of work piece to the required dimensions.
- Again, plain turning operation is performed until the diameter of the work piece reduced to 18mm.
- Using V-cutting tool grooving operation is performed according to the given dimensions and finish the groove using parting tool.
- Swivel the compound slide to the required angle and taper turning operation by rotating the compound slide wheel.
- The angle can be measured by using the formula.
- Finally check the dimensions by using vernier calipers.

Procedure for Thread Cutting and Knurling:



THREAD CUTTING AND KNURLING

ALL DIMENSIONS ARE IN MM

- The work piece is fixed in a 3 – jaw chuck with sufficient overhang.
- Adjust the machine to run the job to required cutting speed.
- Fix the cutting tool in the tool post and centering operation is performed so that the axis of the job coincides with the lathe axis.
- Facing is performed by giving longitudinal depth of cut and cross feed.
- Perform plain turning operation until the diameter of the work piece reduced to 20mm.
- Chamfering operation is done according to the given dimensions.
- Then reverse the work piece in the chuck and plain turning operation is performed according to the given dimensions.
- Using V-cutting tool and parting off tool perform grooving operation to the required dimensions.
- Reduce speed of the spindle by engaging back gear and use Tumbler
- feed reversing mechanism to transmit power through the lead screw.
- And calculate the change gears for the required pitch to be made on the work piece.
- Using half nut mechanism perform thread cutting operation (right hand threading) according to the given dimensions and continues it until required depth of cut is obtained.
- At the same speed knurling operation is performed using knurling tool.
- For every operation check the dimensions using vernier calipers.

Procedure for Groove Cutting:

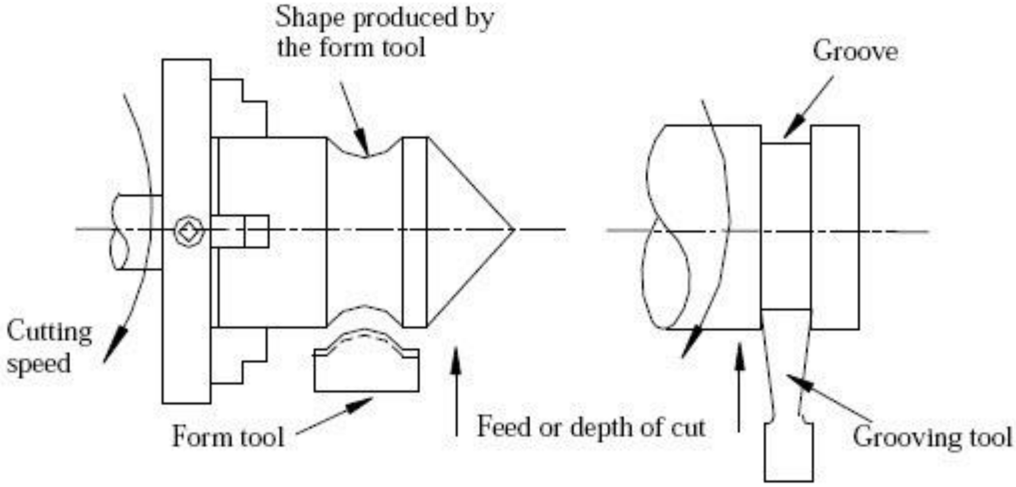


Fig. Groove Cutting

- Mount the workpiece in the 3-jaw chuck; tighten and check run-out.
- Face and rough turn the OD to size if required.
- Mark/locate the groove position (use carriage dial or a stop).
- Mount a grooving/parting tool in the tool post; set tool tip exactly on spindle centre height and square to the work using a try-square or dial indicator.
- Set spindle speed (low–moderate), select fine feed; apply cutting fluid.
- Bring tool to touch the OD; set cross-slide dial (zero) and carriage position reference.

- Feed the tool radially (cross-slide) in small steps (“peck” cuts), clearing chips frequently, until the required groove depth (d) is reached.
- For wider grooves than tool width, retract slightly, shift axially (carriage/compound) and make adjacent passes to achieve groove width (w).
- Break sharp edges, check dimensions (depth & width), stop spindle, deburr, and clean the machine area.

Precautions:

- Before starting the spindle by power, lathe spindle should be resolved by one revolution by hand to make it sure that no fouling is there.
- Tool should be properly ground, fixed at correct height and properly secured, and work also be firmly secured.
- Chips should not be allowed to wind around a revolving job and cleared as often as possible.
- Before operating threading operation, V-tool should be properly ground to the required helix angle.
- Apply cutting fluids to the tool and work piece property.
- No attempt should be made to clean the revolving job with cotton waste.
- On hearing unusual noise, machine should be stopped.

Result:

The job is completed successfully and safely.

Determine the cutting forces during turning of a cylindrical component in lathe machine

Experiment No-04

Aim:

To measure the cutting forces generated during turning of a cylindrical component in lathe machine.

Theory:

Merchant's Circle Diagram and its use

In orthogonal cutting when the chip flows along the orthogonal plane, πO, the cutting force (resultant) and its components PZ and PXY remain in the orthogonal plane. Fig. 1 is schematically showing the forces acting on a piece of continuous chip coming out from the shear zone at a constant speed. That chip is apparently in a state of equilibrium.

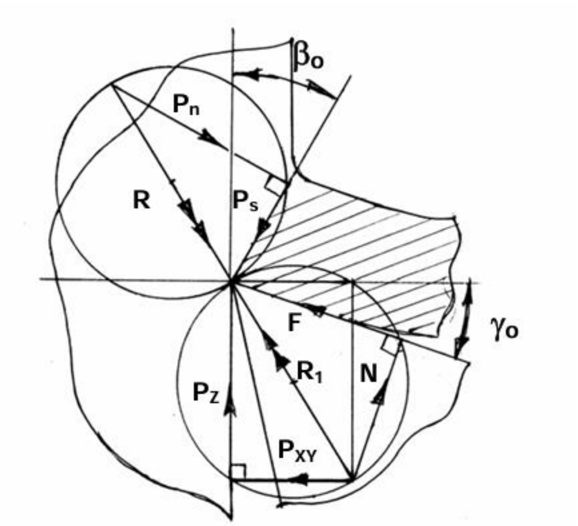


Fig. 1 Development of Merchants Circle Diagram.

The forces in the chip segment are:

- From job-side:
  - P_s – shear force and
  - P_n – force normal to the shear force

  Where, $P_s + P_n = R$(resultant)
- From tool side:
- $R_1=R$ (in state of equilibrium)
- where $R_1=F+N$
- N = force normal to rake face
- F = friction force at chip tool interface
- The resulting cutting force R or R_1 can be resolved further as

$R_1 = P_z + P_{xy}$

where, P_Z = force along the velocity vector

and P_{XY} = force along orthogonal plane.

The circle(s) drawn taking R or R_1 as diameter which contains all the force components concerned as intercepts. The two circles with their forces are combined into one circle having all the forces contained in that as shown by the diagram called Merchant's Circle Diagram (MCD) in Fig. 2.

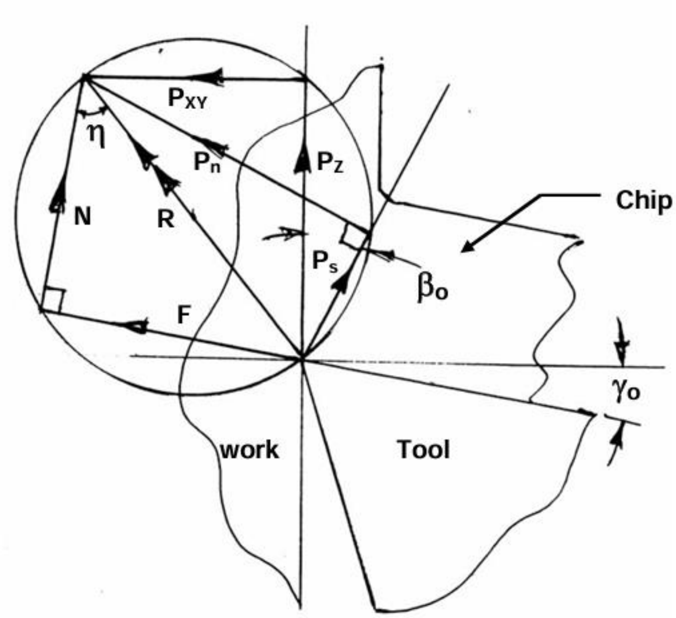


Fig. 2 Merchant' s Circle Diagram with cutting forces.

The significance of the forces displayed in the Merchant's Circle Diagram are:

P_S – the shear force essentially required to produce or separate the chip from the parent body by shear

P_n – inherently exists along with P_S

F – friction force at the chip tool interface

N – force acting normal to the rake surface

P_Z – main force or power component acting in the direction of cutting velocity

P_{xy} - P_x + P_Y

The magnitude of P_S provides the yield shear strength of the work material p under the cutting condition.

The values of F and the ratio of F and N indicate interaction like friction at the chip-tool interface. The force components P_X, P_Y, P_Z are generally obtained by direct measurement. Again, P_Z helps in determining cutting power and specific energy requirement. The force components are also required to design the cutting tool and the machine tool.

Advantageous use of Merchant's Circle Diagram (MCD):

Proper use of MCD enables the followings:

- Easy, quick and reasonably accurate determination of several other forces from a few known forces involved in machining.
- Friction at chip-tool interface and dynamic yield shear strength can be easily determined
- Equations relating the different forces are easily developed.

Some limitations of use of MCD

- Merchant's Circle Diagram (MCD) is valid only for orthogonal cutting
- by the ratio, F/N, the MCD gives apparent (not actual) coefficient of friction
- It is based on single shear plane theory.

The advantages of constructing and using MCD has been illustrated as by an example as follows;

Suppose in a simple straight turning under orthogonal cutting condition with given speed, feed, depth of cut and tool geometry, the only two force components P_Z and P_X are known by experiment i.e., direct measurement, then how can one determine the other relevant forces and machining characteristics easily and quickly without going into much equations and calculations but simply constructing a circle-diagram. This can be done by taking the following sequential steps:

- Determine P_{XY} from $P_X = P_{XY} \sin\varphi$, where P_X and φ are known.
- Draw the tool and the chip in orthogonal plane with the given of γo as shown in Fig. 2
- Choose a suitable scale (P_{XY} in cm value e.g. 100 N = 1 cm) for presenting P_Z and
- Draw P_Z and P_{XY} along and normal to V_C as indicated in Fig. 2
- Draw the cutting force R as the resultant of P_Z and P_{XY}
- Draw the circle (Merchant's circle) taking R as diameter
- Get F and N as intercepts in the circle by extending the tool rake the tool rake surface and joining tips of F and R
- Divide the intercepts of F and N by the scale and get the values of F and N
- For determining P_s (and P_n) the value of the shear angle β_o has to be evaluated from

$$\tan\beta_o = \frac{\cos\gamma_o}{\zeta - \sin\gamma_o}$$

where γ_o is known and ζ has to be obtained from

$$\zeta = \frac{a_2}{a_1} \text{ where } a_1 = s_o \sin\phi$$

S_o and φ are known and a_2 is either known, if not, it has to be measured by micrometer or slide caliper

- Draw the shear plane with the value of β_o and then P_s and P_n as intercepts shown in Fig. 2.
- Get the values of P_s and P_n by dividing their corresponding lengths by the scale
- Get the value of apparent coefficient of friction, μ_a at the chip tool interface simply from the ratio,
- Get the friction angle, η, if desired, either from $\tan_{\eta}= \mu_a$ or directly from the MCD drawn as indicated in Fig. 2.
- Determine dynamic yield shear strength (τ_s) of the work material under the cutting condition

using the simple expression

$$\tau_s = \frac{P_s}{A_s}$$

where, A_s = shear area as indicated in Fig. 3

$$= \frac{a_1 b_1}{\sin\beta_o} = \frac{ts_o}{\sin\beta_o}$$

t = depth of cut (known)

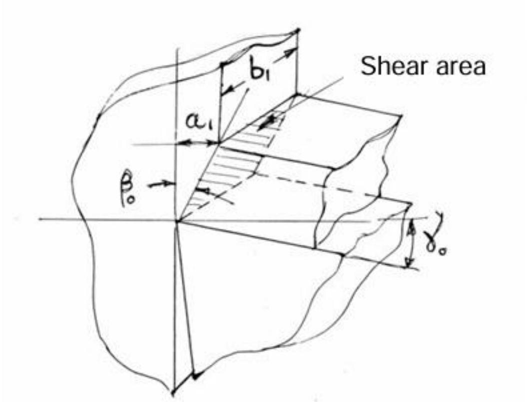


Fig. 3 Shear area in orthogonal turning

Precautions:

- Ensure dynamometer calibration and zeroing before each test.
- Avoid thermal drift: allow tool and workpiece to reach steady temperature or note temperature.
- Keep work holding rigid-minimize runout.
- Use consistent tool geometry and fresh insert/wear state when comparing runs.
- Use proper PPE.

Result:

The cutting forces were successfully measured during turning and it was observed that the main cutting force increases with both feed rate and depth of cut, while the feed and radial forces are smaller in magnitude but follow the same trend.

Perform the gear cutting on milling machine

Experiment No-05

Aim: To produce a gear out of the given work piece using milling machine.

Apparatus Required:

- Horizontal Milling machine
- M10 – End Mill Cutter (HSS)
- Gear tooth Vernier

Materials Required:

- Cast Iron Work piece – 55mm diameter, 20mm thickness

Fig. Milling Machine

Procedure:

- The gear blank is held between the dividing head and tailstock using a mandrel.
- The cutter is mounted on the arbor and the cutter is centred accurately with the gear blank.
- Set the speed and feed for machining. For giving depth of cut, the table is raised till the periphery of the gear blank just touches the cutter.
- The Micrometer dial of vertical feed screw is set to zero at this position. Then the table is raised further to give the required depth of cut.
- The machine is started and feed is given to the table to cut the first groove of the blank.
- After the cut, the table is brought back to the starting position. Then the gear blank is indexed for the next tooth space. This is continued till all the teeth are cut.
- Dimensions of the gear teeth profile are checked using the gear tooth Vernier.

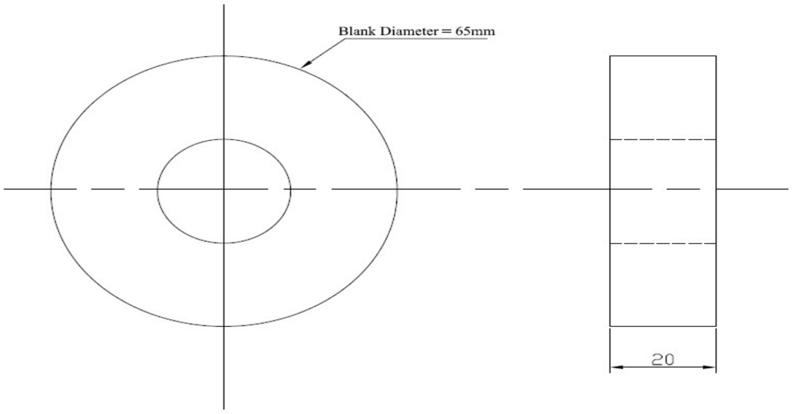


Fig. Before Machining

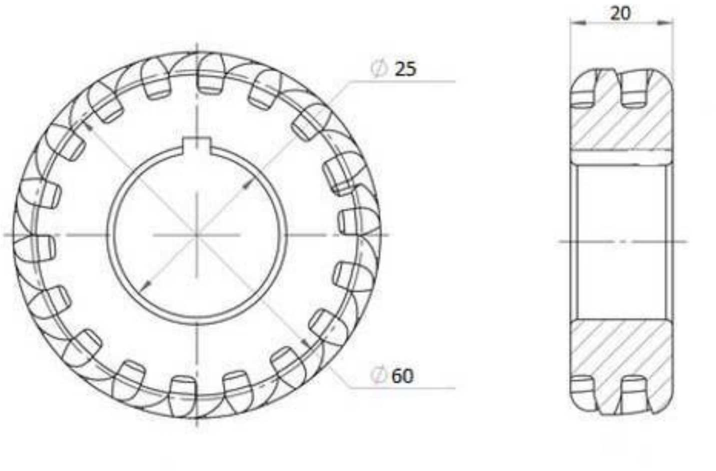


Fig. After Machining

Calculation:

- **Pitch circle Diameter DP** = Diameter of the Blank(**D**) – (2 X Module(**m**)

  = 65-(2X2.5) =60
- **Number of teeth Z** = Pitch circle Diameter / module = 60 / 2.5 = 24
- **Circular Pitch PC** = πDP / Z
- The relationship between **normal pitch** and **transverse pitch** is given by,

$$PN = PC\cos\alpha$$

Helical Gear considerations:

- Helix Angle α is related to Pitch circle diameter (DP) and the lead of the helix (L) by the following relation, Tan α = πDP / L
- With any of the two known values, the third value can be found

Indexing Calculation:

Indexing = 40 / Z

Result:

Thus, a spur gear is made from the given work piece using milling machine

Working with shaper/planer/slotting machine.

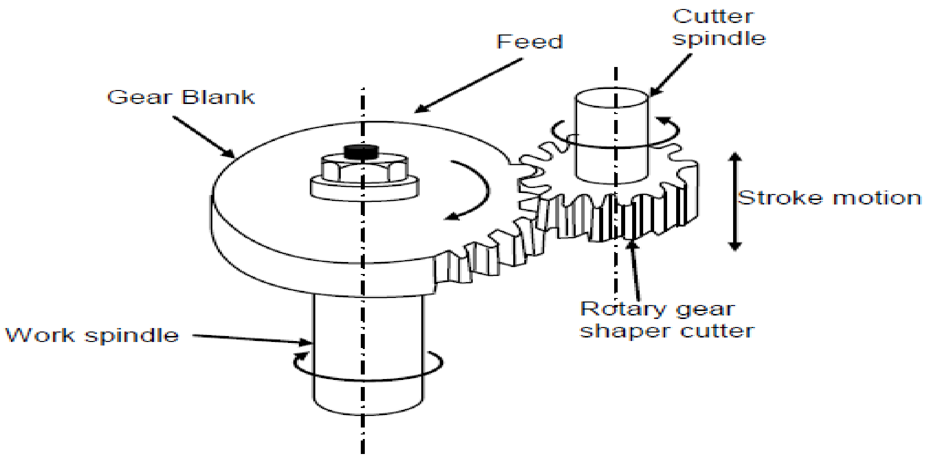
Experiment No-06

Aim:

Study of working principle of shaper/planer/slotting machine.

Study (Shaper Machine):

- This process uses a pinion shaped cutter carrying clearance on the tooth face and sides and a hole at its centre for mounting it on a stub arbor or spindle of the machine.
- The cutter is mounted by keeping its axis in vertical position.
- It is made to reciprocate up and down along the vertical axis up to a pre decided amplitude.
- Both the cutter and the gear blank are set to rotate at a very low RPM about their axis.
- The relative rpm of both (cutter and blank) can be fixed to any of the available value with the help of a gear train.
- This way all the cutting teeth of cutter come is action one-by-one giving sufficient time for their cooling and incorporating a longer tool life.
- The principle of gear cutting by this process as explained above is depicted in the Figure below.



Process of Gear Cutting by Shaper Cutter

SHAPING MACHINE

The main parameters to be controlled in the process are described below

Cutting Speed:

- Shaper cutter can move vertically upward and downward during the operation.
- The downward movement of the cutter is the cutting stroke and its speed (linear) with which it comes down is the cutting speed.
- Length of cutting stroke can be adjusted to any value out of available values on the machine

Indexing motion:

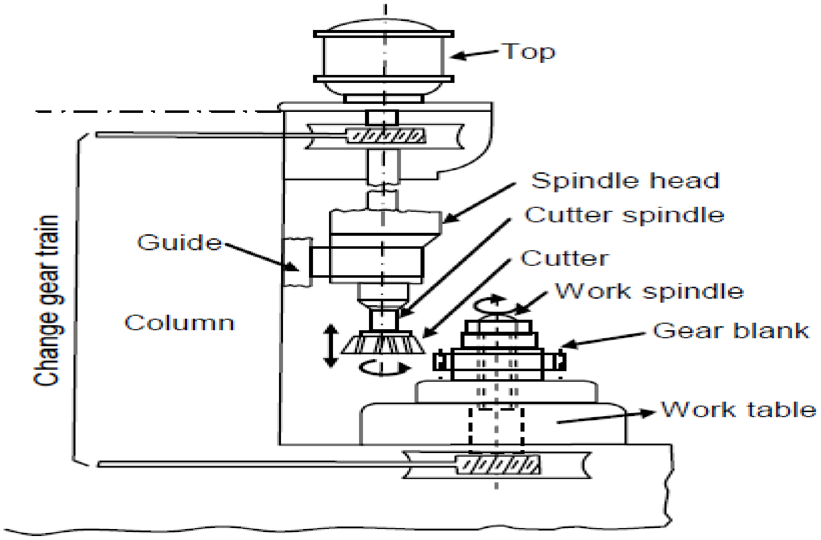
- Indexing motion is equivalent to feed motion in the gear shaping operation.

Slow rotations of the gear cutter and work piece provide the circular feed to the operation.

- These two rpms are adjusted with the help of a gear train.

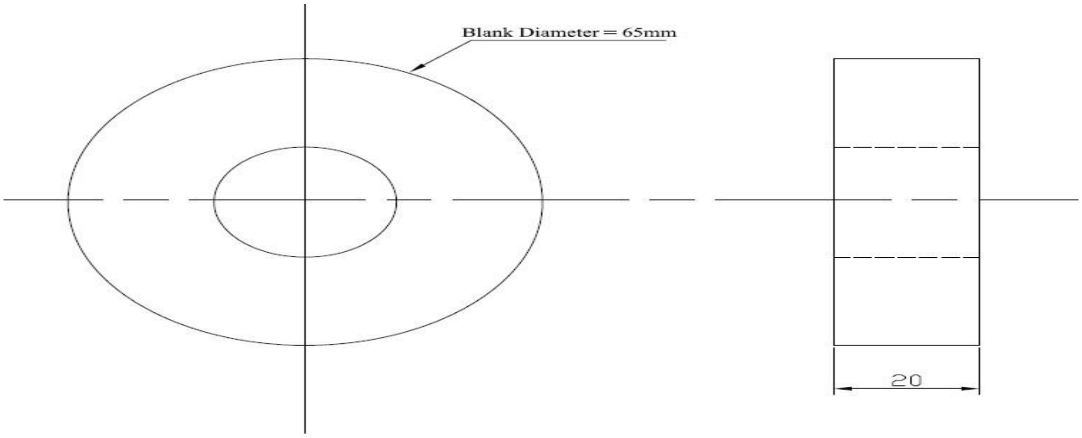
Depth Of Cut:

- The required depth is maintained gradually by cutting the teeth into two or three pass.
- In each successive pass, the depth of cut is increased as compared to its previous path.
- This gradual increase in depth of cut takes place by increasing the value of linear feed in return stroke.

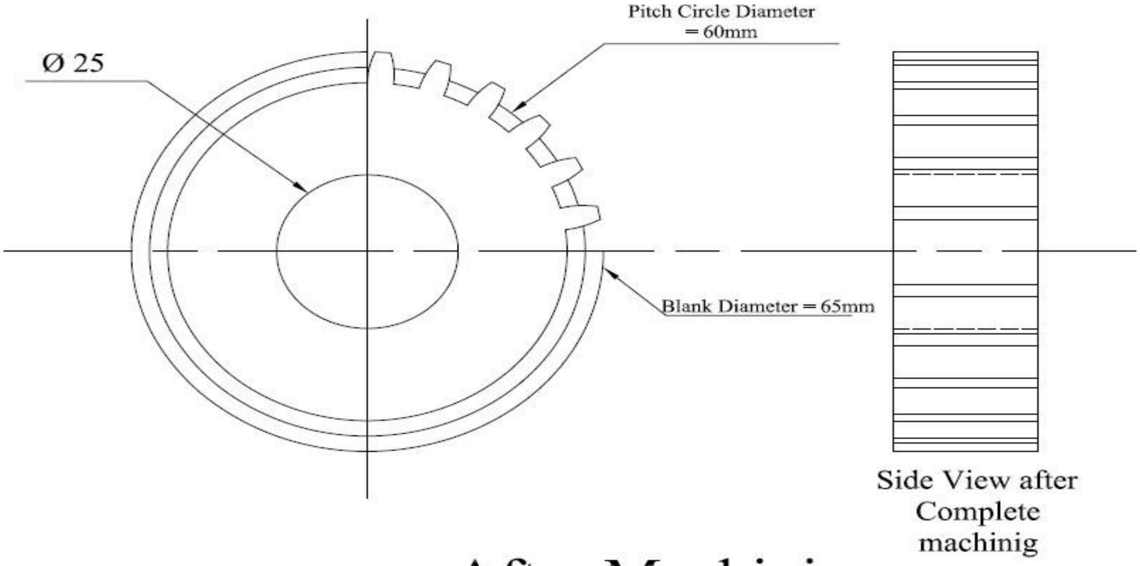


Setup for Gear Shaping Machine

- A Schematic representation of gear shaper is shown above with various parts
- The main advantage of gear shaper is that the process can be used to make a variety of gears and the cycle time for producing one work piece is very less compared to many other processes. Close tolerances can be maintained.
- The main disadvantage is that there is no cutting in the return stroke.
- The process cannot be used to manufacture worm and worm wheel, which is a special type of gear.



Before Machining



After Machining

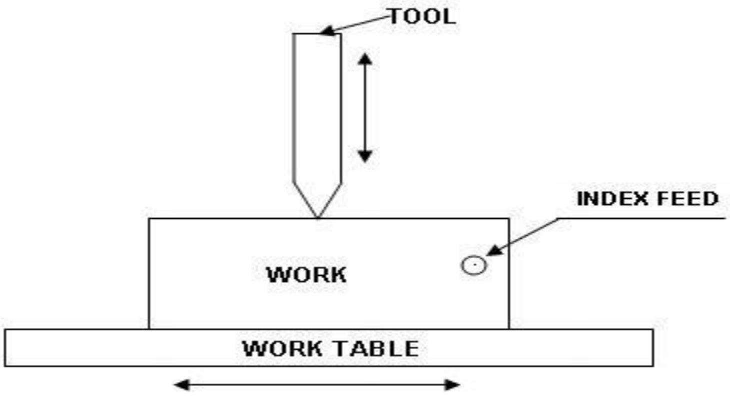
Study (Planing Machine):

Introduction

Planing is one of the basic operations performed in machining work and is primarily intended for machining large flat surfaces. These surfaces may be horizontal, vertical or inclined. In this way, the function of a planning machine is quite similar to that of a shaper except that the former is basically designed to undertake machining of such large and heavy jobs which are almost impractical to be machined on a shaper or milling, etc. It is an established fact that the planning machine proves to be most economical so far as the machining of large flat surfaces is concerned. However, a planing machine differs from a shaper in that for machining, the work, loaded on the table, reciprocates past the stationary tool in a planer, whereas in a shaper the tool reciprocates past the stationary work.

Working Principle of a Planer:

The principle involved in machining a job on a planer is illustrated in fig. Here, it is almost a reverse case to that of a shaper. The work is rigidly held on the worktable or a platen of the machine. The tool is held vertically in the toolhead mounted on the cross rail. The worktable, together with the job, is made to reciprocate past the vertically held tool. The indexed feed after each cut is given to the tool during the idle stroke of the table.



Working Principle of a Planer

Specifications:

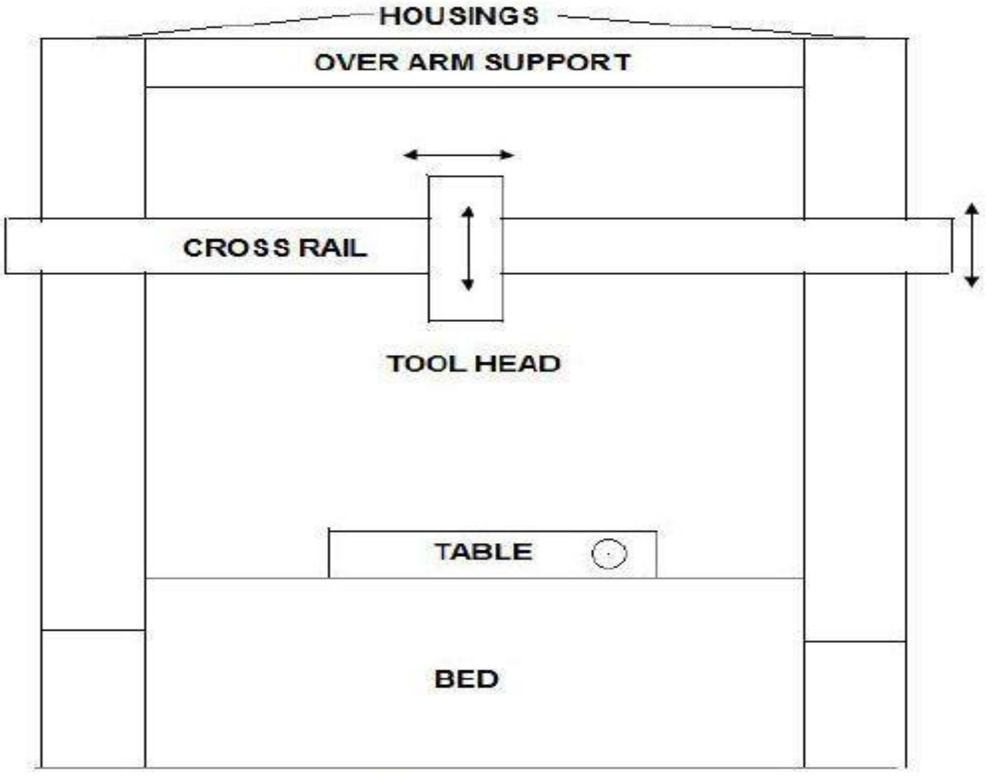
Horizontal distance between two vertical housings:
Vertical distance between tabletop and the cross rail: 800mm
Maximum length of table travel: 1350mm
Length of bed: 2025mm
Length of table: 1425mm
Method of driving – Individual
Method driving table – Geared
H.P. of motor: 3 H.P. & 1 H.P.

Standard or double housing planer:
This is the most commonly used type of planer. It consists of two vertical housings or columns, one on each side of the bed. The housings carry vertical or scraped ways. The cross-rail is fitted between the two housings and carries one or two tool heads. The worktable is mounted over the bed. Some planers may fit with side tool heads fitted on the vertical columns.

Main parts of a planer
A planer consists of the following main parts as illustrated by means of a block diagram in fig.

- Bed
- Table
- Housings or columns
- Cross – rail
- Tool head
- Controls

These machines are heavy duty type and carry a very rigid construction. They employ high speeds for cutting but the size of work they can handle is limited to the width of their table i.e. the horizontal distance between the columns.



MAIN PARTS OF A PLANER

PLANNING
MACHINE

Extremely large and heavy castings, like machine beds, tables, plates, slides, columns, etc., which normally carry sliding surfaces like guide ways or dovetails on their longitudinal faces, are usually machined on these machines. Also because of long table and larger table travel, on either side of the columns, it is possible to hold a number of work pieces in a series over the bed length and machine them together. This will affect a substantial saving in machining time. Further because of no. of tool heads the surfaces can be machined simultaneously. This effects further reduction in machining time. Also because of high rigidity of high rigidity of the machine and robust design of the cutting tools heavier cuts can be easily employed, which leads to quicker metal removal and reduced machining time. Thus, an overall picture emerges that the employment of this type of machine apart from its capacity to handle such heavy and large jobs which are difficult to be handled on other machines, leads to faster machining and reduced machining time and hence to economical machining. However considerable time is used in setting up a planer.

Drive Mechanisms:

Four different methods are employed for driving the table of a planer. They are:

- Crank drive
- Belt drive
- Direct reversible drive
- Hydraulic drive

Study (Slotting Machine):

Theory:

A Slotting machine or slotter has its own importance for a few classes of work. Its main use is in cutting different types of slots and it certainly proves to be most economical so far as this kind of work is concerned. Its other uses are in machining irregular shapes, circular surfaces and other remarked profiles, both internal as well as external. Its construction is similar to that of vertical shaper. Its ram moves vertically and the tool cuts during the downward stroke only.

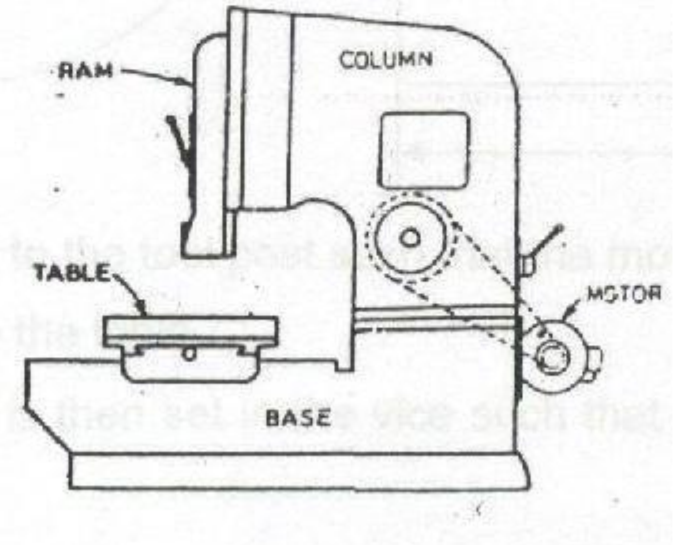


Fig. Main parts of a slotter.

Main Parts of a Slotter:

Base: It is heavy cast iron construction and is also known as bed. It acts as support for the column, the driving mechanism, ram, table and all other fittings. At its top it carries horizontal ways, along which the table can be traversed.

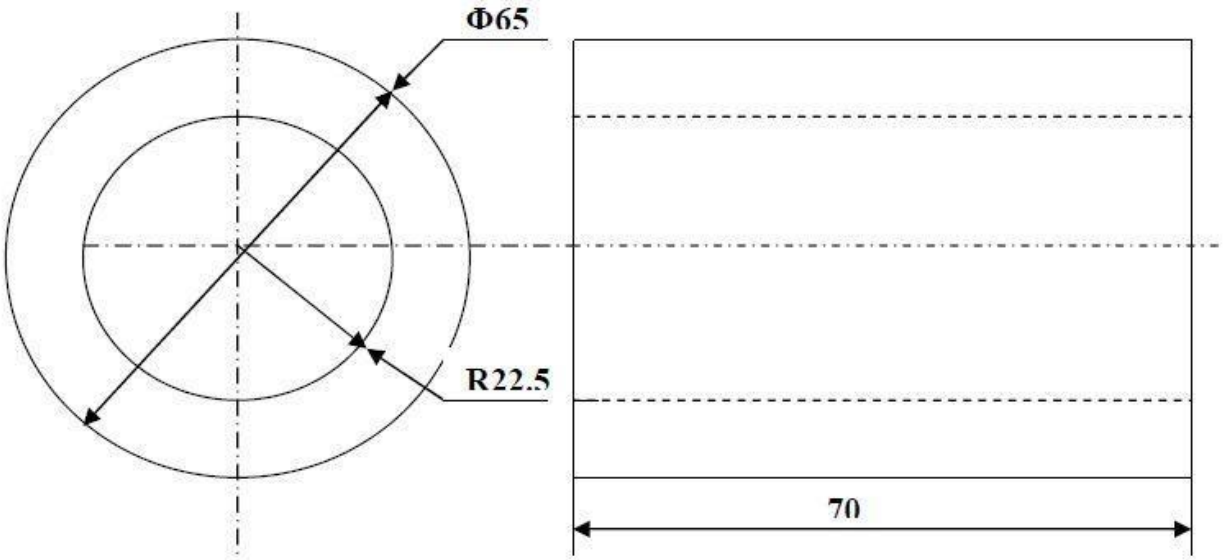
Column: It is another heavy cast iron body which acts as a housing for the driving mechanism. At its front carries vertical ways, along which the ram moves up and down.

Table: Usually, a circular table is provided on slotting machines. In some heavy-duty slotters, either rectangular or circular table can be mounted. On the top of table are provided T- slots to clamp the work or facilitate the use of fixtures etc.

Ram: It moves in vertical direction on the guide ways provided in front of the column. At its bottom, it carries the tool post in which the tool is held. The cutting action takes place during the downward movement of the ram.

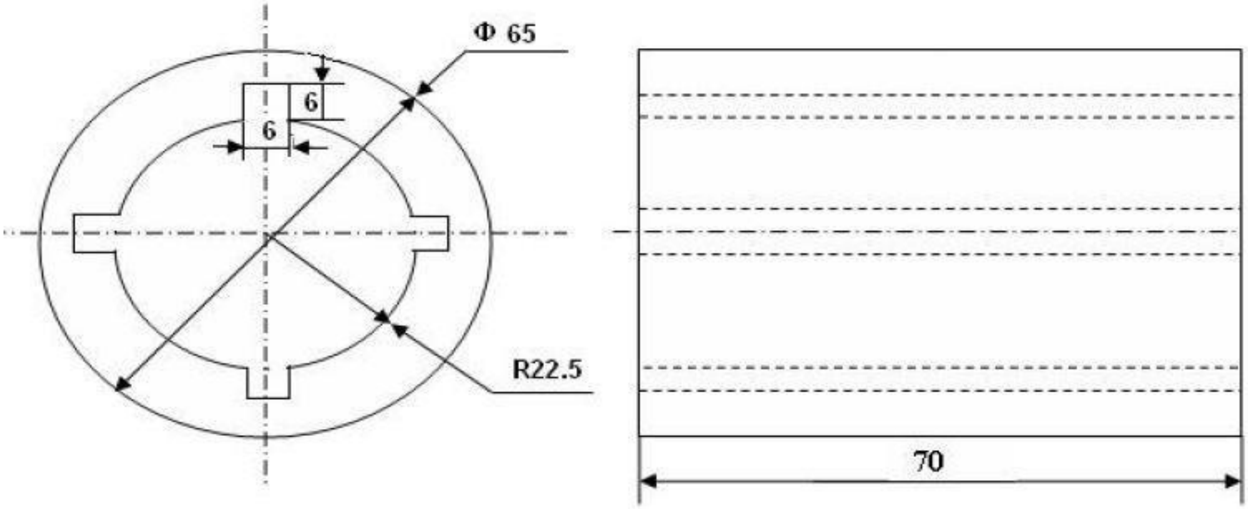
Procedure:

GIVEN WORK PIECE



- The tool is fixed to the tool post such that the movement should be exactly perpendicular to the table.
- The work piece is then set in the vice such that the tool is just above the work piece. Adjust the length of the stroke of the ram.

REQUIRED WORK PIECE



ALL DIMENSIONS ARE IN MM

- Slotting operation is performed and make one slot on the work piece to the required dimensions.

- Then bring the tool to the initial position
- Rotate the worktable by an angle 90_0 and continue the process for the second slot.
- Repeat the process for the remaining slots.

Precautions:

- The work piece should be set securely and rigidly in the vice.
- Before starting the machine make sure that the work, vice, tool, and ram are securely fastened.
- Check that the tool and tool holder will clear the work and also clear the column on the return stroke.
- Make sure that the axis of the work piece is parallel to the line of action of tool.
- Never attempt to adjust a machine while it is in motion.
- Suitable feeds and depth of cut should be maintained uniformly.
- Always feed will be given to the work in the backward stroke only.

Result:

The job is completed successfully and safely.

Working with surface and cylindrical grinding machine.

Experiment No-07

Aim:

To study the working principle of surface and cylindrical grinding machine.

Apparatus required:

- Grinding machine
- Grinding Wheel
- Vernier Caliper

Material Required (Surface Grinding Machine):

- MS / CI plate 12mm X 50mm X 75mm

SURFACE GRINDING MACHINE

Procedure:

- First the work piece is placed on the magnetic chuck.
- The positioning of the work piece is aligned at right angles to the grinding wheel and exactly parallel to the sides of the magnetic chuck by using slip gauges if necessary.
- The magnetic chuck is switched on and the powerful electromagnet holds the job firmly in position.
- Now the spindle is turned on and the grinding wheel is just touched the work piece surface to mark its zero / reference position.
- Now the required feed, either totally or in steps, is given to the grinding wheel and the wheel is traversed all over the work piece.
- Same procedure is repeated until the required dimensions are achieved.
- Care should be taken for maintaining the surface finish.
- Finally, the dimensions are checked using either a Vernier caliper or a screw gauge.

Result:

Thus, plain surface grinding is performed on the given work piece up to the required Dimensions.

Before Grinding

After Grinding

Materials Required (Cylindrical Grinding):

- Cast iron work piece

CYLINDRICAL GRINDING
MACHINE

Procedure:

- First the given work piece is preliminarily finished to the pre-required dimensions on a lathe before beginning the grinding process.
- Now the work piece is fitted in the chuck of the cylindrical grinding machine.
- The grinding wheel is just touched with the work piece and is taken as the zero reference.
- Coolant circulation is switched on and the grinding wheel is engaged with the work piece.
- Both the work piece and the grinding wheel roll on contact with each other like two gears in mesh.
- Now slowly the wheel is moved over the entire length of the work piece to get the grinded finish.
- After one feed is over, the grinding wheel is moved further towards the axis of the work piece and the process is repeated until the required dimensions are achieved.
- Finally, the dimensions are checked using a vernier caliper.

Result:

Thus, cylindrical grinding is performed on the given work piece to the given dimensions.

A study on the importance of coolant during machining

Experiment No-08
Aim:

To study the importance and effect of coolant during the machining process on surface finish, tool life, and temperature rise.

Theory:

It is already realised that the cutting temperature, particularly when it is quite high, is very detrimental for both cutting tools and the machined jobs and hence need to be controlled, i.e., reduced as far as possible without sacrificing productivity and product quality. The methods generally employed for controlling machining temperature and its detrimental effects are:

- Proper selection of cutting tools; material and geometry
- Proper selection of cutting velocity and feed
- Proper selection and application of cutting fluid

o **Selection of material and geometry of cutting tool for reducing cutting temperature and its effects**

Cutting tool material may play significant role on reduction of cutting temperature depending upon the work material.

As for example,

- PVD or CVD coating of HSS and carbide tools enables reduce cutting temperature by reducing friction at the chip-tool and work-tool interfaces.
- In high-speed machining of steels lesser heat and cutting temperature develop if machined by cBN tools which produce lesser cutting forces by retaining its sharp geometry for its extreme hardness and high chemical stability.
- The cutting tool temperature of ceramic tools decrease further if the thermal conductivity of such tools is enhanced (by adding thermally conductive materials like metals, carbides, etc in Al_2O_3 or Si_3N_4) Cutting temperature can be sizeably controlled also by proper selection of the tool geometry in the following ways.
- large positive tool–rake helps in reducing heat and temperature generation by reducing the cutting forces, but too much increase in rake mechanically and thermally weakens the cutting edges compound rake, preferably with chip–breaker, also enables reduce heat and temperature through reduction in cutting forces and friction even for same amount of heat generation, the cutting temperature decreases with the decrease in the principal cutting-edge angle, φ as

  $$\theta_C \;\alpha\; [V_C^{0.5}(s_o \sin\phi)^{0.25}]$$

- nose radiusing of single point tools not only improves surface finish but also helps in reducing cutting temperature to some extent.

o Selection of cutting velocity and feed

Cutting temperature can also be controlled to some extent, even without sacrificing MRR, by proper or optimum selection of the cutting velocity and feed within their feasible ranges. The rate of heat generation and hence cutting temperature are governed by the amount of cutting power consumption, P_C where;

$$P_C = P_Z.V_C = ts_o\tau_s fV_C$$

So apparently, increase in both s_o and V_C raise heat generation proportionately. But increase in V_C, though further enhances heat generation by faster rubbing action, substantially reduces cutting forces, hence heat generation by reducing τ_s and also the form factor f. The overall relative effects of variation of V_C and s_o on cutting temperature will depend upon other machining conditions. Hence, depending upon the situation, the cutting temperature can be controlled significantly by optimum combination of V_C and s_o for a given MRR.

o **Control of cutting temperature by application of cutting fluid**

Cutting fluid, if employed, reduces cutting temperature directly by taking away the heat from the cutting zone and indirectly by reducing generation of heat by reducing cutting forces

(ii) Purposes of application of cutting fluid in machining and grinding.

The basic purposes of cutting fluid application are:

- Cooling of the job and the tool to reduce the detrimental effects of cutting temperature on the job and the tool
- Lubrication at the chip–tool interface and the tool flanks to reduce cutting forces and friction and thus the amount of heat generation.
- Cleaning the machining zone by washing away the chip – particles and debris which, if present, spoils the finished surface and accelerates damage of the cutting edges
- Protection of the nascent finished surface – a thin layer of the cutting fluid sticks to the machined surface and thus prevents its harmful contamination by the gases like SO_2, O_2, H_2S, N_xO_y present in the atmosphere.

However, the main aim of application of cutting fluid is to improve machinability through reduction of cutting forces and temperature, improvement by surface integrity and enhancement of tool life.

(iii) Essential properties of cutting fluids

To enable the cutting fluid, fulfil its functional requirements without harming the Machine – Fixture – Tool – Work (M-F-T-W) system and the operators, the cutting fluid should possess the following properties:

- **For cooling:**
    - high specific heat, thermal conductivity and film coefficient for heat transfer
    - spreading and wetting ability
- **For lubrication:**
    - high lubricity without gumming and foaming
    - wetting and spreading
    - high film boiling point
    - friction reduction at extreme pressure (EP) and temperature
    - Chemical stability, non-corrosive to the materials of the M-F-T-W system
    - less volatile and high flash point
    - high resistance to bacterial growth
    - odourless and preferably colorless
    - nontoxic in both liquid and gaseous stage
    - easily available and low cost. life.

iv) Principles of cutting fluid action

The chip-tool contact zone is usually comprised of two parts; plastic or bulk contact zone and elastic contact zone as indicated in Fig. 1.

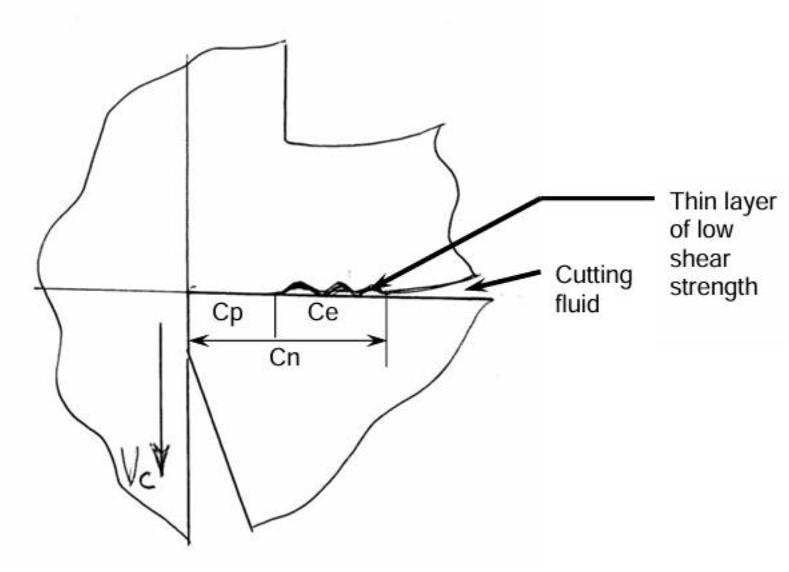


Fig. 1 Cutting fluid action in machining.

The cutting fluid cannot penetrate or reach the plastic contact zone but enters in the elastic contact zone by capillary effect. With the increase in cutting velocity, the fraction of plastic contact zone gradually increases and covers almost the entire chip-tool contact zone as indicated in Fig. 2. Therefore, at high-speed machining, the cutting fluid becomes unable to lubricate and cools the tool and the job only by bulk external cooling.

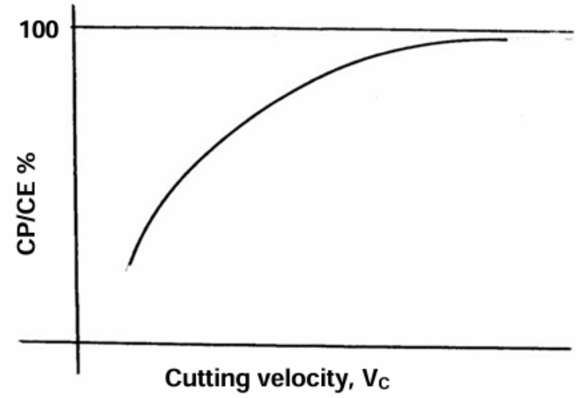


Fig. 2 Apportionment of plastic and elastic contact zone with increase in cutting velocity.

The chemicals like chloride, phosphate or sulphide present in the cutting fluid chemically reacts with the work material at the chip under surface under high pressure and temperature and forms a thin layer of the reaction product. The low shear strength of that reaction layer helps in reducing friction. To form such solid lubricating layer under high pressure and temperature some extreme pressure additive (EPA) is deliberately added in reasonable amount in the mineral oil or soluble oil. For extreme pressure, chloride, phosphate or sulphide type EPA is used depending upon the working temperature, i.e. moderate ($200^oC \sim 350^oC$), high ($350^oC \sim 500^oC$) and very high ($500^oC \sim 800^oC$) respectively.

(v) Types of cutting fluids and their application

Generally, cutting fluids are employed in liquid form but occasionally also employed in gaseous form. Only for lubricating purpose, often solid lubricants are also employed in machining and grinding. The cutting fluids, which are commonly used, are:

- **Air blast or compressed air only.**

Machining of some materials like grey cast iron become inconvenient or difficult if any cutting fluid is employed in liquid form. In such case only air blast is recommended for cooling and cleaning

- **Water**

For its good wetting and spreading properties and very high specific heat, water is considered as the best coolant and hence employed where cooling is most urgent.

- **Soluble oil**

Water acts as the best coolant but does not lubricate. Besides, use of only water may impair the machine-fixture-tool-work system by rusting So oil containing some emulsifying agent and additive like EPA, together called cutting compound, is mixed with water in a suitable ratio (1 ~ 2 in 20 ~ 50). This milk like white emulsion, called soluble oil, is very common and widely used in machining and grinding.

- **Cutting oils**

Cutting oils are generally compounds of mineral oil to which are added desired type and amount of vegetable, animal or marine oils for improving spreading, wetting and lubricating properties. As and when required some EP additive is also mixed to reduce friction, adhesion and BUE formation in heavy cuts.

• **Chemical fluids**

These are occasionally used fluids which are water based where some organic and or inorganic materials are dissolved in water to enable desired cutting fluid action. There are two types of such cutting fluid.

– Chemically inactive type – high cooling, anti-rusting and wetting but less lubricating

– Active (surface) type – moderate cooling and lubricating.

• **Solid or semi-solid lubricant**

Paste, waxes, soaps, graphite, Moly-disulphide (MoS_2) may also often be used, either applied directly to the workpiece or as an impregnant in the tool to reduce friction and thus cutting forces, temperature and tool wear.

• **Cryogenic cutting fluid**

Extremely cold (cryogenic) fluids (often in the form of gases) like liquid CO_2 or N_2 are used in some special cases for effective cooling without creating much environmental pollution and health hazards.

Precautions:

- Ensure proper mixing ratio of coolant.
- Do not direct coolant on electrical parts.
- Use safety googles while machining.
- Clean the machine and working area after completion.

Result:

The use of coolant reduces tool temperature significantly, improves surface finish, and increases tool life. Therefore, coolant plays a vital role in improving machining efficiency and quality.