Q.P. Code : 50603

(3 Hours) **[Total Marks : 80]**

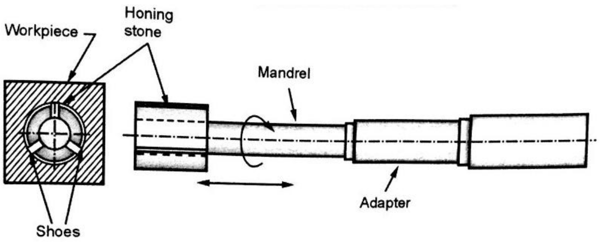
N.B. **(1)** Question no. **1** is **compulsory.**
(2) Attempt any **three** questions out of remaining **five** questions.
(3) Illustrate your answer with **necessary** sketch wherever **necessary**.
(4) Figures to the **right** indicate full **marks**.

1. Attempt any **FOUR** of the following : **(20)**

(a) Write short note on Honning Machine.

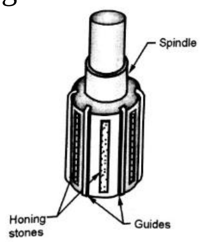
Honing is an abrading process, used for finishing already machined surfaces.

- Mostly, honing is used for finishing internal cylindrical surfaces such as drilled holes.
- The tool used during the process is called as hone which is bonded with abrasive stones and made in the form of a stick.
- Honing can be done by hand or using machines.
- Figure shows a hand honing tool and honing process



Hand honing tool and honing process

- The honing stones are held in holder or mandrel and forced outwards hydraulics or mechanical pressure.
- To form the honing stones aluminium oxide, silicon carbide or diamond grains of suitable grit are bonded in resinoid, vitrified or shellac bond.
- In some cases vertical honing machines are also used.
- Figure shows the honing tool used in vertical honing machines.



Honing tooi head for vertical machines

- Out of all the surface finishing operations, honing enables maximum stock removal, still it is not a metal removing process..
- Due to this ability, honing is used for correcting slight out of roundness or taper.
- The general amount of stock removed by honing process is from 0.1 mm to 0.25 mm, though it is capable of removing the stock upto 0.75 mm.
- The speed used for honing process is in the range of 10 to 30 m/min.

(b) What are the features of a horizontal CNC machine?

involves the replacement of convention hardwired NC controller unit by a small computer which performs some or all of the basic NC functions by storing the programme in its memory.

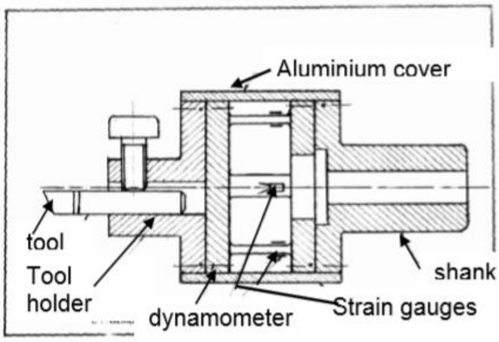
Control features:

- program editing
- part program storage
- program storage in RAM
- controlled flexibility
- fixed cycles and programming subroutine

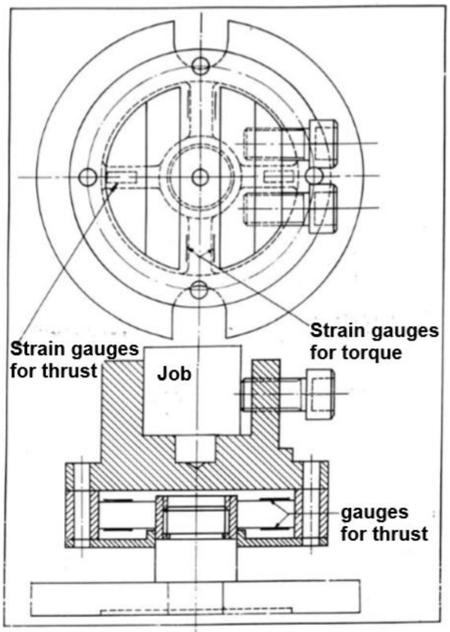
constructional features: spindle drives

- axes feed drives
- slideways for machine
- recirculating ball screws

(c) Explain what is a tool dynamometer with a neat sketch.



Schematic view of a strain gauge type 2 – D turning dynamometer.



Schematic view of construction of a strain gauge type drilling dynamometer.

(d) State the factors for selection of grinding wheel.

The factors on which the selection of grinding wheel depends are as follows

Constant Factors:

- Workpiece material: It affects the selection of abrasive, grain, size, grade, structure and bond.
- Amount of material to be removed.
- Contact area: For small contact area finer and harder wheels are used whereas, for large contact area coarse and softer wheels are used.
- Type of grinding machine

Variable factors:

- Speed of the wheel: It affects the selection of grade and bond.
- Speed of the workpiece: Higher workpiece speed indicates more material is to be ground in given time hence, greater wear on the wheel. For this, harder wheel is selected.
- Machine condition

(e) Explain the steps for designing the broach tools.

The main elements to be decided in designing a broach are as follows:

- tool pitch
- total rise
- length of cutting portion of broach
- load on broach

Let,

n = total number of teeth in the broach

L = Effective length of a broach in mm

p = tooth pitch in mm

l = length to broached in mm

t = rise per tooth in mm

n_s = no. of safety teeth preceding cutting teeth (if any)

n_f = no. of finishing teeth (generally 3 to 6)

(1) Tooth pitch (p):

Pitch is given by $p = 1.75\sqrt{l}$

(2) Rise per tooth (t):

Rise per tooth depends upon:

- Shape and size of hole
- Type of material being broached
- Force available at the machine

Total rise = No. of teeth in the broach x Rise per tooth

Total rise = n x t

(3) Total number of teeth in the broach (n):

Total number of teeth in the broach are given by,

n = Roughing teeth + finishing teeth

$$n = \left(\frac{Depth\ of\ cut}{Cut\ per\ tooth}\right) + \left(n_s + n_f\right)$$

$$n = \left(\frac{Total\ rise}{Rise\ per\ tooth}\right) + \left(n_s + n_f\right)$$

(4) Effective length (L):

Effective length of a broach is given by,

L = No. of teeth in the broach x pitch

$$\therefore L = n \times p$$

(5) Tooth load on a broach (F):

Calculation of the load on a broach helps in judging the capability of the machine for pulling or pushing the broach. It also helps in ascertaining whether the broach will stand during the operation or fail. This tooth load is given by,

a. For round holes:

F = Hole circumference x N x t x K

$$\therefore F = \pi d \times N \times t \times K$$

b. For square holes:

F = Hole perimeter x N x t x K

$$\therefore F = 4H \times N \times t \times K$$

Where, d = finished hole diameter in mm

N = maximum number of teeth cutting at a time,

t = rise per tooth in mm

K = force required to cut 1 mm^2 of metal at a given rise per tooth

H = finished length of one side of the square hole, in mm

(6) Total Effective length (L_T):

L_T = length of roughing teeth + length of finishing teeth

2. (a) How is gear manufactured? and also explain the limitations of the different processes. **(10)**

Several processes for making gears

- Form cutting (form-milling)
- Gear generating (Hobbing, Shaping)

Gears are cut from gear blanks (disks)

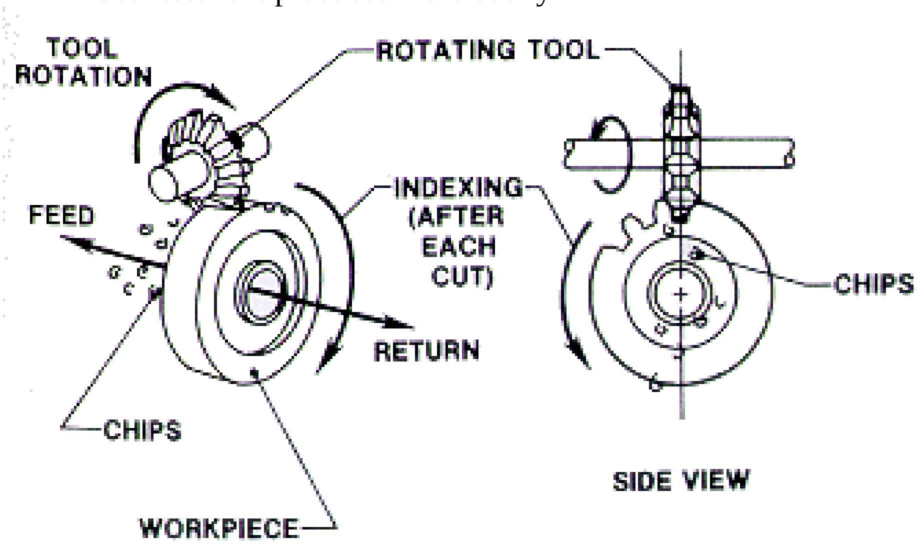
The following may be used to cut gear teeth:

- A form cutter
- Broaches (often for internal teeth)
- A single-point cutting tool guided by a template
- A pinion-shaped cutter
- A rack-shaped cutter
- A hob

Gear milling

- Uses a rotating form cutter
- Gear blanks are indexed after each cut

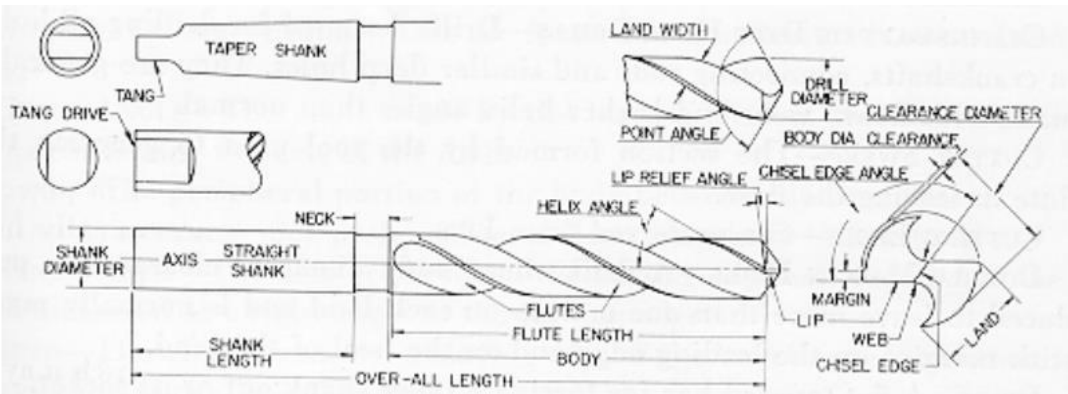
- Is a low production process
- Gear teeth are produced individually

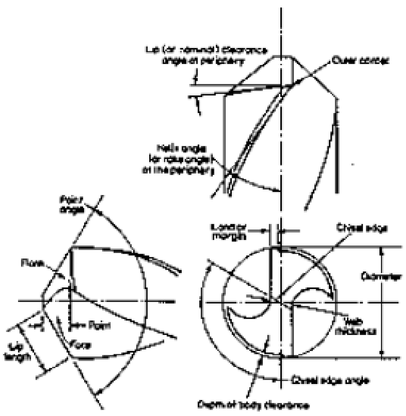


(b) Draw and explain the different terms of a twist drill. **(6)**

Axis: The imaginary straight line which forms the longitudinal center line of the drill
Back Taper: A slight decrease in diameter from front to back in the body of the drill
Body: The portion of the drill extending from the shank or neck to the outer corners of the cutting lips
Body Diameter Clearance: That portion of the land that has been cut away so it will not rub against the walls of the hole
Built-Up Edge: An adhering deposit of nascent material on the cutting lip or the point of the drill
Cam Relief: The relief from the cutting edge to the back of the land, produced by a cam actuated cutting tool or grinding wheel on a relieving machine
Chip Breaker: Nicks or Grooves designed to reduce the size of chips; they may be steps or grooves in the cutting lip or in the leading face of the land at or adjacent to the cutting lips
Chip Packing: The failure of chips to pass through the flute during cutting action
Chipping: The breakdown of a cutting lip or margin by loss of fragments broken away during the cutting action
Chisel Edge: The edge at the end of the web that connects the cutting lips
Chisel Edge Angle: The angle included between the chisel edge and the cutting lip, as viewed from the end of the drill
Clearance: The space provided to eliminate undesirable contact between the drill and the workpiece
Clearance Diameter: The diameter over the the cut away portion of the drill lands
Axis: The imaginary straight line which forms the longitudinal center line of the drill
Back Taper: A slight decrease in diameter from front to back in the body of the drill
Body: The portion of the drill extending from the shank or neck to the outer

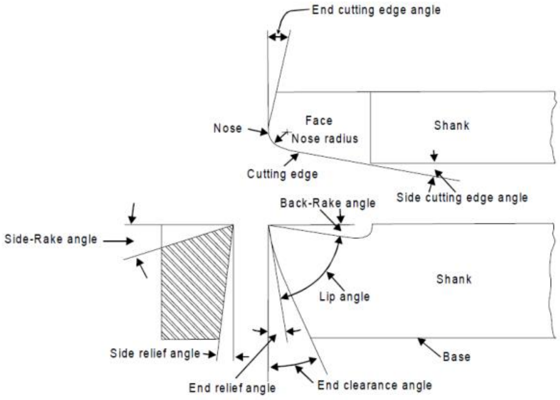
corners of the cutting lips
Body Diameter Clearance: That portion of the land that has been cut away so it will not rub against the walls of the hole
Built-Up Edge: An adhering deposit of nascent material on the cutting lip or the point of the drill
Cam Relief: The relief from the cutting edge to the back of the land, produced by a cam actuated cutting tool or grinding wheel on a relieving machine
Chip Breaker: Nicks or Grooves designed to reduce the size of chips; they may be steps or grooves in the cutting lip or in the leading face of the land at or adjacent to the cutting lips
Chip Packing: The failure of chips to pass through the flute during cutting action
Chipping: The breakdown of a cutting lip or margin by loss of fragments broken away during the cutting action
Chisel Edge: The edge at the end of the web that connects the cutting lips
Chisel Edge Angle: The angle included between the chisel edge and the cutting lip, as viewed from the end of the drill
Clearance: The space provided to eliminate undesirable contact between the drill and the workpiece
Clearance Diameter: The diameter over the the cut away portion of the drill lands





(c) Write in brief about tool signature. **(4)**

Convenient way to specify tool angles by use of a standardized abbreviated system is known as tool signature. It indicates the angles that a tool utilizes during the cut. It specifies the active angles of the tool normal to the cutting edge. This will always be true as long as the tool shank is mounted at right angles to the workpiece axis.



Elements of tool signature

The seven elements that comprise the signature of a single point cutting tool are always stated in the following order:

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

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

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

3. (a) In an orthogonal cutting, the following observations were made, **(10)**

Rake angle = 10°, Cutting speed = 50 m/min, chip thickness = 0.4mm, uncut chip thickness = 0.148 mm, depth of cut = 2mm, cutting force = 1500N, Thrust force = 1000N. Calculate (i) chip reduction coefficient, (ii) shear angle (iii) shear force (iv) force normal to the shear plane (v) frictional force (vi) normal to frictional force (vii) shear stress (viii) shear strain (ix) coefficient of friction (x) resultant force.

Given:

$\alpha = 10°$

r = 0.4

Ft = FH = 1000 N

Fc = FV = 1500 N

(i) Chip reduction coefficient (k)

$$r = \frac{t_1}{t_2} = \frac{0.148}{0.4} = 0.37$$

$$k = \frac{1}{r} = \frac{1}{0.37} = 2.7$$

(ii) Shear plane angle (Φ)

$$\tan\varphi = \frac{r\ cos\ \alpha}{1 - r\ sin\ \alpha} = \frac{0.37\cos 10}{1 - 0.37\sin 10}$$

φ = 21.27°

(iii) Shear force along the rake face (Fs):

$$F_s = F_c \cdot \cos\varphi - F_t \cdot \sin\varphi = 1500\cos 21.27 - 1000\sin 21.27 = 1035.05\ N$$

(iv) Normal force on the rake face (Fn)

$$F_n = F_t \cdot \cos\varphi + F_c \cdot \sin\varphi = 1000\cos 21.27 + 1500\sin 21.27 = 1476.03\ N$$

(v) Frictional force (F):

$$F = F_c \cdot \sin\alpha + F_t \cdot \cos\alpha = 1500\sin 10 + 1000\cos 10 = 1245.28\ N$$

(vi) Normal reaction:

$$N = F_c \cdot \cos\alpha - F_t \cdot \sin\alpha = 1500\cos 10 - 1000\sin 10 = 1303.56\ N$$

(vii) Shear stress (τ_s):

$$\tau_s = \frac{F_s}{t_1 \cdot w}\sin\varphi = \frac{1035.05}{0.148 \times 2} \times \sin 21.27 = 1268.51\ N/mm^2$$

(viii) Shear strain (γ):

$$\gamma = \frac{\cos\alpha}{\cos(\varphi - \alpha)\sin\varphi} = \frac{\cos 10}{\cos(21.27 - 10)\sin 21.27} = 2.768$$

(ix) Coefficient of friction

Coefficient of friction (μ)

$$\mu = \frac{F_c \cdot \sin\alpha - F_t \cdot \cos\alpha}{F_c \cdot \cos\alpha - F_t \cdot \sin\alpha} = \frac{1500\sin 10 + 1000\cos 10}{1500\cos 10 - 1000\sin 10} = 0.955$$

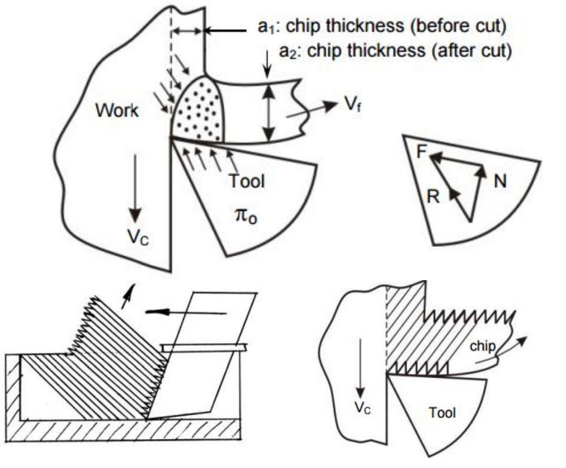
(x) Resultant force (R)

$$R = \sqrt{{F_t}^2 + {F_c}^2} = \sqrt{1000^2 + 1500^2} = 1802.77\ N$$

(b) Explain the mechanism of chip formation. **(6)**

Machining is a semi-finishing or finishing process essentially done to impart required or stipulated dimensional and form accuracy and surface finish to enable the product to • fulfill its basic functional requirements • provide better or improved performance • render long service life. Machining is a process of gradual removal of excess material from the preformed blanks in the form of chips. The form of the chips is an important index of machining because it directly or indirectly indicates : • Nature and behaviour of the work material under machining condition • Specific energy requirement (amount of energy required to remove unit volume of work material) in machining work • Nature and degree of interaction at the chip-tool interfaces. The form of machined chips depend mainly upon : • Work material • Material and geometry of the cutting tool • Levels of cutting velocity and feed and also to some extent on depth of cut • Machining environment or cutting fluid that affects temperature and friction at the chip-tool and work-tool interfaces.

During continuous machining the uncut layer of the work material just ahead of the cutting tool (edge) is subjected to almost all sided compression as indicated in Fig



(a) Shifting of the postcards by partial sliding against each other (b) Chip formation by shear in lamella.

Fig. Piispanen model of card analogy to explain chip formation in machining ductile materials

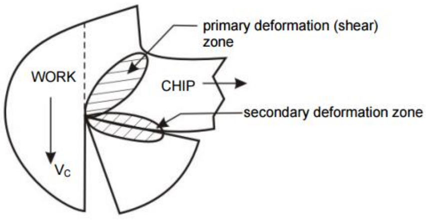


Fig. Primary and secondary deformation zones in the chip.

The basic two mechanisms involved in chip formation are • Yielding – generally for ductile materials • Brittle fracture – generally for brittle materials During machining, first a small crack develops at the tool tip as shown in Fig

At the sharp crack-tip stress concentration takes place. In case of ductile materials immediately yielding takes place at the crack-tip and reduces the effect of stress concentration and prevents its propagation as crack. But in case of brittle materials the initiated crack quickly propagates, under stressing action, and total separation takes place from the parent workpiece through the minimum resistance path as indicated in Fig.

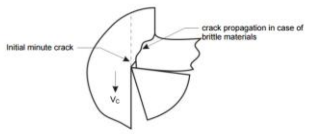


Fig. Development and propagation of crack causing chip separation.

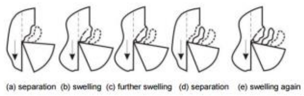


Fig. Schematic view of chip formation in machining brittle materials.

(c) Explain orthogonal rake system in detail. **(4)**

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: πR - πC - πO and Xo - Yo - Zo which are taken in respect of the tool configuration as indicated in Fig.

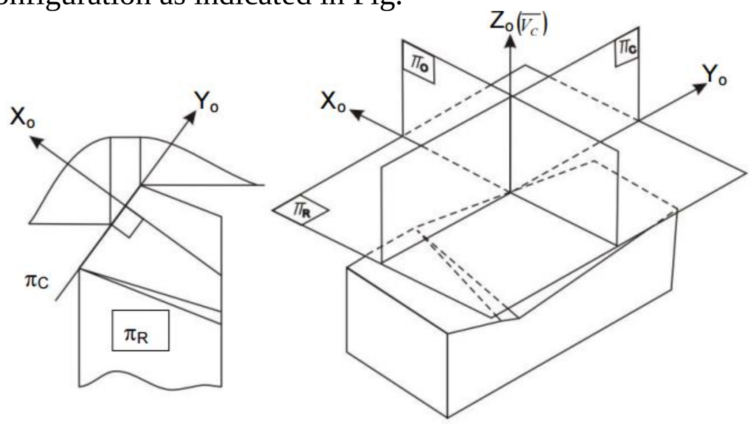


Fig. Planes and axes of reference in ORS

where, πR = Refernce plane perpendicular to the cutting velocity vector, VC πC = cutting plane; plane perpendicular to πR and taken along the principal cutting edge πO = Orthogonal plane; plane perpendicular to both πR and πC and the axes; Xo = along the line of intersection of πR and πO Yo = along the line of intersection of πR and πC Zo = along the velocity vector, i.e., normal to both Xo and Yo axes. The main geometrical angles used to express tool geometry in Orthogonal Rake System (ORS) and their definitions will be clear from Fig.

4. (a) What are the functions of cutting fluid? Explain different types of cutting fluid. **(10)**

Cutting fluids are used in metal machining for a variety of reasons such as improving tool life, reducing workpiece thermal deformation, improving surface finish and flushing away chips from the cutting zone. Practically all cutt ing fluids presently in use fall into one of four categories:

Straight oils
Soluble oils
Semisynthetic fluids
Synthetic fluids

Straight oils are non-emulsifiable and are used in machining operations in an undiluted form. They are composed of a base mineral or petroleum oil and often contains polar lubricants such as fats, vegetable oils and esters as well as extreme pressure additives such as Chlorine, Sulphur and Phosphorus. Straight oils provide the best lubrication and the poorest cooling characteristics among cutting fluids.

Synthetic Fluids contain no petroleum or mineral oil base and instead are formulated from alkaline inorganic and organic compounds along with additives for corrosion inhibition. They are generally used in a diluted form (usual concent ration = 3 to 10%). Synthetic fluids often provide the best cooling performance among all cutting fluids.

Soluble Oil Fluids form an emulsion when mixed with water. The concentrate consists of a base mineral oil and emulsifiers to help produce a stable emulsion. They are used in a diluted form (usual concentration = 3 to 10%) and provide good lubrication and heat transfer performance. They are widely used in industry and are the least expensive among all cutting fluids.

Semi-synthetic fluids are esentially combination of synthetic and soluble oil fluids and have characteristics common to both types. The cost and heat transfer performance of semi-synthetic fluids lie between those of synthetic and sol uble oil fluids.

The principal methods of cutting fluid application include:
Flood Application of Fluid:
a flood of cutting fluid is applied on the workpiece

Jet Application of Fluid:
a jet of cutting fluid is applied on the workpiece directed at the cutting zone

Mist Application of Fluid:
cutting fluid is atomised by a jet of air and the mist is directed at the cutting zone

Cutting Fluid Effects in Machining

The primary functions of cutting fluids in machining are :
Lubricating the cutting process primarily at low cutting speeds
Cooling the workpiece primarily at high cutting speeds
Flushing chips away from the cutting zone
Secondary functions include:
Corossion protection of the machined surface
enabling part handling by cooling the hot surface
Process effects of using cutting fluids in machining include:
Longer Tool Life
Reduced Thermal Deformation of Workpiece
Better Surface Finish (in some applications)
Ease of Chip and Swarf handling

Cutting Fluid Selection Criteria
The principal criteria for selection of a cutting fluid for a given machining

operation are:
Process performance :

Heat transfer performance
Lubrication performance
Chip flushing
Fluid mist generation
Fluid carry-off in chips
Corrosion inhibition
Fluid stability (for emulsions)
Cost Performance
Environmental Performance
Health Hazard Performance

(b) In an orthogonal cutting, the following observations were made, **(6)**
Rake angle = 10°, Cutting speed = 60 m/min, chip thickness = 0.4mm, uncut chip thickness = 0.148 mm, depth of cut = 2mm, cutting force = 2000N, Thrust force = 1250N. Calculate (i) chip reduction coefficient, (ii) shear angle (iii) shear force (iv) force normal to the shear plane (v) frictional force (vi) normal to frictional force (vii) shear stress (viii) shear strain (ix) coefficient of friction (x) resultant force.

Given:
$\alpha = 10°$
r = 0.4
Ft = FH = 1250 N
Fc = FV = 2000 N

(i) Chip reduction coefficient (k)

$$r = \frac{t_1}{t_2} = \frac{0.148}{0.4} = 0.37$$

$$k = \frac{1}{r} = \frac{1}{0.37} = 2.7$$

(ii) Shear plane angle (Φ)

$$\tan\varphi = \frac{r\,cos\,\alpha}{1 - r\,sin\,\alpha} = \frac{0.37\cos 10}{1 - 0.37\sin 10}$$

$\varphi = 21.27°$

(iii) Shear force along the rake face (Fs):

$$F_s = F_c \cdot \cos\varphi - F_t \cdot \sin\varphi = 2000\cos 21.27 - 1250\sin 21.27 = 1410.31\ N$$

(iv) Normal force on the rake face (Fn)

$$F_n = F_t \cdot \cos\varphi + F_c \cdot \sin\varphi = 1250\cos 21.27 + 2000\sin 21.27 = 1890.38\ N$$

(v) Frictional force (F):

$$F = F_c \cdot \sin\alpha + F_t \cdot \cos\alpha = 2000\sin 10 + 1250\cos 10 = 1578.30\ N$$

(vi) Normal reaction:

$$N = F_c \cdot \cos\alpha - F_t \cdot \sin\alpha = 2000\cos 10 - 1250\sin 10 = 1752.55\ N$$

(vii) Shear stress (τ_s):

$$\tau_s = \frac{F_s}{t_1 \cdot w}\sin\varphi = \frac{1410.31}{0.148 \times 2} \times \sin 21.27 = 1728.41\ N/mm^2$$

(viii) Shear strain (γ):

$$\gamma = \frac{\cos\alpha}{\cos(\varphi - \alpha)\sin\varphi} = \frac{\cos 10}{\cos(21.27 - 10)\sin 21.27} = 2.768$$

(ix) Coefficient of friction

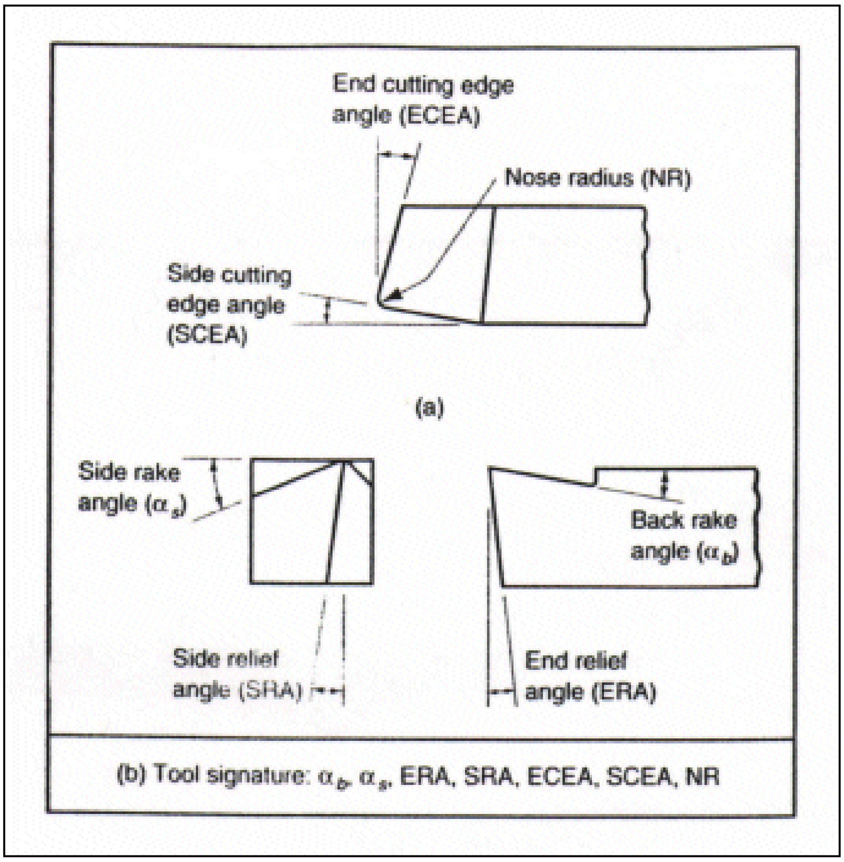
Coefficient of friction (μ)

$$\mu = \frac{F_c \cdot \sin\alpha - F_t \cdot \cos\alpha}{F_c \cdot \cos\alpha - F_t \cdot \sin\alpha} = \frac{2000\sin 10 + 1250\cos 10}{2000\cos 10 - 1250\sin 10} = 0.900$$

(x) Resultant force (R)

$$R = \sqrt{{F_t}^2 + {F_c}^2} = \sqrt{1250^2 + 2000^2} = 2358.49\ N$$

(c) Write notes on single point cutting tools. **(4)**



5. (a) State various machining centers. Describe any one in detail. **(10)**

- It carries a vertical machining spindle (head) which can slide along vertical guide ways provided on the column.
- The vertical head can be tilted in either direction.
- These machines are suitable for machining flat surfaces with deep cavities like the manufacturing of moulds, dies, etc.
- These machines are also very heavy in construction. Figure shows the principal parts of VMC which are similar to HMC.

i. Bed: It is a heavy structure which supports the complete machine and carries guide-ways over its top surface. It is generally made of cast iron.

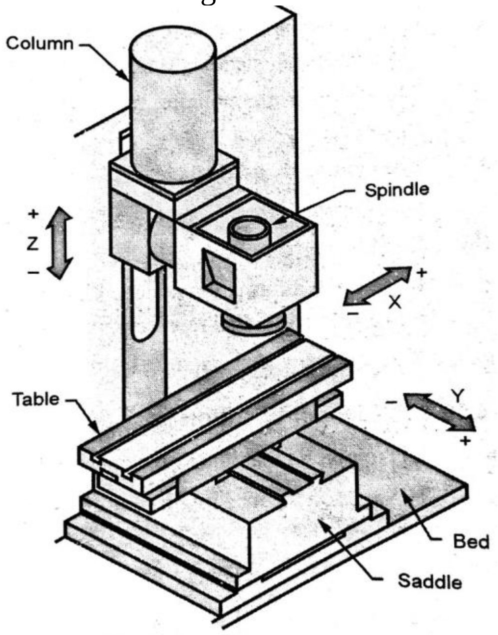
ii. Saddle: It is mounted over the guide ways on the bed and also carries column over it. It provides y axis movement to the machining center.

iii. Table: It is mounted over the guide was provided on the saddle. It is made up of cast iron. For mounting the work holding devices, T slots are provided on the tale. It provides X axis movement to the machining centers.

iv. Column: It is mounted over the saddle. It provides z axis movement to the machining center.

v. Automatic tool changer: It is used to change the tool from the machine spindle, rapidly.

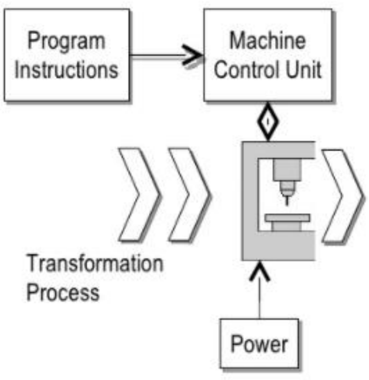
vi. Spindle and servo system: Spindle is mounted on the headstock and it provides z axis movement to the machining center. Servo system consists of servo motors and feedback system. It provides accurate and rapid movement along all the axes.



Vertical machining centre

(b) Explain NC, CNC and DNC machine with block diagram. **(6)**

NC Machine



BASIC BLOCK DIAGRAM

CNC Machine Tool

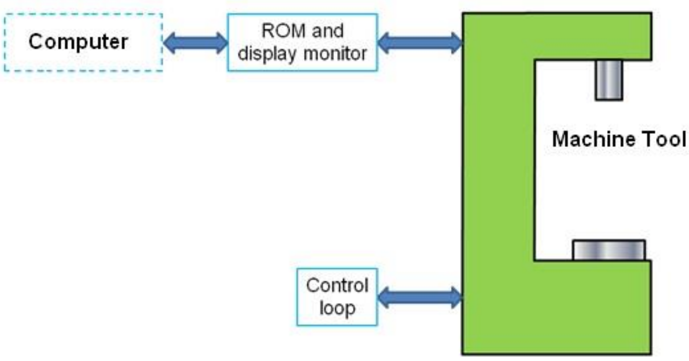


Figure Schematic of a CNC machine Tool

Figure shows a schematic of a machine tool controlled by a computer. It consists of a Machine Control Unit (MCU) and machine tool itself. MCU, a computer is the brain of a CNC machine tool. It reads the part programs and controls the machine tools operations. Then it decodes the part program to provide commands and instructions to the various control loops of the machine axes of motion. The details regarding the construction and working of mechatronics based system have already been studied in last lectures.

CNC systems have a limitation. If the same NC program is used on various machine tools, then it has to be loaded separately into each machine. This is time consuming and involves repetitive tasks. For this purpose direct numerical control (DNC) system is developed. Figure 2 shows the schematic of a DNC system. It consists of a central computer to which a group of CNC machine tools are connected via a communication network. The communication is usually carried out using a standard protocol such as TCP/IP or MAP. DNC system can be centrally monitored which is helpful when dealing with different operators, in different shifts, working on different machines.

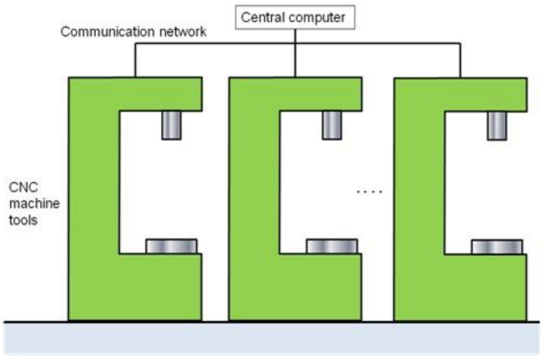


Figure Direct numerical control (DNC) system

(c) Explain automatic tool changer. **(4)**

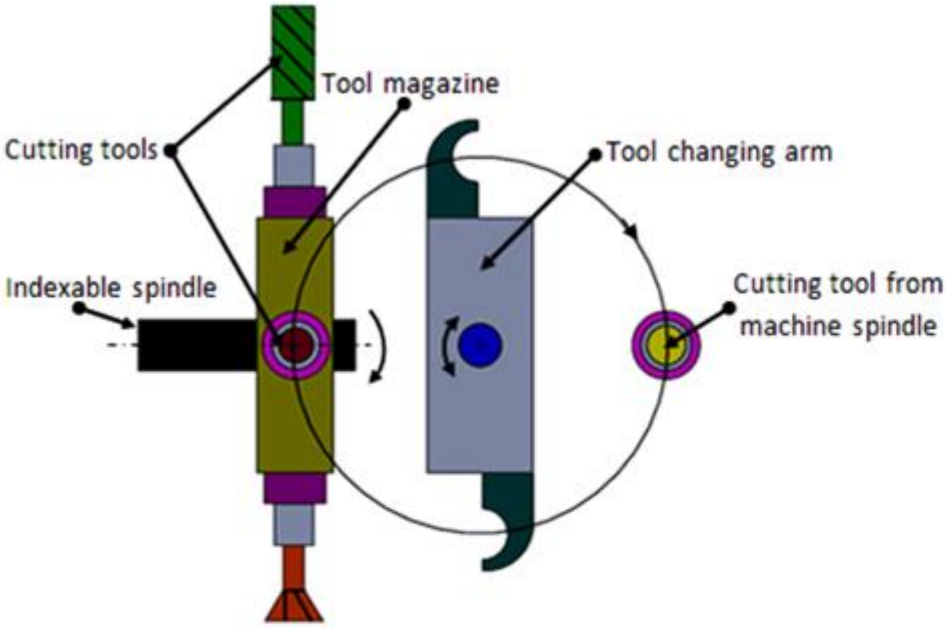


Fig. Automatic tool changer

The tools from the magazines and spindle are exchanged by a tool changer arm (Fig.). The tool change activity requires the following motions:

a. The spindle stops at the correct orientation for the tool change arm to pick the tool from the spindle.
b. Tool change arm moves to the spindle.
c. Tool change arm picks the tool from the spindle.
d. Tool change arm indexes to reach the tool magazine.
e. Tool magazine indexes so that the tool from the spindle can be placed.
f. The tool is placed in the tool magazine.
g. The tool magazine indexes to bring the required tool to the tool change position.
h. Tool change arm picks the tool from the tool magazine.
i. Tool change arm indexes to reach the spindle.
j. New tool is placed in the spindle.
k. Tool change arm moves back to its parking position.

3.1 Advantages of automatic tool changer

- Increase in operator safety by changing tools automatically
- Changes the tools in seconds for maintenance and repair
- Increases flexibility
- Heavy and large multi-tools can easily be handled
- Decreases total production time

6. Write short notes on any **FOUR** : **(20)**

(a) Machinability.

- Machinability of a material indicates how machinable the material is.
- the parameters which affect the machinability of a material are:
  1. physical and mechanical properties of a material
  2. chemical composition and microstructure of a material
  3. cutting conditions
- the general criteria for evaluating machinability are as follows:
  1. Tool life - longer tool life at given cutting speed indicates better machinability
  2. Surface finish - better the surface finish, higher is the machinability
  3. Power consumption - If for metal removal power consumption is low then it indicates better machinability
  4. Cutting Force - Lesser the cutting force for removing the higher volume of metal, higher will be machinability
  5. Shear angle - larger shear angle gives better machinability

(b) Type of coolants.

The purpose of a coolant is to cool the workpiece and grinding wheel during machining and to lubricate the ground surface. Cooling of the workpiece and tool is necessary to eliminate the unwanted effects of heat on both the workpiece and tool.

Lubrication is needed to reduce friction between the tool and workpiece and protect the workpiece from corrosion.

Types of coolants

The following agents are used as coolants:

• oil for machining where very smooth surface is required,

• mineral, white emulsion - emulsion of oil in water with added emulsifiers and antibacterial additives; universally applicable,

• synthetic, clear emusion - emulsion of synthetic oils in water; resistant to bacteria,

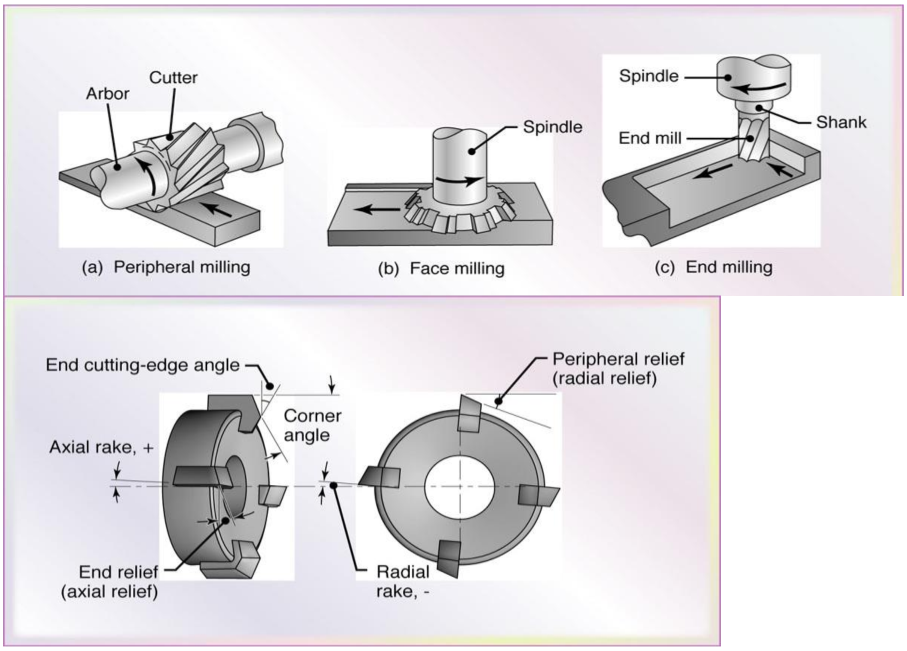
• synthetic coolants.

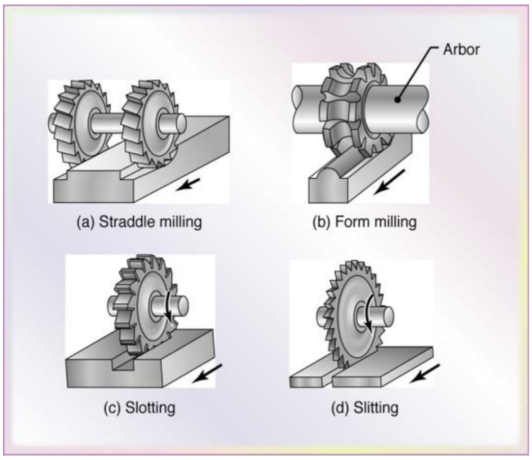
The coolant type and concentration depend on the machining method and type of material worked. It is recommended that the user consult the producer of coolant regarding the type of coolant and its concentration.

Most important properties of coolants:

• Good cooling and lubrication,
• prevention of corrosion,
• physical, chemical and technological stability during use,
• no harmful effects on human health,
• no excessive foaming.

(c) Geometry of milling cutter.





(d) Carbide inserts.

- Various types of cemented (sintered) carbides developed to suit different materials and machining operations
  - Good wear resistance
  - Operate at speeds ranging 150 to 1200 sf/min
- Can machine metals at speeds that cause cutting edge to become red hot without loosing harness
- Products of powder metallurgy process
  - Tantalum, titanium, niobium
- Operations
  - Blending
  - Compaction
  - Presintering
  - Sintering

Types of Carbide Lathe Cutting Tools

- Blazed-tip type
  - Cemented-carbide tips brazed to steel shanks
  - Wide variety of styles and sizes
- Indexable insert type
  - Throwaway inserts
  - Wide variety of shapes: triangular, square, diamond, and round
    - Triangular: has three cutting edges
  - Inserts held mechanically in special holder

(e) GM codes in CNC machines.

G-codes, also called preparatory codes, are any word in a CNC program that begins with the letter G. Generally it is a code telling the machine tool what type of action to perform, such as:

- Rapid movement (transport the tool as quickly as possible through space to the location where it will cut)
- Controlled feed in a straight line or arc

- Series of controlled feed movements that would result in a hole being bored, a workpiece cut (routed) to a specific dimension, or a profile (contour) shape added to the edge of a workpiece
- Set tool information such as offset
- Switch coordinate systems

CNC G codes

G00 - Positioning at rapid speed; Mill and Lathe
G01 - Linear interpolation (machining a straight line); Mill and Lathe
G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe
G03 - Circular interpolation, counter clockwise; Mill and Lathe
G04 - Mill and Lathe, Dwell
G09 - Mill and Lathe, Exact stop
G10 - Setting offsets in the program; Mill and Lathe
Lathe
G43 - Tool length compensation; Mill
G44 - Tool length compensation cancel; Mill (sometimes G49)
G50 - Set coordinate system and maximum RPM; Lathe
G52 - Local coordinate system setting; Mill and Lathe
G53 - Machine coordinate system setting; Mill and Lathe
G54~G59 - Workpiece coordinate system settings #1 t0 #6; Mill and Lathe
G61 - Exact stop check; Mill and Lathe
G65 - Custom macro call; Mill and Lathe
G70 - Finish cycle; Lathe
G71 - Rough turning cycle; Lathe
G72 - Rough facing cycle; Lathe
G73 - Irregular rough turning cycle; Lathe

CNC M Codes

M00 - Program stop; Mill and Lathe
M01 - Optional program stop; Lathe and Mill
M02 - Program end; Lathe and Mill
M03 - Spindle on clockwise; Lathe and Mill
M04 - Spindle on counterclockwise; Lathe and Mill
M05 - Spindle off; Lathe and Mill
M06 - Toolchange; Mill
M08 - Coolant on; Lathe and Mill