1

Process of Metal Cutting

1.0. Metal Cutting

Metal cutting process consists in removing a layer of metal from blank to obtain a machine part of the required shape and dimensions and with the specified quality of surface finish.

A metal cutting tool is the part of a metal cutting machine tool that, in the cutting process, acts directly on the blank from which the finished part is to be made. The metal cutting process accompanied by deformation in compression, tension and shear by a great deal of friction and heat generation is governed by definite laws. In order to cut the material from blank the cutting tool should be harder than material to be cut, the tool should penetrate the blank and the tool should be strong enough to withstand the forces developed in cutting.

Metals are given different usable forms by various processes. These processes are of two types.

(*a*) Chip removal processes.

(*b*) Non-chip removal processes (Chipless processes).

1.1. Chip Removal Processes

In chip removal processes the desired shape and dimensions are obtained by separating a layer from the parent workpiece in the form of chips. The various chip forming processes are as follows :

(*i*) Turning	(*ii*) Shaping
(*iii*) Boring	(*iv*) Drilling
(*v*) Milling	(*vi*) Honing
(*vii*) Grinding.	

During the process of metal cutting there is a relative motion between the workpiece and cutting tool. Such a relative motion is produced by a combination of rotary and translatory movements either of the workpiece or of cutting tool or of both. This relative motion depends upon the type of metal cutting operation. Table 1.0 indicates the nature of relative motion for various continuous cutting operations.

Table 1.0

Operation	*Motion of Workpiece*	*Motion of Cutting Tool*
Shaping	Fixed	Translatory
Turning	Rotary	Translatory
Drilling	Fixed	Rotary as well as translatory
Milling	Translatory	Rotary

1.2. Chipless Processes

In chipless processes the metal is given the desired shape without removing any material from the parent workpiece. Some of the chipless processes are as follows :

(*i*) Rolling
(*ii*) Forging
(*iii*) Spinning
(*iv*) Stamping.

1.3. Metal Cutting Principle

Metal cutting is one of the most important processes carried out in an industry. The purpose of any metal cutting operation commonly called machining is to produce a desired shape, size and finish of a component by removing the excess metal in the form of chips from a rough block of material. Consequently the primary objective in metal cutting is the production of chips, although these chips are only a means to an end and are discarded. They may constitute more than 50% of the initial workpiece material. The machining process should be carried at high speeds and feeds and least cutting effort and at lowest cost. The cutting operation whether being carried out on lathe, milling machine or any other machine tool is based on theory which is same for all processes. A number of inter-related factors affect metal cutting, the more important factors being as follows :

(*i*) The properties of work material.

(*ii*) The properties and geometry of the cutting tool.

(*iii*) The interaction between the tool and the work during metal cutting.

The exact mechanism of metal cutting briefly stated is that a cutting tool exerts a compressive force on the workpiece. Under this compressive force the material of the workpiece is stressed beyond its yield point causing the material to deform plastically and shear off. The plastic flow takes place in a localised region called shear plane (Fig. 1.1) which extends from the cutting obliquely up to the uncut surface ahead of tool (Fig. 1.2). The sheared material begins to flow along the cutting tool face in the form of small pieces called chips. The compressive force applied to form the chips is called cutting force. The flowing chips cause wear of cutting tool. Heat is produced during shearing action. The heat generated raises the temperature of the work, cutting tool and chips. The temperature rise in the cutting tool tends to soften it and causes loss of keenness in the cutting edge leading to its failure. The cutting force, heat and abrasive wear are thus the basic features of the material cutting process.

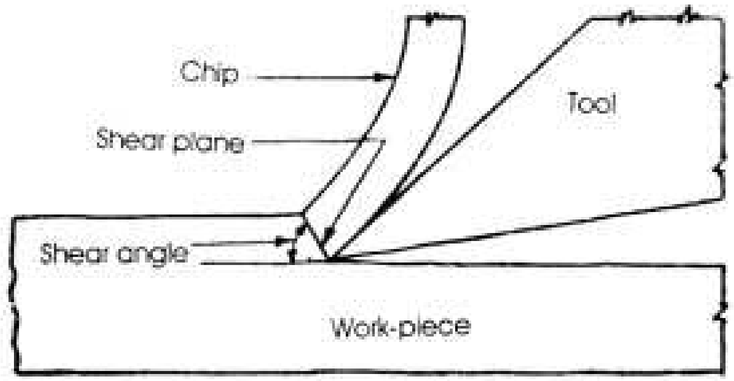


Fig. 1.1.

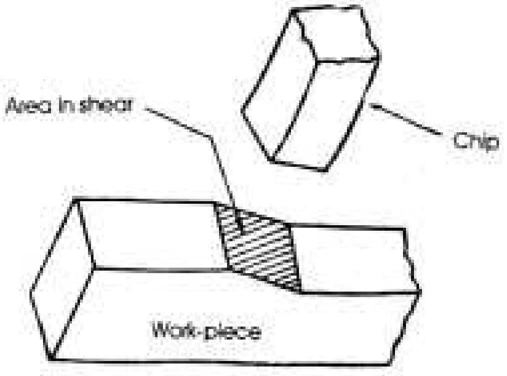


Fig. 1.2.

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

(*i*) Hardness
(*ii*) Toughness
(*iii*) Inherent hard spots and surface inclusions
(*iv*) Abrasive qualities
(*v*) Tendency to weld.

On the other hand the tool material should be hard, strong, tough and wear resistant.

1.3.1. Basic Elements of machining. The basic elements of all machining operations are as follows :

(*i*) Workpiece.
(*ii*) Cutting tool.
(*iii*) Chip.

The workpiece provides the parent metal from which the unwanted material is removed by the cutting action of tool to obtain desired size and shape. For providing the cutting action a relative motion between tool and workpiece is necessary.

1.4. Classification of Cutting Tools

Depending upon the number of cutting edges the cutting tools used in metal cutting are classified as follows :

(*i*) Single point cutting tool
(*ii*) Multipoint cutting tool.

(*i*) **Single point cutting tool.** This type of tool has an effective cutting edge and removes excess material from the workpiece along the cutting edge. Single point cutting tool is of the following types:

(*a*) Ground type
(*b*) Forged type
(*c*) Tipped type
(*d*) Bit type.

In ground type the cutting edge is formed by grinding the end of a piece of tool steel stock. Whereas in forged type the cutting edge is formed by rough forging before hardening and grinding. In tipped type cutting tool the cutting edge is in the form of a small tip made of high grade material which is welded to a shank made up of lower grade material. In bit type, a high grade material of a square, rectangular or some other shape is held mechanically in a tool holder. Single point tools are commonly used in lathes, shapers, planers, boring machines and slotters.

Single point cutting tool may be left handed or right handed type. A tool is said to be right/left hand type, if the cutting edge is on the right or left side when viewing tool from the point end. Lathe tools, shaper tools, planner tools and boring tools are single point tools.

(*ii*) **Multipoints cutting tools.** They have more than one effective cutting edge to remove the excess material from the workpiece. Milling cutters, drills, reamers broaches and grinding wheels are multipoint cutting tools.

1.5. Tool Geometry

The desirability of getting the maximum use from a tool before it needs regrinding is one of the objectives of tool technology. Tool life is defined as the length of time, a tool will operate before its failure occurs. There are many factors that contribute to cutting tool efficiency. Among the most important of these are the following :

(*i*) The shape of the cutting edge that removes the excess material.

(*ii*) The correct selection of the type of cutting tool for the material to be machined.

(*iii*) The correct choice of cutting speed and feed.

(*iv*) The proper setting of cutting tool relative to work.

(*v*) The correct choice and proper application of coolants.

Unless the cutting tools is ground to the correct shape with correct angles and unless it is ground with a keen, smooth cutting edge, time will be wasted, accuracy will be impossible and a poor finish will result.

The optimum tool geometry depends upon the following factors:

(*i*) Workpiece material

(*ii*) Machining variables

  (*a*) Cutting speed

  (*b*) Feed

  (*c*) Depth of cut.

(*iii*) Material of the tool point

(*iv*) Type of cutting.

Tool geometry refers to the tool angles, shape of the tool face and the form of the cutting edges.

Fig. 1.3 shows a typical single point tool. The most important features are the cutting edges and adjacent surfaces. Fig. 1.6 shows various angles of single point tool.

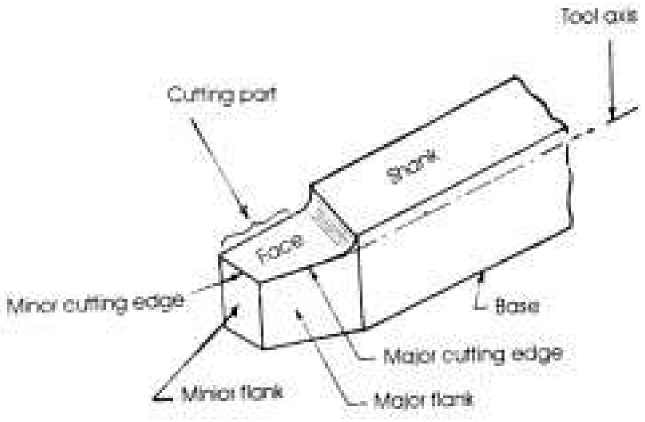


Fig. 1.3.

Fig. 1.4 shows the principal plane and cutting plane with reference to a single point cutting tool with respect to velocity taken along. Z axis whereas Fig. 1.5 shows principal and cutting planes in relation to working surface.

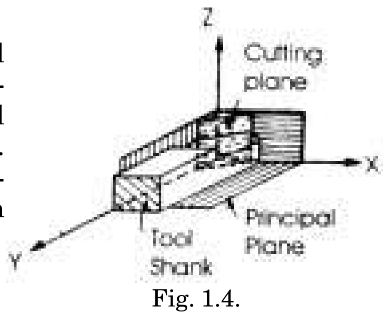


Fig. 1.4.

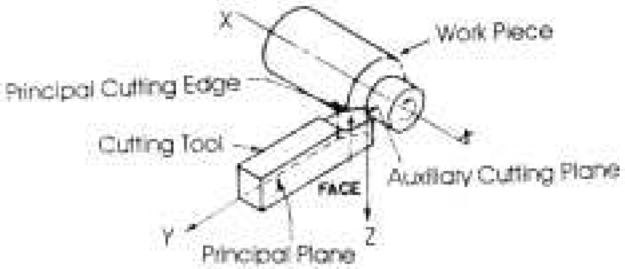


Fig. 1.5.

1.5.1. Terms and definitions.

(*i*) **Face**. The face is the surface over which the chip flows.

(*ii*) **Cutting edge.** The cutting edge carries out the cutting.

(*iii*) **Nose.** The nose is the corner, arc or chamfer at the junction of the major and minor cutting edges.

(*iv*) **Flank.** Flank of the tool is the surface below the cutting edge.

(*v*) **Tool angles.** The various angles influence tool performance to a considerable extent and therefore their value should be selected with great care and consideration.

Fig. 1.6 shows the geometry of single point tool.

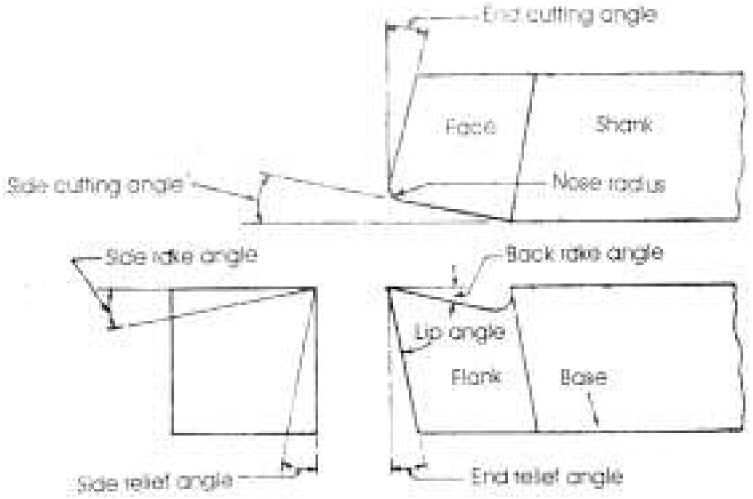


Fig. 1.6.

The various functions of the rake or slope of the tool face are as follows :

(*i*) It allows the chip to flow in a convenient direction.
(*ii*) It gives sharpness to the cutting edge.
(*iii*) It increases tool life.
(*iv*) An improved surface finish is obtained.

(*v*) The cutting force required to shear the metal is reduced, and therefore power required during cutting is reduced.

The amount of rake angle to be provided on a cutting tool depends upon the following factors :

(*i*) Material of the workpiece
(*ii*) Material of cutting tool
(*iii*) Rigidity of the machine tool
(*iv*) Depth of cut.

(*vi*) **Back rake angle.** It measures the downward slope of the top surface of the tool from the nose to the rear along the longitudinal axis. Its purpose is to guide the direction of the chip flow. It also serves to protect the point of the cutting tool. (Fig. 1.7). The size of the angle depends upon the material to be machined, the softer the material the greater should be positive rake angle. Aluminium requires more back rake angle than cast iron or steel. Back rake angle may be positive, neutral or negative. Fig. 1.8 shows a tool with a positive back rake angle. Fig. 1.9 shows a tool with a neutral back rake angle and Fig. 1.10 shows a tool with a negative back rake angle.

Fig. 1.7.

Fig. 1.8.

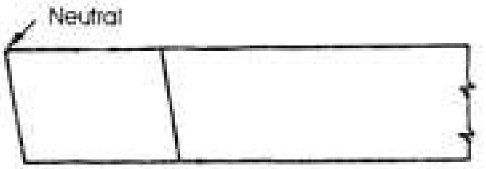


Fig. 1.9.

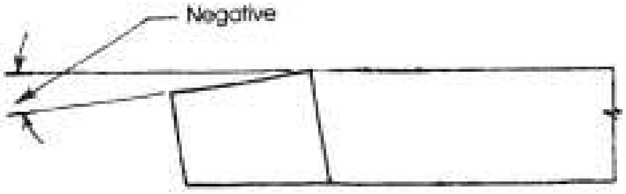


Fig. 1.10.

Higher value of rake angle weakens the cutting edge. A negative rake angle increases tool forces to some extent but this minor disadvantage is off set by the added support to the cutting edge. This is particularly important in making intermittent cuts and in absorbing the impact during the initial engagement of the tool and work.

Cemented carbide cutting tools are normally given negative rake because cemented carbide possess very high compressive strength and comparatively very low tensile and shear strength.

The advantages of negative rake cutting when using sintered carbide or ceramics tool under suitable conditions are as follows :

(*i*) Higher cutting speeds may be used.

(*ii*) There is a less tendency for a built up edge to form on the tool.

(*iii*) The heat conductivity of the working part of the tool is improved.

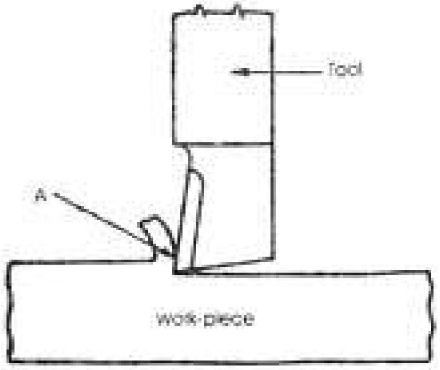


Fig. 1.11.

(*iv*) Favourable combination of conditions in that at high cutting speeds and corresponding high chip velocity, heat is carried away rapidly by the chip. Negative rake cutting with sintered carbide cutting tools can be applied with advantage on ductile steels. The machine tools using such tools must possess considerable rigidity and be equipped with high quality bearing.

(*v*) The main cutting force is directed towards strongest part of the tool.

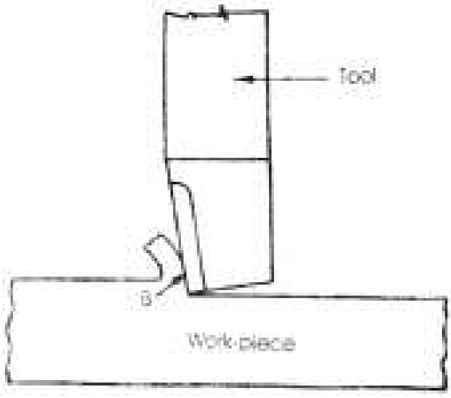


Fig. 1.12

A cutting tool with positive rake (Fig. 1.11) results in a force component in the direction of arrow *A* which tends to shear the edge of tool. When the cutting tool is provided with a negative rake (Fig. 1.12), the main force component is directed at the strongest rake section of the tool, as shown by arrow *B*. Cutting tools with negative rake angle are stronger and are used to cut high strength alloys.

However the use of an increased negative rake angle leads to increased cutting force during machining. This causes vibrations, reduces machining accuracy and raises power consumed in cutting. Therefore tools with negative rake should be used only when absolutely necessary.

General rules for rake angles are as follows :

1. Use positive rake angles when

(*i*) Machining work hardening materials
(*ii*) Machining low strength ferrous and non-ferrous metals
(*iii*) Using small machine tools with low horse power
(*iv*) Turning long shafts of small diameters
(*v*) Set up and work piece lack rigidity and strength

(*vi*) It is necessary to machine below recommended cutting speeds.

2. Use negative rake angles when

(*i*) Machining high strength alloys

(*ii*) Interrupted cuts and heavy feed rates are to be used

(*iii*) Machine tools are more rigid.

(*vii*) **Side rake angle.** It measures the slope of the top surface of the tool to the side in a direction perpendicular to the longitudinal axis. It also guides the direction of the chip away from the job. The amount that a chip is bent depends upon this angle. With increase in side rake angle, the amount of chip has to bend decreases and hence the power required to part and bend the chip decreases. Larger side rake angle produces smooth surface finish. Fig. 1.13 shows side rake angle.

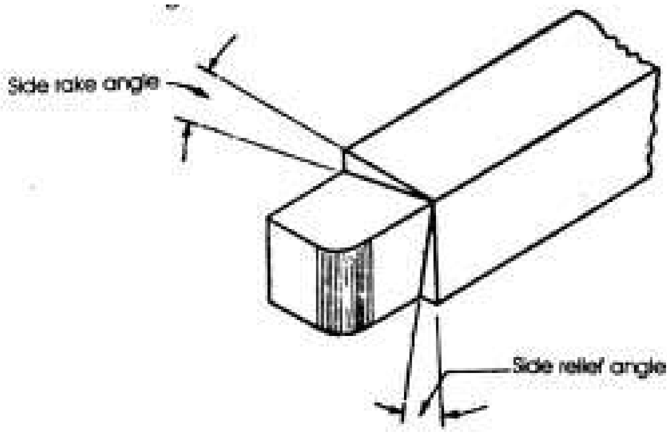


Fig. 1.13.

(*viii*) **End cutting edge angle.** It is the angle between the face of the tool and a plane perpendicular to the side of the shank. It acts as a relief angle that it allows only small section of the end cutting edge to contact the machined surface and prevents chatter and vibration. Normally it varies from 5 to 15 degrees.

(*ix*) **Side cutting edge angle.** It is the angle between the side cutting edge and the longitudinal axis of the tool. It avoids formation of built up, edge, controls the direction of chip flow and distributes the cutting force and heat produced over larger cutting edge.

(*x*) **Side relief angle.** It is the angle made by the flank of the tool and a plane perpendicular to the base just under the side cutting edge. This angle permits the tool to be fed side-way into the job so that it can cut without rubbing. If the side relief angle is very large the cutting edge of the tool will break because of insufficient support whereas if the side relief angle is very small the tool can not be fed into the job, it will rub against the job and will get overheated and become blunt and the finish obtained on the job will be rough.

(*xi*) **End relief angle.** It is the angle between a plane perpendicular to the base and the end flank. This angle prevents the cutting tool from rubbing against the job. If the angle is very large the cutting edge of tool will be unsupported and will break off. Whereas if this angle is very small the tool will rub on the job, cutting will not be proper and poor finish will be obtained on the job. Its value varies from 6 to 10°.

(*xii*) **Nose radius.** The nose radius is provided to increase finish and strength of the cutting tip of the tool. Small radii will produce smooth finishes and are used on thin cross-section of work. Large radii strengthen the tool and are used on cast iron and castings, where the cuts are interrupted.

Recommended tool angles for H.S.S. single point tools for some of the commonly used materials are indicated in Table 1.1.

Table 1.1

Material	*Relief Angle (Degrees)*		*Rake Angle (Degrees)*	
	Side	*End*	*Back*	*Side*
Aluminium	12-14	8-10	30-35	14-16
Brass, (Free cut)	10-12	8-10	0	1-3
Cast Iron, (Gray)	8-10	6-8	3-5	10-12
Copper (Commercial)	12-14	12-14	14-16	18-20
Steel	7-9	6-8	5-7	8-10

Recommended tool angles for carbide single point tool are indicated in Table 1.2.

Table 1.2

Material	*Relief Angle*		*Rake Angle*	
	(degrees)		*(degrees)*	
	Side	*End*	*Back*	*Side*
Aluminium	6-10	6-10	0-10	10-20
Copper	6-8	6-8	0-4	15-20
Brass	6-8	6-8	0-(-5)	8-(-5)
Cast Iron	5-8	6-8	0-(-7)	6-(-7)
Steel	5-10	5-10	0-(-7)	6-(-7)

1.5.2. Methods of holding tool. There are different ways of holding single point cutting tools. Tool directly fitted into tool holder is shown in Fig. 1.14 whereas Fig. 1.15 shows a tool fitted in a tool post.

The tool hold by a clamp is shown in Fig. 1.16 and tool insert brazed to a tool shank is shown in Fig. 1.17.

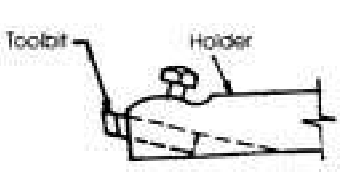


Fig. 1.14

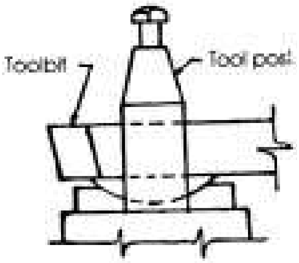


Fig. 1.15

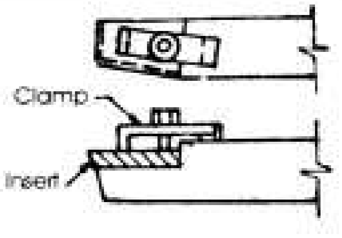


Fig. 1.16

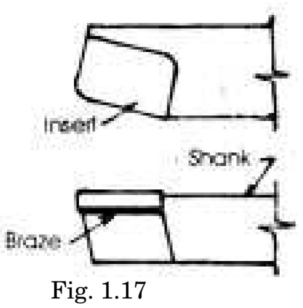
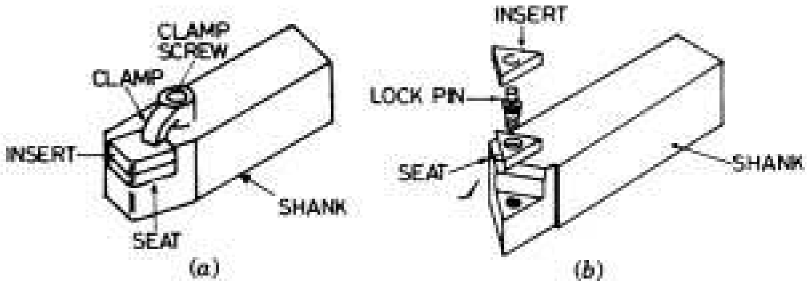


Fig. 1.17

1.5.3. Carbide inserts clamping methods. Fig. 1.17.1 shows types of seats for cemented carbide inserts.



(*a*) (*b*)

Fig. 1.17.1.

1.5.4. High Speed Steel (H.S.S.) Tools Bits. The hardened and ground H.S.S. bits are available in the following general sizes.

$6 \times 6 \times 100$ *mm.*

$8 \times 8 \times 100$ *mm.*

$10 \times 10 \times 100$ *mm.*

$12 \times 12 \times 150$ *mm.*

$16 \times 16 \times 150$ *mm.*

$20 \times 20 \times 150$ *mm.*

$25 \times 25 \times 150$ *mm.*

H.S.S. tool bits have an average chemical composition as follows :

C = 0.75%

W (Tungsten) = 18%

Cr. (Chromium) = 4%

V (Vanadium) = 1%

Cobalt = 5 or 10%

(5% cobalt H.S.S. is used for free machining steel whereas 10% cobalt H.S.S. is used for high tensile steel).

1.5.5. Specifying a Single Point Tool. The following details should be specified while ordering a single point tool.

(*i*) Shape of the tool.

(*ii*) Size of the shank.

(*iii*) Grade of the carbide tip.

(*iv*) Direction of cutting.

(*a*) Right Hand (R.H.)

(*b*) Left Hand (L.H.)

1.6. Tool Signature

The tool angles have been standardised by the American Standards Association (ASA). The seven important elements comprise the signature of the cutting tool and are stated in the following order.

Back rake angle, side rake angle, end relief angle, side relief angle, end cutting edge angle, side cutting edge angle and nose radius. It is usual to omit the symbols for degrees and mm, simply stating the numerical value of each element. For example a tool having tool signature as 10, 10, 6, 6, 8, 8, 2 will have the following angles :

Back rake angle = 10°
Side rake angle = 10°
End relief angle = 6°
Side relief angle = 6°
End cutting edge angle = 8°
Side cutting edge angle = 8°
Nose radius = 2mm

1.6.1. Requirements of a cutting tool. A cutting tool intended for high production machining should possess the following requirements :

(*i*) It should be amply strong and rigid.

(*ii*) It should be keenly sharpened with a high class finish.

(*iii*) It should have optimum geometry.

(*iv*) It should be producible in manufacture and convenient in use.

1.7. Two Systems of Defining the cutting Angles of a Single Point Tool

The cutting angles of a single point tools are commonly defined in two systems called *L-M-N* planes and *X-Y-Z* planes. The *L-M-N* plane system is chosen in reference to three mutually perpendicular planes. The plane *L* called as cutting plane being a vertical plane is tangent to the cutting edge of the tool. The plane *M* called as orthogonal plane being a vertical plane is perpendicular to *L* plane. The plane *N* called as base plane is a horizontal plane being perpendicular to L and M planes. The various angles according to this shown in Fig. 1.18 are as follows :

α = Side relief angle.
β = Wedge angle.
γ = Orthogonal rake angle.
δ = Cutting angle.
$= \alpha + \beta$
ϕ_1 = Auxiliary cutting edge angle.
θ = Nose angle.
ϕ = Plan approach angle. (Principal cutting edge angle)
λ_1 = Inclination angle.
α_1 = End relief angle.
β_1 = Side wedge angle.
γ_1 = Side rake angle.

This system is also known as International Orthogonal System (I.O.S.) of designating tool angles.

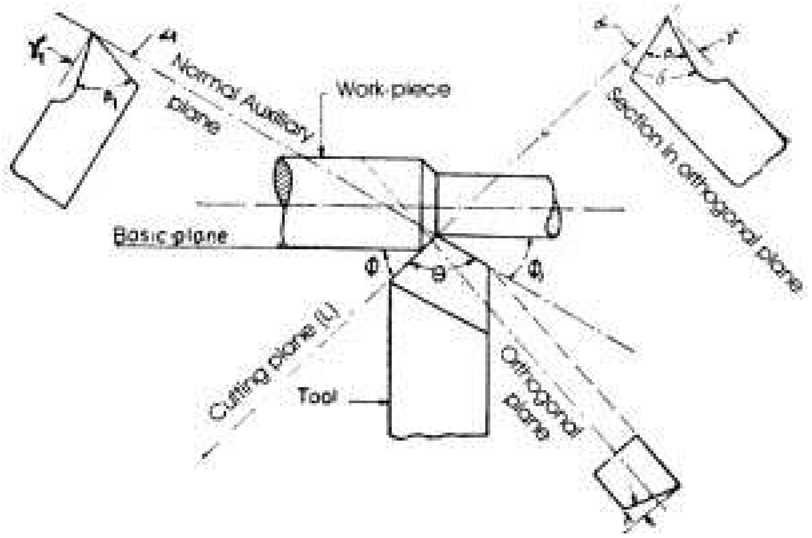


Fig. 1.18. Tool angles in L–M–N plane.

The various angles in X–Y–Z systems are shown in Fig. 1.19. The system is the most popular system. This system is also called American system of tool nomenclature.

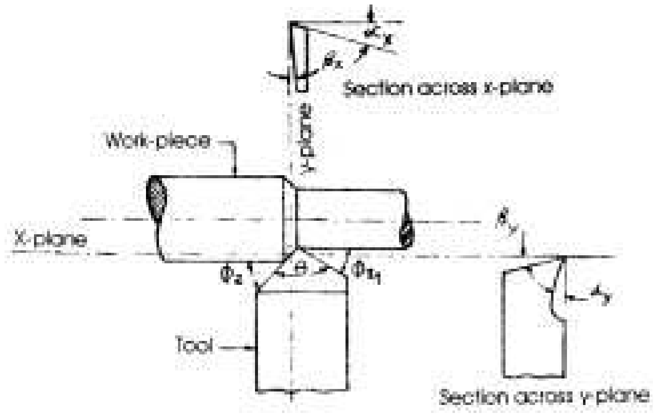


Fig. 1.19. Tool angles in *X–Y–Z* planes.

α_y = Back rake angle.
β_y = End relief angle.
α_x = Side rake angle.
β_x = Side relief angle.
θ = Nose angle.
ϕ_z = Side cutting edge angle.
ϕ_{z1} = End cutting edge angle.

Inter-relationship between different angles of system *L–M–N* and system *X–Y–Z* is as follows :

Tan $\lambda = \tan \alpha_y \sin \phi - \tan \alpha_x \cos \phi$

Tan $\alpha_x = \sin \phi \tan \gamma - \cos \phi \tan \lambda$

Tan $\alpha_y = \cos \phi \tan \gamma + \sin \phi \tan \lambda$

Tan $\gamma = \tan \alpha_y \cos \phi + \tan \alpha_x \sin \phi$

Normal rake angle (γ_n) is given by

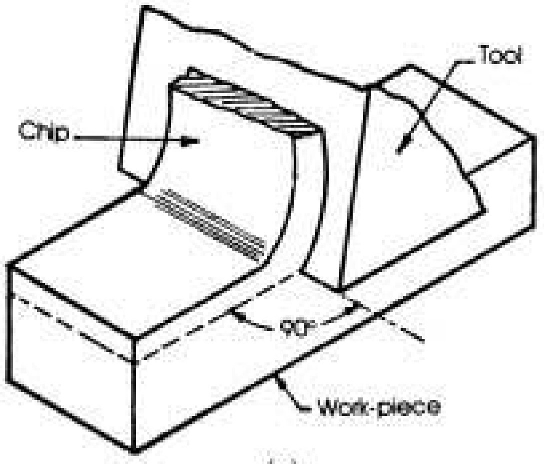
Tan $\gamma_n = \tan \gamma . \cos \lambda$

1.8. Types of Metal Cutting Process

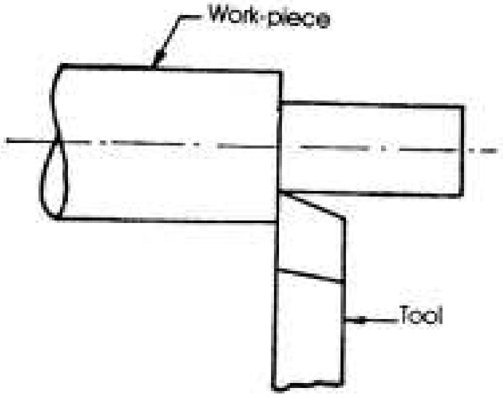
The metal cutting processes are of two types :

(*i*) 2Orthogonal cutting process (two dimensional cutting).
(*ii*) Oblique cutting process (three dimensional cutting).

Orthogonal cutting. Orthogonal (two dimensional) cutting occurs when the major cutting edge of the tool is presented to the work piece perpendicular to the direction of feed motion. Orthogonal cutting involves only two forces and this makes the analysis of cutting motion much easier. (Fig. 1.20).



(*a*)



1.20 (*b*) Orthogonal cutting

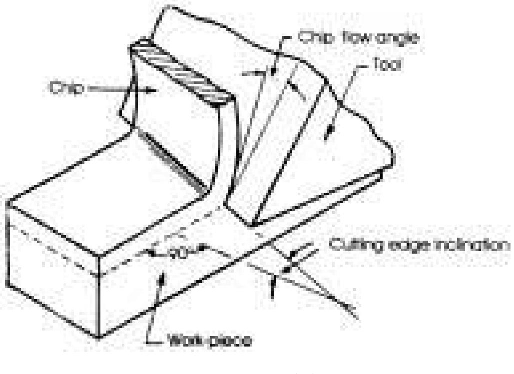
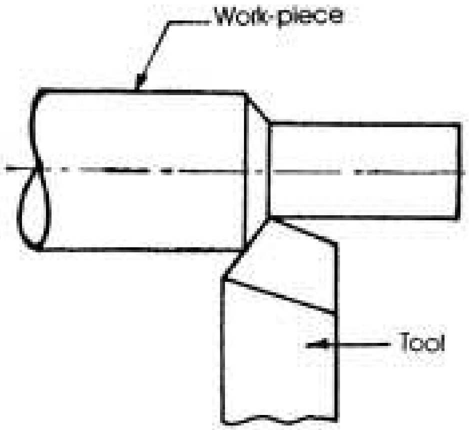


Fig. 1.21(*a*)



(b)
1.21. Oblique cutting.

Oblique cutting. This form of cutting occurs when the major edge of cutting tool is presented to the work piece at an angle which is not perpendicular to the direction of feed motion. (Fig. 1.21).

1.8.1. Comparison of orthogonal and oblique cutting. These two methods are compared as follows :

Orthogonal cutting	*Oblique cutting*
(*i*) The cutting edge of the tool remains normal to the direction of tool feed.	The cutting edge of the tool is inclined at an acute angle to the direction of tool feed
(*ii*) The direction of chip flow velocity is normal to the cutting edge of the tool.	The direction of chip flow velocity is at an angle with the normal to the cutting edge of the tool.
(*iii*) The cutting edge clears the width of the work-piece on either ends.	The cutting edge may or may not clear the width of the workpiece.
(*iv*) Only two components of cutting forces act on the tool. These two components are prependicular to each other.	Three mutually perpendicular components of cutting forces act at the cutting edge of the tool.

1.9. Chip Formation

The type of chip produced during metal cutting depends upon the machining conditions and material being cut. The variables which influence the type of chip produced are as follows :

(*i*) Properties of material cut especially ductility

(*ii*) depth of cut

(*iii*) feed rate

(*iv*) effective rake angle of tool

(*v*) cutting speed

(*vi*) type and quantity of cutting fluid.

Factors like surface finish of tool faces, coefficient of friction between tool and chip, and temperature reached in the region of cutting also have some influence on chip formation but are generally less significant as compared to variables listed above.

Three different type of chips formed during metal cutting are as follows :

(*i*) Continuous chips

(*ii*) Discontinuous chips

(*iii*) Continuous chip with a built up edge.

Continuous chip. A continuous chip is obtained when cutting ductile materials such as low carbon steel, aluminium and copper. This chip is severly deformed and either comes off in the form of a long string, or curls into a tight roll. (Fig. 1.22).

Some very soft and ductile materials with a low strength tend to tear away form the parent metal of the workpiece rather than shear clearly. This results in a rough surface that has to be cleaned up by a very keen cutting edge. In addition to ductile workpiece material the other conditions which favour their formation are fine feed, sharp cutting edge, higher cutting speeds and larger rake angles.

Discontinuous chip. Brittle materials such as grey cast iron lack the ductility necessary for appreciable plastic chip formation. Consequently the compressed material ahead of tool fails in a brittle manner along the shear zone producing small fragments. Such chips are called discontinuous chips. (Fig. 1.23). Lower cutting speeds and insufficient rake angles cause the formation of such chips.

Continuous chips with a built up edge. When during cutting, the temperature and pressure is quite high it causes the chip material to weld itself to the tool face near the nose (Fig. 1.24). This is called "Built up edge". This accumulated build up of chip material will then break away, part adhearing to the underside of the chip and part to the workpiece. This process gives rise to a poor finish on the machined surface and accelerated wear on the tool face. High friction at tool face, coarse feed, low take angle and ineffective use of cutting fluid produce such chips.

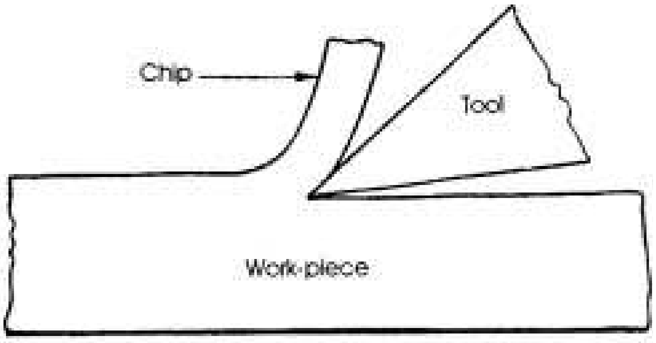


Fig. 1.22. Continuous chip.

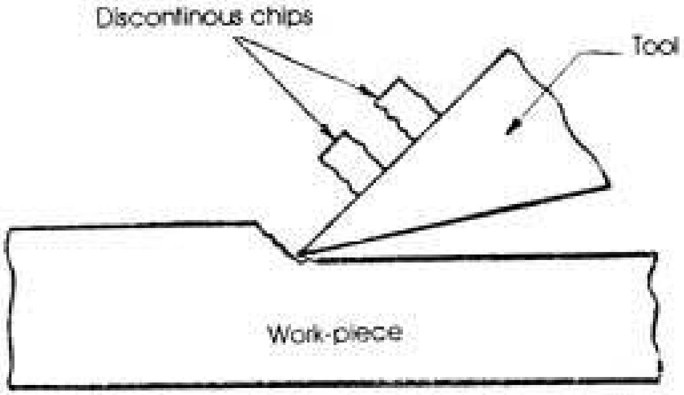


Fig. 1.23. Discontinuous chip.

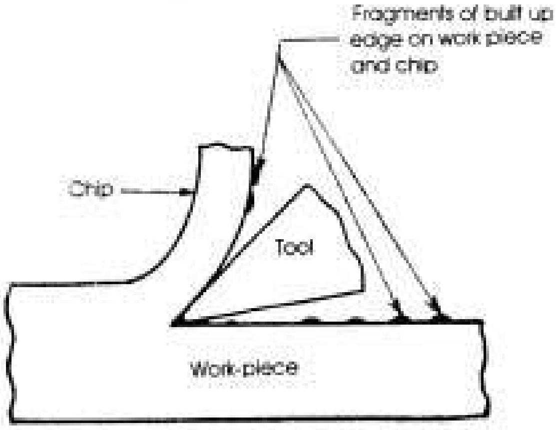


Fig. 1.24.

1.10. Chip Thickness Ratio

The outward flow of the metal causes the chip to be thicker after separation from the parent metal. Metal prior to being cut is much longer than the chip which is removed (Fig. 1.25).

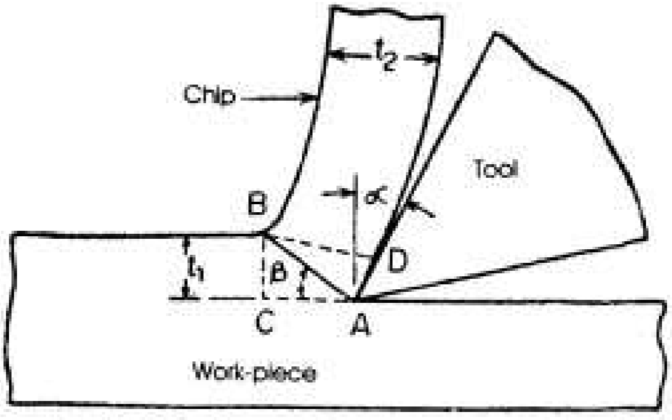


Fig. 1.25.

Let t_1 = chip thickness before cutting

t_2 = chip thickness after cutting

chip thickness ratio, $r = \frac{t_1}{t_2}$

The chip thickness ratio or cutting ratio is always less than unity. If the ratio r is large, the cutting action is good. A ratio of 1 : 2 yields good results.

k = chip reduction coefficient

$= \frac{1}{r}$

When metal is cut there is no change in the volume of the metal cut.

$$t_1 \cdot b_1 \, l_1 = t_2 \cdot b_2 \cdot l_2$$

t_1 = chip thickness before cutting (Depth of cut)

where b_1 = width of cut

l_1 = length of chip before cutting

t_2 = chip thickness after cutting

b_2 = width of chip after cutting

l_2 = length of chip after cutting

It is observed that $b_1 = b_2$

$$\therefore \qquad t_1 \cdot l_1 = t_2 \cdot l_2$$

$$\frac{t_1}{t_2} = \frac{l_2}{l_1} = r.$$

Although chip thickness ratio can be obtained by measuring chip thickness (t_2) and depth of cut (t_1) this is not the most precise procedure. Chip thickness ratio can be easily obtained by measuring l_1 and l_2.

From the right angle triangle *ABC* (Fig. 1.25) we have

$$\frac{BC}{AB} = \sin \beta$$

$$AB = \frac{BC}{\sin \beta} = \frac{t_1}{\sin \beta} \qquad \text{...}(i)$$

From right angle triangle *ABD*

$$\frac{BD}{RB} = \sin(90 - \beta + \alpha) = \cos(\beta - \alpha)$$

or

$$\frac{t_2}{AB} = \cos(\beta - \alpha)$$

$$\frac{t_2}{\cos(\beta - \alpha)} = AB \qquad \text{...}(ii)$$

From (*i*) and (*ii*) we get

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

$$\frac{t_1}{t_2} = r = \frac{\sin \beta}{\cos(\beta - \alpha)}$$

$$= \frac{\sin \beta}{\cos \beta \cos \alpha + \sin \beta \sin \alpha}$$

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

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

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

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

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

Shear angle can be measured by measuring chip thickness, depth of cut and rake angle of tool. This can be most conveniently solved with the help of nomograph shown in Fig. 1.26 for determining shear angles.

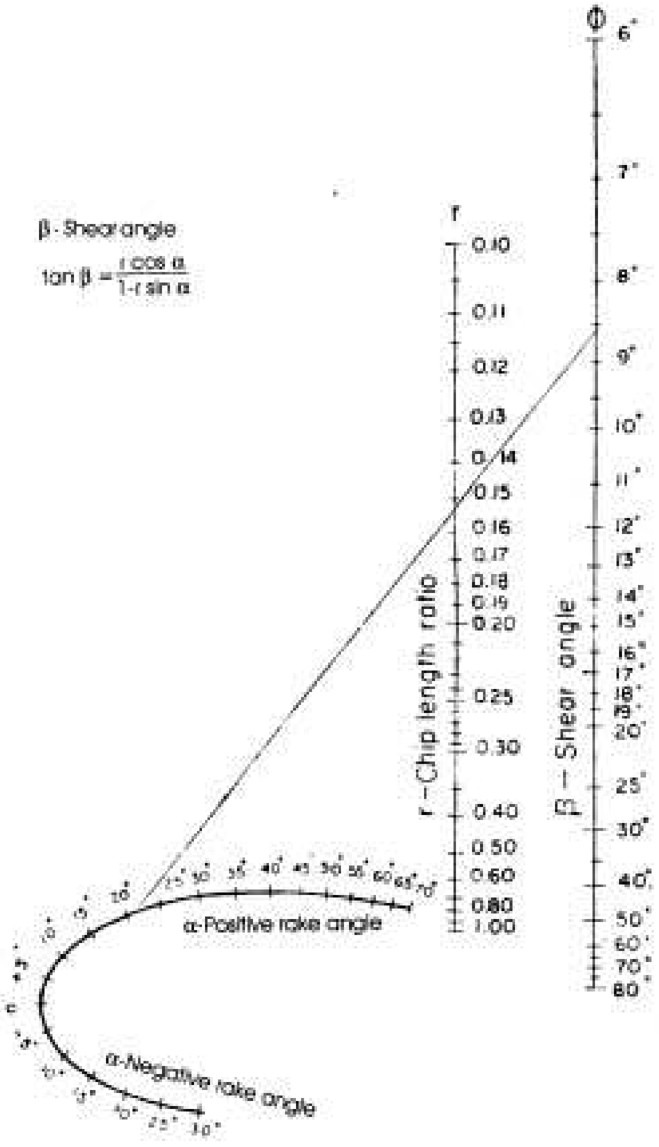


Fig. 1.26. Nomograph for Shear Angles.

1.11. Chip Breaker

Chip breakers are provided to control the continuous ribbonlike chips that are formed at high cutting speeds. Continuous chips are dangerous to the operator of the machine. These chips are hard, sharp and hot. They become entangled around the revolving job and the cutting tool. The chip breaker deflects the chip at a sharp angle and causes it to break into small pieces. So that they are easily removed by the coolant or air or simply allowed to fall into the chip pan of the machine.

There are basically two types of chip breakers :

(*i*) Groove type

(*ii*) Obstruction type.

Fig. 1.27 shows the groove type chip breaker.

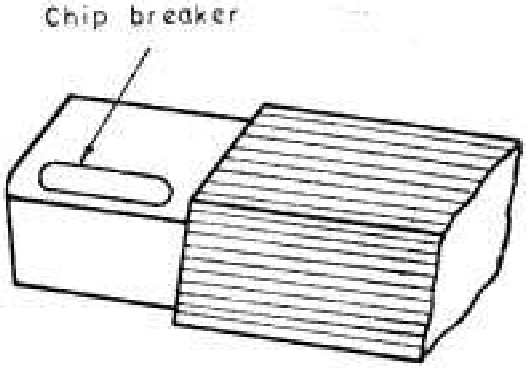


Fig. 1.27.

Fig. 1.28 shows an obstruction type chip breaker.

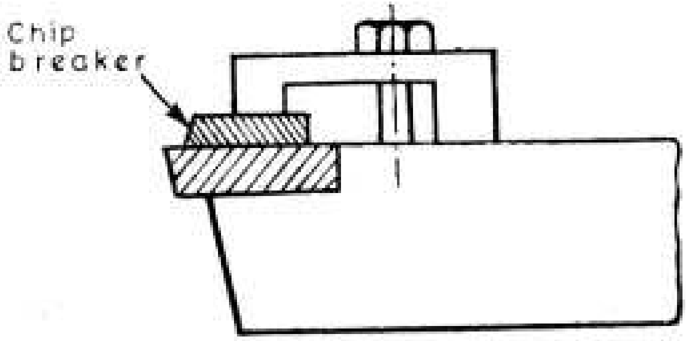


Fig. 1.28.

1.12. Radius of Chip Curvature

In an obstruction type chip breaker the chip starts to curl away from the tool face at the end of chip tool contact region and that the chip then maintains a constant radius of curvature until it clears the chip breaker. (Fig. 1.29). The radius of the chip curvature can be found by the following formula.

$$R = \frac{(S - H)^2}{2h} + \frac{h}{2}$$

where $R =$ Radius of chip curvature

$S =$ Chip breaker distance

$H =$ Length of chip tool contact

$h =$ Chip breaker height.

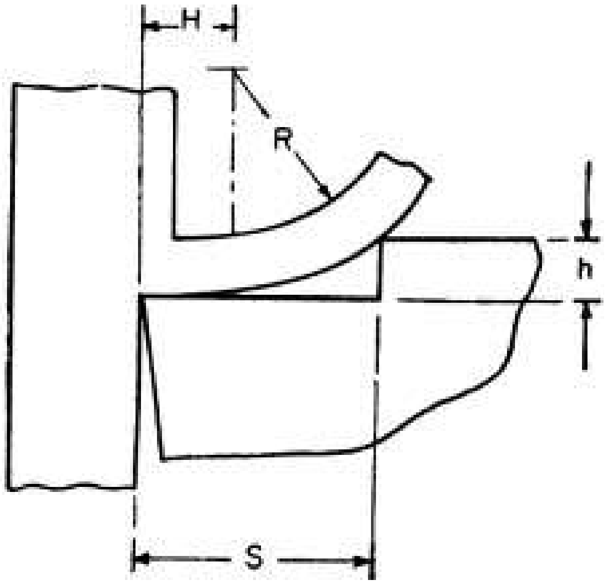


Fig. 1.29.

It is observed that larger shear angles require less cutting forces than smaller shear angles because smaller the shear angles the greater thickness of the material which the fracture plane need to traverse. (Fig. 1.30) the shear angle is a function of the rake of the tool, the material being cut and the friction at the face of the tool. As shown $t_3 > t_2 > t_1$ and shear angle $\beta_2 < \beta_1$. The chip thickness is t_2 when shear angle is β_1 and chip thickness is t_3 when shear angle is β_2.

1.13. Curling of Chip.

The chip curls into a spiral because the layer adjoining the tool is deformed. It can be seen in Fig. 1.31 that on the side facing the force P the layer becomes thicker and acquires a wedge shape as a result of which the curvature (curling) is produced. Another reason for curling the chip is that in meeting the tool face a change in its direction of flow away from tool face takes place curling is also produced by the non-uniform cooling of the chip throughout its thickness. Depending upon the machining conditions the chip may curl into a flat (logarithmic) spiral or into a helix.

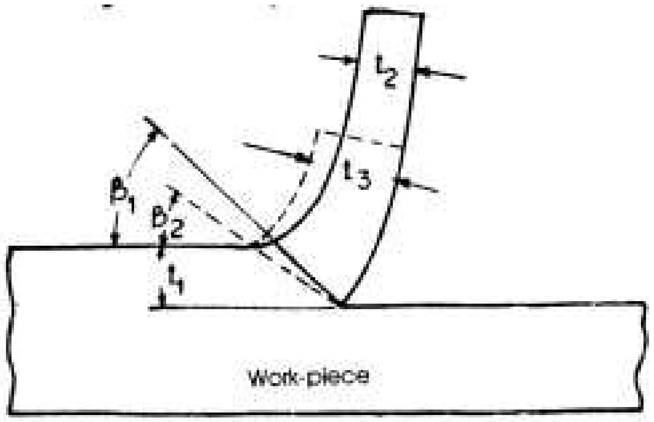


Fig. 1.30.

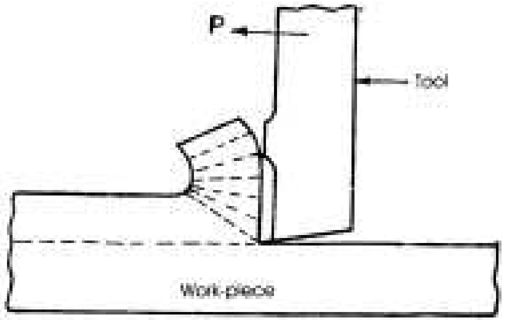


Fig. 1.31.

1.14. Chip Reduction Coefficient

The chip produced during metal cutting turns out to be shorter than the part of the work from which it has been cut (Fig. 1.32) because of the plastic compression of the layer of metal being cut. This shortening of the chip is known as longitudinal chip contraction and its magnitude is characterised by the coefficient of contraction.

Let, K = Chip reduction coefficient
L_1 = Length of the section of the work from which chip was removed in mm

$L_2 =$ Length of chip cut from the section in mm

$$K = \frac{L_1}{L_2}$$

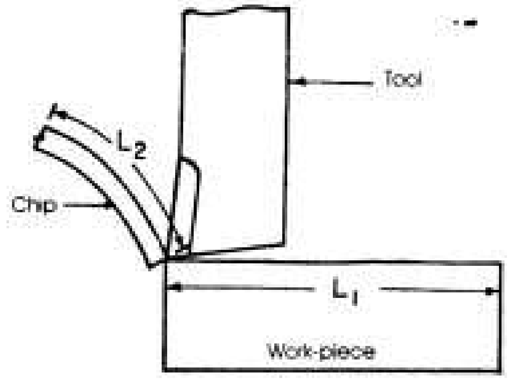


Fig. 1.32. Chip contraction.

The value of K may be as high as 6 to 8 depending upon the machining conditions. The coefficient of chip contraction is a certain quantitative measure of the degree of plastic deformation in metal cutting smaller value of chip contraction coefficient indicates less plastic deformation in cutting process and this will require less power consumption to machine the given workpiece. Chip contraction depends upon the following factors :

(*i*) Cutting variables such as cutting speed, feed and depth of cut.

(*ii*) Geometry of cutting tool.

(*iii*) Type of cutting fluid used.

(*iv*) Type of material of work piece and its mechanical properties.

It is observed that the greater the cutting angle (or the smaller the positive rake angle) the greater the contraction. An increase in nose radius leads to greater chip contraction. Fig. 1.33 shows the effect of cutting speed (V) on chip reduction coefficient K. With an increase in cutting speed contraction is first reduced, reaching a minimum and then increases reaching a maximum after which it drops again. Fig. 1.34 shows variation of chip reduction coefficient with feed upon an increase in feed coefficient of chip reduction is usually reduced.

The effect of surface active cutting fluids is clearly manifested on the reduction of chip contraction (Fig. 1.35).

1.15. Cutting Speed, Feed and Depth of Cut

Cutting speed. It is the travel of a point on the cutting edge relative to the surface of the cut in unit time in the process of accomplishing the primary cutting motion. For example in lathe work when a workpiece of diameter D rotates at a speed N revolutions per minute the cutting speed (V) is given by the relation

$$V = \frac{\pi \cdot D \cdot N}{1000} \text{ m per min.}$$

where D = Diameter of workpiece in mm.

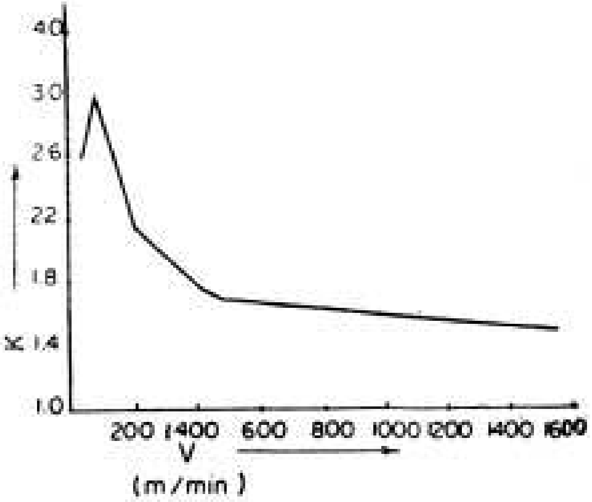


Fig. 1.33

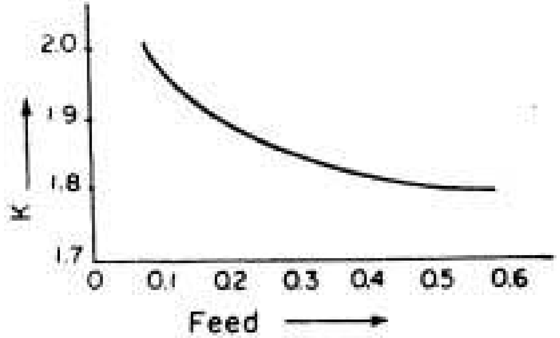


Fig. 1.34

Feed. The feed or more precisely rate of feed is the amount of tool advancement per revolution of job parallel to the surface being machined. Feed is expressed either as the distance moved by the tool in one minute. It is expressed in millimeters per revolution. On

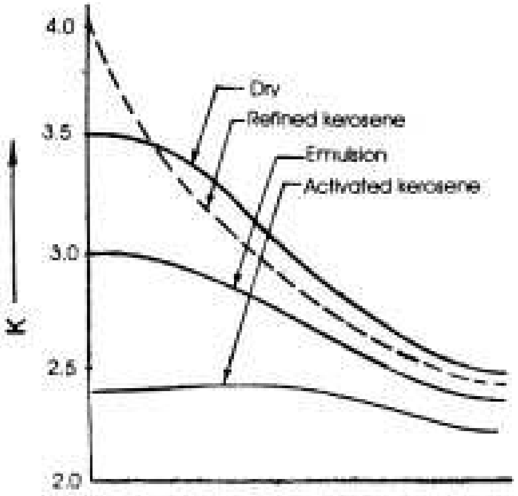


Fig. 1.35.

a shaper feed is the distance the work is moved relative to the tool for each cutting stroke. Feed is expressed as millimeters per tooth for milling cutters and broaches. Feed depends on depth of cut, rigidity of cutting tool and type of cutting tool material. Higher feeds are used in roughing cuts, rigid set-ups, soft materials, rugged cutters and heavy machine tools. Lower feeds are used for finishing cuts, frail set-ups, hard work materials and weak cutters. Normally feed varies from 0.1 to 1.5 mm.

Table 1.3 shows cutting speed and feed for H.S.S. turning tools for machining some of the materials.

Table 1.3

Material to be machined	*Cutting speed (m/min)*	*Feed (mm/rev.)*
Aluminium	70 - 100	0.2 - 1.00
Brass (Free cutting)	70 - 100	0.2 - 1.5
Copper	35 - 70	0.2 - 1.00
Grey Cast Iron	25 - 40	0.15 - 1.7
Mild Steel	35 - 50	0.2 - 1.00

In lathe work distinction is made between longitudinal feed when the tool travels in a direction parallel to the work axis and cross feed when the tool travels in a direction perpendicular to the work axis.

Feed of a lathe and similar machine tools is the distance the cutting tool is fed for each revolution of work. The feed of drilling machine is the rate at which drill is advanced into the work piece per revolution of drill. The feeds for reaming operation is usually higher than that used for drilling because reamers have more teeth and a reaming operation is used for sizing and finishing. The feed of shapers and planers is the distance the cutting tool advances at the end of each cutting stroke. Feed of a milling machine is expressed in millimeter per minute of table movement.

The maximum feed is limited by the following factors :

(*i*) Cutting edge strength

(*ii*) Rigidity and allowable deflection

(*iii*) Surface finish required

(*iv*) Tool chip space.

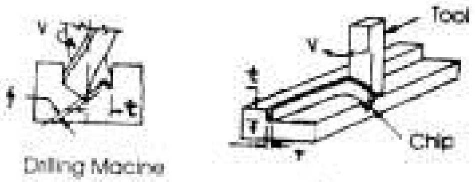
Use of proper cutting fluid permits higher feeds and increased cutting speeds as well as attainment of better surface finish.

Depth of cut. It is the thickness of the layer of metal removed in one cut or pass measured in a direction perpendicular to the machined surface. The depth of cut is always perpendicular to the direction of the feed motion.

In external longitudinal turning it is half the difference between the work diameter (D_1) and the diameter of the machined surface (D_2) obtained after one pass.

Lathe
(a)

Milling
(b)



Drilling Macine
(c)

Shaper
(d)

Fig. 1.36.

$$t = \text{Depth of cut} = \frac{D_1 - D_2}{2} \text{ mm}$$

Fig. 1.36 shows cutting speed (v), depth of cut (t) and feed (f) for lathe, milling machine, drill machine and shaper.

In general speed and feed depend upon the following factors :

(*i*) Type of material of workpiece.
(*ii*) Type of material of cutting tool.
(*iii*) Quality of finish desired.
(*iv*) Type of coolant used.
(*v*) Rigidity of the machine tool.

Table 1.4 shows typical lathe turning speeds for ceramic tools.

Table 1.4

Material to be cut	*Roughing (m/min)*	*Finishing (m/min)*
Upto 0.2 Carbon Steel	130 - 260	320 - 420
0.2 to 0.3 Carbon Steel	100 - 200	260 - 320
0.3 to 0.4 Carbon Steel	80 - 160	230 - 300
0.4 to 0.6 Carbon Steel	65 - 100	130 - 200
Cast Iron (BHN 217)	160 - 320	320 - 500

To achieve higher machining efficiency it is desired to use high feed that will allow :

(*a*) Required surface finish.
(*b*) Use of maximum depth of cut consistant with available power and rigidity of machine tool.
(*c*) Maximum cutting speed depending on tool life.

1.15.1.Economical Cutting Speed. The best cutting speed is which reduces the total cost of machining and at the same time gives maximum production rate. Fig. 1.36.1 shows a graph between cutting speed (V) and production rate and production cost. It is observed that at point M production cost is minimum and at point N production rate is maximum. Therefore the best cutting speed will be between point M and point N.

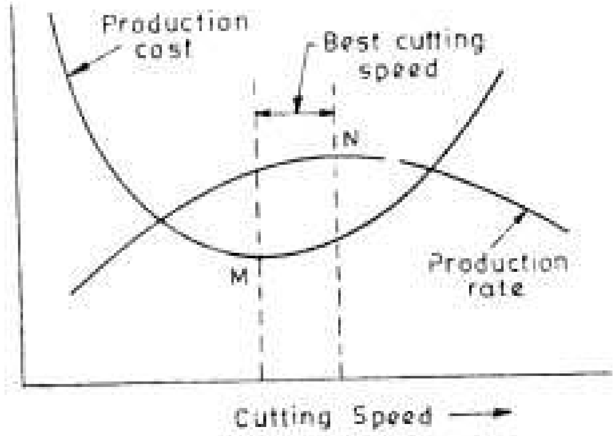


Fig. 1.36.1.

Tool design depends on the following factors :

(1) Type of machining operation such as
  (*i*) Turning
  (*ii*) Milling
  (*iii*) Drilling
  (*iv*) Shaping
  (*v*) Slotting etc.

(2) Type of material of tool such as
  (*i*) H.S.S.
  (*ii*) Cemented Carbide.

(3) Type of work piece material

(4) Machining variables such as
  (*i*) Cutting speed
  (*ii*) Feed
  (*iii*) Depth of cut

(5) Cutting force

(6) Accuracy to be obtained on work piece.

(7) Heat treatment and surface finish of tool.

1.16. Objectives of a Good Tool Design

A cutting tool should be made up of proper material and should be accurately designed in order to achieve the following objectives.

(*i*) Easy chip flow.

(*ii*) Proper surface finish and accuracy.
(*iii*) High productivity.
(*iv*) Less amount of input power.
(*v*) Minimum overall cost.

The tool designed should have sufficient strength to maintain a sharp cutting edge during cutting and should have sufficient hardness to prevent picking up the chips.

During machining the cutting forces should be low and the cost of production should be minimum. As shown in Fig. 1.36.2 .

In zone A cutting forces are very high due to the low cutting speed. Therefore cutting edges of brittle material like carbide may break. In zone C the cutting edge temperature will be very high due to the high cutting speed which will lead to rapid plastic deformation of the cutting edge. Zone B is, therefore, ideal for working with carbides where cutting forces are more or less independent of change in cutting speed. Cutting speeds vary with the machineability factor of the material being machined. Normally we employ higher cutting speeds for softer, and lower cutting speeds for harder materials. Therefore, it is advisable to use lower speeds within Zone B combined with higher feeds to achieve low cutting tool costs.

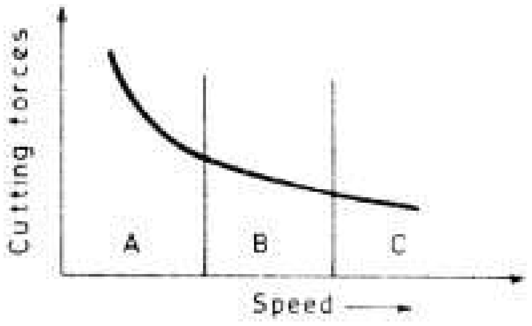


Fig. 1.36.2.

Though the tool life can be increased by using lower cutting speeds, the objective generally is to have that cutting speed which will give us the lowest cost of production. The cutting speed so selected is called economical cutting speed as shown in Fig. 1.36.3.

1.16.1. Re-sharpening Allowance. Allowance for re-sharpening of single point tools is indicated in Table 1.4.A.

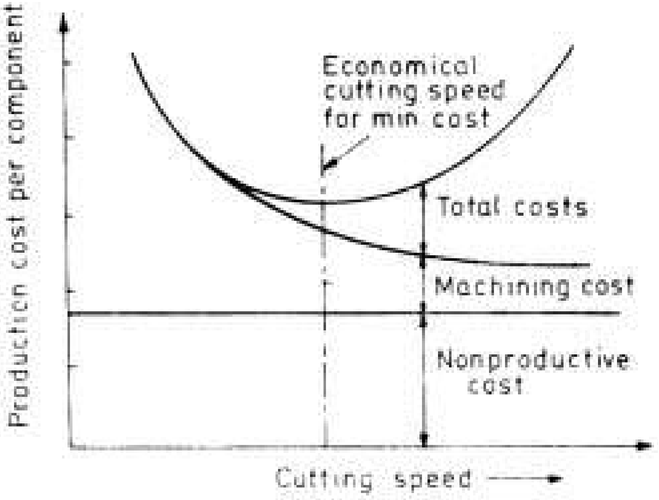


Fig. 1.36.3.

Table 1.4.A.

			Resharpening allowance, millimeters						
Tool Types	Tool material	Surface to be ground	Tool cross section, mm × mm						
			10 × 16	12 × 20	16 × 25	20 × 30	25 × 40	30 × 45	40 × 60
For single tool lathes	Cemented carbid	Face and flank	0.2	0.2	0.3	0.3	0.4	0.4	0.5
	High speed steel	Face and flank	0.3	0.4	0.5	0.5	0.6	0.6	0.6
For multi tool lathes	Cemented carbide	Flank	0.6	0.6	0.7	0.7	0.8	0.9	--
	High-speed steel	Flank	--	0.7	0.8	0.8	0.9	--	--
For circular foming tools	High-speed steel	Face	1.1	1.1	1.1	1.1	1.1	1.1	1.1

Tools used on multiple lathes are ground only on flanks whereas forming tools are ground on their faces.

1.17. Cutting Tools Materials

The purpose of cutting tool material is to remove metal under controlled conditions. Therefore the tool must be harder than the material which it is to cut. The cutting tools are made up of different materials. The cutting tool material should possess the following requirements :

(*i*) It should be strong enough to withstand the forces being applied due the cutting *i.e.* bending compression, shear etc.

(*ii*) It should be tough (resistant to shock loads). It is quite important when tool is used for intermittent cutting.

(*iii*) It should be sufficient harder (resistant to wear, abrasion and indentation) than the material being cut.

(*iv*) It should be able to resist high temperature.

(*v*) It should be capable of withstanding the sudden cooling effect of coolant used during cutting.

(*vi*) The coefficient of friction between the chip and the tool should be as low as possible in the operating range of speed and feed.

(*vii*) It should be easily formed to the required cutting shape.

Material chosen for a particular application will depend on the material being machined, the quantity of components to be produced, the cost of cutting tool materials, the type of machining operation (intermittent or continuous roughing or finishing, high or low speed etc.), the tool design details (cutting and clearance angles method of holding rigidity etc.) and the general conditions of machine tool.

Several materials exhibiting above properties in varying degrees have been developed for use in cutting tools.

The following metals suitably heat treated wherever required (tungsten carbide and ceramics are not heat treatable) are used in the manufacture of cutting tools :

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

1.17.1. Carbon tool steels. Carbon steels are limited in use to tools of small section operating at lower speeds. A typical plain carbon steel used for cutting tool has the following composition.

C = 0.8 to 1.3%

SI = 0.1 to 0.4%

Mn = 0.1 to 0.4%

Such steels when hardened and tempered have good hardness, strength and toughness and can be given a keen cutting edge. Tools made up of plain carbon steels can be used for machining soft materials such as free cutting steels and brass. This material starts loosing its hardness at about 250°C and is therefore not used when the operational temperature is more.

Table 1.5(*a*) indicates percentage composition of typical medium alloy tool steels.

Table 1.5(a)

C	*Mn*	*Si*	*Cr*	*W*	*Mo*
1.2	0.6	0.3	0.5	-	-
1.2	0.7	0.3	0.5	-	0.5
1.2	0.3	0.3	0.7	1.5	0.3
1.3	0.3	0.3	0.7	4.0	-

Medium alloy tool steels are similar to carbon tool steel. However in medium alloy tool steels elements like chromium, molybdenum and tungsten are added. Chromium and molybdenum are added to increase hardenability of steel whereas tungsten is added to improve wear resistance.

1.17.2. Alloys and their effect on tool steels. Plain carbon steels when used as cutting tools lack certain necessary characteristics such as

(*i*) Red hardness
(*ii*) Impact strength
(*iii*) Abrasive resistance.

Addition of alloying elements such as

(*i*) Chromium
(*ii*) Vanadium
(*iii*) Tungsten
(*iv*) Cobalt
(*v*) Titanium
(*vi*) Molybdenum
(*vii*) Nickel
(*viii*) Manganese
(*ix*) Silicon

Helps in achieving the following properties :

(*i*) Improving toughness
(*ii*) Improving tensile and impact strength
(*iii*) Improving wear resistance
(*iv*) Improving hardenability
(*v*) Increasing high temperature resistance (red hardness)
(*vi*) Improving corrosion resistance.

Table 1.4.2. indicates case hardening temperature (T) and hardness (HRC) obtainable for some of Die Steels.

Table 1.4.2.

Type	T (°C)	HRC
En - 8	830 - 840	38 - 48
En - 9	830 - 860	45 - 55
En - 24	850 - 870	55 - 60
En - 30	900 - 920	50 - 60
En - 31	850 - 860	58 - 60
En - 36	900 - 920	50 - 60
En - 56	860 - 870	55 - 58
Spring steel	840 - 850	60 - 64
OHNS	860-880	60-62

1.17.3. High speed steels. High speed steel tools give improved cutting performance and higher metal removal rates. High speed steel is widely used for drills, many types of general purpose milling cutters and single-point tools.

Although several types of high speed steels are in use but 18-4-1 high speed steel containing tungsten 18%, chromium 4% and vanadium 1% is quite commonly used. This type of material gives excellent performance over a great range of materials and cutting speeds and it retains its hardness upto around 600°C. The coolant should be used freely to increase tool life. The chief characteristics of high speed steels are superior hardness and wear resistance.

Sometimes, the surface of high speed steel tools are treated by the following methods to reduce friction and to increase wear resistance.

(*i*) Super finishing (to reduce friction).

(*ii*) Chromium of electrolytic plating (to reduce friction).

(*iii*) Oxidation (to reduce friction).

(*iv*) Nitriding (to increase wear resistance).

Table 1.5 indicates some of the carbon tool steels.

1.17.4. Cemented carbides. They consist of tungsten, tantalum and titanium carbides together with a binder usually cobalt all mixed together as fine powders. These powders are compacted (compressed) into the required shape and subjected to a high temperature treatment known as sintering. During this process the cobalt binder is fused to the carbides, producing a hard, dense substance.

Tools made up of cemented carbides are extremely hard having Rockwell hardness varying from 90-93 HRC. They can be used at cutting speeds 200 to 500% greater than those used for high speed steel. They have virtually replaced high speed steels in high speed and high production machining. Cemented carbides are very hard and the usual practice is to confine the size to a relatively small shape known as an "insert" which is clamped to a tough steel shank or holder. This has the advantage that the tool bit is well supported to resist the cutting forces. The insert may have three to eight edges and is so designed that each of its cutting edges can be used in turn. It may then be discarded and replaced with a new tool bit giving low maintenance and breakdown time.

The general characteristics of carbide tools are as follows.

(*i*) They have high thermal conductivity, low specific heat and low thermal expansion.

(*ii*) They have high hardness over a wide range of temperature (upto 900°C).

(*iii*) Their compressive strength is more than tensile sthrength.

(*iv*) They are very stiff. Their Young's modulus is about three times than steel.

Tungsten carbide cutting tools containing only tungsten carbide and cobalt with no other elements added are referred to as the straight grades. The only variables in this case are the grain size of the tungsten carbide particles and the percentage of cobalt. Any change of these variables affects the hardness and transverse rupture strength (toughness) of the carbide insert. In turn, any reduction of hardness would reduce the abrasion resistance of the tool, and a reduction in toughness would result in a more brittle tool. In other words, the tougher the grade, the lower its wear resistance and vice versa.

Table 1.5. Carbon Tool Steels

IS : 1570 / 1961	*Nominal Chemical Composition*						
Designation	*C%*	*Mn%*	*Si%*	*S%*	*P%*	*Cr%*	*V%*
T 70	0.70	0.25	0.20	0.03	0.03	—	—
T 80	0.80	0.25	0.20	0.03	0.03	—	—
T 90	0.90	0.25	0.20	0.03	0.03	—	—
T 103	1.00	0.25	0.20	0.03	0.03	—	—
T 118	1.15	0.25	0.20	0.03	0.03	—	—
T 133	1.30	0.25	0.20	0.03	0.03	—	—

Table 1.6 shows some of popular grades of high speed steels used for making tools.

Table 1.6. High Speed Steel

IS:1570 / 1961	*Nominal Chemical Composition*									
Designation	*C%*	*Mn%*	*Si%*	*S%*	*P%*	*Cr%*	*Ni%*	*Mo%*	*W%*	*V%*	*CO%*
T70W18Cr4VI	0.70	0.40	0.35	0.25	0.025	4.15	—	—	18.25	1.20	—
T75W18Co6 Cr4VIMo75	0.75	0.40	0.35	0.25	0.025	4.20	—	0.75	18.25	1.25	5.50
T83Mo6W6Cr4V2	0.83	0.40	0.35	0.25	0.025	4.10	—	5.50	6.00	1.90	—

The effect of the variables in the straight carbide grades is as follows :

An increase in grain size

(*a*) lowers the hardness

(*b*) increases the transverse rupture strength (toughness).

An increase in cobalt content

(*a*) lowers the hardness

(*b*) increases the transverse rupture strength.

Maximum wear resistance is obtained from using a small grain size and a low cobalt content. When chipping of the tool occurs, the toughness is increased by increasing the grain size and cobalt content at the expense of wear resistance.

1.17.5. Ceramics. Aluminium oxide and boron nitride powders mixed together and sintered at 1700°C form the ingredients of ceramic tools. These materials are very hard with good compressive strength. Ceramics are usually in the form of disposable tips. They can be operated at from two to three times the cutting speeds of tungsten carbide and cutting speeds on cast iron in the order of 1000 m/min are not uncommon. They resist cratering, usually require no coolant. However in order to take full advantage of their capabilities special and more rigid machine tools are required.

Ceramic and cemented carbide cutting tools should be sharpened to a high degree of surface finish to minimise friction between tool and chip. This will improve the efficiency of the cutting process and less power will be absorbed and there will be a less tendency for a built up edge to form on the cutting face of the tool.

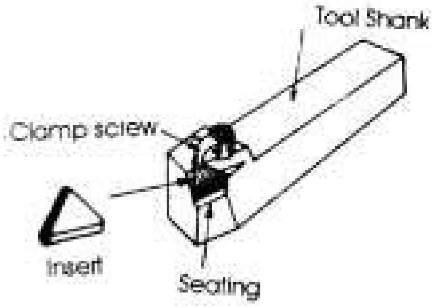


Fig. 1.37.

Cemented carbide or ceramic tips can be brazed to medium carbon steel shank and resharpened by grinding when necessary. For many applications it is more economical to clamp the tip to the shank (Fig. 1.37). The technique enables a tip to be thrown away (called throw away tips) at the end of its useful life and new tip to be clamped to the existing shank.

The various conditions for the effective use of carbide or ceramic tool are as follows :

(*i*) Rigidity of tool and workpiece.
(*ii*) Highly finished surface on cutting tool.
(*iii*) High cutting speeds.
(*iv*) Use of machine tool having high quality bearings.
(*v*) Use of effective chip bearings.
(*vi*) Elimination of any unbalanced forces.
(*vii*) Chip guards are essential when using these tools.

Appropriate reliefs and clearances are ground on carbide inserts as shown in Fig. 1.37(*a*).

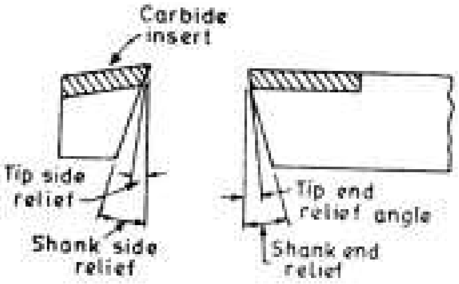


Fig. 1.37.(*a*).

Shank sizes for single point carbide tools vary from 12 mm to over 50 mm. The shank size should be large enough to support the load of the tool for a given tool over hang. A typical shank size for a single point carbide tool is as follows.

Shank size = 12 mm × 24 mm.
when depth of cut = 3 mm.
Feed = 0.375 mm.
Over hang = 36 mm.

1.17.6. Diamonds. Diamond is the hardest material known. It has a low coefficient of friction, high compressive strength and is extremely wear resistant. It is used mainly for cutting very hard materials, such as glass, plastics, ceramics etc. Diamond tools produce a very good surface finish at high speeds with good dimensional accuracy. Diamond tools are small and are best suited for light cuts and finishing operations. Other application for diamond is that it is used for truing the grinding wheels. The general properties of diamond may be summarised as follows :

1. It is the hardest known substance.
2. It has lowest thermal expansion of any pure substance.
3. It has high heat conductivity.
4. It has poor electrical conductivity.
5. It has very low coefficient of friction against metals.

Since very high hardness is always accompanied by brittleness a diamond tool must be cautiously used to avoid rupturing the point. This usually limits the use of diamond tools to light continuous cut in relatively soft metals and low values of rake angle are usually used to provide a stronger cutting edge.

The main disadvantages of diamonds are their brittleness and high cost.

1.17.7. Multi-coated cemented carbide inserts. Cemented carbide inserts coated with layers of optimum thickness of Titanium Carbide, Titanium Carbonitride, and Titanium nitride have more life. Layer coating is carried out by chemical vapour deposition process.

Various advantages of multiple coatings are as follows :

1. A hard, binderless layer of TIN, acts as a diffusion barrier and eliminates the possibility of chip welding and chemical diffusion on insert Rake face, thereby **increasing crater resistance.**
2. Reduced Flank wear.
3. Lower frictional resistance and reduction in cutting forces upto 10-15%. This also allows higher cutting speed.
4. Higher Thermal conductivity.
5. Superior surface finish on machined parts.

1.18. Variation of Hardness with Temperature

Fig. 1.38 shows the variation of hardness with temperature for the tools made up of different materials.

1. A plain carbon tool steel tool.
2. An 18-4-1 high speed steel tool.
3. A cast alloy tool.
4. A cast iron grade cemented tungsten carbide tool.

1.19. Economic Comparison of Tool Materials

The principal tool materials are high speed steel, cemented carbide and ceramics in increasing order of hardness and reducing order of toughness. High speed steel is quite commonly used as it can be used to manufacture complicated shaped cutting tools such as drills, taps, reamers, dies and milling cutters. Cemented carbide because of its high cost and need to form it in its final shape has limited applications and is generally used for rough or finish turning and face milling operations. Ceramic tool material is brittle, cannot withstand high force and is generally used in finish turning operations.

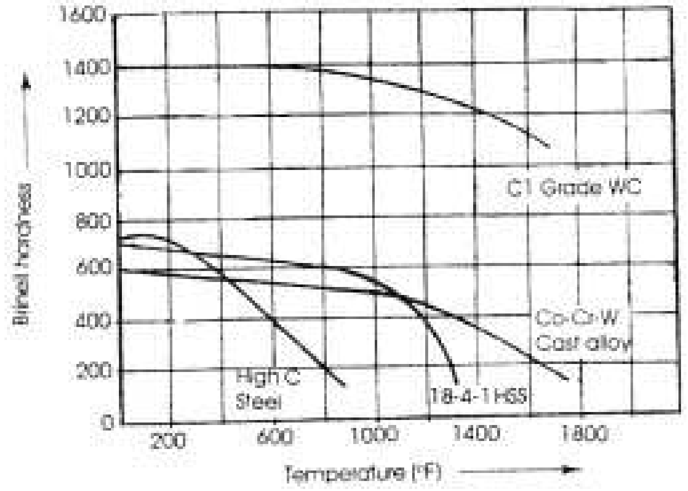


Fig. 1.38.

Properties of tool materials are basically governed by the following :

(*i*) Chemical composition of the materials.

(*ii*) Type and size of grains of materials before manufacture.

(*iii*) Manufacturing process.

(*iv*) Finishing treatments.

1.20. Multi-edge Cutting Tools

Multi-edge cutting tools present more than one cutting edge to the work. They have a higher metal removal rate than single edged cutting tools and also the life of the multi-edge cutting tool is raised by increasing the number of cutting edges. The different multi-edge cutting tools are divided into two groups as shown in Table 1.7.

A grinding wheel is also a form of multi-edge cutting tool.

Table 1.7

Group	*Machines*	*Cutting Tools*
A	Drills	Twist drills
	Capstan	Reamers
	Lathe	Multi flute core drills
	Turret	Counter bores spot
	Lathe	Facing cutters.
B	Milling	Saws and slotting
	Machines	Cutters
		Side and face cutters
		Slab mills
		End mills
		Face mills
		Form-relieved cutters.

The tools in group *A* have their cutting edges in continuous engagement with the work and are fed axially at a uniform feed per revolution. The underformed chip thickness is therefore constant and directly proportional to the feed/rev. The geometry and mechanics of cutting are then identical with those discussed for single point tools. The various tools in group *B* have their cutting edges intermittently engaged with the work and are fed in a plane parallel or perpendicular to the cutter axis of rotation.

1.21. The Twist Drill

It is used to produce a hole in the workpiece. Its sole purpose is to remove the maximum volume of metal in a minimum period of time. The finish obtained by a drill is not so fine. If a hole of accurate size and good finish is required the drilled hole should be finish machined by means of a reamer or by single point boring tool. Twist drills are usually made of high speed steel. High carbon steel can also be used to manufacture drills. Drills with cemented carbide cutting tips (tips are brazed) are used at very high speeds for drilling operations on non-ferrous metals but are not recommended for ferrous metal particularly steels because the tips are not supported as effectively as in case of single point cutting tools or milling cutters. A twist drill consists of a cylindrical body carrying two helical grooves into it to from the flutes. The flutes run the full length of body of twist drill and perform the following functions.

(*i*) They provide the rake angle.

(*ii*) They form the cutting edges.

(*iii*) They provide a passage of the coolant.

(*iv*) They facilitate swarf removal.

1.22. Elements of a Twist Drill

A twist drill is shown in Fig. 1.39. The various elements of a twist drill are as follows :

(*i*) **Body.** It is the part of the drill that is fluted and relieved.

(*ii*) **Shank.** It is the part held in the holding device.

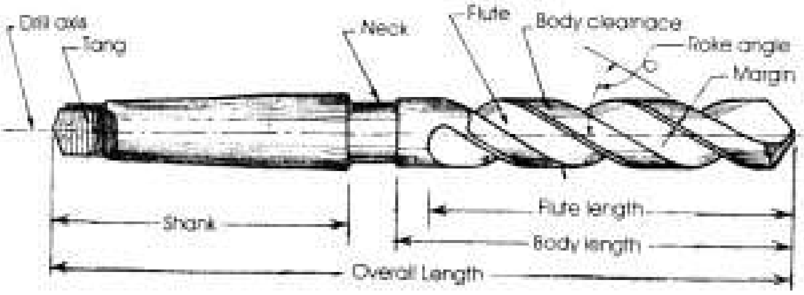


Fig. 1.39. Elements of a twist drill.

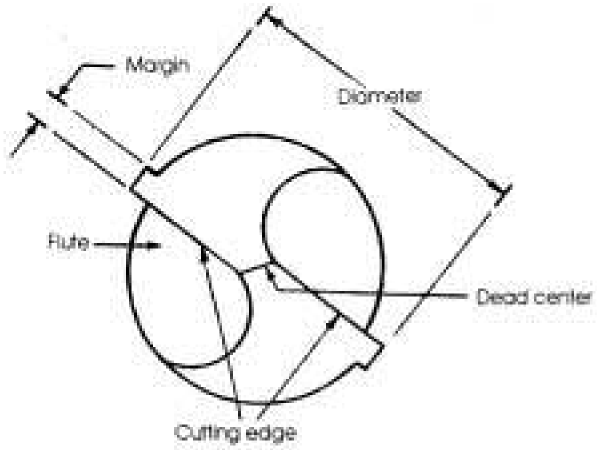


Fig. 1.40. The point of twist drill.

The most common types of shanks are the taper shank and the parallel shanks. Small drills up to about 12.7 mm diameter are provided with parallel shanks and the larger size drill have tapered shanks. Tapered shank drill carry a tang at the end of the shank to ensure a positive grip. The straight shank drills are held in a drill machine by a chuck. The jaws of the chuck are tightened around the drill by means of a key or wrench. The point of a drill is the entire cone shaped surface at the cutting end of the drill. (Fig. 1.40).

(*iii*) **Dead centre.** The dead centre or chisel edge of the drill is the sharp edge at the extreme tip end of the drill. It should always be in the exact centre of the axis of the drill.

(*iv*) **Lip.** Lip or cutting edge is formed by the intersection of the flank and face. Both the lips of the drill should be of equal length and should be at the same angle of inclination with the drill axis. This will enable to produce a perfectly round smooth and accurate hole. Unequal lips will result in an oversize hole.

(*v*) **Flank.** Flank is the surface on a drill point which extends behind the lip to the following flute.

(*vi*) **Chisel edge corner.** The corner formed by the interection of lip and the chisel edge is called chisel edge corner.

(*vii*) **Flutes.** The grooves in the body of the drill which provide lips.

1.22.1. Twist drill angles. Lake any other cutting tool the twist drill is provided with correct tool angles (Fig. 1.41). The various angles are as follows :

(*i*) **Rake angle or helix angle.** It is the angle of flute in relation to the work.

Smaller the rake angle greater will be the torque required to derive the drill at a given speed. Its usual value is 30° although it may vary up to 45° for different materials.

The rake angle also partially governs the tightness with which the chips curl and hence the amount of space which the chips occupy. Other conditions being the same, a very large rake angle makes a tightly rolled chip, while a rather small rake angle makes a chip tend to curl into a more loosely rolled helix.

(*ii*) **Lips clearance angle.** It is the angle formed by the flank and a plane at right angle to the drill axis. Lip clearance is the relief that is given to the cutting edges in the order to allow the drill to enter the metal without interference. This angle is 12° in most cases. In order to allow strength and rigidity to the cutting edge the clearance angle should be kept minimum.

(*iii*) **Cutting angle or point angle.** It is the angle included between the two lips projected upon a plane parallel to the drill axis and parallel to the two cutting lips. It is observed that the best point angle is 118°.

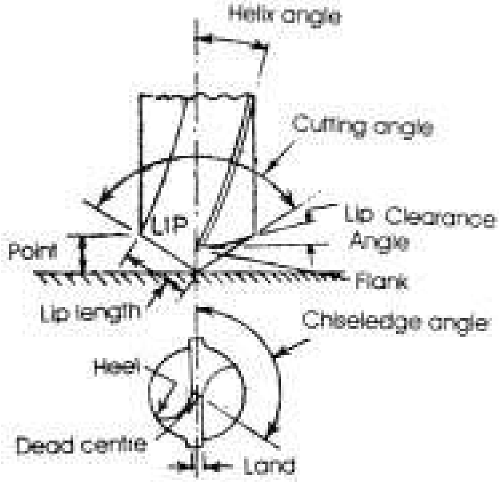


Fig. 1.41.

(*iv*) **Chisel edge angle.** It is the angle included between the chisel edge and the lip as seen from the end of the drill.

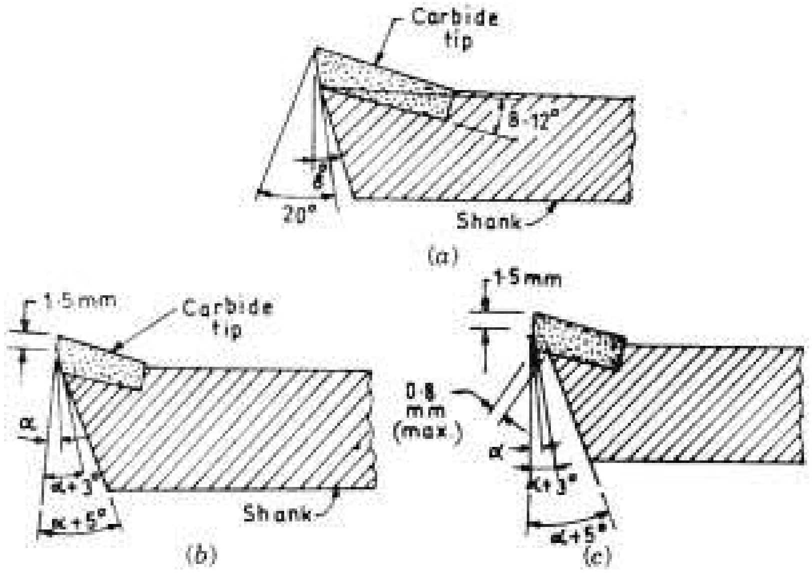
The various angles for a drill are shown in table 1.8.

Table 1.8

Material	*Point angle (Degrees)*	*Lip clearance angle (Degrees)*	*Chisel edge angle (Degrees)*	*Helix angle (Degrees)*
Aluminium	90-140	8-12	120-135	24-48
Brass	111	8-15	120-135	0-27
Copper	100-118	8-15	120-135	28-40
Cost iron hard	118	8-12	120-135	24-40
Steel	118	8-12	120-135	24-32
Stainless Steel	125-135	10-12	120-135	24-32

Fig. 1.41(*a*) shows a carbide tip. embedded in the shank. The tool flank must be designed in such a way as to minimise sharpening

costs. The flanks of almost all carbide tips are bevelled to an angle of 20°. Fig. 1.41(*b*) shows shape of flank surfaced of carbide tipped tool without tip over hang whereas Fig. 1.41(*c*) shows shape of flank surface of carbide tipped tool with tip overhang.



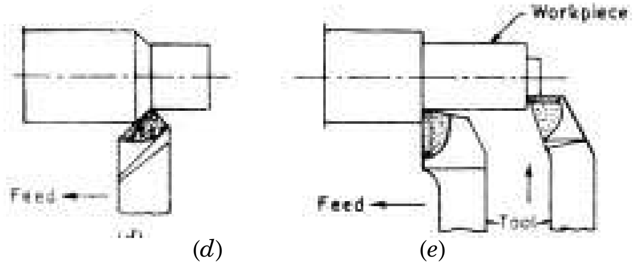
(*b*) (*c*)

Fig. 1.41

Tipped tools are preferable when expensive cemented carbides or high speed steels are used : then, the point is made of a metal-cutting material, while the shank is of structural carbon steel.

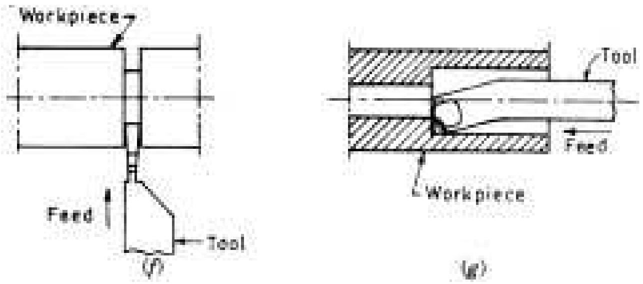
Cemented-carbide tips are brazed or clamped mechanically, while tips of highspeed steel are clamped.

An as-brazed tip may protrude from the shank for as much as 1.5 millimetres, but in tool sharpening this over-hang should be reduced to within 0.8 millimetre. This will enable carbide tips to be sharpened by diamond wheels, keeping the latter out of touch with the shank.



(*d*) (*e*)

Fig. 1.41(*d*) shows a turning tool whereas a facing tool is shown in Fig. 1.41(*e*). A parting tool used to cut off the material is shown in Fig. 1.41(*f*). A boring tool is shown in Fig. 1.41(*g*).



(*f*) (*g*)

Fig. 1.41.

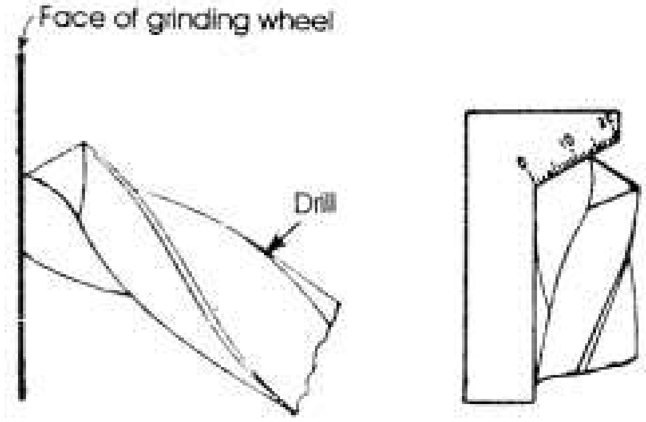
1.23. Cutting Fluids for Drilling

Cutting coolants and oils are used during drilling to carry away the heat from the drill point preventing it from overheating. This permits higher cutting speeds and longer drill life. The action of chips coming out of the hole tends to restrict the entry of the fluid. A continuous supply of cutting fluid should be maintained in order to obtain proper cooling. For severe conditions drills containing oil holes have a considerable advantage. In such case not only the fluid is supplied near the cutting edges but the flow of fluid aids in chip removal from the hole.

Practically all metals require the use of coolant when being drilled except cast iron which may be machined dry.

1.24. Twist Drill Grinding

It is very important that drills are properly ground. The drill point must be in the centre of rotation of the drill. The lips should be of equal length and at correct angle with the axis. The twist drill is ground on the side of grinding wheel of the grinding machine as shown in Fig. 1.42 (*a*). The flat surface that this provides makes it much easier to generate a true point. The straight cutting lip should lie vertically against the side of the wheel when the drill has been ground it should be checked on a point gauge. Fig. 1.42 (*b*) shows a twist drill point angle and lip length gauge. If the lips are of unequal length the result will be that both the point and lip will be off-centre (Fig. 1.43). This will cause the hole to be large than the drill. If a drill is ground with its tip on the centre but with the cutting edges at different angles the drill will bend on one side of the hole (Fig. 1.44). Only one lip or cutting edge will do the work resulting in rapid wear on that edge and the hole will be larger than drill.



(*a*) (*b*)

Fig. 1.42.

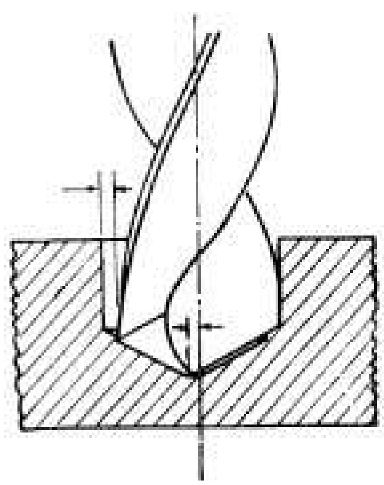


Fig. 1.43.

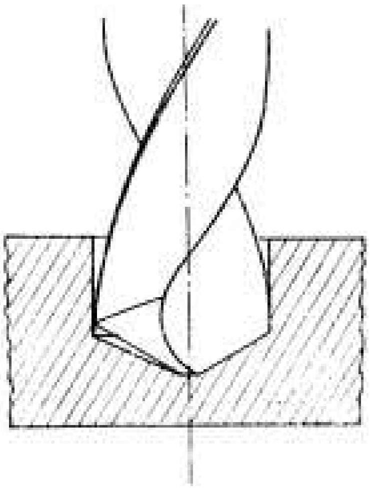


Fig. 1.44

1.24.1. Twist drill failure. A twist drill will suffer an early failure or produce holes that are dimensionally inaccurate, out of round and poor finish for the following general reasons :

(*i*) Incorrect speed and feeds

(*ii*) Incorrect grinding of the point

(*iii*) Mishandling

(*iv*) Lips of unequal length.

1.25. Rake and Relief Angles of a Twist Drill

Rake Angle. The rake angle (α) is the angle between the tangent to the face (in the flute) at the point of lip (cutting edge) being referred to and the normal at the same point to the surface of revolution described by the lip about the drill axis. The rake angle is measured in a plane perpendicular to the lip (plane MM in Fig. 1.45). This angle varies gradually along the lip. It can be found by using the following relation :

$$\tan \alpha_x = \frac{r_s \tan \beta}{R \sin \theta}$$

where β = Helix angle of flutes

R = Radius of drill.

θ = One half of the point angle.

rx = Radius of the circle on which the point being considered is located.

R = Radius of the drill

Angle α acquires its maximum value at the periphery of the drill. The variation of rake angle along the lip is shown by a graphical construction.

Refer Fig. 1.46. When a helix is developed on a plane it will become the hypotenuse of a right triangle of which one side is the lead (pitch of the flutes and the other side is the circumference of a circle of a diameter (*D*) on which the helix was formed.

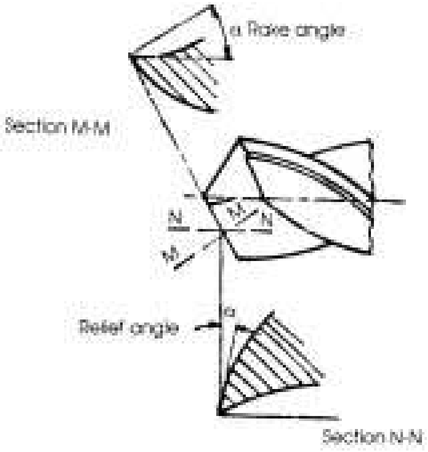


Fig. 1.45.

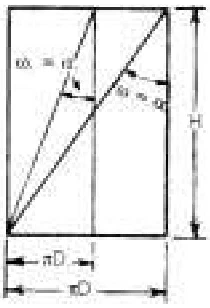


Fig. 1.46.

Relief Angle γ. It is the angle between a tangent to the flank or lip relief surface at the point being considered on the lip and a tangent at the same point to the circle the point describes as it rotates about the drill axis. This angle is measured in plane *NN* (Fig. 1.45) which is tangent to the cylindrical surface on which the above mentioned point on the lip lies.

1.26. Drill Specifications

According to Indian Standard System, twist drills are specified by the diameter, the I.S. number, the material and the series to which they belong. The drills are made in three types :

(*i*) Type *N*--for normal low carbon steel.

(*ii*) Type *H*--for hard materials.

(*iii*) Type *S*--for soft and tough materials.

For example a parallel shank twist drill 12 mm diameter and made up of high speed steel and conforming to IS : 5101 with a point angle of 118° and of N-type is designated as 12.00--IS : 5101 HS-N-118.

Unless otherwise mentioned in the designation it should be assumed that drill type is N and point angle is 118°.

1.26.1. Drill Size. In metric system the drills are made in diameters from 0.2 mm to 100 mm.

1.27. Cutting Speed, Feed and Depth of Cut

(*i*) **Cutting speed.** It is the peripheral speed of a point on the surface of the drill in contact with the work piece. It is usually expressed in metres per minute.

Let D = Diameter of drill in mm.

N = R.P.M. of drill spindle.

V = Cutting speed in metre per minute.

$$V = \frac{\pi DN}{1000} \text{ metre per minute.}$$

Cutting speed mainly depends upon the following factors :

(*i*) Type of material to be drilled.

(*ii*) Type of material of drill. Twist drill made of high speed steel can be operated at about twice the speed of drill made of high carbon steel.

(*iii*) Type of finish required.

(*iv*) Type of coolant used.

(*v*) Capacity of machine and tool life.

Table 1.9 shows cutting speed for high speed steel drill.

Table 1.9

Material being drilled	*Cutting speed (m/min)*
Aluminium	70--100
Brass	35--50
Phosphor Bronze	20--35
Grey Cast Iron	25--40
Copper	35--45
Mild Steel	30--40
Alloy Steel (High tensile)	5--8

(*ii*) **Feed.** It is the distance the drill moves into the work at each revolution of the spindle. It is expressed as millimeter per revolution.

It may also be expressed as feed per minute.

Let N = R.P.M. of drill spindle.

f = feed in mm /rev.

f_1 = feed in mm per minute.

$= N.f.$

Table 1.10 shows feed for high speed steel drill of various diameters.

Table 1.10

Drill diameter (mm)	*Feed (mm/rev.)*
1.0--2.5	0.04--0.06
2.6--4.5	0.05--0.10
4.6--6.0	0.075--0.15
6.1--12.00	0.75--0.25
12.1--15.0	0.20--0.30
15.1--18.0	0.23--0.33
18.1--21.0	0.26--0.36
21.1--25	0.28--0.39

A twist drill gives satisfactory performance if it is run at correct cutting speed and feed. The following factors help in running the drill at correct cutting speed and feed.

(*i*) The work is rigidly clamped.

(*ii*) The machine is in good condition.

(*iii*) A coolant is used if required.

(*iv*) The drill is correctly selected and ground for the material being cut. The selection of drill depends upon the following.

(*a*) Size of drill hole
(*b*) Material of workpiece
(*c*) Point angle of drill.

The rates of feed and cutting speed for twist drill are lower than most other machining operations because of the following reasons.

(*i*) The twist drill is weak compared with other cutting tools.

(*ii*) It is relatively difficult for the drill to eject chips.

(*iii*) It is difficult to keep the cutting edges cool when they are enclosed in the hole.

(*iii*) **Depth of cut.** It is equal to one half of the drill diameter.

D = Diameter of drill in mm.

t = Depth of cut in mm.

$$= \frac{D}{2} \text{ mm.}$$

1.28. Machining Time

Machining time in drilling is calculated as follows (Fig. 1.47)

Let T = Machining time in minutes

l = Thickness of workpiece in mm.

h = Approach of drill = 0.3 D in mm.

D = Diameter of drill in mm.

L = Length of axial travel of drill = $l + h$.

f = Feed/rev. in mm.

$$T = \frac{L}{N \times f}.$$

1.29. Forces Acting on a Drill

The various forces acting on a drill are shown in Fig.1.48. All the elements of a drill are subject to certain forces in drilling. Resolving the resultant forces of resistance to cutting at each point

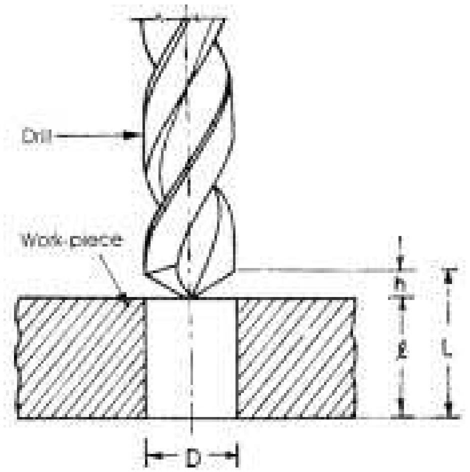


Fig. 1.47

of the lip we obtain three forces F_Z F_V, and F_H acting in directions mutually perpendicular to each other. The horizontal forces F_H acting on both lips are considered to counter balance each other. The vertical force F_V also called as thrust force comprises of the forces F_{V1}, F_{V2}, F_C and F_m. (The forces F_C and F_M are not shown). The force F_{V1}, acts on the web. This force is quite large and is about 60% of the total thrust force. This force F_{V2} act on each of the two lips and forms the real cutting force which depends upon the work material, cutting variables and cutting point geometry and is about 37% of total thrust force. The F_c and E_m are of smaller magnitude. The force F_c is due to the rubbing of the chips, flow from the hole against the sides of the hole and flutes on the drill. This force is about 1% of the total thrust force. The force F_M is due to the rubbing action of margin of the drill against the sides of the hole and is about 2% of thrust force.

In order that the drill is to penetrate into the work piece the thrust force F applied to it by the machine must over come the sum of resistances acting along the drill axis.

$$F > \Sigma\,(F_{V_1} + 2Fv_2 + F_C + F_m)$$

The force F_Z sets up the moment of resistance (M_r)

$$M_r = F_Z \,.\, S$$

The total moment of the forces of resistance (M) to cutting is made up of the following moments.

(*i*) Moment of forces F_Z *i.e.* M_r.

(*ii*) Moment of forces due to scraping and friction on the chisel edge (M_c).

(*iii*) Moment of the friction forces on the margins (M_m).

(*iv*) Moment of the forces of friction of the chip on the drill and on the machined surface.(M_d)

$$M = M_r + M_e + M_m + M_d$$

The total moment of resistance should be overcome by the available torque of the drilling machine.

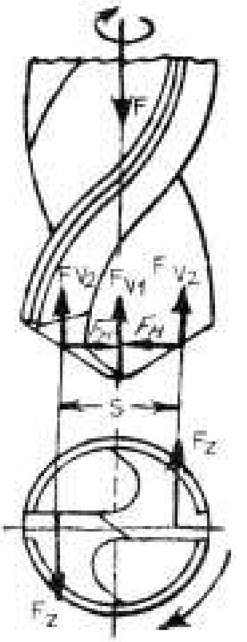


Fig. 1.48.

1.30. Power of Drilling

When a drill is cutting it has to overcome the resistance offered by the metal and a twisting effort is necessary to turn it. The effort is called turning moment or torque on the drill. The torque required to operate a drill depends upon various factors. The relationship between torque, diameter of drill and feed is as follows :

$$T_1 = C.f^{0.75}\,D^{1.8} \text{ newton metres}$$

where T_1 = Torque in newton metre.

f = Drill feed in mm/rev.

D = Diameter of drill in mm.

C = Constant depending upon the material being drilled. The values C are given In table 1.11 for different materials.

Table 1.11

Materials to be drilled	*Value of C*
Aluminium	0.11
Soft brass	0.084
Cast Iron	0.07
Mild steel	0.56
Corbon tool steel	0.4

Power $(P) = \dfrac{2\pi N T_1}{60,000}$ kW

where N = Drill speed in R.P.M.

Table 1.11 (a) shows the approximate equivalents of tool geometry.

Table 1.11(a)

Lathe and planer	*Milling cutter*	*Drill*	*Broach*
Back rake	Axial rake	Helix Angle	Hook angle
Side rake	Radial rake	Radial rake	Side rake
Side cutting edge angle	Corner Angle	Half point Angle	
End cutting edge angle	End cutting angle		
Side relief	Peripheral relief		Clearance
End relief	End relief	Lip relief	Clearance

1.31. Milling Cutters

Milling is an operation in which metal cutting is carried out by means of multi-teeth rotating tool called cutter. The cutters are manufactured in a variety of shapes and size each made for a specific purpose. In milling cutters each tooth after taking a cut comes into operation again after some interval of time. This allows the tooth to cool down before the next cutting operation is done by it. This minimises the effect of heat developed in cutting on cutting edges. A coolant is helpful when milling steel for producing a good surface finish. The coolant may be oil or an emulsion of oil and water. The form of each tooth of cutter is same as that of a single point tool.

Many types of milling cutters are used on a milling machine. Most of them are considered standard and are available in many sizes. High speed steel is the material commonly favoured for cutters. They are also available with carbide tips at cutting edges. Cutters should be kept sharp. Cutters which are sharpened frequently usually last longer than those which are allowed to become dull.

A milling cutter is a multiple tooth tool and therefore higher requirements are made to its construction, sharpening and operation. The cutter should be reliably and rigidly clamped on the spindle. Milling cutters may be solid, inter locking, carbide tipped solid cutters and inserted blade cutters with high speed steel or carbide tipped blades.

The various advantages of carbide tipped cutters either solid or inserted blade type are as follows :

(*i*) They have high production capacity.

(*ii*) They produce surface finish of high quality.

(*iii*) They can machine hardened steels.

(*iv*) Their use leads to reduction in machining costs.

1.30.1. Milling operations. With cylindrical cutters the two methods of milling operations commonly used are as follows :

(*i*) **Up or conventional milling.** In this process the cutter rotates against the direction in which the work is feeding (Fig. 1.49).

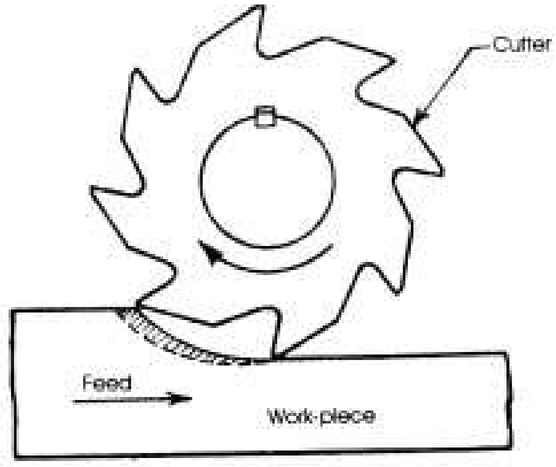


Fig. 1.49. Up milling.

(*ii*) **Down milling or Climb milling.** In this process the cutter rotates in the same direction as that in which the work is feeding (Fig. 1.50). The machine in which high velocity milling is to done should not only be capable of operating at high speeds and feeds but should also possess ample rigidity. The cutter should be reliable and rigidly clamped on the spindle.

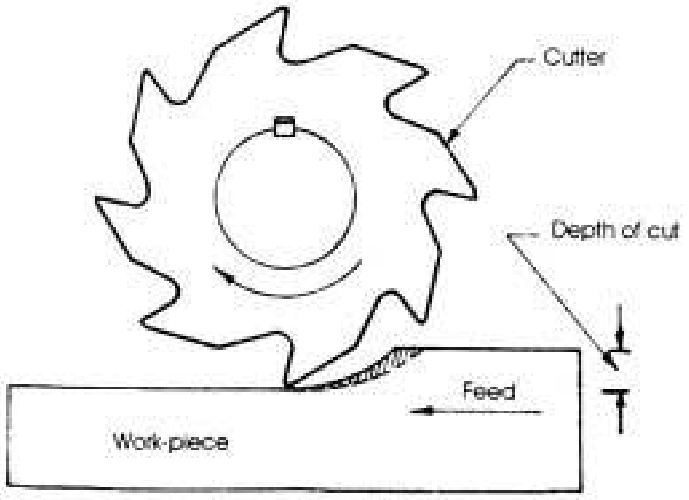


Fig. 1.50. Down milling.

1.32. Elements of a Plain Milling Cutter

A milling cutter may have either straight teeth *i.e.* parallel to the axis of ratation or in helical shape. A milling cutter can be considered as a built up unit of a number of single point cutting tools

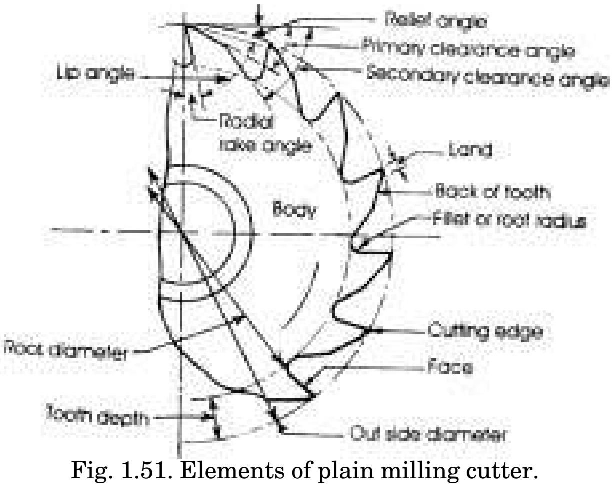


Fig. 1.51. Elements of plain milling cutter.

such that each tooth of the cutter is a single point cutting tool. Fig. 1.51 shows a plain milling cutter. The various parts of the cutter teeth are face, cutting edge, fillet, and body.

The various elements of the cutter are as follows :

(*i*) **Lip angle.** It is the angle included between the land and the face of the tooth or it is the angle between the tangent to the back at the cutting edge and the face of the tooth.

(*ii*) **Relief angle.** The angle in a plane perpendicular to the axis which is the angle between the land of a tooth and the tangent to the outside diameter of cutter at the cutting edge of that tooth.

The main function of relief angle is to prevent interference between land and work surface. The relief angle varies with the type of material being milled. Table 1.12 indicates the relief angles for different materials.

Table 1.12

Material	*Relief angle (Degrees)*
Cast Iron	4-7
Mild Steel	3-6
Brass	10-12
Aluminium alloys	10-12

(*iii*) **Rake angle (Radial).** It is the angle measured in the diameteral plane between the face of the tooth and radial line passing through the tooth cutting edge. It facilitates free cutting by tool allowing the chip to flow smoothly. This ensures less power consumption and better surface finish, less wear and hence more life of tool. It may be positive, negative or zero.

Table 1.13 indicates the rake angles in degrees for different materials to be milled.

Table 1.13.

Material	*Cemented carbide cutter*	*H.S.S. Cutter*
Aluminium alloys	10-12	20-35
Cast Iron Hard	0-3	8-10
Soft cast iron	3-6	10-15
Mild Steel	0-(-5)	10-15
Brass	3	10

(*iv*) **Primary clearance angle.** It is the angle between a line passing through the surface of the land and a tangent to the periphery at the cutting edge. This angle is provided to prevent the back of tooth from rubbing against the work. It is 4 to 7° for C.I. and 10 to 12 degrees for aluminium and brass.

(*v*) **Secondary clearance angle.** This angle is generally ground back of the land to keep the width of land within proper limits. It is usually 3 degrees greater than primary clearance angle.

(*vi*) **Land.** It is the narrow surface back of the cutting edge resulting from providing a clearance angle.

(*vii*) **Face.** Tooth face is the surface upon which the chip is formed when the cutter is cutting. It is may be flat or curved.

(*viii*) **Back of tooth.** It is created by the fillet and secondary clearance angle.

A milling cutter is a multiple tooth tool and therefore higher requirements are made to its construction, sharpening and operation. The cutter should be railably and rigidity fixed on the spindle. The machine in which high velocity milling is to be done should not only be capable of operating at high speeds and feeds but possess ample rigidity.

Figure 1.51(*a*) shows a high-speed steel milling cutter.

Relief and rake angles are applied to milling cutters in the same manner in which they were applied to tool bits. The milling cutter is actually a multiple tool bit arrangement.

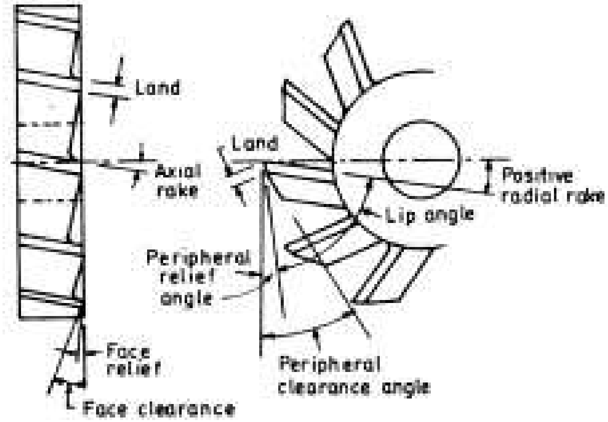
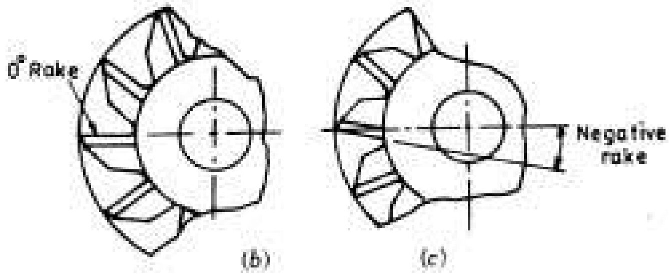


Fig. 1.51. (*a*)

Fig. 1.51(*b*) shows zero degree rake angle and Fig. 1.51(*c*) shows negative rake angle on tooth.



(b) (c)

Fig. 1.51.

1.32.1. Milling Cutter Teeth

Following formula may be used to find number of teeth on a milling cutter $Z = \frac{D}{12} + 8$ provided D is more than 50 mm.

where D = cutter diameter in mm.

1.32.2. Feed for milling cutter

Feed per tooth for milling steel with H.S.S. cutters and cemented carbide cutters is indicated in table 1.14*a*. The feed can be increased 1.5 to 2 times the feed values for milling steel.

Table 1.14(*a*)

	Feed in mm.tooth for a depth of cut			
Types of cutter		up to 3 mm	3 to 5 mm	5 to 8 mm
Plain milling cutter	H.S.S.	0.08-0.05	0.08-0.05	0.05-0.03
	Carbide	0.15-0.12	0.12-0.10	0.10-0.05
Face milling cutter	H.S.S.	0.12-0.10	0.10-0.05	0.06-0.03
	Carbide	0.25-0.20	0.20-0.15	0.10-0.12
End milling cutter	H.S.S.	0.05-0.10	0.02-0.050	0.08-0.04
	Carbide	0.06-0.04	0.05-0.03	0.04-0.03

Milling machine feed mechanism is used for

(*i*) Logitudinal motion.

(*ii*) Cross motion.

(*iii*) Vertical up and down motion.

(*iv*) Rapid traverses for all the above motions.

Feed motions may be provided from

(*i*) the main motor through a telescopic shaft.
(*ii*) a sparate motor placed at the knee.

1.33. Milling Cutter Sharpening

Milling cutters are usually sharpened on universal tool and cutter grinder. During grinding the cutter should be so set that the tooth can be advanced to the grinding wheel. Grinding is done with the face of a cup wheel. The cutter tooth should be positioned in respect to the wheel so that required relief angle (γ) is obtained. The tooth face bears on a stationary tooth rest. Fig. 1.52 shows sharpening a face milling cutter by grinding the tooth flanks. The amount of offset (S) is determined by using the following relation.

$$S = \frac{D}{2}\sin\gamma$$

where D is the cutter diameter in mm.

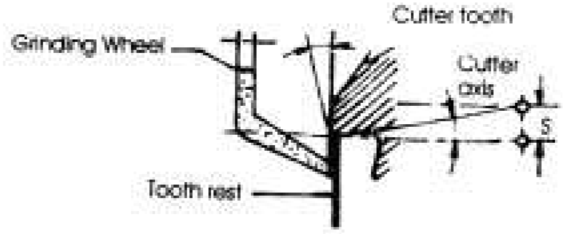


Fig. 1.52.

1.34. Materials for Milling Cutters

The common materials used for manufacturing milling cutters are as follows :

(*i*) High carbon steel
(*ii*) High speed steel
(*iii*) Cemented carbides
(*iv*) Stellite
(*v*) Ceramics.

High carbon steel is not commonly used except for a few cutters used for small scale production. High speed steel containing 18% tungsten is quite commonly used. Cemented carbides are very commonly used when milling cutters are to be run at higher speeds. Stellite is very useful material for milling cutters, when particularly machining hard metals, forgings and castings. Tips made of ceramics are also used in milling cutters. Such cutters, because of their brittleness, are used for finishing operations and smaller cuts. They are commonly used to machine cast iron and bronze.

Cemented carbide tipped cutters have the following advantages

(*i*) Their production capacity is high.

(*ii*) They can produce surface finish of high quality.

(*iii*) Hardened steels can be machined by them.

(*iv*) Their use leads to reduction in machining cost.

1.35. Number of Teeth in a Cutter

Milling cutters may be coarse tooth *i.e.* cutters with a large pitch and a small number of teeth or fine tooth *i.e.* cutters with a small pitch and a large number of teeth. The teeth are stronger in cutters with a coarse pitch. Coarse tooth straight flute cutters do not operate smoothly. Fine tooth cutters remove chips of relatively less thickness and are commonly used for finishing operation whereas coarse tooth remove thicker chip, resulting in removal of more metal in each cut and are used for roughing milling operation various empirical formulas are used to determine number of teeth in a cutter. A typical formula for calculating the number of teeth suitable for a cutter is as follows :

$$Z = A\sqrt{D}$$

where Z = Number of teeth

D = Diameter of cutter in mm

A = Constant

It depends upon the construction of cutter and machining conditions. Values of A are given in Table 1.14

Table 1.14

Type of cutter	θ*(Helix angle)*	*A*
Solid cutters		
(*a*) Fine tooth	15 to 20°	2
(*b*) Coarse tooth	Upto 30°	1
Inserted blade cutters		
(*a*) Find tooth	20°	0.9
(*b*) Coarse tooth	45°	0.8

1.36. Cutting Speed and Feed for Milling Cutter

Cutting Speed. It is the distance travelled per minute by the cutting edges of the cutter. It is expressed in metre per minute.

The cutting speed for milling cutter is found in the same way as for turning and drilling.

Let, D = Diameter of cutter in mm

N = R.P.M. of cutter

V = Cutting speed

$$= \frac{\pi DN}{1000} \text{ metre/minute}$$

The cutting speed should be as high as possible constant with economical cutter life before it needs regrinding.

Feed. It is the rate at which the workpiece advances past the rotating cutter. Three types of feed are distinguished.

(*a*) Feed per tooth (f_1.)

(*b*) Feed per cutter revolution (f_2).

(*c*) Table feed per minute (f_3).

These feeds are related by following equations.

Let T = Number of teeth on cutter

N = R.P.M. of cutter

f_1 = feed per tooth in mm

$f_2 = f_1 \times T$

$f_3 = f_2 \times N = f_1 \cdot T \cdot N$ mm/min.

Table 1.15 shows feed per tooth possible with various types of milling cutters.

Table 1.15. Feed for H.S.S. Cutters

Cutter	*Feed per tooth (mm)*
Form cutter	0.05 to 0.2
Slotting cutter	0.5 to 0.15
End mill	0.1 to 0.25
Face mill	0.1 to 0.5
Spiral mill	0.05 to 0.2

The average value of cutting speed in metre per minute for H.S.S. cutter and cemented carbide cutter are shown in Table 1.16.

Table 1.16

Material to be machined	*Tool material,*	
	H.S.S.	*Cemented carbide*
Aluminium	180-240	400-450
Brass soft	45-55	140
Copper	45-55	140
Gray cast iron	18-24	60
Cast iron hard	10-12	45
Mild steel	18-25	60
Hard steel	10-12	30

For a multipoint cutting tool such as milling cutter the maximum feed per tooth is limited by the following factors :

(*i*) Cutting edge strength.
(*ii*) Rigidity and allowable deflection.
(*iii*) Surface finish required.
(*iv*) Tool chip space.

With increase in feed, the cutting force gets increased and this causes greater deflection between tool and work piece. Accuracy can not be maintained if deflection is large. Proper cutting fluid can permit higher feeds and increased speeds as well as attainment of better surface finish. Cutting fluid should be directed to the exact point where cutting is taking place.

For setting up the operating conditions for any multipoint cutting tool there are three important variable that can be adjusted. They are as follows :

(*i*) Cutting speed.
(*ii*) Feed per cutting edge.
(*iii*) Cutting fluid.

1.37. Power Required at the Cutter

The horse power ($H.P_c$) required at the cutter depends on the volume of metal to be removed by the cutter per minute and varies for different materials.

$$H.P_c = \frac{Kfdb}{1,00,00}$$

where f = Feed in $^{\text{mm}}\!/_{\text{min.}}$

d = Depth of cut in mm.

b = Width of cut in mm.

K = Constant varying according to thematerial being cut.

The values of K are shown in Table 1.17.

Table 1.17

Material to be cut	*Constant (K)*
Light alloy (Positive rake)	1.3
Copper (Positive rake)	2.6
Brass (Positive rake)	3.0
Cast iron (Positive rake)	3.0
Bronze (Positive rake)	5.0
Steel (Negative rake),	
Low carbon	5.0
High carbon	8.5

$$H.P_m = \text{ Horse power at motor } = \frac{HP_c}{\eta}$$

The efficiency (η) of various milling machines is shown in Table 1.18.

Table 1.18

H.P. of the machine	*Efficiency (%)*
3	40
5	48
10	52
15	52
20	60
25	65
30	70
40	75

1.38. Machining Time

The machining time for plain milling and face milling is calculated at follows.

(*i*) **Plain Milling.** Fig. 1.53 shows milling operation being carried out by using plain milling cutter.

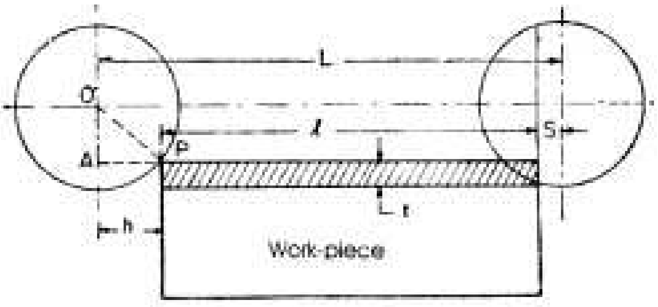


Fig. 1.53.

Let h = approach in mm.

l = Length of workpiece in mm.

s = Overrun of cutter in mm.

L = Total cutter travel in mm. = $h + l + s$

D = Diameter of cutter

t = Depth of cut

f = Feed/rev. in mm.

N = R.P.M.

Then T = Machining time

$$= \frac{L}{N \times f}$$

The approach h can be calculated as follows.

In triangle AOP

$$OP^2 = OA + AP^2$$

$$\left(\frac{D}{2}\right)^2 = \left(\frac{D}{2} - t\right)^2 + h^2$$

$$h = \sqrt{t(D - t)}$$

(*ii*) **Face milling.** Fig. 1.54 shows milling operation being carried out by face milling cutter.

l = Length of workpiece in mm.

h = Approach in mm.

s = Overrun of cutter.

$L = l + h + s$

f = Feed in mm /rev.

N = R.P.M.

$$T = \frac{L}{f \cdot N}$$

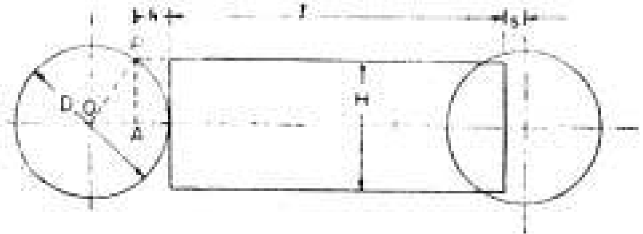


Fig. 1.54.

The approach h is can be calculated as follows.

H = Width of workpiece

D = Diameter of cutter

$$OP^2 = OA^2 + Ap^2$$

$$\left(\frac{D}{2}\right)^2 = \left(\frac{D}{2} - h\right)^2 + \left(\frac{H}{2}\right)^2$$

$$h = \tfrac{1}{2}[D - \sqrt{D^2 - H^2}]$$

1.38 Turning/Milling data sheet. This data sheet indicates the following details of machine tool such as lathe, milling machine and cutting tool, workpiece etc.

1. Machine tool details
   - (*i*) Type
   - (*ii*) Make
   - (*iii*) Power
   - (*iv*) Rigidity (*a*) Good (*b*) Fair (*c*) Poor
   - (*v*) Stiffness
   - (*vi*) Coolant.
2. Workpiece details
   - (*i*) Material
   - (*ii*) Strength
   - (*iii*) Hardness

(*iv*) Dimensions
(*v*) Tolerance
(*vi*) Surface conditions.

3. Tool details
(*i*) Material (*a*) Insert being used.
(*ii*) Tool geometry/approach angle
(*iii*) Tool holder condition
  (*a*) Good
  (*b*) Fair
  (*c*) New
(*iv*) Tool overhang
  (*a*) Normal
  (*b*) Excessive.

4. Operation details
(*i*) Nature
  (*a*) Turning
  (*b*) Copy turning
  (*c*) Face milling
  (*d*) End milling etc.
(*ii*) Stage
  (*a*) Roughing
  (*b*) Semi-finishing
  (*c*) Finishing.

5. Machining data
(*i*) Spindle speed
(*ii*) Cutting speed
(*iii*) Feed (*a*) Handfeed (*b*) Autofeed
(*iv*) Depth of cut
(*v*) Metal removal rate
(*vi*) Surface finish
(*vii*) Machining time
(*viii*) Number of parts made for insert
(*ix*) Wear pattern.

6. Chips

- (*i*) Type
  - (*a*) Short
  - (*b*) Long
  - (*c*) Broken
  - (*d*) Curved
  - (*e*) Ribbon type
- (*ii*) Colour
  - (*a*) Grey
  - (*b*) Blue
  - (*c*) Burnt.

1.38.1. Broaching Tool. A broaching tool or a broach has a series of multiple teeth. The main elements of a broaching tool are shown in Fig. 1.55. The first teeth are designed to do the heaviest cutting and are called roughing teeth. Next teeth are semi-finishing teeth which are followed by finishing teeth. The finishing teeth carry out finishing operation Fig. 1.56 shows teeth details. The rake angle (face angle) of the tooth depends on the material to be cut and its hardness, toughness and ductility.

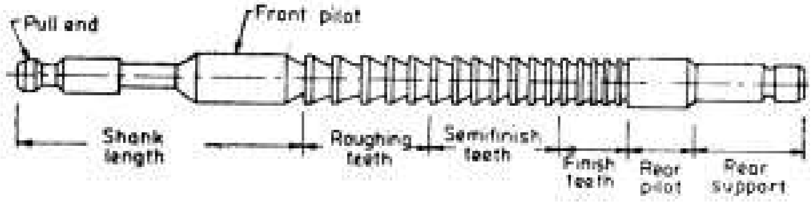


Fig. 1.55.

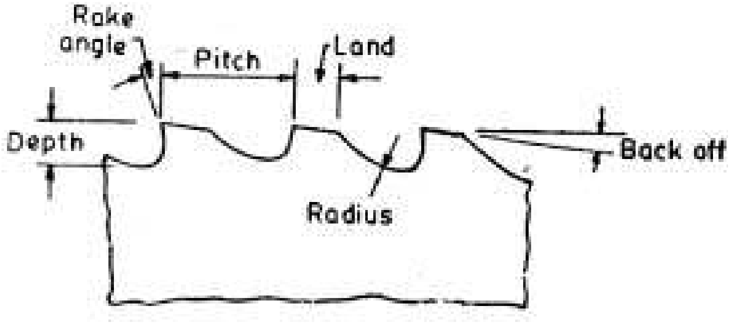


Fig. 1.56.

The values of face angle for different materials are as follows :

Material	*Face angle (Degrees)*
Aluminium	10
Brass	-5 -5
Cast iron	6 -8
Steel (hard)	8 -12
Steel (soft)	16 -20

The land of tooth determines its strength. The pitch determines the length of cut and chip thickness which a broach can handle.

During broaching operation the broach is either pulled or pushed by the broaching machine past the surface of the workpiece. In doing so each tooth of broach takes a small cut through the metal surface. Most broaches are made from 18-4-1 high speed steel, ground after hardening. Carbide broaches are used extensively in broaching cast iron.

1.38.2. Design elements. (*i*) *Length of broach.* The length of broach is calculated as follows :

$$L = L_1 + L_2 + L_3 + L_4 + L_5$$

where L_1 = Length of cutting portion

$= p_1 \cdot Z_1$

where p_1 = pitch of cutting teeth

Z_1 = Number of cutting teeth

L_2 = Length of finishing teeth

$= p_2 \cdot Z_2$

where p_2 = pitch of finishing teeth

Z_2 = Number of finishing teeth

L_3 = Length of pull end

L_4 = Length of neck

L_5 = Length of rear pilots

Values of L_3, L_4, L_5 depend on the standards, broaching fixture and work piece.

(*ii*) *Number of teeth.* The teeth provided on a broach are calculated as follows :

Z = Total number of teeth

$$= \frac{A}{d} + Z_1 + Z_2$$

where A = Amount of material to be removed

d = Depth of cut

Z_1 = Number of finishing teeth

= 3 or 4

Z_2 = Number of safety teeth

(*iii*) *Pitch of teeth.* It is determined as follows

p = pitch

$= (1.25 \text{ to } 1.50)\sqrt{L}$ for plain broach

$= (1.45 \text{ to } 1.90)\sqrt{L}$ for progressive broach

where L = Length of broached surface in mm.

(*iv*) *Cutting force* (P_C). The cutting force in broaching is found as follows

$$P_C = F\Sigma M$$

where P_C = Cutting force

F = Cutting force per mm of cutting edge length (kgf /mm)

ΣM = Total length of cutting edges of all the teeth in operation simultaneously (mm)

$$\Sigma M = \frac{\pi D Z_s}{Z_g} \text{ for cylindrical holes.}$$

where D = Maximum diameter of teeth in mm

Z_g = Number of teeth in a group.

Z_s = Number of teeth in operation simultaneously

$$\Sigma M = \frac{BN}{Z_g} \text{ for spline holes and key ways.}$$

where

B = Width of broached surface on work piece in mm.

N = Number of splines on key ways

The maximum broaching force is found as follows

$$P_{max} = k_1 \cdot k_2 \cdot d \cdot b \cdot \frac{l}{p}$$

where k_1 = Coefficient that takes into account friction

= 1.1 to 1.3

k_2 = Specific cutting resistance (kg/cm^2)

d = Depth of cut per tooth in cm.

b = Width of cut in cm.

l = Length of tool entry in cm.

p = Pitch of cutting edges in cm.

(*v*) *Power required for broaching.* The power required during broaching is determined as follows

$$P = \text{Power} = \frac{P_C\,V}{7200} \text{ k. Watt.}$$

where P_C = Cutting force in N

V = Cutting speed in m/min.

(*vi*) *Cutting speed.* The cutting speed is the speed of broach movement. It depends on material and type of surface being broached.

$$V = \frac{EK}{T^x S^y} \text{ m /min.}$$

where

E = Co–efficient characterising the processing conditions

K = Co–efficient that takes into account broach material

T = Tool life in minutes

S = Cut per tooth in mm.

Values of x, y depend on broach material and other processing conditions.

For broaching round holes in steel

$E = 12,\ x = 0.6,\ \ y = 0.6$

K = 1 to 1.5 and T = 80 to 100 minutes.

Value of cutting Speed in broaching usually varies from 1.5 m to 20 m per minute.

(*vii*) *Rise per tooth.* Rise per tooth depends on the following factors.

(*a*) Shape of hole

(*b*) Type of material of work piece

(*c*) Force available at the machine

(*d*) Size of hole.

h = Rise per tooth.

= 0.025 mm to 0.16 mm.

(*viii*) *Depth of tooth (H)*

$$H = 1.13\ \sqrt{fL\,k}$$

where

f = Difference in height between two successive cutting teeth
= 0.02 to 0.12 mm.
L = Length of work piece
K = Constant
= 3 to 5 for surface broaches
= 6 to 10 for internal broaches.

(*ix*) *Neck diameter.* It is 0.3 to 1.0 mm less than shank diameter.

(*x*) *Width of land.* It varies between 2.5 to 4.5 mm.

(*xi*) *Groove radius.* It varies between 2.5 to 5 mm.

(*xii*) *Broaching allowance.* It is calculated as follows

A = Broaching allowance
$= 0.005D + (0.1 \text{ to } 0.2)\sqrt{L}$

where D = Basic diameter of hole in mm.
L = Length of hole to be broached in mm.

In terms of pitch the following commonly used values may be adopted.

Tooth height = (0.35 to 0.40) p.
Land width = (0.3 to 0.4) p.
Groove radius = (0.2 to 0.25) p.
Flute angle = 30° to 36°
Groove angle = 45° to 50°.

1.39. Improving Cutting Efficiency

The cutting efficiency during machining can be improved by using following methods.

(*i*) Reduction of tool-face friction.

(*ii*) By reducing chip length by means of chip breakers.

(*iii*) By improving surface finish of tool face by honing or chrome plating.

(*iv*) By increasing cutting speeds.

(*v*) Increased depth of cut increases cutting efficiency by a small amount.

1.40. Power Distribution

Out of total power supplied to the machine tool the power available at the cutter depends upon the type of machine tool. A typical power distribution for a milling machine and a lathe is as follows.

Milling Machine

Electrical power input = 100%
Losses in motor = 16%
Losses in spindle transmission = 18%
Power to feed = 21%
Power available at cutter
= 45%

Lathe

Electrical power input = 100%
Losses in motor = 11%
Losses in machine = 18.5%
Power to feed = 0.5
Power available at tool = 70%

1.41. To Determine Power Rating of Electric Motor

The required power of main drive is determined on the basis of useful power calculated for the most effective cutting conditions. In designing general purpose machine tools the useful power is calculated for the maximum cutting speeds and feeds.

$$N_m = \text{Required power rating of electric motor}$$

$$= N + N_f$$

where N = Useful power

N_f = Power lost in overcoming friction

η = Efficiency of main drive.

$$= \frac{N}{N_m}$$

The efficiency (η) may range from 0.7 to 0.85 for machine tools with a rotary primary motion having a single motor drive.

1.42. Cutting Tool Design

The design of a cutting tool means determination of all the dimensions and shapes of all the elements of a cutting tool by carrying out calculations and graphical construction. The cutting of metals with a single point cutting tool is fully applicable to any kind of cutting tool since all tools remove a certain layer of stock and impart the required shape, size, and surface finish to the machined part. The cutting teeth of all cutting tools whatever may be their shapes and purpose resemble to the point of single point tool.

The common procedure carried out during the design of cutting tool consists of following calculations :

(*i*) To determine forces acting on cutting surface of the tool.
(*ii*) To find out optimum tool geometry.
(*iii*) To select suitable material for making cutting elements of the tools.
(*iv*) To find suitable shapes of cutting and mounting elements of tool and to determine the tolerance on the dimensions of cutting and mounting elements of tool depending on machining accuracy required on work piece.
(*v*) To determine strength and rigidity of mounting and cutting elements of tool.
(*vi*) To prepare a working drawing of tool.

1.43. Friction between Chip and Tool

Friction between the chip and tool plays a significant role in cutting process. It should be as low as possible friction can be reduced in the following ways :

(*i*) Improved tool finish and sharpness of the cutting edge.
(*ii*) Use of low friction tool material.
(*iii*) Improved tool geometry.
(*iv*) Use of cutting fluids.
(*v*) High sliding speeds.

1.44. Design of a Single point Lathe Tool

The design on the cutting tool depends upon the cutting forces acting on the cutting surfaces of the tool and rigidity of the machine on which cutting tool is used. The shank of a single point tool may be rectangular, square or rounding cross-section. The rectangular cross-section is commonly used because the reduction in strength of shank (at section YY of a single point cutting tool as shown in (Fig. 1.56 (*a*) is less than for a square shank when a seat is cut for a tip.

F_H = Cutting force

Let B = Width of shank

H = Height of shank.

For rectangular cross-section the relations between B and H are as follows :

$$\frac{H}{B} = 1.25 \text{ for roughing operations}$$

$$\frac{H}{B} = 1.6 \text{ for semifinishing and finishing operations.}$$

The minimum permissible size of the shank cross-section on strength basis is calculated by equating the actual bending moment to the maximum moment permitted by cross-section of the shank.

M_1 = Bending moment due to cutting force F_H

$= F_H \times l$

where l = Cutting tool overhang.

$(l = 1 \text{ to } 1.5\ H)$

M_2 = Maximum moment permitted by the cross–section of the shank $= f \times Z$

where f = Permissible bending stress for the shank material

Z = Section modulus of the tool shank

$$\therefore \quad M_1 = M_2$$

$$F_H \times l = f \times Z$$

For rectangular cross-section of tool shank

$$Z = \frac{BH^2}{6}$$

$$\therefore \quad F_H \times l = f \times \frac{BH^2}{6}$$

$$BH^2 = \frac{6\, F_H \cdot l}{f}$$

$$H = \sqrt{\frac{6F_H \cdot l}{f.B}}$$

For square cross-section shank

$$B = H.$$

$$F_H \times l = f \cdot \frac{BH^2}{6}$$

$$= f \cdot \frac{B \cdot B^2}{6}$$

$$B^3 = \frac{6F_H\, l}{f}$$

$$B = \sqrt[3]{\frac{6F_H \cdot l}{f}}$$

For round cross-section shank

$$Z = \frac{\pi}{32} D^3$$

where D = Diameter of shank.

$$l \times F_H = fZ. = f \cdot \frac{\pi}{32} D^3$$

$$D^3 = \frac{32 \cdot F_H \cdot l}{\pi \times f}$$

$$D = \sqrt[3]{\frac{32F_H \cdot l}{\pi f}}$$

Square shank tools are commonly used for boring machines, screw machines and turret lathes. Round shank tools are generally used for boring and thread cutting operations.

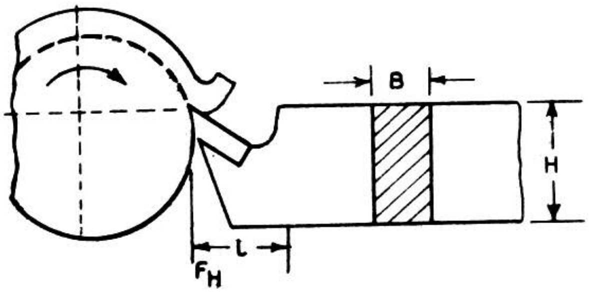


Fig. 1.56 (*a*)

Example 1.0. *In orthogonal cutting of a mild steel bar on a lathe the feed (f_1) used is 0.3 mm per revolution and the depth of cut (t) is 2 mm. Determine the cross-section of a rectangular tool shank if the* *allowable stress in the shank material is 7* kg/m^2 *and the cutting force* (F_H) *can be calculated by the relation* $F_H = 200 \times f_1^{0.75} \times t$.

Solution. Let $H =$ Height of a shank

$B =$ Width of shank

$$\frac{H}{B} = 1.16 \text{ (assume)}$$

we know $H = \frac{\sqrt{6F_H \cdot l}}{f \cdot B}$

where $F_H =$ cutting

$$= 200 \times f_1^{0.75} \times t = 200 \times 0.3^{0.75} \times 2$$

$$= 170 \text{ kg.}$$

$l =$ Cutting tool overhang.

$= 1.25\,H$ (assume)

$f = 7$ kg /mm^2

$$H = \sqrt{\frac{6 \times 170 \times 1.25H}{7 \times \frac{H}{1.6}}}$$

$H = 17$ mm.

$$B = \frac{H}{1.6} = \frac{17}{1.6} = 10.6 \text{ mm.}$$ **Ans.**

Example 1.1. *Determine the spindle speed for a high steel drill 12 mm diameter cutting medium carbon steel at 28 m/min.*

Solution. $V = 28$ m/min

$D =$ Diameter of drill = 12 mm.

$N =$ R.P.M. of spindle

$$V = \frac{\pi DN}{1000}$$

$$28 = \frac{\pi \cdot 12 \cdot N}{1000}$$

$N = 743$ R.P.M. **Ans.**

Example 1.2. *Determine the machining time to drill a hole of 20 mm diameter in a workpiece 25 mm thick by a drill at a cutting speed of 30 metre/min with a feed of 0.2 mm/rev.*

Solution. $L =$ Length of axial travel of drill in mm.

$=$ Thickness of workpiece + 0.3 D

where $D =$ Hole diameter

$L = 25 + 0.3 \times 20 = 25 + 6 = 31$ mm

$V =$ Cutting speed = 30 metre per min.

$$V = \frac{\pi D N}{1000}$$

$$30 = \frac{\pi \times 20 \times N}{1000}$$

$N = 477$ R.P.M.

$T =$ Machining time

$f =$ feed/rev = 0.2 mm /rev.

$$T = \frac{L}{N \times f} = \frac{31}{477 \times 0.2} \text{ minute}$$

$$= \frac{31 \times 60}{477 \times 0.2} = 19.5 \text{ seconds.}$$ **Ans.**

Example 1.3. *A 30 mm H.S.S. drill is used to drill a hole in a cast iron block 100 mm thick. Determine the time required to drill the hole if feed is 0.3 mm/rev. Assume an over travel of drill as 4 mm. The cutting speed is 20 metre/min.*

Solution. $D =$ diameter of drill = 30 mm

$$V = \text{cutting speed} = \frac{\pi D N}{1000}$$

$$20 = \frac{\pi \cdot 30 \cdot N}{1000}$$

$N = 202$ R.P.M.

$h =$ approach

$= 0.3\, D$ (assume) $0.30 \times 30 = 9$ mm

$s =$ over travel = 4 mm

$L =$ Total length

$= h + S +$ Thickness of work piece

$= 9 + 4 + 100 = 113$ mm.

$$T = \text{Machining Time} = \frac{L}{fN} = \frac{113}{0.3 \times 202}$$

where $f =$ feed

$T = 1.85$ minutes.

Example 1.4. *Calculate the power required to drill 20 mm diameter hole in mild steel at a feed of 0.25 mm/rev and at a drill* *speed 300 R.P.M. Determine also the volume of metal removed per unit of energy.*

Solution. $D =$ hole diameter = 20 mm

$f =$ feed = 0.25 mm/rev.

$N =$ 300 R.P.M.

$T_1 =$ Torque $= C \cdot f^{0.75} D^{1.8}$

where, $C =$ Constant = 0.36 for mild steel

$T_1 = 0.36 \times 0.25^{0.75} \times 20^{1.8} = 27.96$ newton–m.

$P =$ Power

$$= \frac{2\pi N T_1}{60{,}000} = \frac{2\pi \times 300 \times 27.96}{60{,}000} = 0.879 \text{ kW.}$$

Volume of metal removed per minute

= Area of hole × Feed × Speed

$$= \frac{\pi}{4} \times 20^2 \times 2.25 \times 300$$

$= \mathbf{23550\ mm^3.}$

Energy consumption $= \dfrac{23550}{879}$ **Ans.**

$= 21\ \text{mm}^3$/watt minute.

Example 1.5. (*a*) *A carbide face milling cutter of 200 mm diameter is used take one cut across the face of a block of aluminium which is 200 mm wide. The length of block is 450 mm. If a feed of 0.75 mm/rev is used how long will it take to machine one cut on the block. The total travel is 12 mm. The cutting speed is 320 meter/minute.*

(*b*) *Calculate the time taken if the diameter of cutter is 300 mm.*

Solution.

(*a*) $D =$ Diameter of cutter = 200 mm.

$H =$ Width of block = 200 mm

$l =$ Length of block = 450 mm

$f =$ Feed = 0.75 mm/rev.

$s =$ Overrun of cutter = 12 mm.

As the diameter of cutter is equal to width of block, (Fig. 1.57).

∴ $h =$ Approach

$$= \frac{D}{2} = \frac{200}{2} = 100 \text{ mm.}$$

$N =$ R.P.M. of cutter

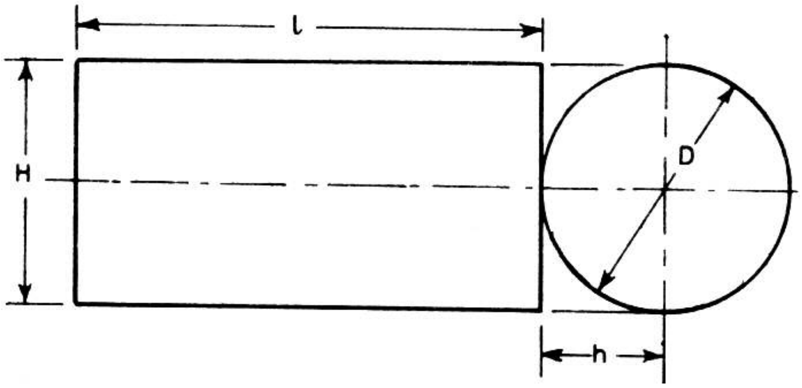


Fig. 1.57.

$$V = \text{ cutting speed} = \frac{\pi DN}{1000}$$

$$320 = \pi \times \frac{200}{1000} \times N$$

$$N = 510 \text{ R.P.M.}$$

Total length, $L = l + h + s = 450 + 100 + 12 = 562$ mm.

$$T = \text{machining time} = \frac{562}{0.75 \times 510}$$

$$= 1.45 \text{ minute}$$ **Ans.**

(*b*) D = Diameter of cutter = 300 mm.

In this case the diameter of cutter is more than the width of block

$$\therefore \quad h = \text{Approach} = ½[D - \sqrt{D^2 - H^2}]$$

$$= ½[300 - \sqrt{300^2 - 200^2}] = 38 \text{ mm.}$$

$$\therefore \quad L = \text{Total length} = l + h + S$$

$$= 450 + 38 + 12 = 500 \text{ mm.}$$

$$V = \frac{\pi DN}{1000}$$

$$320 = \frac{\pi \times 300 \times N}{1000}$$

$$N = 340$$

$$T = \text{Machining time}$$

$$T = \frac{L}{f.N.} = \frac{500}{0.75 \times 340}$$

$$= 1.97 \text{ minutes}$$ **Ans.**

Example 1.6. *A carbide milling cutter 250 mm in diameter is used to cut a block of mild steel with a plain cutter .The block is 500 mm long. If the feed is 0.50 mm/rev and depth of cut is 1.2 mm determine the time required to take one cut. The over travel is 16 mm. The cutting speed is 80 metre per minute.*

Solution. D = Diameter of cutter = 250 mm.

f = feed = 3.5 mm/rev.

l = Length of block = 500 mm.

t = Depth of cut = 1.2 mm

s = Over travel = 16 mm

V = cutting speed = 80 metre/minute

N = R.P.M.

$$V = \frac{\pi DN}{1000}$$

$$80 = \frac{\pi \cdot 250 \cdot N}{1000}$$

N = 102 R.P.M.

h = Approach = $\sqrt{t(D-t)}$

$= \sqrt{1.2(250 - 1.2)} = 17.3$ mm.

L = Total length = $l + h + s$

$= 500 + 17.3 + 16 = 533.3$ mm.

T = Machining time

$$= \frac{L}{f \cdot N} = \frac{533.3}{0.5 \times 102}$$

= 10.45 minute **Ans.**.

Example 1.7. *Determine the horse power required to face mill a block of grey cast iron 125 mm wide, using a tungsten carbide cutter 150 mm in diameter and having eight tooth. The depth of cut is 3 mm and a chip thickness of 0.25 mm is to be used. The cutting speed is 60 m/min and the specific horse power required for cast iron is 0.30 H.P./cm^3/min.*

Solution. V = cutting speed = 60 m/min.

Now $$V = \frac{\pi DN}{1000}$$

$$60 = \frac{\pi \times 150 \times N}{1000}$$

where D = cutter diameter

N = R.P.M. of cutter

$\therefore \quad N = 128$

Now $\quad f_1$ = feed per tooth (chip thickness)

= 0.25 mm.

f_2 = feed per cutter revolution

= $f_1 \times$ Number of teeth

$= f_1 \times 8 = 0.25 \times 8$

f_3 = feed per minute = $f_2 \times N$

= $0.25 \times 8 \times 128$ mm/minute

= 256 mm/minute = 25.6 cm/minute.

The metal removal rate (w)

w = Width of workpiece × Depth of cut × feed per minute

$= 125 \times 0.3 \times 25.6 \text{ cm}^3\text{/minute}$

$= 96 \text{ cm}^3\text{/minute}$

Horse power required

$= 96 \times 0.03 = 2.88.$ **Ans.**

Example 1.8. *The feed of an 8-tooth face mill cutter is 0.325 mm per tooth at 200 R.P.M. The material cut is 300 BHN Steel.*

If the depth of the cut is 3 mm and width is 100 mm calculate the following :

(*a*) *B.P. at the cutter.*

(*b*) *H.P. at the motor.*

Solution . $$H.P_c = \frac{K \cdot f \cdot d \cdot b}{10,0000}$$

where K = Constant = 8.5 (From table 1.11)

f = Feed = $0.325 \times 8 \times 200$

= 520 mm/minute.

d = Depth of cut in mm

= 3 mm

b = Width of cut = 100 mm.

$$HP_c = \frac{85 \times 520 \times 3 \times 100}{10^5} = 12.48 \qquad \textbf{Ans.}$$

Let the efficiency of the machine be 50%

HP_m = Horse power at the motor

$$= \frac{HP_c}{0.5} = \frac{12.48}{0.5} = 25. \qquad \textbf{Ans.}$$

Example 1.9. *Determine the power required by a milling cutter to take a cut 100 mm wide 3 mm deep at 75 mm/min feed for an alloy steel. If the cutter diameter is 100 mm and cutting speed is 15 meter/min, find the mean torque at the arbor.*

Solution. The horse power required at the cutter is given by

$$HP_c = \frac{K \cdot f \cdot d \cdot b}{10,0000}$$

Now $K = 8.5$ (From table 1.11.)

$f = 75$ mm/min.

$b = 100$ mm.

$d = 3$ mm.

$$HP_c = \frac{8.5 \times 75 \times 100 \times 3}{100,000} = 1.9.$$

Mean force at the periphery of the cutter,

$$F = \frac{HP_c}{V} \times 4500$$

where V = cutting speed = 15 metre / min.

$$F = \frac{1.9 \times 4500}{15} = 570 \text{ kg.}$$

Mean torque at the arbor.

$$T = F \times \text{ Radius of cutter}$$

$$T = 570 \times \frac{50}{1000} = 28.5 \text{ kg–m.}$$ **Ans.**

Example 1.10. *During turning a mild steel component with a 0-10-7-7-8,-9-1.5 mm shaped orthogonal shaped tool a depth of cut of 1.8 mm is used. If feed is 0.18 mm/rev. and a chip thickness of 0.36 mm is obtained determine the following :*

(*a*) *Chip thickness ratio.*

(*b*) *Shear angle.*

Solution. r = chip thickness ratio $= \frac{t_1}{t_2}$

where t_1 = chip thickness before cutting

= 0.18 mm.

t_2 = chip thickness after cutting

= 0.36 mm.

$$r = \frac{0.18}{0.36} = 0.5$$

(*b*) β = Shear angle

α = Back rake angle = 0° (given)

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

$$= \frac{0.5 \times \cos\theta}{1 - 0.5\sin\theta} = \frac{0.5}{1} = 0.5$$

$$\beta = 26^{\circ} - 34$$

Example 1.11. *Determine the power required for milling a mild steel workpiece with a cutter of 80 mm diameter having 9 teeth and rotating at 120 R.P.M. The workpiece has a width of 60 mm. Depth of cut is 4 mm and tooth feed is 0.03 mm.*

Solution. P = Power in kW /cm^2 /min.

It is given by the following formula

$$P = \frac{C}{\left[f\sqrt{\frac{d}{D}}\right]^{n}}$$

where f = feed per tooth

C = Material constant

d = Depth of cut in mm.

The values of C and n depends upon material to be cut. The values are as follows :

Mild Steel : $C = 0.02$ *to* 0.03.

$n = 0.28$

Bronze : $C = 0.006–0.007$

$n = 0.4$

Now $C = 0.02$ (say)

$$f = \text{Feed per tooth} = \frac{F}{T}$$

where F = Feed rate, mm /min.

T = Number of teeth on cutter

$f = 0.03$ mm.

d = Depth of cut = 4 mm.

D = Diameter of cutter = 80 mm.

$n = 0.28$

$$P = \frac{0.02}{\left[0.03\sqrt{\frac{4}{80}}\right]^{0.28}} = 0.245$$

P_1 = Power in kW

$= P \times b \cdot d \cdot f \cdot T \cdot N$

where b = width of workpiece = 6 cm.

N = R.P.M. = 120

$P^1 = 0.245 \times 6 \times 0.4 \times 0.003 \times 9 \times 120$

$= 1.9$ kW.

Example 1.12. *Determine the power required to cut a brass bar on a lathe when the cutting speed is 18 metres per minute, feed is 0.06 mm per revolution and depth of cut is 0.058 cm. Assume that the power lost in friction is 30%.*

Solution. H.P. = Total horse power required $P_1 + P_2$

where P_1 = Power required for cutting action

P_2 = Power lost in friction

= 30% *of* P_1

∴ H.P. = 1.3 P_1

$$= \frac{1.3 \times K \cdot d \cdot f \cdot V}{4500}$$

where K = constant = 12000 for brass

d = depth of cut in cm.

= 0.058 cm. = 0.58 mm

f = 0.06 mm per revolution

V = cutting speed = 18 metres per minute.

The value of K depends on material being machined. The values of K are as follows.

Metal to be cut	K
Aluminium	6700
Brass	12000
Cast iron	8600
Steel	11000

$$\text{H.P.} = \frac{1.3 \times 12000 \times 0.58 \times 0.06 \times 18}{4500}$$

= 2.17 **Ans.**

Example 1.13. *Estimate the power of electric motor for a drilling machine to drill a hole 15 mm diameter in cast iron (soft) work piece at 450 R.P.M. and 0.2 mm feed. The specific power is 0.03 kW and efficiency of motor is 80%.*

Solution. D = Hole diameter

= 15 mm

N = 450 R.P.M.

F = feed = 0.2 mm/rev.

S = specific power

= Power required to remove material of one cubic centimeter per minute.

= 0.03 kW

η = Efficiency of motor = 0.8

V = volume of metal to be cut during drilling

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

$$= \frac{\pi}{4}\left(\frac{15}{10}\right)^2 \times \frac{0.2}{10} \times 450$$

= 15.9 cu cm/min

P_1 = Power required at the drill

= $V \times S$ = 15.9 × 0.03

= 0.477 kW

P = Power of electric motor

$$= \frac{0.477}{\eta} = \frac{0.477}{0.8}$$

= 0.6 kW **Ans.**

Example 1.14. *A broach is used to cut a key way 8 mm wide, 5 mm deep in a boss 64 mm long. Determine*

(*a*) *cutting length of broach*

(*b*) *Number of teeth on broach.*

Solution. L = Length to be broached in mm = 64 mm

p = pitch

= $1.5\sqrt{64}$ = 12 mm.

d = Depth of cut = 5 mm

h = rise per tooth

= 0.0875 mm (Assume)

Z_1 = Number of teeth

$$= \frac{d}{h} = \frac{5}{0.0875} = 57$$

Z_2 = Number of finishing teeth

= 13 (say)

Z = Total number of teeth

$= Z_1 + Z_2 = 57 + 13 = 70.$

$p_1 = p$ Pitch for finishing teeth

$$= \frac{p}{2} = \frac{12}{2} = 6 \text{ mm.}$$

(The pitch for finishing teeth is usually half of that adopted for cutting teeth)

L = Total effective length (cutting length) of broach = $L_1 + L_2$

where L_1 = length of roughing teeth

L_2 = length of finishing teeth

$$L = 57 \times p + 13 \times p_1$$

$$= 57 \times 12 + 13 \times 6$$

$$= 762 \text{ mm.}$$

Example 1.15. *Calculate the cutting times for each of the following maching operations :*

(*a*) *Work piece-grey cast iron, cutting speed 20 m / min, depth of cut is 2.5 mm, dimensions of surface to be machined 50 mm long* × 70 *mm wide over run of cutter = 10 mm*

cutter (i) Helical mill 76 mm diameter and 45 degree helix

(ii) Face mill 100 mm diameter.)

Solution. (*a*) *For helical milling*

D = cutter diameter

= 76 mm

α = Helix angle

$= 45^\circ$

d = Depth of cut

= 2.5 mm

Z = Number of teeth in cutter

$$= \frac{4\pi D}{D + 4d} \times \cos\alpha = \frac{4\pi \times 76}{76 + 4 + 2.5} \times \cos 45$$

$$= 8$$

f = feed/tooth

= 0.25 mm (From table 1.15)

h = Approach

$= \sqrt{d(D-d)}$

$= \sqrt{2.5(76-2.5)}$

= 13.6 mm

N =R.P.M. of spindle

V = cutting speed = 20

$$20 = \frac{\pi DN}{1000}$$

$$= \frac{\pi \times 76 \times N}{1000}$$

$N = 84$ *R.P.M.*

S = over run of cutter = 10 mm

F = Table feed

$= N \times Z \times f$

$= 84 \times 8 \times 0.25$

= 168 mm/min

L = length of cut

= length of workpiece + h + s

= 50 + 13.6 + 10 = 73.6 mm.

$$T = \text{Time for one cut} = \frac{L}{F}$$

$$= \frac{73.6}{168}$$

= 0.44 min.

(*b*) *For face milling*

b = width of component (cut) = 70 mm.

$$Z = \frac{2\pi D}{b}$$

$$= \frac{2\pi \times 100}{70} = 9$$

d = Depth of cut

= 0.3 mm (From table 1.15)

D = cutter diameter = 100 mm.

$$h = \text{Approach}$$

$$= \frac{D}{2} - \sqrt{\frac{D^2}{4} - \frac{b^2}{4}}$$

$$= \frac{100}{2} - \sqrt{\frac{100^2}{4} - \frac{70^2}{4}}$$

$$= 14.3 \text{ mm}$$

$$V = 20 = \frac{\pi DN}{1000}$$

$$20 = \frac{\pi \times 100 \times N}{1000}$$

$$N = 64 \text{ R.P.M.}$$

$$F = \text{Table feed}$$

$$= N \times Z \times f$$

$$= 64 \times 9 \times 0.3$$

$$= 172.8 \text{ mm/min}$$

$$L = \text{Length of component} + h + s$$

$$= 50 + 14.3 + 10$$

$$= 74.3 \text{ mm}$$

$$T = \text{Time for one cut}$$

$$= \frac{L}{F}$$

$$= \frac{74.3}{172.8}$$

$$= 0.43 \text{ minutes.}$$

Example 1.16. *The bore of an alloy steel component is to be finish broached to* $21.75^{+0.01}$ *mm diameter, the bore prior to broaching being* $21.24^{+0.05}$ *mm diatermeter-Determine*

(*a*) *Pitch of teeth*

(*b*) *Length of cutting portion*

(*c*) *Force to pull broach through work if length of hole is 36 mm and rise per tooth is 0.025.*

Number of finishing teeth is 3 and force to remove 1 mm^2 *of metal is 4800 N.*

Sketch the broach.

Solution. $L_1 = \text{hole length}$

$$= 36 \text{ mm}$$

p = pitch

$= 1.5\sqrt{36}$

$= 9$ mm.

Z = Number of finishing teeth = 5

L = Length of cutting portion of broach

$$= P\left(\frac{T}{h} + Z\right)$$

where T = metal to be removed by roughing teeth

$$= \frac{21.76 - 21.24}{2}$$

$= 0.26$ mm.

h = Rise per tooth = 0.025 mm.

$$L = p\left(\frac{T}{h} + Z\right)$$

$$= 9\left(\frac{0.26}{0.025} + 5\right)$$

$= 144$ mm.

The first cutting tooth will be 21.24 mm diameter, second will be 21.26 mm diameter. The rest of cutting teeth will rise in steps of 0.05 mm on diameter and finishing teeth will be 21.76 mm diameter.

F = Force to pull broach = $\pi D Z_1 h K$

where D = Finish diameter

$= 21.76$ mm.

Z_1 = Maximum number of teeth cutting at a time

$= 3$

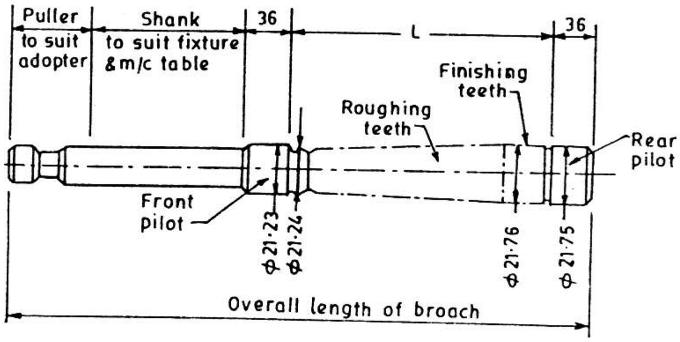


Fig. 1.58.

K = Force required to remove 1 mm^2 of metal at given rise / tooth

$= 4800\ N$

h = Rise per tooth = 0.025 mm.

$F = \pi \times 21.76 \times 3 \times 0.025 \times 4800$

$= 25581 \text{ N} = 25.581 \text{ kN}$

Fig. 1.58 shows sketch of broach.

Example 1.17. *A mild steel rod is machined on a lathe. Find power of motor using following data.*

Cutting speed = 90 m /min

Feed = 0.5 mm / rev.

Depth of cut = 2.54 mm

Correction factor = 0.5

Solution. The formula for calculating power of motor is as follows:

P = Power (kW)

$$= \frac{v \cdot f \cdot d \cdot F}{6000 \times \eta}$$

where v = cutting speed = 90 mpm

f = Feed = 0.5 mm / rev.

F = specific resistance

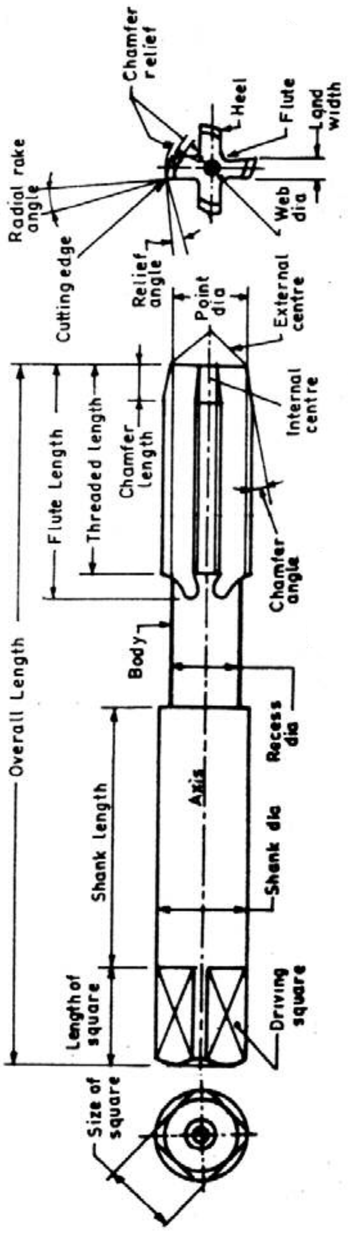
$= 270 - 300\ kg / \text{mm}^2$ (assume) for steel

d = Depth of cut = 2.54 mm

η = correction factor

$$p = \frac{90 \times 0.5 \times 2.54 \times 280}{6000 \times 0.5}$$

$$= 10.7 \text{ kW.}$$



1.45. Screwing Tap

Fig. 1.59 shows various elements of a screwing tap. Fig. 1.60 shows flutes with right hand (RH) helix and flutes with spiral point. Fig. 1.61 shows various types of rake for the tap teeth.

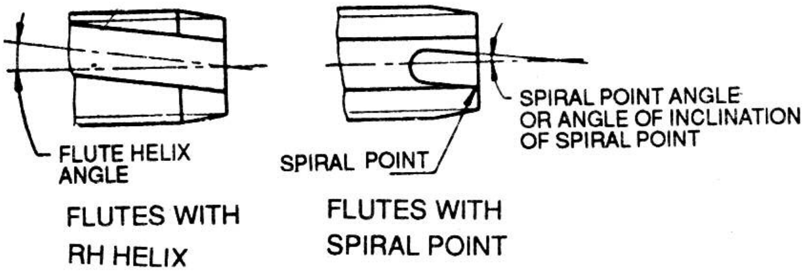


Fig. 1.60.

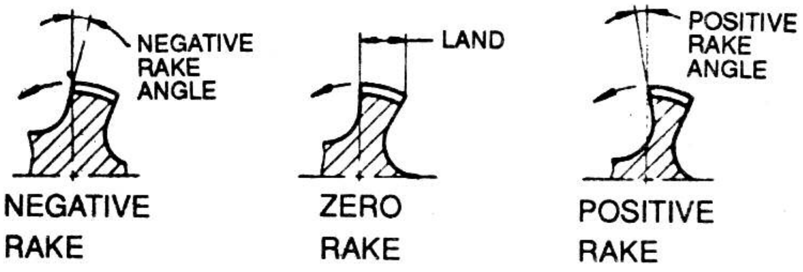


Fig. 1.61.

1.46. Force and Torque during Drilling

The drilling thrust (P) and drilling torque (T) depend on the following factors :

(*i*) Workpiece material (*ii*) Hardness

(*iii*) Point angle of drill (*iv*) Drill diameter

(*v*) Feed rate

P = Thrust force during drilling as shown in Fig. 1.62.

$= 9.80665.\, C_1 \,.\, K_1 \,.\, K_2 \,.\, d^x \,.\, s^y$ Newton.

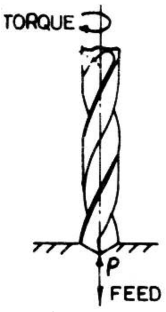


Fig. 1.62.

where C_1 = Material constant.

K_1 = Brinell hardness coefficient.

K_2 = Drill point angle coefficient.

d = Drill dia meter in mm.

S = Feed rate in mm /Rev.

T = Drilling torque.

$= 9.80665.\ C_2 \ .\ K_1 \ .\ K_2 \ .\ d^m\ s^n$ (N–mm)

where x, y, m and n are constants.

The values of c_1, c_2, x, y, m and n are shown in Table 1.19.

Table 1.19

Materials	C_1	C_2	x	y	m	n
Carbon steel						
σ_u = 735 N /mm^2	85	34	1	1.9	0.7	0.8
C.I. *HB* = 190	52	20	1	1.9	0.8	0.8
Bronze	31	12	1	1.9	0.8	0.8
Alloy steel :						
σ_u = 112 N /mm^2	270	75	1.15	2.1	0.77	0.76
σ_u = 686 N /mm^2	163	55	1.0	2.0	0.62	0.76

The values of K_1 is indicated in Table 1.20

Table 1.20

HB	130	150	170	190	210	220	250	270
K_1	0.80	0.87	0.94	1.00	1.96	1.13	1.18	1.25

The values of K_2 are indicated in Table 1.21.

Table 1.21

½ point angle	*P*			*T*		
degree	45	60	75	45	60	75
Steel	0.97	1.0	1.04	1.23	1.0	0.82
Cast iron (K_2)	0.73	1.0	1.27	1.25	1.0	0.88

1.47. Brazing of Carbide Tools

For carbide tools a suitable shank material is carbon steel with a tensile strength of about 70 kg/mm^2. So that the least possible stresses shall occur, the ratio between the thickness of the carbide tip and the height of the shank material beneath the tip should be about 1 : 3 as shown in Fig. 1.63.

Grind the brazing areas of the tip on a silicon carbide or diamond wheel. Another reliable method is hard blasting with steel shot. De-grease the tip, and when necessary the tip-seating of the shank as well, with, for instance, trichlorethylene, acetone or carbon tetrachloride.

Arrange the seating of the tip so that the brazing stresses are as small as possible. 'Open' seatings of the tip generally give the best result. Onlt the underside of the tip and one side face (max. on-third of the tip's thickness) should come into contact with the tool shank. Then carbide tip is brazed on the seat.

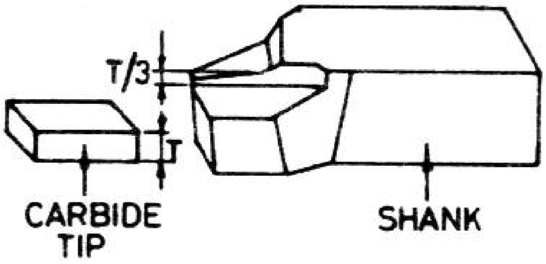


Fig. 1.63.

Re-grinding is facilitated and the carbide tip's thickness is better preserved if the tip is set at an inclination greater than that which the top-rake angle requires. Tip-seating angle for most turning tools is greater by 2° than the top rake angle.

The brazed joint has two functions. It has to hold the carbide tip in position despite heavy shearing forces and high temperatures. It must also be capable of minimizing the stresses which occur during the heating and cooling of the tool material.

1.48. Cutting Tool Wear

For a given metal cutting operation the metal removal rate and the tool life are controlled by cutting parameters. By a judicious selection of cutting data one could get the maximum rate of metal removal and optimum tool life.

Under usual cutting conditions when the form stability of the cutting edge has been achieved, wear due to interaction between the chip and the tool and between the workpiece and the tool are the main processes by which the cutting tool wears out. These types of wear are called crater wear and flank wear respectively.

1.48.1. Tool wear mechanism. Tool wear is a complex phenomenon in which the following actions take place simultaneously :

(*i*) Molecular diffusion between workpiece material and tool material. Carbon and cobalt move from tool surface to flow zone of the workpiece. Iron moves from work material to tool surface. Structural changes in the surface region of a carbide tool lead to its deterioration. By providing thin diffusion barriers the wear can be reduced.

(*ii*) Chemical affinity between the nascent layer of the workpiece material and the cutting tool material.

(*iii*) Mechanical abrasion between the micro constituents and matrix of the workpiece material and the layer of the cutting tool.

(*iv*) Pressure welding of workpiece material to cutting tool surface followed by micro transfer of small particles from the cutting tool matrix adhering to workpiece material.

All the above actions in tool wear phenomena result in two predominant types of tool wear-crater wear and flank wear.

1.48.2. Tool Wear Types. (*i*) *Flank Wear*. This occurs on the relief side and nose radius of the cutting edge. It is due to the action of abrasion and resultant heat generated between the tool and the workpiece. This increases the cutting forces, ruins the surface finish and dimensional accuracy.

(*ii*) *Crater Wear*. When turning long chipping materials like steel, the hot chips under pressure have a tendency to weld to the top face as the chip moves over it. These welded granules in turn get loosened along with some carbide particles and are carried away by passing chips. This triple action of welding, uprooting and carrying away leads to cratering. The development of a crater results in premature cutting edge failure. Higher cutting speed wear out tools faster than higher cutting feeds.

1.49. Carbide Tool Failures and Remedies

Premature tool failure problems should be checked to increase tool life. Some of the commonly observed porblems and their solutions are as follows :

Problem	**Reommended Solution**
1. Wear	(*i*) Decrease speed
	(*ii*) Increase feed
	(*iii*) Decrease lead angle
	(*iv*) Check that tool is not above centre height.
	(*v*) Tool should be of proper grade and hardness.
	(*vi*) Increases one or both relief and clearance angle.
2. Poor surface finsih	(*i*) Increase nose radius
	(*ii*) Increase lead angle
	(*iii*) Tool should be properly clamped
	(*iv*) Increase speed
	(*v*) Decrease feed
3. Chatter	(*i*) Decrease nose radius
	(*ii*) Decrease rake angle
	(*iii*) Increase relief and clearanance angle
	(*iv*) Decrease overhang.
	(*v*) Check the condition of spindles and slides of the machine.
	(*vi*) Decrease cutting speed
	(*vii*) Increase feed
	(*viii*) Check that workpiece is properly held and clamped.
4. Crater	(*i*) Use coolant of good lubricating qualities
	(*ii*) Use harder grade tool
	(*iii*) Increase speed
	(*iv*) Decrease feed
	(*v*) Positive rake angle may be used.
5. Built up edge	(*i*) Increase speed

	(*ii*) Use positive rake
	(*iii*) Use coolant with good lubricating properties
6. Chipping Cracking and Breakage	(*i*) Increase nose radius
	(*ii*) Increase lead angle if possible
	(*iii*) Use negative rake.
	(*iv*) Increase speed
	(*v*) Decrease feed
	(*vi*) Check the tool for looseness
	(*vii*) Check rigidity of workpiece
	(*viii*) Decrease tool overhang

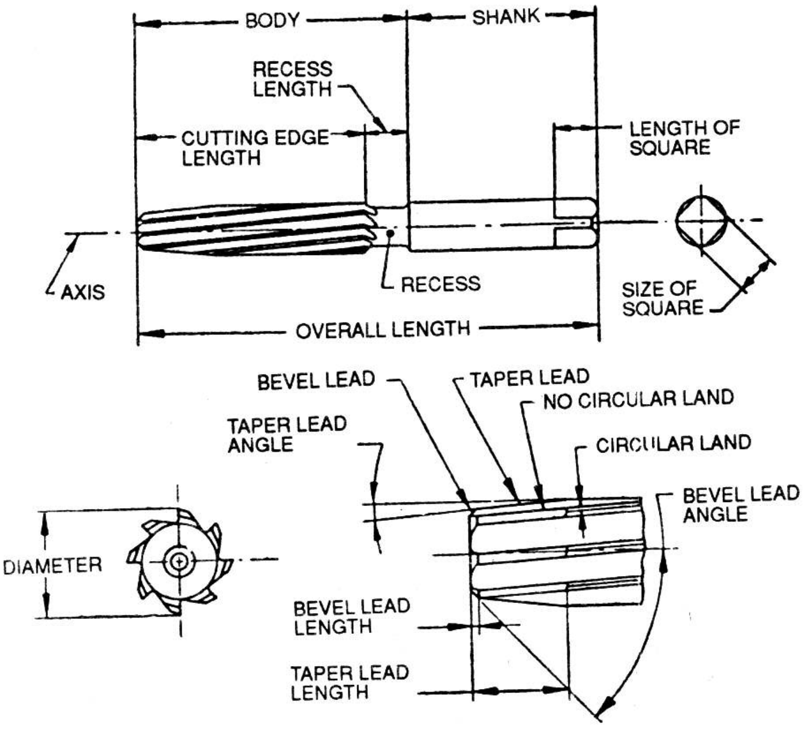


Fig. 1.64.

Only one solution or number of solutions listed above may be used to run the tool smoothly to increase life of tool, to obtain proper surface finish, production rate etc.

1.50. Cast Alloy Tools

They contain no iron and are cast into their final shape . They consists of cobalt, chromium, tungston, and carbon, the carbide phase being about 25 to 30 per cent by volume of the matrix. These tools can be used at slightly higher speeds than high speed steels and have less tendency to form a built up edge during machining. They are commonly used in machining of cast iron, malleable iron and hard bronzes.

1.51. Reamer

It is used to produce an accurate hole. Reaming is carried out after drilling.

Fig. 1.64 shows various elements of a reamer.

PROBLEMS

1.1. (*a*) What is metal cutting ? Define chip removal and non-chip removal process. Give examples.

(*b*) Discuss basic elements of metal cutting.

1.2. Explain orthogonal cutting and oblique cutting.

1.3. (*a*) How are tools classified ? State three examples of each.

(*b*) Explain with the help of neat sketch the complex geometry of a single point cutting tool.

1.4. (*a*) Define ‘tool signature’.

(*b*) Explain each term of a tool designated as 8, 12, 10, 7, 0, 15, 1.5 mm.

1.5. (*a*) What is chip formation ?

(*b*) Name the different types of chips formed in metal cutting. Describe each type with the help of neat sketches.

State the conditions which favour the production of each type.

1.6. (*a*) List the main requirements of cutting tool material.

(*b*) Name and explain the various materials used for cutting tools.

1.7. Write short notes on the following :

(*a*)Chip breakers
(*b*)Advantages of negative rake angle
(*c*)Curling of chip
(*d*)Cutting speed
(*e*)Feed

(*f*)Depth of cut

(*g*)Throw-away tips.

1.8. (*a*) What is chip reduction coefficient ?

(*b*) What are the effects of cutting variable on the chip reduction coefficient.

1.9. (*a*) What is a multipoint cutting tool ?

(*b*) Sketch a twist drill and describe its various elements.

1.10. (*a*) Make a neat sketch of a twist drill, showing various angles.

(*a*)Define the following angles of a twist drill.

(*i*)Rake angle

(*ii*)Lip clearance angle

(*iii*)Point angle

(*iv*)Chisel edged angle.

1.11. Write short notes on the following :

(*a*)Cutting fluids for drilling

(*b*)Twist drill grinding

(*c*)Twist drill failure

(*d*)Drill specifications.

1.12. Analyse the forces acting on a drill.

1.13. What are the parameters which affect the drilling torque.

1.14. What factors decide the cutting speed of a drill.

1.15. (*a*)What is a milling cutter

(*b*)Sketch and explain

(*i*) Up milling

(*ii*) Down milling.

1.16. Sketch a plain milling cutter and explain the various elements of the cutter.

1.17. Write short notes on the following :

(*a*)Milling cutter sharpening

(*b*)Number of teeth on a cutter

(*c*)Cutting speed

(*d*)Coarse tooth and fine tooth milling cutters.

1.18. Explain how to calculate machining time in

(*a*)Plain milling

(*b*)Face milling.

1.19. A stab milling operating is carried out with a 60 inch diameter inserted carbide tooth cutter having 10 teeth at a cutting speed of 400 feet/min and a feed of 40 inches per minute. If the depth of cut is 0.02 inch and width of cut is 4.5 inch and the following relation is assumed :

$$F = -32000\, b \cdot d^{0.8} f^{0.7}$$

where F = Tangential force on cutter

d = Depth of cut in inch

b = Width of cut in inch

f = Feed in inches per tooth

Determine the following :

(*a*)Tangential force exerted by the cutter.

(*b*)Torque transmitted by the cutter.

(*c*)Total H.P. required at the drive end of milling machine at a mechanical efficiency of 7%.

(*d*)Specific H.P. required at the arbor.

1.20. Determine the depth of cut taken on a lathe while turning brass at a feed of 0.625 mm per revolution and cutting speed of 24 metre/min if 3 horse power is available and 20% of power is lost in friction.

1.21. Estimate the time required for a single cut in each of the following machining operations.

(*a*) To turn 25 mm dia 100 mm long with cutting speed 30 m/min and a feed of 0.25 mm/rev.

(*b*) To turn 3 mm deep recess in a 50 mm dia and shaft using a 6 mm wide form tool with a cutting speed of 12 m/min and a feed of 0.05 mm.

1.22. (*a*) State the characteristics of carbide tools.

(*b*) Describe the power distribution in a milling machine and a lathe.

1.23. Write short notes on the following :

(*a*) Requirements of cutting tool.

(*b*) Power required at milling cutter.

1.24. Write short notes on the following :

(*a*)Improving metal cutting efficiency.

(*b*)Power distribution in the following machines.

(*i*) Lathe

(*ii*) Milling machine

1.25. Discuss the design of a cutting tool.

1.26. Explain the following terms with reference to a plain milling cutter :

(*i*)Radial rake angle

(*ii*)Lip angle

(*iii*)Relief angle

(*iv*)First clearance angle

(*v*)Second clearance angle.

1.27. Sketch a broaching tool and describe its construction.

1.28. Write short notes on the following :

(*a*)Friction between chip and tool.

(*b*)Power rating of electric motor.

(*c*)Improvement of cutting efficiency during machining.

(*d*)Design of a single point cutting tool.

(*e*)Common sizes of H.S.S. tool bits.

(*f*)Specification of a single point tool.

1.29. Determine the total production time for a batch of 400 components to be milled two at a time in a fixture on a horizontal milling machine given that

Length of workpiece = 180 mm

Approach = 25 mm
Over run = 10 mm
Feed/tooth = 0.25 mm
Number of teeth on cutter = 10
R.P.M. of cutter = 80

Time allowed for loading, unloading and cleaning is 1 1/2 min/cycle and other allowances amount to 1/2 min/cycle.

1.30. Determine the production quantity at which the manufacturing costs of the following two alternative manufacturing methods break-even :

Numerically controlled miller and Conventional miller
(*i*) Computer cost Rs. 80, -
(*ii*) Programming cost Rs. 60, -
(*iii*) Machining time $\frac{1}{2}$h and $1\frac{1}{2}$h
(*iv*) Setter operator rate Rs. 12/h and Rs. 10/h
(*v*) Set up time $\frac{1}{2}$h and 1 h
(*vii* Over heads Rs. 9/h and Rs. 7/h
(*vii*) Depreciation Rs. 15/h and Rs. 8/h

1.31 Write short notes on the following :
(*i*) Tool wear
(*ii*) Carbide tool failure and remedies
(*iii*) Cast alloy tools
(*iv*) Sketch a reamer and name its parts
(*v*) Brazing of carbide tools
(*vi*) Screwing tap.