SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AUTOMOBILE ENGINEERING

SMEA 1505 _ MANUFACTURING TECHNOLOGY II

UNIT I THEORY OF METAL CUTTING

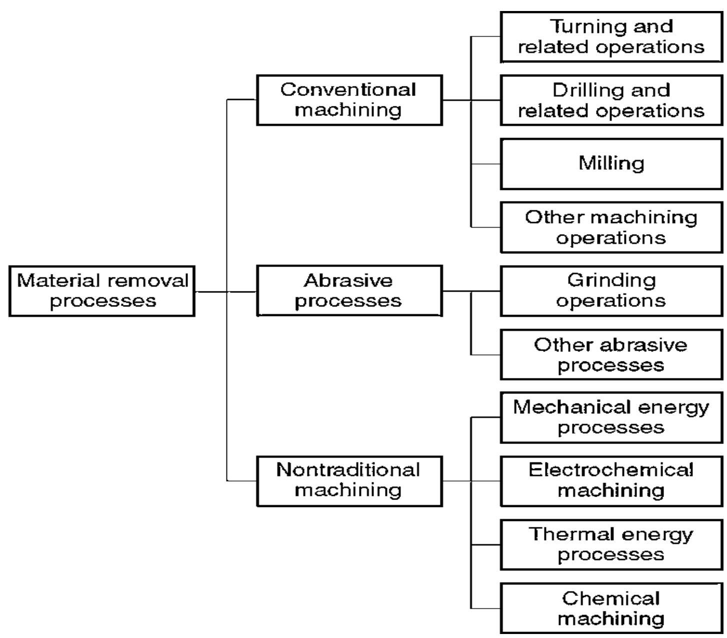
UNIT I

THEORY OF METAL CUTTING

The family of shaping operations, the common feature of which is removal of material from a starting workpart so the remaining part has the desired shape.

Categories:

- *Machining* – material removal by a sharp cutting tool, e.g., turning, milling, drilling
- *Abrasive processes* – material removal by hard, abrasive particles, e.g., grinding
- *Non-traditional processes* - various energy forms other than sharp cutting tool to remove material



A typical cutting tool in a simplified form is shown in Fig. 1 removing metal. The important features to be observed are:

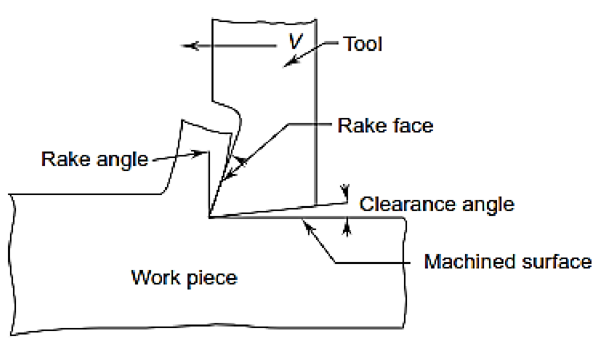


Fig.1 The general characteristics of a metal cutting tool

Rake angle

It is the angle between the face of the tool called the rake face and the normal to the machining direction. This angle specifies the ease with which a metal is cut. Higher the rake angle better is the cutting and less is the cutting force. Increasing the rake angle reduces the metal back up available at the tool rake face. This reduces the strength of the tool tip as well as the heat dissipation through the tool. Thus, there is a maximum limit to the rake angle and is generally of the order of 15° for high speed steel tools cutting mild steel. It is possible to have rake angle as zero or negative.

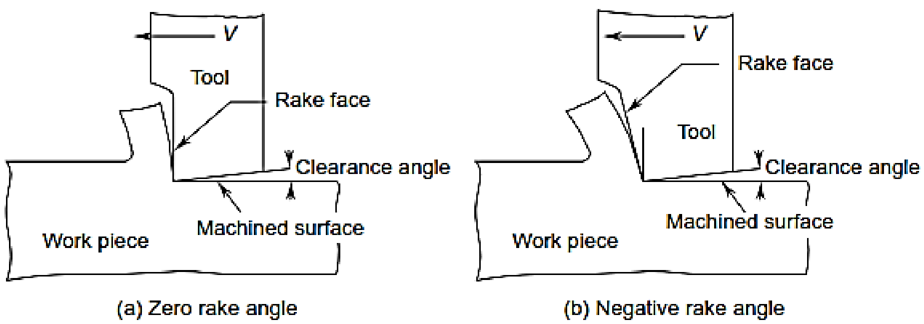


Fig. 2 Tool cutting at different rake angles

Clearance angle

This is the angle between the machined surface and the underside of the tool called the flank face. The clearance angle is provided such that the tool will not rub or spoil the machined surface, but at the same time will increase the cutting forces. A very large clearance angle reduces the strength of the tool tip; hence normally an angle of the order of 5 to 6° is generally used.

The conditions which have a predominant influence on the metal cutting are: work material, cutting tool material, cutting tool geometry, cutting speed, feed rate, depth of cut and cutting fluid used. The cutting speed, V, is the speed with which the cutting tool moves through the work material. This is generally expressed in metres per second.

Feed rate, f, may be defined as the small relative movement per cycle (per revolution or per stroke) of the cutting tool in a direction usually normal to the cutting speed direction.

Depth of cut, d, is the normal distance between the unmachined surface and the machined surface.

Chip Formation

Metal cutting process is one of the most complex processes. Fig shows the basic material removal operation schematically. The metal in front of the tool rake face gets immediately compressed first elastically and then plastically. This zone is traditionally called the shear zone, in view of the fact the material in the final form would be removed by shear from the parent metal. The actual separation of the metal starts from the cutting tool tip as yielding or fracture, depending upon the cutting conditions. Then the deformed metal (called chip) flows over the tool (rake) face. If the friction between the tool rake face and the underside of the chip (deformed material) is considerable, then chip gets further deformed, which is termed as secondary deformation. The chip after sliding over the tool rake face would be lifted away from the tool, and the resultant curvature of the chip is termed as chip curl. Plastic deformation can be caused by yielding, in which case strained layers of material would get displaced over other layers along the slip-planes which coincide with the direction of maximum shear stress.

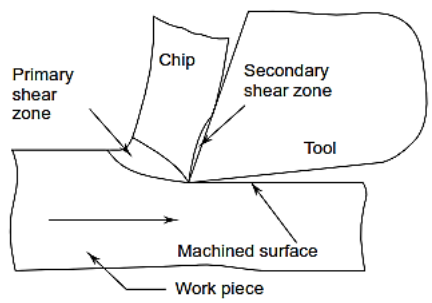


Fig.3 The possible deformations in metal cutting

The chip used in actual manufacturing practices is variable both in size and shape. Study of the chip is one of the most important things in metal cutting. As will be seen later, the mechanics of metal cutting are greatly dependent on the shape and size of the chips produced.

The chip formation in metal cutting could be broadly categorised into three types:

- Discontinuous chip
- Continuous chip, and
- Continuous chip with BUE

Discontinuous Chip

When brittle materials like cast iron are cut, the deformed material gets fractured very easily and thus the chip produced is in the form of discontinuous segments as shown in Fig. In this type the deformed material instead of flowing continuously gets ruptured periodically. Discontinuous chips are easier from the chip disposal view point. However, the cutting force becomes unstable with the variation coinciding with the fracturing cycle as shown in Fig. Also they generally provide better surface finish. However, in case of ductile materials they cause poor surface finish and low tool life. Higher depths of cut (large chip thick ness), low cutting speeds and small rake angles are likely to produce dis continuous chips.

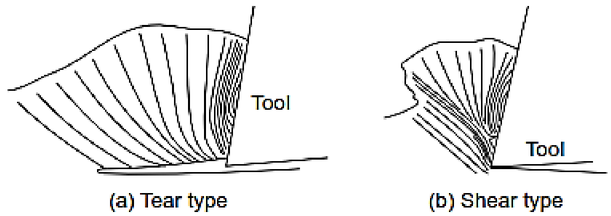


Fig. Possible discontinuous chip formations

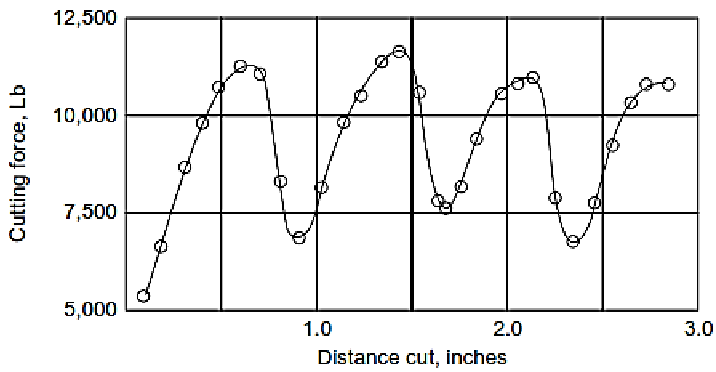


Fig.4 The variation of cutting force in discontinuous chip formation (Medium carbon steel, Rake angle = 30°, Cutting speed = 1.65 ft/min, Depth of cut = 0.41 in, Feed rate = 0.125 in/rev)

Continuous Chip

Continuous chips are normally produced when machining steel or ductile metals at high cutting speeds. The continuous chip, which is like a ribbon flows along the rake face. Continuous chip is possible because of the ductility of metal (steel at high temperature generated due to cutting) flows along the shear plane instead of rupture. Thus on a continuous chip you do not see any notches. It can be assumed that each layer of metal flows along the slip plane till it is stopped by work hardening. Each of these layers gets welded to the previous ones because of the high temperature, thus forming a continuous chip. Some ideal conditions that promote continuous chips in metal cutting are: sharp cutting edge, small chip thickness (fine feed), large rake angle, high cutting speed, ductile work materials and less friction between chip tool interfaces through efficient lubrication.

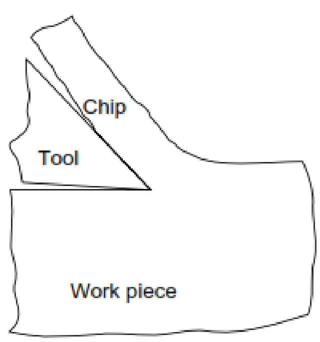


Fig.5 Continuous Chip Formation

Continuous Chip with BUE

When the friction between tool and chip is high while machining ductile materials, some particles of chip adhere to the tool rake face near the tool tip. When such sizeable material piles up on the rake face, it acts as a cutting edge in place of the actual cutting edge. This is termed as built up edge (BUE). By virtue of work hardening, BUE is harder than the parent work material. As the size of BUE grows, it becomes unstable and parts of it get removed while cutting. The removed portions of BUE partly adhere to the chip underside and partly to the machined surface. This causes the finished surface to be rough. However, since the cutting is carried by the BUE and not the actual tool tip, the life of the cutting tool increases while cutting with BUE. In this way BUE is not harmful during rough machining. The conditions that normally induce the formation of BUE are low cutting speed, high feed and low rake angle. One of the prerequisites for the formation of BUE is the work hardenability of the work piece material. Higher the work hardenability, rougher is the machined surface produced.

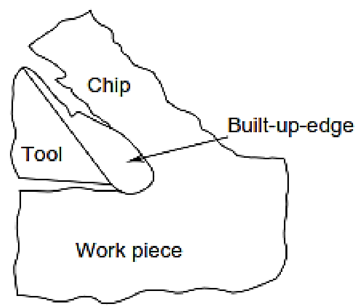


Fig.6 Continuous Chip with BUE

BUE

The formation of a BUE on the tool is brought about by the high normal loads on the tool rake face leading to adhesion between the chip and the tool. This adhesion may be so severe that instead of the chip sliding over the tool face, considerable plastic flow and eventual rupture occurs within the chip. Further layers build up, leading to a large nose of the material projecting from the cutting edge.

The adhesion at the chip tool interface is very strong and different from the conventional adhesion characteristics of the material pair concerned. The conditions of machining are more extreme compared to most other deformation processes.

(i) It is a plastic flow process with exceptionally large strains. There is high compressive stress acting on the plastic zone and this prevents rupture until the strain is well above the rupture value, in say, a tensile test.

(ii) The deformation is localised to an extremely small plastic zone. The strain rate is unusually high.

(iii) The chip material rubbing over the tool face is freshly formed from the body of the work material and is in a chemically clean condition. This makes it more chemically active than the usual oxidised surfaces encountered in most sliding situations, a feature which increases the tendency for adhesion and so gives a higher friction force.

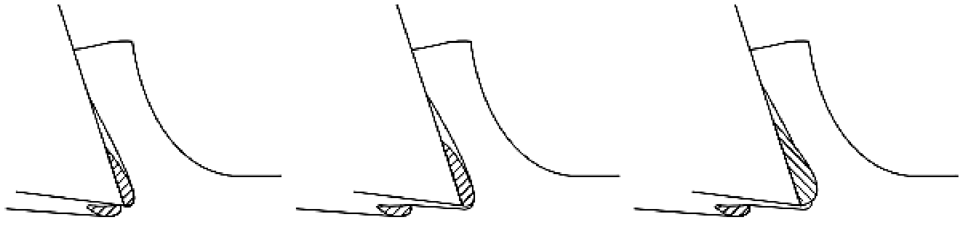


Fig.7 BUE cycle

Shear Zone

There are basically two schools of thought to analyse the metal removal process. One school of thought is that the deformation zone is very thin and planar. The other school thinks that the actual deformation zone is a thick one with a fan shape.

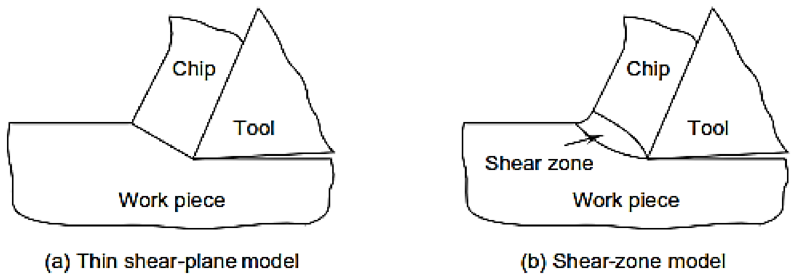


Fig.8 (a) Thin shear plane model, (b) Thick shear zone model

Orthogonal cutting

Investigators in the metal cutting field have attempted to develop an analysis of the cutting process which gives a clear understanding of the mechanisms involved and which enables the prediction of the important cutting parameters, without the need for testing. But none of the models developed so far could be fully substantiated and definitely stated to be the correct solution. It is worth examining them because they will be qualitatively explaining the phenomenon.

A general purpose metal cutting operation such as turning or milling is three-dimensional and is normally termed as oblique cutting. The obliquity comes from the angle between the cutting speed vector and the cutting edge of the tool

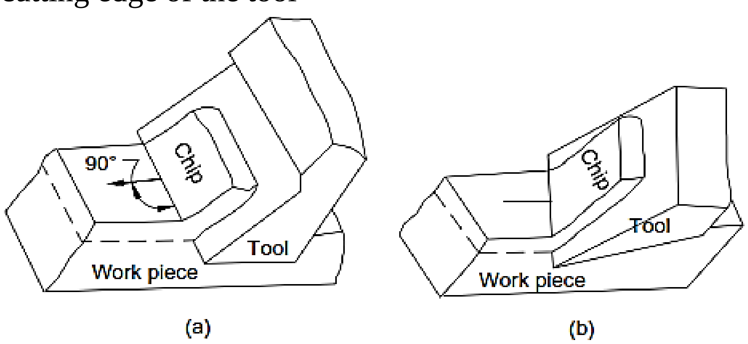


Fig.9 (a) Orthogonal cutting, (b) Oblique cutting

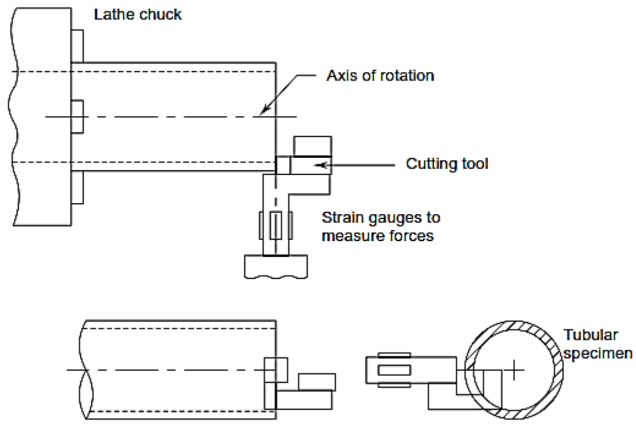


Fig.10 Realisation of orthogonal cutting in practice while turning a tube from the end

Mechanics of Orthogonal Metal Cutting

The current analysis is based on Merchant's thin shear plane model considering the minimum energy principle. This model would be applicable at very high cutting speeds, which are generally practised in production.

Assumptions:

(i) The tool is perfectly sharp and has no contact along the clearance face.
(ii) The surface where shear is occurring is a plane.
(iii) The cutting edge is a straight line extending perpendicular to the direction of motion and generates a plane surface as the work moves past it.
(iv) The chip does not flow to either side or no side spread.
(v) Uncut chip thickness is constant.
(vi) Width of the tool is greater than the width of the work.
(vii) A continuous chip is produced without any BUE.
(viii) Work moves with a uniform velocity.
(ix) The stresses on the shear plane are uniformly distributed.

F_V – Force perpendicular to the primary tool motion (thrust force)

F_s – Force along the shear plane

N_s – Force normal to the shear plane

F – Frictional force along the rake face

N – Normal force perpendicular to the rake face

$$F_S = F_H \cos\varphi - F_V \sin\varphi$$

$$N_S = F_V \cos\varphi + F_H \sin\varphi$$

$$= F_S \tan(\varphi + \beta - \alpha)$$

From Fig. 2.15 and 2.17, we can write

$$F = F_H \sin\alpha + F_{V} \cos\alpha$$

$$N = F_H \cos\alpha - F_{V} \sin\alpha$$

If μ is the coefficient of friction along the rake face, then

$$\mu = \tan\beta = \frac{F}{N} = \frac{F_{V} + F_H \tan\alpha}{F_H - F_{V} \tan\alpha}$$

where β is the friction angle, and

ϕ is the shear angle

This friction is not similar to the usual sliding case since F and N are not uniformly distributed sliding area. This aspect is discussed later.

Now, the area of shear plane, A_s is given by

$$A_s = \frac{b\,t}{\sin\phi}$$

The shear force is given by

$$F_s = \tau A_S = \frac{\tau b t}{\sin\varphi}$$

where τ is the mean shear stress in the shear plane.

b is the width of cut

and t is the uncut chip thickness

$$\sigma = \frac{N_S}{A_S} \quad \text{or} \quad N_S = \frac{\sigma b t}{\sin\varphi}$$

Where σ is the mean normal stress in the shear plane.

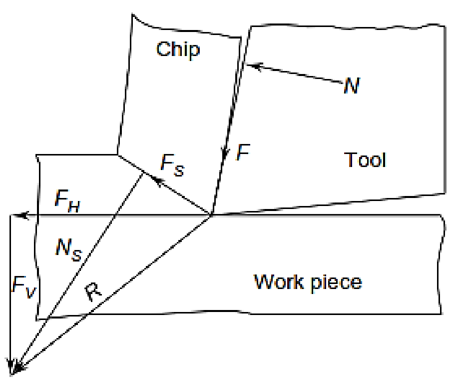
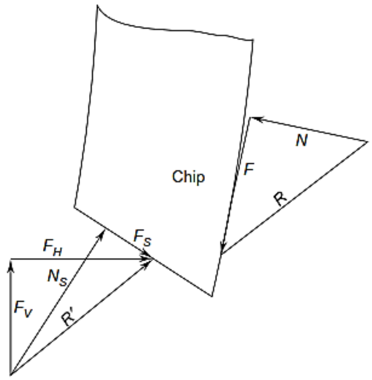


Fig.11 Various forces acting in orthogonal cutting



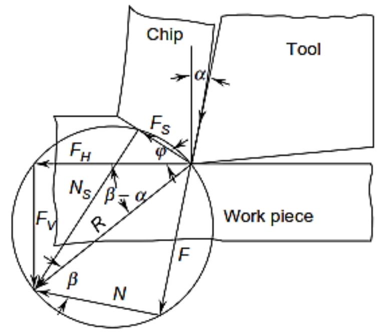


Fig. Forces acting on an isolated chip in metal cutting

Fig. Merchant's cutting force circle in orthogonal cutting

We can show that by resolving

$$F_H = F_s \cos\varphi + N_s \sin\varphi$$

$$F_V = N_s \cos\varphi - F_s \sin\varphi$$

Substituting Eq. (3) in (10), we get

$$F_H = F_s\,[\cos\varphi + \sin\varphi \tan(\varphi + \beta - \alpha)]$$

Similarly,

$$F_V = F_s\,[\cos\varphi \tan(\varphi + \beta - \alpha) - \sin\varphi]$$

Rearranging, we get

$$F_H = F_s\left[\frac{\cos(\alpha - \beta)}{\cos(\varphi + \beta - \alpha)}\right]$$

$$F_H = \frac{\tau b t \cos(\beta - \alpha)}{\sin(\varphi)\cos(\varphi + \beta - \alpha)}$$

$$F_V = \frac{\tau b t \sin(\beta - \alpha)}{\sin(\varphi)\cos(\varphi + \beta - \alpha)}$$

Merchant considered that τ would have the value of the yield shear stress for the work material and that μ would have the usual value for any dry sliding friction. To determine Φ he assumed that the minimum energy principle applied in metal cutting, so that the deformation process adjusted itself to a mini mum energy condition, or

$$\frac{dF_H}{d\varphi} = \frac{\tau b t \cos(\beta - \alpha)\cos(2\varphi + \beta - \alpha)}{\sin^2\varphi\,\cos^2(\varphi + \beta - \alpha)} = 0$$

or $$\cos(2\varphi + \beta - \alpha) = 0$$

or $$2\varphi + \beta - \alpha = \frac{\pi}{2}$$

$$\varphi = \frac{\pi}{4} - \frac{1}{2}(\beta - \alpha)$$

Substituting back, we can show that

$$F_H = 2\,\tau\,b\,t\cot\varphi$$

$$F_v = \tau\,b\,t\,(\cot^2\varphi - 1)$$

Example

A bar of 75 mm diameter is reduced to 73 mm by a cutting tool while cutting orthogonally. If the mean length of the cut chip is 73.5 mm, find the cutting ratio. If the rake angle is 15°, what is the shear angle?

Solution Length of uncut chip, $l = \dfrac{\pi(75 + 73)}{2} = 232.4779$ mm

Cutting ratio, $r = \dfrac{t_c}{t} = \dfrac{73.9}{232.4779} = 0.3179$

Shear angle, $\varphi = \tan^{-1}\left[\dfrac{r\cos\alpha}{1 - r\sin\alpha}\right] = \tan^{-1}\left[\dfrac{0.3179\cos 15}{1 - 0.3179\sin 15}\right]$

Shear angle, $\varphi = \tan^{-1}(0.3346) = 19°$

Example

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

Chip thickness ratio = 0.3
Horizontal component of the cutting force = 1290 N
Vertical component of the cutting force = 1650 N

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

Solution Given $r = 0.3$ $\alpha = 10°$

The shear plane angle, ϕ is

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} = \frac{0.3\cos 10}{1 - 0.3\sin 10} = 0.311679$$

shear angle, $\phi = \tan^{-1}(0.311679) = 17.31°$

Given $F_V = 1650$, $F_H = 1290$

The friction force along rake face is

$$F = F_H \sin\alpha + F_V \cos\alpha = 1290\sin 10 + 1650\cos 10 = 1848.94 \text{ N}$$

the normal force on the rake face is

$$N = F_H \cos\alpha - F_V \sin\alpha = 1290\cos 10 - 1650\sin 10 = 983.88 \text{ N}$$

The coefficient of friction, μ, at the chip tool interface is given by

$$\mu = \frac{F}{N} = \frac{1848.94}{983.88} = 1.8792$$

The friction angle, β is given by

$$\beta = \tan^{-1}\mu = \tan^{-1}(1.8792) = 62°$$

The resultant cutting force, R is given by

$$R = \sqrt{1650^2 + 1290^2} = 2094.42 \text{ N}$$

The shear force along the shear plane is

$$F_s = F_H \cos\varphi - F_{V} \sin\varphi = 1290 \cos 17.31 - 1650 \sin 17.31 = 740.63 \text{ N}$$

The normal force on the shear plane is

$$N_s = F_{V} \cos\varphi + F_H \sin\varphi = 1650 \cos 17.31 + 1290 \sin 17.31 = 1959.10 \text{ N}$$

The area of the shear plane is given by

$$A_s = \frac{bt}{\sin\varphi} = \frac{6 \times 0.10}{\sin 17.31} = 2.0165 \text{ mm}^2$$

To verify the shear angle from the relation suggested by Merchant:

$$\varphi = \frac{\pi}{4} - \frac{1}{2}(\beta - \alpha) = \frac{\pi}{4} - \frac{(62 - 10)}{2} = 19°$$

It can be seen that the actual value of the shear angle is 17.31, whereas the value calculated from the shear angle relation of Merchant's is 19. The error is 9.76%.

Cutting tool materials

A large variety of cutting tool materials has been developed to cater to the variety of materials used in these programmes. Before we proceed to know these materials, let us look at the important characteristics expected of a cutting tool material:

(i) Higher hardness than that of the work piece material being machined, so that it can penetrate the work material.

(ii) Hot hardness, which is the ability of the material to retain its hardness at elevated temperatures, in view of the high temperatures existing in the cutting zone. This requirement becomes more and more stringent with the increasing emphasis on higher cutting speeds to bolster productivity.

(iii) Wear resistance – The chip-tool and chip-work interfaces are exposed to such severe conditions that adhesive and abrasion wear is very common. The cutting tool material should therefore have high abrasion resistance to improve the effective life of the tool.

(iv) Toughness – Even though the tool is hard, it should have enough toughness to withstand the impact loads at the beginning of the cut or to force fluctuations due to imperfections in the work material. This requirement is more useful for interrupted cutting, e.g. milling.

(v) Low friction – The coefficient of friction between chip and tool should be low which would allow for lower wear rates and better chip flow.

(vi) Better thermal characteristics – Since a lot of heat is generated at the cutting zone, it is necessary that the tool material should have higher thermal conductivity to dissipate this heat in the shortest time, otherwise the tool temperature will become too high thus reducing its life.

Table 1 Comparative properties of cutting tool materials

Cutting Tool Material	Hardness, R_A			Transverse Rupture Strength $\times 10^3$ MPa
	Room Temperature	540°C	760°C	
High-speed steel	85 to 87	77 to 82	Very low	3.8 to 4.5
Cast cobalt	82 to 85	75 to 82	70 to 75	1.4 to 2.8
Carbides	89 to 94	80 to 87	70 to 82	to 2.4
Ceramics	94	90	87	0.5 to 0.4
Diamond	7000 Knoop	7000 Knoop	7000 Knoop	—

Carbon Tool Steels

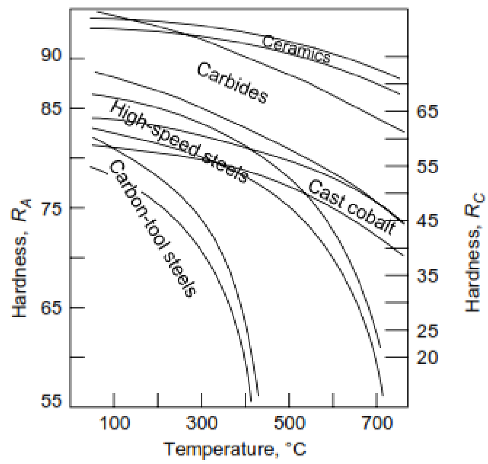
These are the earliest tool materials used. These are essentially plain carbon steels with carbon percentages between 0.6 to 1.5% and some very small alloy additions such as manganese, silicon, tungsten, molybdenum, chromium and vanadium. The major disadvantage with this range of cutting tool materials is their inability to withstand high temperatures. Beyond 200°C they lose their hardness and cease to cut. Thus these are useful only for very low cutting speeds (about 0.15 m/s) and can be used with low temperature generating operations such as machining wood, magnesium, brass and aluminium. They are easy to prepare and ground, as a result they are used for form tool making to be used for low quantity production.

High speed steel

They were able to significantly improve the cutting speeds by 3 to 5 times (about 0.5 m/s) than the speed prevalent at that time, using carbon tool steels. Because of this high cutting speed capability they were termed as high speed steels or more popularly called HSS.

This class of tool materials have significant quantities of tungsten, molybdenum, chromium and vanadium. The complex carbides of tungsten, molybdenum and chromium distributed throughout the metal matrix provide very good hot hardness and abrasion resistance. The major alloying elements, which contribute to the hardness is tungsten and molybdenum. Tungsten is expensive, while molybdenum is cheap but has higher toughness. For the same hardness, less amount of molybdenum needs to be added, however more care need to be exercised in hardening as decarburizing takes place in molybdenum steels. Also they have narrow temperature range for heat treatment. Molybdenum tool steels are more popular.

The main advantages of high speed steels are their high hardness, hot hardness, good wear resistance, high toughness and reasonable cost. Toughness of high speed steels is highest among all the cutting tool materials.



Graph 1 Variation of hardness with temperature for various cutting tool materials

Table 2 Typical compositions of high speed steel materials

AISI Steel Type	% Chemical Composition						
	C	*Cr*	*V*	*W*	*Mo*	*Co*	W_{eq}
T1	0.70	4.0	1.0	18.0			18.0
T6	0.80	4.25	1.5	2.0	0.90	12.0	21.8
M1	0.80	4.0	1.0	1.5	8.0		17.5
M6	0.80	4.0	1.50	4.0	5.0	12.0	14.0
M30	0.85	4.0	1.25	2.0	8.0	5.0	18.0
M42	1.10	3.75	1.15	1.50	9.50	8.25	20.5

Cast Cobalt Alloys

These, termed as stellites, are normally produced by the powder metallurgy method, though casting is also used by some manufacturers. Fine powders of a number of non-ferrous metals (having compositions as shown in Table 2.5) are thoroughly mixed and compacted to the final shape under hot isostatic pressure. They are then ground to their final geometry. They retain their hardness even at elevated temperatures better than HSS and consequently are used at cutting speeds higher (25% higher) than HSS. Because of their formability these are used for making form tools. They have higher toughness and higher stiffness.

Table 3 Typical compositions and uses of cast non-ferrous alloys

Nominal % Composition								Grade
Cr	W	Mo	C	Mn	Si	Ni	Co	
30	4.5	1.5	1.1	1.0	1.5	3.0	rest	Roughing
31	10.5	—	1.7	1.0	1.0	3.0	rest	General purpose
32	17.0	—	2.5	1.0	1.0	2.5	rest	Finishing

Cemented carbides

Cemented carbides are produced by the cold compaction of the tungsten carbide powder in a binder such as cobalt, followed by liquid-phase sintering. These have a very large number of advantages compared to the other cutting tool materials.

(i) High hot hardness. These can retain their hardness to much higher temperatures and as a result the cutting speeds used are 3 to 6 times (about 5 to 6 m/s) than that of HSS.

(ii) Higher Young's modulus. This results in stiffer cutting tools with less tendency towards chatter.

However, carbides are more brittle and expensive. It is possible to vary the composition of carbides to get a range of properties. The variations achieved are based on the amount of Co binder, different types of carbides and the grain size of carbide. Increasing the cobalt binder decreases the hot hardness and wear resistance while increasing the strength. The usual composition of the straight grade carbides is 6 wt. % Co and 94 wt. % WC with the cobalt composition ranging from 5 to 12 wt. %. For heavy interrupted and roughing operations high cobalt (Co) content is required while medium coarse grain tungsten carbide is used to withstand the shock. For finishing applications lower cobalt content is required as hardness becomes the important requirement. Addition of titanium carbide (TiC) increases the hot hardness, wear resistance, and resistance to thermal deformation, but decreases the strength. The usual composition is about 5–25 wt. %. Similarly the presence of tantalum carbide (TaC) increases the hot hardness and resistance to thermal deformation while decreasing the wear resistance and strength.

Table 4 ISO Classification of cemented carbide tools

Main Groups of Chip Removal			Group of Application			Direction of Increase	
Symbol	**Broad Categories of Materials to be Machined**	**Colour**	**Designation**	**Material to be Machined**	**Use and Working Condition**	**of Cutting Speed**	**of Feed Rate**
P	Ferrous metals with long chips	BLUE	P01	Steel, steel castings	Finish turning and boring, high cutting speeds, small chip section, accuracy of dimensions and finish, vibration free operation	↑	↓
			P10	Steel, steel castings	Turning, copying, threading and milling, high cutting speeds, small or medium chip sections		
			P20	Steel, steel castings, malleable cast iron with long chips	Turning, copying, milling, medium cutting speeds and chip sections, planing with small chip sections		
			P30	Steel, steel castings, malleable cast iron with long chips	Turning, milling, planing, medium or low cutting speeds, medium or large chip sections, and machining in unfavourable conditions		
			P40	Steel, steel castings with sand inclusion and cavities	Turning, planing, slotting, low cutting speeds, large chip sections, with possibilities of large cutting angles for machining in unfavourable conditions and work on automatic machines		
			P50	Steel, steel castings of medium or low strength with sand inclusion and cavities	For operations demanding very tough carbides, turning, planing, slotting, low cutting speeds, large chip sections, with possibilities of large cutting angles for machining in unfavourable conditions and work on automatic machines		

Main Groups of Chip Removal			Group of Application			Direction of Increase	
Symbol	**Broad Categories of Materials to be Machined**	**Colour**	**Designation**	**Material to be Machined**	**Use and Working Condition**	**of Cutting Speed**	**of Feed Rate**
M	Ferrous metals with long or short chips and nonferrous metals	YELLOW	M10	Steel, steel castings, manganese steel, grey cast iron, alloy cast iron	Turning medium or high cutting speeds, small or medium chip sections	↑	↓
			M20	Steel, steel castings, austenitic or manganese steel, grey cast iron	Turning, milling, medium or cutting speeds and chip sections		
			M30	Steel, steel castings, austenitic steel, grey cast iron, high-temperature resistant steels	Turning, milling, planing, medium or cutting speeds and medium or large chip sections		
			M40	Mild free cutting steel, low tensile steel, non-ferrous metals and light alloys	Turning, parting off particularly on automatic machines		
K	Ferrous metals with short chips, non-ferrous metals and non-metallic materials	RED	K01	Very hard grey cast iron, chilled castings of over 85 shore, high silicon aluminium alloys, hardened steel, highly abrasive plastics, hard cardboard, ceramics	Turning, finish turning, boring, milling, scraping	↑	↓
			K10	Grey cast iron over 220 BHN, malleable cast iron with short chips, silicon aluminium alloys, hardened steel, copper alloys, plastics, glass, hard rubber, hard cardboard, porcelain, stone	Turning, drilling, boring, milling, broaching, scraping		
			K20	Grey cast iron up to 220 BHN, non-ferrous metals, copper, brass, aluminium	Turning, planing, boring, milling, broaching, demanding very tough carbides		
			K30	Low hardness grey cast iron, low tensile steel, compressed wood	Turning, planing, milling, slotting, for machining in unfavourable conditions and with the possibility of large cutting angles		
			K40	Soft wood or hard wood, nonferrous metals	Turning, planing, milling, slotting, for machining in unfavourable conditions and with the possibility of large cutting angles		

Coated carbides

The range of work materials is large; there is a need for hard and refractive coatings on conventional tool materials, so that the same could be used in diverse situations. Thus several coatings and coating methods have been developed for cutting tools. Since late 60's thin (about 5 mm) coating of TiN has been used on cemented carbide tools. Ceramic coatings used are hard materials and therefore provide a good abrasion resistance. They also have excellent high temperature properties such as high resistance to diffusion wear, superior oxidation wear resistance, and high hot hardness. Further the good lubricating properties of the coatings minimise friction at the tool–chip and tool–work piece interfaces, thereby lowering the cutting temperature. All these translate into lower forces generated during machining compared to uncoated tools.

The substrate is a normal cemented carbide tool that has the necessary strength and toughness. The coating on the top, as shown in Fig, provides the required hardness and refractoriness that prolongs the life of the tool. The life of the coated tools is often two to three times of the uncoated, while these can be used at higher cutting speeds, thus increasing productivity.

The coatings need to be metallurgical bonded to the substrate. These coatings such as titanium carbide, titanium nitride, aluminium oxide, hafnium nitride and hafnium carbide or multiple coatings of the above are deposited on the carbide tool bits by the chemical vapour deposition (CVD) process. The chemical reaction necessary to deposit the required coating takes place close to the substrate. The coating is deposited literally atom by atom onto the surface thereby providing a very strong adhesion between the coating and the substrate.

Typical coating materials used include TiC, TiN, Al2O3, TiCN, TiAlN, TiZrN, TiB and diamond. Typical physical properties of the coating materials. The TiCN coating has the highest room temperature hardness, but above 750°C the TiAlN coating.

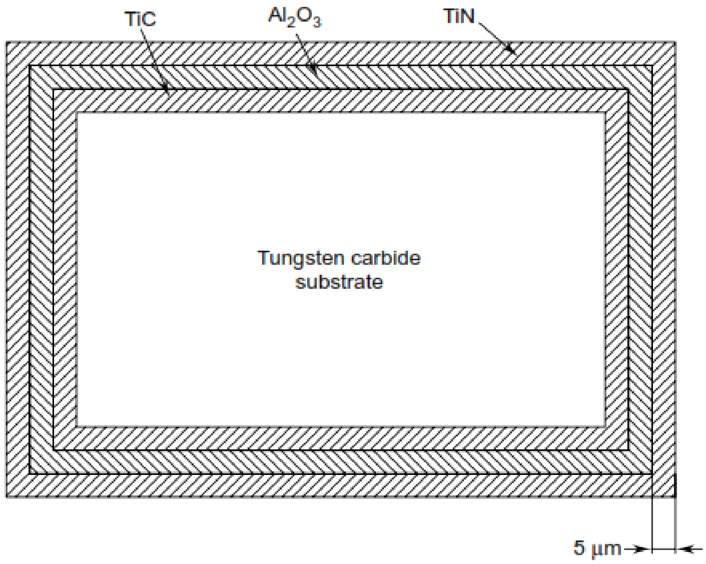


Fig.12 Schematic representation of a multicoated carbide tool bit

Table 5 Properties of some coating materials

Coating	Room Temperature Hardness (HV)	Oxidation Resistance, °C	Coefficient of Friction
TiN	1930–2200	600	0.4–0.5
TiCN	2730–3000	400	0.3
TiAlN	3000–3500	800	0.7
TiN/AlN	4000	950	—
TiAlCN	3200	600	—

Ceramics

Ceramics are essentially alumina (Al2O3) based high refractory materials introduced specifically for high speed machining of difficult to machine materials and cast iron. These can withstand very high temperatures, are chemically more stable and have higher wear resistance than the other cutting tool materials. In view of their ability to withstand high temperatures, they can be used for machining at very high speeds of the order of 10 m/s. It is possible to get mirror finish on cast iron using ceramic turning. The main problems of ceramic tools are their low strength, poor thermal characteristics and the tendency to chipping. About 2 to 5 weight% of zirconium oxide (ZrO2) is added to alumina that increases the fracture toughness of the tool without affecting its wear resistance. The machine tools used for ceramic machining have to be extremely rigid to provide smooth machining conditions for machining with ceramics and should be able to provide high cutting speeds. They are not suitable for intermittent cutting or for low cutting speeds.

Table 6 Properties of ceramic materials

Base System	Density g/cm^3	Hardness		Transverse Rupture Strength, MPa
		25°C (HRA)	1000°C (HV)	
Al_2O_3	3.98	93.9	710	50
Al_2O_3 + TiC	4.24	94.3	770	80
Si_3N_4	3.27	92.6	1100	100

Ceramic tools should be used with very high cutting speeds on steels. They are not suitable for low cutting speeds or for intermittent cutting. Cutting fluid if applied should be in flooding with copious quantity of fluid to thoroughly wet the entire machining zone, since ceramics have very poor thermal shock resistance.

Ceramic tools cannot machine some materials such as aluminium, titanium, since they have strong affinity towards them, as a result of which chemical reactions are likely to take place. Among other things, some of the vital requirements when machining with ceramics are:

- Use the highest cutting speed recommended and preferably Select Square or round inserts with large nose radius.
- Use rigid machine with high spindle speeds and safe clamping angle.
- Machine rigid work pieces.
- Ensure adequate and uninterrupted power supply.
- Use negative rake angles so that less force is applied directly to the ceramic tip.
- The overhang of the tool holder should be kept to a minimum; not more than 1.5 times the shank thickness.
- Large nose radius and side cutting edge angle on the ceramic insert to reduce the tendency of chipping.
- Always take a deeper cut with a light feed rather than a light cut with heavy feed; ceramic tips are capable of cuts as deep as one-half the width of the cutting surface on the insert.
- Avoid coolants with aluminium oxide based ceramics.
- Review machining sequence while converting to ceramics and if possible introduce chamfer or reduce feed rate at entry.

Diamond

Diamond is the hardest known (Knoop hardness ~ 8000 kg/mm2) material that can be used as a cutting tool material. It has most of the desirable properties of a cutting tool material such as high hardness, good thermal conductivity, low friction, non-adherence to most materials, and good wear resistance. However, the factors that weigh against its use are the high cost, possibility of oxidation in air, allotropic transformation to graphite above temperatures of 700°C, very high brittleness and difficulties associated in shaping it to suitable cutting tool form. Natural diamond tools could be used for relatively light cuts where these provide extremely high tool life, which can easily justify the high cost of diamond. However, natural diamond is unreliable in performance because of the impurities present and easy cleavage. Artificial diamonds are basically polycrystalline (PCD) in nature. These are extensively used in industrial application because they can be formed for any given shape with a substrate of cemented carbide.

Cubic Boron Nitride (CBN)

Cubic Boron Nitride (CBN) is next in hardness only to diamond (Knoop hardness ~ 4700 kg/mm2). It is not a natural material but produced in the laboratory using a high temperature/ high pressure process similar to the making of artificial diamond. CBN is less reactive with materials like hardened steels, hard chill cast iron, and nickel base and cobalt based super alloys, and hence is used effectively for machining these alloys. These are more expensive than cemented carbides but in view of the higher accuracy and productivity for difficult to machine materials, they are used in special applications.

Table 7 Cutting data for Polycrystalline Diamond (PCD) tool bits

Material	Cutting Speed, m/min	Depth of Cut, mm	Feed Rate, mm/rev
Aluminium alloys > 11% Si	300–3000	Up to 4	0.10–0.50
MMC SiC-particles (15–30%)	200–800	Up to 3	0.10–0.50
Copper, Brass and Bronze	600–1200	Up to 3	0.10–0.50
Carbon and graphite	100–400	Up to 3	0.10–0.50
Sintered carbide	10–40	Up to 3	0.10–0.50
Green carbide	80–200	Up to 0.5	0.10–0.50
Green ceramic	100–600	Up to 2	0.05–0.20
Plastic composites	100–1000	Up to 3	0.10–0.50

Table 8 Summary of applications for various cutting tool materials

Tool Material	Work Materials	Remarks
Carbon steels	Low strength, softer materials, nonferrous alloys, plastics	Low cutting speeds, low strength materials
Low/medium alloy steels	Low strength, softer materials, nonferrous alloys, plastics	Low cutting speeds, low strength materials
HSS	All materials of low and medium strength and hardness	Low to medium cutting speeds, low to medium strength materials
Cemented carbides	All materials up to medium strength and hardness	Not suitable for low speed application
Coated carbides	Cast iron, alloy steels, stainless steels, super alloys	Not for Titanium alloys, not for non-ferrous alloys as the coated grades do not offer additional benefits over uncoated.
Ceramics	Cast iron, Ni-base super alloys, nonferrous alloys, plastics	Not for low speed operation or interrupted cutting. Not for machining Al, Ti alloys.
CBN	Hardened alloy steels, HSS, Ni-base super alloys, hardened chill cast iron, commercially pure nickel	High strength, hard materials
Diamond	Pure copper, pure aluminium, Al-Si alloys, cold pressed cemented carbides, rock, cement, plastics, glass-epoxy composites, non-ferrous alloys, hardened high carbon alloy steels (for burnishing only), fibrous composites	Not for machining low carbon steels, Co, Ni, Ti, Zr.

Tool Wear and Tool Life

With the usage of tools over a long time, they are subjected to wear. The type of wear found in cutting tools

is shown in Fig. There are two major types of wear found in tools. They are:

Crater wear:

The crater is on the rake face and is more or less circular. The crater does not always ex tend to the tool tip, but may end at a distance from the tool tip. It increases the cutting forces, modifies the tool geometry, and softens the tool tip.

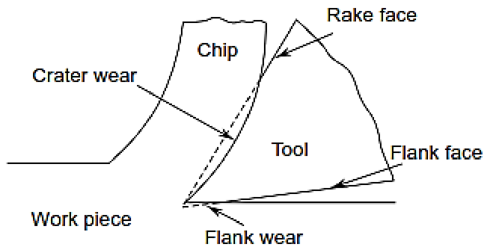


Fig.13 Typical wear patterns present in cutting tools

Flank wear:

Flank wear or wear land is on the clearance surface of the tool. The wear land can be characterised by the length of wear land, w. It modifies the tool geometry and changes the cutting parameters (depth of cut).

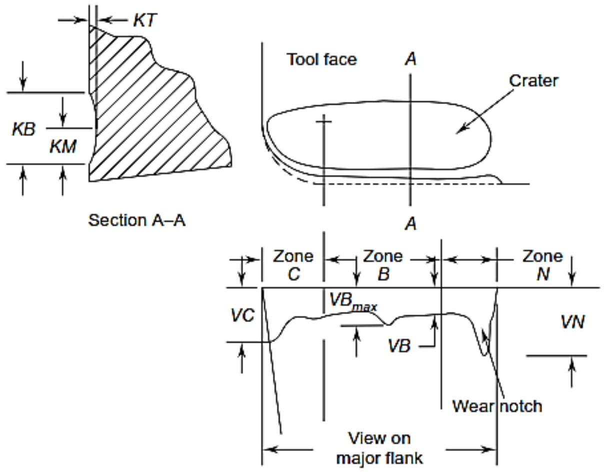


Fig.14 The wear parameters and their characterisation

Cutting tools are subjected to extremely severe cutting conditions such as the following:

- metal to metal contact with work and chip
- very high stress
- very high temperature
- virgin metal
- very high temperature gradients
- very high stress gradients

A number of wear mechanisms as follows have been proposed to explain the observed tool wear phenomenon.

- Adhesion
- Abrasion
- Diffusion
- Fatigue

Tool Life

Tool life represents the useful life of the tool, generally expressed in time units from the start of a cut to an end point defined by a failure criterion.

A tool that no longer performs the desired function is said to have failed and hence reached the end of its useful life. At such an end point the tool is not necessarily unable to cut the work piece but is merely unsatisfactory for the purpose. The tool may be re-sharpened and used again.

The tool life values as suggested by ISO are:

VB = 0.3 mm if the flank is regularly worn in zone B, or

VBmax = 0.6 mm if the flank is irregularly worn, scratched, chipped or badly grooved in zone B.

The tool life can be specified by any of the following measurable quantities:

- Actual cutting time to failure
- Length of work cut to failure
- Volume of metal removed to failure
- Number of components produced
- Cutting speed for a given time to failure.

All these various parameters are related and are used depending upon the final function of a given operation. However, the problem normally faced is in the definition of the failure criterion.

The following are some of the possible tool failure criteria that could be used for limiting tool life.

Based on tool wear:

(i) Chipping or fine cracks developing at the cutting edge.

(ii) Wear land size

(iii) Crater depth, width or other parameters

(iv) A combination of the above two

(v) Volume or weight of material worn off the tool

(vi) Total destruction of the tool

Based on consequences of worn tool

(i) Limiting value of surface finish

(ii) Limiting value of change in component size

(iii) Fixed increase in cutting force or power required to perform a cut

Table 9 The various possible tool life criteria

Wear Land, mm	Tool Material	Remarks
0.75	Carbides	Roughing
0.25 to 0.38	Carbides	Finishing
1.50	HSS	Roughing
0.25 to 0.38	HSS	Finishing
0.25 to 0.38	Oxide	Roughing and finishing

Tool Life Equation

Taylor thought that there is an optimum cutting speed for maximum productivity. He reasoned this from the fact that at low cutting speeds, the tools have higher life but productivity is low, and at higher speeds the reverse is true. This inspired him to check the relationship between tool life and cutting speed. Based on his experimental work he proposed the formula for tool life

$$V\,T^{n} = C$$

where T is the tool life in minutes

V is the cutting speed, m/min

C and n are constants

Though this is a fairly good formula, but it does not take all the effecting parameters into account. As a result the applicability of the above formula is restricted to very small regions of cutting process parameters. This formula was extended by a number of researchers to reduce this deficiency, as given below.

$$T\,\theta^{\beta} = C$$

$$VT^{\frac{0.5-2x}{1-2x}} = \left[\frac{T_C\,H^{0.5}}{C'u_s\,A^{x}}\right]^{\frac{1}{1-2x}}$$

where H = specific heat × thermal conductivity

θ = tool temperature

A = area of cut

u_s = specific cutting energy/unit cutting force

C and x are constants

$$\theta = \frac{c_o\,u_s\,V^{0.44}\,A^{0.22}}{k^{0.44}\,\tau^{0.56}}$$

where k = thermal conductivity of work

τ = specific heat of work

$$V\,T^{n}\,f^{n1}\,d^{n2} = C$$

Machinability

Machinability is the characteristic of the work material expressing its ease of machining. However convenient it looks, it is a characteristic which is difficult to quantify. Unfortunately like other characteristics of the material it is not a simple property. For example, hard work materials are difficult to machine. However, hardness alone would not be able to specify the machinability, since it also depends on the other characteristics such as tool materials used, process parameters, etc.

Variables Affecting Tool Life

The variables that affect tool life are:

(i) cutting conditions,

(ii) tool geometry,

(iii) tool material,

(iv) work material, and

(v) cutting fluid

The effect of cutting speed, feed and depth of cut is represented in the above tool life equation. If any of them increases, the tool life decreases as exemplified by the exponents in the above tool life equation.

Increasing the rake angle de creases the cutting forces and the heat produced at the tool tip, therefore increases tool life. However, increasing the rake angle to a large value reduces the tool material available at the tool tip for conducting heat generated, thus increasing the tool tip temperature. This would decrease the tool life, thus explaining the existence of an optimum value for the rake angle.

Surface Finish

Machining operations are performed in order to achieve better surface finish as compared to other manufacturing operations. Thus it is important to know the effective surface finish that can be achieved in a machining operation. The surface finish in a given machining operation is a result of two factors:

(i) the ideal surface finish, which is a result of the geometry of the manufacturing process, can be determined by considering the geometry of the machining operation.

(ii) the natural component, which is a result of a number of the uncontrollable factors in machining, is difficult to predict.

Cutting Fluids

The functions of cutting fluids, which often are erroneously called coolants are:

(i) to cool the tool and work piece

(ii) to reduce the friction

(iii) to protect the work against rusting

(iv) to improve the surface finish

(v) to prevent the formation of built up edge

(vi) to wash away the chips from the cutting zone

However, the prime function of a cutting fluid in a metal cutting operation is to control the total heat. This can be done by dissipating and reducing the heat generated. The mechanisms by which a cutting fluid performs these functions may be listed as follows:

- cooling action
- lubricating action

Cooling Action

Originally it was assumed that the cutting fluid improves the cutting performance by its cooling properties alone. That is why the name coolant was given to it. Since most of the tool wear mechanisms are thermally activated, cooling the chip tool interface helps in retaining the original properties of the tool and hence prolongs its life. However, a reduction in temperature of the work piece may under certain conditions increase the shear flow stress of the work piece thereby decreasing tool life. It has been shown through a number of investigations that cooling, in fact, is one of the major factors in improving the cutting performance.

Lubricating Action

The best improvement in cutting performance can be achieved by the lubricating action since this reduces the heat generated, thus reducing the energy input to the metal cutting operation. However, if the cutting fluid is to be effective, it must reach the chip tool interface.

Types of Cutting Fluids

There are three basic types of cutting fluids used in metal cutting. They are:

Water based emulsions: Pure water is by far the best cutting fluid available because of its highest heat carrying (high specific heat) capacity. Besides this it is cheap and easily available. Its low viscosity makes it flow at high rates through the cutting fluid system and also penetrates the cutting zone. However, water corrodes the work material very quickly, particularly at high temperatures prevalent in the cutting zone as well as the machine tool parts on which it is likely to spill.

Hence other materials would be added to water to improve its wetting characteristics, rust inhibitors and any other additives to improve lubrication characteristics. These are also called as water soluble oils. The concentrated oil is normally diluted in water to any desired concentration, such as 30:1 to 80:1.

Straight Mineral oils:

These are the pure mineral oils without any additives. Their main function is lubrication and rust prevention. These are chemically stable and lower in cost. However, their effectiveness as cutting fluids is limited and therefore would be used for light duty application only.

Mineral oils with additives (Neat oils):

This is by far the largest variety of cutting fluids available commercially. A number of additives have been developed which when added to the mineral oils would produce the desirable characteristics for the different machining situations. Many difficult to machine situations would be helped by the use of these cutting fluids. These are generally termed as neat oils. The additives generally improve the load carrying capacity as well as chemical activity. Fatty oils are generally used for adding the load carrying properties. Other class of additives termed as EP (Extreme Pressure) additives are used for more difficult to machine situations. These EP agents come into effect whenever minute high-spots on the mating surfaces break through the oil film and rub together to setup localised high temperature spots. This high temperature causes the EP additives to react with the adjacent metals and create an anti-welding layer of solid lubricant precisely where it is required. The layer is continuously broken by the severe rubbing action between the chip and the tool.

EP additives are basically chlorine, sulphur or a combination of both of them. As a result, the anti-welding compounds formed in the cutting zone are iron chloride and iron sulphide, both of which have very low shear strengths.

Cutting Fluid Selection

The selection of cutting fluid for a given application requires the examination of a number of parameters such as

(i) work piece material
(ii) machining operation
(iii) cutting tool material, and
(iv) other ancillary factors

Table 10 Selection of cutting fluid based on tool material

Tool Material	Characteristics	Cutting Fluid Requirements
High carbon steel	Not widely used. They should be well cooled.	Water based coolants are generally used
High speed steels	Have better hot hardness characteristics	For general machining water based cutting fluids can be used. For heavy duty work EP neat oils are preferable.
Nonferrous materials	During a cut they should never be overcooled or subjected to intermittent cooling because they are brittle and are likely to suffer thermal shock.	Neat cutting oils are the most suitable choice for most applications.
Carbides, Ceramics and diamond	These are used for very high speeds. Hence the requirement is to reduce the large amount of heat produced to reduce the thermal distortion of work piece.	Water based coolant is recommended. In low speed applications EP based oils could reduce the problem of adhesion of chip with tool.

Table 11 Selection of cutting fluid based on work material

Material	Characteristics	Cutting Fluid to be Used
Grey Cast iron	Grey cast iron could be machined dry since the graphite flakes act as solid lubricants in cutting.	Soluble oils and thinner neat oils are satisfactory for flushing swarf and metal dust.
Copper alloys	Better machinability. Some could be machined dry.	Water based fluids could be conveniently used. Tougher alloys a neat oil blended with fatty or inactive EP additive is used.
Aluminium alloys	They are generally ductile and can be machined dry. However in combination with steel they have a high friction coefficient. generally BUE forms on the tool and prevents the chip flowing smoothly away from the work.	It is necessary to have the tool surface highly polished. Generally a soluble oil to which an oiliness agent has been added. For more difficult application straight neat oil or fatty oil or kerosene could be used.
Mild steel and low to medium carbon steels	Easiest to machine. Low carbon steels have lower tensile strength and hence may create problems because of their easy tearing.	Milky type soluble oil or mild EP neat cutting oil could be used.
High carbon steels	They are tough and pressures and temperatures in the cutting zone are higher.	EP cutting oil is used. In some applications milky soluble can be used.
Alloy steels	Particularly with chromium and nickel, the steels are tough. They are similar to high carbon steels.	EP cutting oil is used. In some applications milky soluble can be used.
Stainless steels and heat resistant alloys	They have great toughness, corrosion resistance and pronounced work hardening. Very difficult to machine. Because of their toughness they create very high pressure in the work area.	Use high performance neat oils with high concentration of chlorinated additives. Sulphur additives are to be avoided. Some high performance EP soluble oils may sometimes be useful.

Economics

The ultimate objective of the manufacturing engineer is to produce the objects at the most economical cost. To do this he should be able to analyse the machining process for all the possible costs, so that he is able to optimise the process to get the minimum possible costs, after satisfying all the requirements.

The various costs associated with machining process are:

(i) the manpower cost, C_l which is measured in per unit time, generally hours that operator is employed

(ii) the machine tool operating (overhead) cost, C_m which includes machine depreciation, and other costs associated with the running of the machine tool such as power consumed, maintenance overheads, consumables such as oils, etc. This may also include the other overhead costs, which takes care of all the fixed overheads such as buildings, land and administrative overheads. Combining the above two costs we can have an overall overhead cost as, C_o

$$C_o = C_m + C_l$$

(iii) the job handling cost, which arises because of the time spent in loading and unloading of the job, during which time the machine tool is kept idle, and also requires the operator to attend to the job. It is also possible that some special equipment such as crane, etc. may be used for heavy jobs.

(iv) the tool cost, C_t which is the cost of the cutting tool for the given operation.

The three optimisation criterion that are generally considered are:

(i) Minimisation of the machining cost,

(ii) Maximizing the production rate, and

(iii) Maximizing the profit rate.

Time for machining is given by

$$T_m = \frac{L}{f\,N}$$

where N is the spindle RPM which is related to cutting speed by the following relation

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

Substituting this into the previous equation, we get

$$T_m = \frac{\pi\,D\,L}{1000\,f\,V}$$

For a given job, the cost of all overheads is given by

$$C_l = C_o\,[\,p\,(t_l + t_{ul} + t_a) + t_o\,]$$

where t_a = tool advance and withdrawal time

t_l = job loading time

t_{ul} = job unloading time

t_o = initial setup time of the machine for a batch of components

and p = number of parts produced per batch

Example A 600 mm long job of diameter 150 mm of AISI 4140 steel is to be turned with a depth of cut of 1.5 mm and feed rate 0.25 mm/rev. The following data is applicable for the problem:

Labour cost per hour = ₹12.00
Machine overhead per hour = ₹40.00
Grinding cost per hour = ₹15.00
Grinding machine overhead per hour = ₹50.00
Idle time = 5 minutes
The Taylor's tool life equation is given by

$$V\,T^{0.22} = 475$$

The operation can be carried out using tungsten carbide tools either as brazed tools or throw away tools.

For brazed tools:

Initial cost = ₹60
Grinding time = 5 minutes/edge
Tool change time = 2 minutes
9 grinds per tool before salvage

For throw away tips:

Initial cost = ₹40
Tool change time = 1.5 minutes
Total cutting edges = 8

Find the optimum cutting speed, tool life and the cost of the operation for both the brazed tip and the away type using the following criteria:

(a) Minimum production cost, and
(b) Maximum production rate.

Comment on the results obtained.

Solution Data given:

$n = 0.22$; $C = 475$: $r = 9$;
Idle time = $t_1 + t_{ul} + t_a = 5$ minutes
Overhead cost $C_o = 12 + 40 = 52$ ₹/hour = 0.8667 ₹/minute

Brazed Tools:

$$\text{Tool-grinding cost} = \frac{5(15+50)}{60} = 5.417\text{ ₹}$$

$$\text{Tool cost, } C_e = \frac{60 + 5.417 \times 9}{10} = 10.875\text{ ₹}$$

Tool change time, $t_c = 2$ minutes

Minimum Cost criterion:

$$\text{Optimum cutting speed, } V = C\left[\frac{C_o}{C_e + t_c C_o}\left(\frac{n}{1-n}\right)\right]^n$$

$$= 475\left[\frac{0.8667}{10.875 + 20.8667}\left(\frac{0.22}{1-0.22}\right)\right]^{0.22}$$

$$= 199.5 \text{ m/min}$$

Optimum tool life, $T = \left[t_c + \frac{C_e}{C_o}\right]\left[\frac{1-n}{n}\right]$

$$= \left[2 + \frac{10.875}{0.8667}\right]\left[\frac{1-0.22}{0.22}\right]$$

$$= 51.58 \text{ minutes}$$

Machining time, $T_m = \dfrac{\pi D l}{1000 f V}$

$$= \frac{\pi \times 150 \times 600}{1000 \times 0.25 \times 199.5}$$

$$= 5.669 \text{ minutes}$$

Total time $= t_l + t_{ul} + t_a + \dfrac{t_o}{p} + T_m\left[1 + \dfrac{t_c}{T}\right]$

$$= 5 + 5.669\left[1 + \frac{2}{51.58}\right] = 10.89 \text{ minutes}$$

Cost of operation $= C_o\left(t_l + t_{ul} + t_a + \dfrac{t_o}{p}\right) + T_m\,[C_e + t_c C_o]$

$$= 0.8667 \times 5 + 5.669\,[(10.875 + 2 \times 0.8667)]$$

$$= 10.63 \text{ ₹}$$

Maximum Production Rate criterion:

Optimum cutting speed, $V = C\left[\dfrac{n}{t_c\,(1-n)}\right]^n$

$$= 475\left[\frac{0.22}{2 \times (1-0.22)}\right]^{0.22} = 308.7 \text{ m/min}$$

Optimum tool life, $T = \dfrac{t_c\,(1-n)}{n}$

$$= \frac{2\,(1-0.22)}{0.22} = 7.091 \text{ minutes}$$

Machining time, $T_m = \dfrac{\pi \times 150 \times 600}{1000 \times 0.25 \times 308.7} = 3.664$ minutes

Total time $= 5 + 3.664\left[1 + \dfrac{2}{7.091}\right] = 9.697$ minutes

Cost of operation $= C_o\left(t_l + t_{ul} + t_a + \dfrac{t_o}{p}\right) + T_m\,[C_e + t_c C_o]$

$$= 0.8667 \times 5 + 3.664\ [(10.875 + 2 \times 0.8667)]$$
$$= 14.02\ ₹$$

Throw away Tools:

Tool cost, $C_e = \dfrac{40}{8} = 5\ ₹$

Tool change time, t_c = 1.5 minutes

Minimum Cost criterion:

$$\text{Optimum cutting speed, } V = 475\left[\frac{0.8667}{5 + 1.5 \times 0.8667}\left(\frac{0.22}{1 - 0.22}\right)\right]^{0.22}$$
$$= 232.4 \text{ m/min}$$

$$\text{Optimum tool life, } T = \left[1.5 + \frac{5}{0.8667}\right]\left[\frac{1 - 0.22}{0.22}\right] = 25.77 \text{ minutes}$$

$$\text{Machining time, } T_m = \frac{\pi \times 150 \times 600}{1000 \times 0.25 \times 232.4} = 4.866 \text{ minutes}$$

$$\text{Total time} = 5 + 4.866\left[1 + \frac{1.5}{25.77}\right] = 10.149 \text{ minutes}$$

$$\text{Cost of operation} = 0.8667 \times 5 + 4.866\ [(5 + 1.5 \times 0.8667)]$$
$$= 9.74\ ₹$$

Maximum Production Rate criterion:

$$\text{Optimum cutting speed, } V = 475\left[\frac{0.22}{1.5 \times (1 - 0.22)}\right]^{0.22} = 328.87 \text{ m/min}$$

$$\text{Optimum tool life, } T = \frac{1.5(1 - 0.22)}{0.22} = 5.318 \text{ minutes}$$

$$\text{Machining time, } T_m = \frac{\pi \times 150 \times 600}{1000 \times 0.25 \times 328.87} = 3.439 \text{ minutes}$$

$$\text{Total time} = 5 + 3.439\left[1 + \frac{1.5}{5.318}\right] = 9.409 \text{ minutes}$$

$$\text{Cost of operation} = 0.8667 \times 5 + 3.439\ [(5 + 1.5 \times 0.8667)]$$
$$= 11.388\ ₹$$

The above results are summarised in the following table:

Parameters	**Brazed Tools**		**Throw Away Tools**	
	Minimum Cost	**Maximum Production Rate**	**Minimum Cost**	**Maximum Production Rate**
Cutting speed, m/min	199.5	308.7	232.4	328.87
Tool life, Minutes	51.58	7.09	25.77	5.318
Machining time, minutes	5.669	3.664	4.866	3.439
Operation time, Minutes	10.89	9.679	10.149	9.409
Cost, ₹	10.63	14.02	9.74	11.388

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AUTOMOBILE ENGINEERING

SMEA 1505 _ MANUFACTURING TECHNOLOGY II

UNIT II CENTRE LATHE AND SPECIAL PURPOSE LATHES

UNIT II

CENTRE LATHE AND SPECIAL PURPOSE LATHES

The principal form of surface produced in a lathe is the cylindrical surface. This is achieved by rotating the work piece while the single point cutting tool removes the material by traversing in a direction parallel to the axis of rotation. A large number of variants of lathes are used in manufacturing shops. The variations are:

1. Centre lathe
    - Bench Lathe
2. Tool room lathe
3. Special purpose lathes
    - Copying lathe
    - Gap bed lathe
    - Hollow spindle lathes
4. Capstan and turret lathes
5. Automatic lathes

The centre lathe is the most common of the lathes, which derives its name from the way a work piece is clamped by centres in a lathe, though this is not the only way in which the job is mounted. This is sometimes also called engine lathe in view of the fact that early lathes were driven by steam engines. This is used for more general applications and thus the construction of the machine tool is more rigid.

The tool room lathe is generally meant for applications of tool making, where the accuracy desired is much higher than is normally required for general production work. Also the range of sizes and materials handled would normally be large. Thus the machine would have a higher range of speeds and feeds along with greater rigidity. Also the range of accessories and attachments would generally be larger. The special purpose lathes are developed from the centre lathe, to cater to special forms of application which cannot be handled by the conventional centre lathe. Capstan and turret lathes and automatic lathes are the form of lathes to cater for high rate production and thus would be used for very special applications.

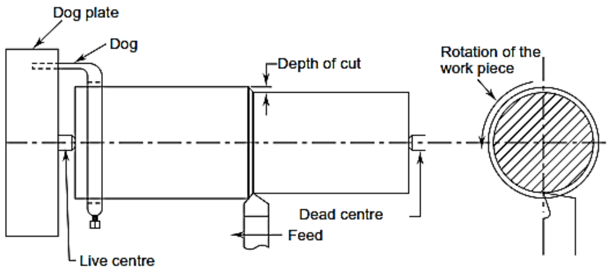


Fig.1 Cylindrical turning operation in lathe

Construction

The headstock houses the power source, all the power transmission, gear box and the spindle. The headstock is fixed at the left-most end on the bed. The spindle is hollow and should be sufficiently rigid to provide accurate rotary motion and maintains perfect alignment with the lathe axis. A live centre fits into the Morse taper in the spindle hole for the purpose of locating the work piece axis. The main gear box provides the necessary spindle speeds considering the range of materials to be turned in the lathe. The headstock also houses the feed gear box to provide the various feed rates and thread cutting ranges.

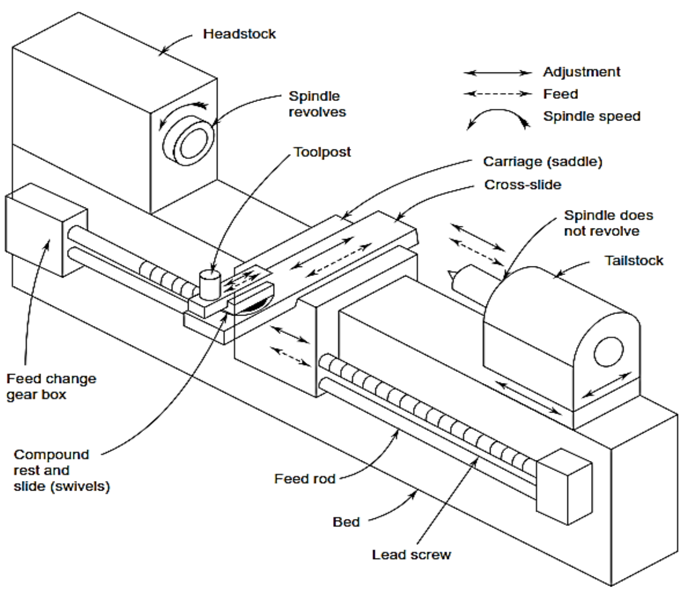


Fig.2 Centre lathe

The tailstock is towards the right-most end on the bed, and houses the tailstock spindle for the purpose of locating the long components by the use of centres. The tailstock is movable on the inner guide ways provided on the bed to accommodate the different lengths of work pieces. It also serves the purpose of holding tools such as centre drill, twist drill, reamer, etc. for making and finishing holes in the components, which are located in line with the axis of rotation.

The third major element in the lathe mechanism is the carriage, which provides the necessary longitudinal motion for cutting tool to generate the necessary surfaces. This also houses three parts: the cross slide for giving motion (cross feed) to the cutting tool in a direction perpendicular to the axis of rotation, the compound slide which provides an auxiliary slide to get the necessary special motion for specific surface generations, and the tool post which allows for the mounting of the cutting tool.

The motion from the spindle motor is communicated to the carriage through a lead screw. Engagement of the lead screw with the carriage is through the use of a half nut. Though the lead screw could be used for feeding the cutting tool in a direction parallel to the axis of rotation, many a times a separate feed rod is provided for this function. The main reason is that the lead screw would be more accurate and would be sparingly used only for thread cutting, such that it maintains its accuracy. For routine feeding, the feed rod is used.

Lathe Specifications

In order to specify a lathe, a number of parameters could be used based on the specific application. However, the major elements used for specification should invariably be based on the components that would be manufactured in the lathe. Thus the following are the basic elements generally specified for the capability of the lathe machine

● distance between centres—this specifies the maximum length of the job that can be turned in the lathe

● Swing over the bed—this specifies the maximum diameter of the job that can be turned in the lathe machine, generally restricted to small length jobs.

● Swing over the cross slide—this specifies the maximum diameter of the job that can be turned in the lathe machine with the job across the cross slide there are a number of other factors that should also be specified to fully describe the lathe machine. They are:

- horse power of the motor
- cutting speed range
- feed range
- screw cutting capacity
  - Accuracy achievable
  - spindle nose diameter and hole size

Table 1 Specifications of Centre Lathes

Centre Height, mm	250,300	375,450	525,600	750,900
Bed Width, mm	325,375	450,550	650,750	900,1050
Sizes Available, mm	1650 to 4200	2400 to 9600	2400 to 9600	2400 to 9600
Distance Between Centres (mm)	500 to 3100	1000 to 8200	800 to 8000	
Power Capacity, H.P.	3	5	7.5/10	10/15
Spindle speed Range	30 to 550	30 to 350	15 to 200	15 to 200

The work holding devices normally used should have the following provisions:

- suitable location
- effective clamping
- support when required

The most common form of work holding device used in a lathe is the chuck. Chucks come in various forms with a varying number of jaws. Of these the three jaw chuck or the self-centring chuck as shown the most common one. The main advantage of this chuck is the quick way in which the typical round job is centred. All the three jaws would be meshing with the flat scroll plate. Rotating the scroll plate through a bevel pinion would move all the three jaws radially inward or outward by the same amount. Thus, the jaws will be able to centre any job, whose external locating surface is cylindrical or symmetrical, like hexagonal. Though it is good for quick centring, it has limitations in terms of the gripping force and also the accuracy is gradually lost due to the wear of the mating parts.

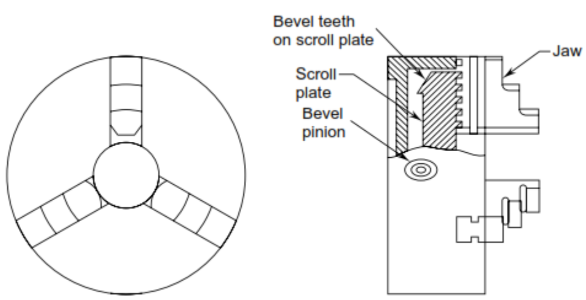


Fig. 3-jaw chuck and principle of operation

The independent jaw chuck has four jaws, which can be moved in their slots independent of each other, thus clamping any type of configuration. Since each of these jaws could move independently any irregular surface could be effectively centred. Better accuracy in location could be maintained because of the independent movement. However more time is spent in fixturing a component in a 4-jaw chuck compared to the 3-jaw chuck. This is generally used for heavy work pieces and for any configuration.

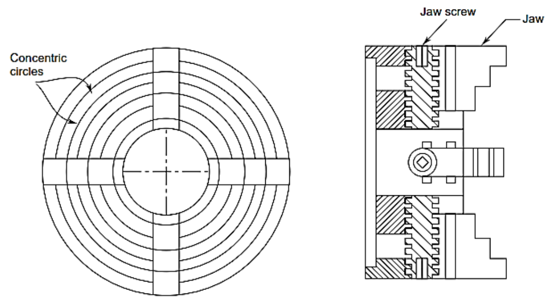


Fig. 4 Jaw Chuck

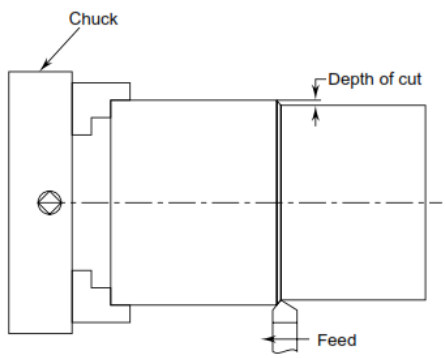


Fig.5 Chuck and reverse jaw usage

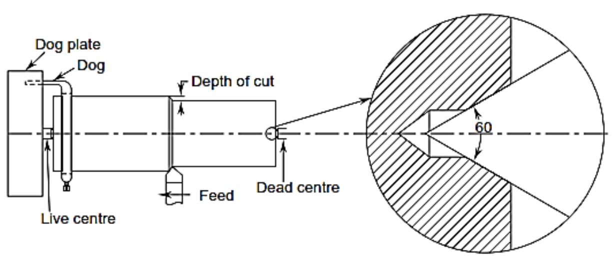


Fig.6 Centre hole, locating between centres

The centre located in the spindle is termed live centre while that in the tailstock is termed the dead centre. The shank of the centre is generally finished with a Morse taper which fits into the tapered hole of the spindle or tailstock.

Live centre rotates with the work piece, and hence it remains soft. Whereas the dead centre does not rotate, it is hardened as it forms the bearing surface. However, in case of heavier work pieces the relative movement between the work piece and the dead centre causes a large amount of heat to be generated. In such cases, a revolving centre is used. In this the centre is mounted in roller bearings thus it rotates freely reducing the heat generation at the tailstock end.

In cases where a facing operation is to be carried out with centres, a half centre would sometimes be used.

Some of the precautions to be observed during the use of centres are:

● The centre hole in the work must be clean and smooth and have an angle of 60° bearing surface, large enough to be consistent with the diameter of the work. For heavier work this may be changed to 75° or 90°.

● The bearing must take place on the countersunk surfaces and not on the bottom of the drilled hole. When the job becomes very long, it is likely to deflect because of its own weight as well as due to the cutting force acting away from the supports provided at both the ends. A steady is used for supporting the work piece at the maximum deflection point.

Sometimes a steady is fixed to the carriage, so that it moves with the tool thus effectively compensating for the acting cutting force. For odd shaped components a faceplate is more widely used where the locating and clamping surfaces need not be circular. This has radial slots on the plate for the purpose of locating the component and clamped by means of standard clamps.

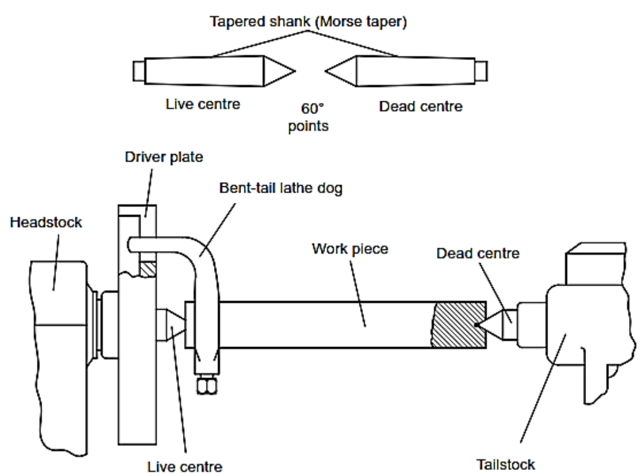


Fig.7 Dog carrier and revolving centre

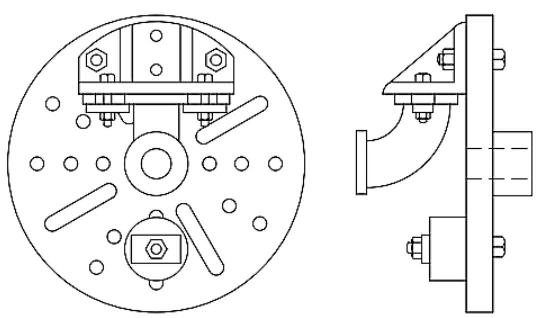


Fig.8 Face plate

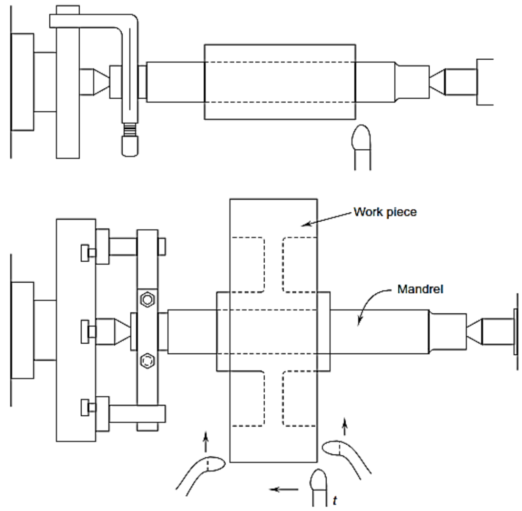


Fig. 9 types of mandrels used for work holding

Collet has a sleeve as the holding part, which is slit along the length at a number of points along the circumference. When uniform pressure is applied along the circumference of the sleeve, these segments would elastically deflect and clamp the component located inside. Since the deflection of the sleeve is in the elastic range, it would spring back once the clamping pressure is removed, thus releasing the component located inside. This clamping method is very accurate and fast in operation.

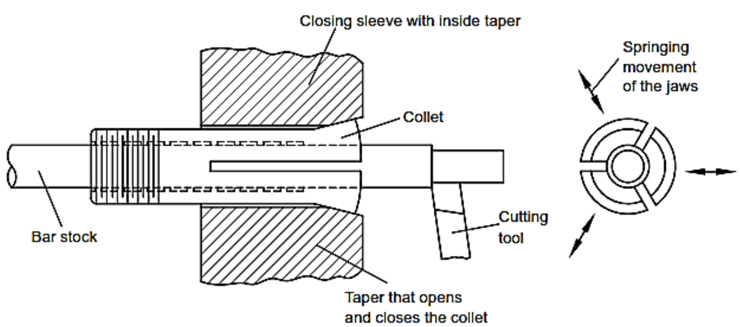


Fig.10 Collect chuck principle

Steady rest

The steady rest, also called a centre rest, should be used when turning or boring long work pieces. It is also used for internal threading operations where the work piece projects a considerable distance from the chuck or faceplate. The steady rest is clamped to the lathe bed at the desired location (where the maximum deflection is likely to occur) and supports the work piece with three adjustable jaws. The jaws must be carefully aligned to properly locate the axis of rotation of the work piece. The area of contact must be lubricated frequently. The top section of the steady rest swings away from the bottom section to permit removal of the work piece without disturbing the jaw setting. For machining with very high cutting speeds, steady rests as shown will generate substantial heat, so they will be provided with a ball or roller bearings built into the jaws. Another problem with this type of rest is that since the carriage cannot pass it, the work piece needs to be turned in two set-ups by reversing after the first portion is machined.

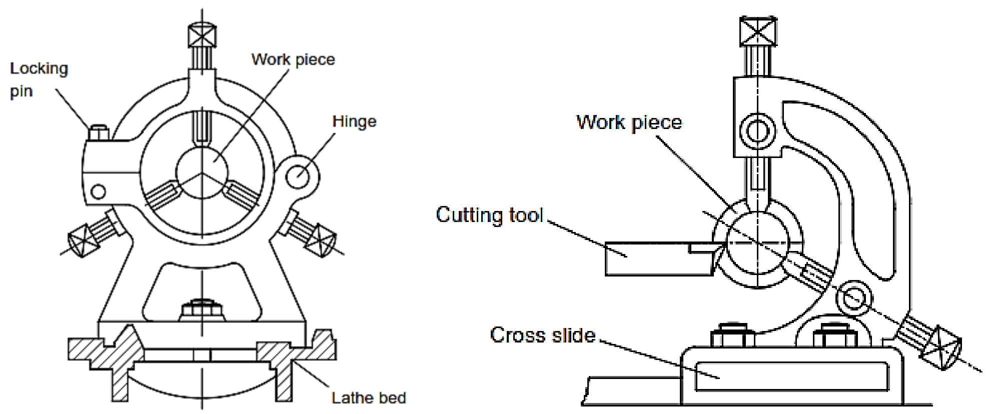


Fig.11 Steady rest and follower rest

Follower rest

The follower rest on the other hand has two jaws that bear against the work piece. The follower rest is fastened to the lathe carriage so that it will follow the cutting tool and bear upon the portion of the work piece that has just been turned. The cut must first be started and continued for a short longitudinal distance before the follower rest may be applied. The rest is generally used only for straight turning and for threading long, thin work pieces. Steady rests and follower rests can be equipped with ball-bearing surfaces on the adjustable jaws. These types of rests can be used without excessive lubricant or having to machine a polished bearing surface.

Cutting Tool

Cutting Tool Geometry

The size of the tool is generally square or rectangular in cross section. The shank is that part of the tool on one end of which the cutting point is formed. It is supported in the tool post of the lathe. The base is that part of the shank which bears against the support and bears the tangential force of the cut. These individual angles have considerable influence on the cutting performance. They have to be judicially chosen for a given application. For example the side cutting edge angle controls the width and thickness of the chips produced. A very large angle means that the uncut chip thickness reduces resulting in higher specific cutting resistance. When it approaches zero, the radial component of the cutting force is minimum while the axial component is maximum. This is generally the preferred condition since the vibration resistance is at its best in this condition. The recommended tool angles for various types of work and tool material combinations.

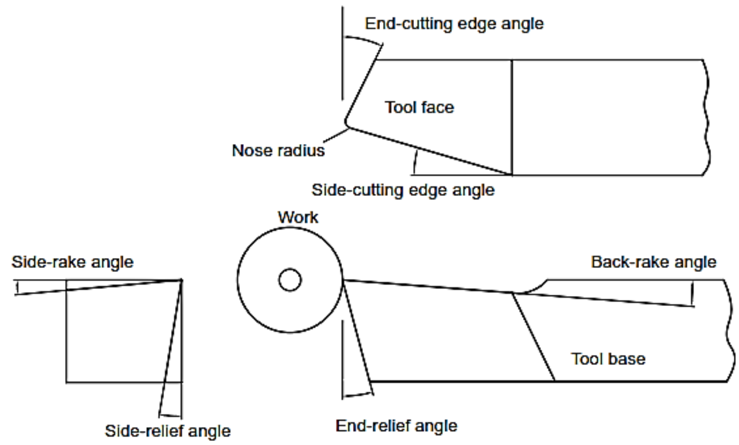


Fig.12 Turning tool geometry

Table 2. Recommended tool angles in degrees for high speed steel cutting tools

Work Material	Back Rake Angle	Side Rake Angle	Side Relief Angle	Front Relief Angle	Side Cutting Edge Angle	End Cutting Edge Angle
Steel	8 – 20	8 – 20	6	6	10	15
Cast steel	8	8	6	6	10	15
Cast iron	0	4	6	6	10	15
Bronze	4	4	6	6	10	10
Stainless steel	8 – 20	8 – 20	6	6	10	15

Table 3. Recommended tool angles in degrees for carbide cutting tools

Work Material	Back Rake Angle	Side Rake Angle	Side Relief Angle	End Relief Angle
Aluminium and magnesium alloys	0 – 10	10 – 20	6	6
Copper	0 – 4	15 – 20	6 – 8	6 – 8
Brass and bronze	0 – 5	–5 – 8	6 – 8	6 – 8
Cast iron	–7 – 0	–7 – 6	5 – 8	5 – 8
Plain carbon steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Alloy steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Stainless steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Titanium alloys	–5 – 6	–5 – 0	5 – 8	5 – 8

Different Types of Tools Used

A large variety of tools are used in centre lathes in view of the large types of surfaces that are generated. The actual type of tool used depends upon the surface of job being generated as well as the work piece. A variety of tools used for normal generation of external surfaces.

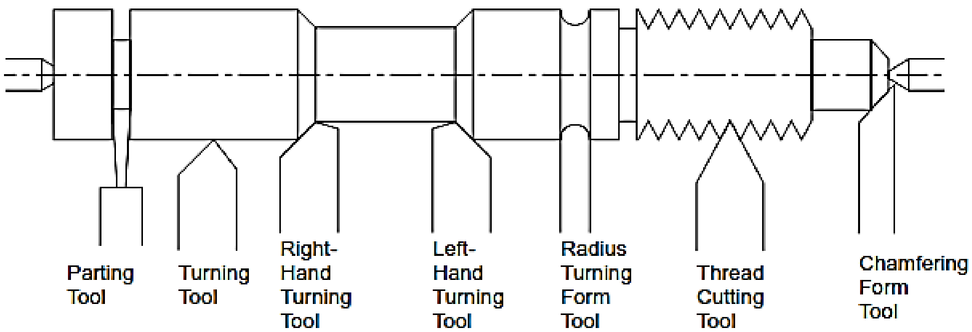


Fig.13 Different kinds of tools used for external surfaces

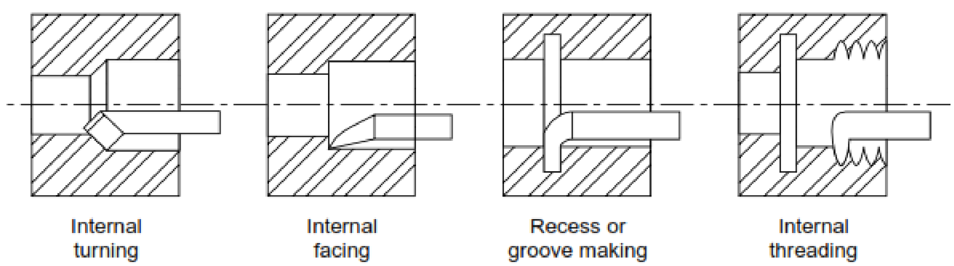


Fig.14 Different kinds of tools used for internal surfaces

The tools have primary cutting edge by means of which the direction of the movement of the tool for removing the metal is indicated. The direction is termed as right or left depending upon the movement direction. The tool is termed right when it cuts during the movement towards the head stock. It is derived by the fact that when the right palm is placed on the tool, the direction of the thumb indicates the direction of tool motion. Similarly the left hand tool cuts during its motion in the direction of tailstock.

The variations in the type of tools are indicative of the variety applications for which these tools are used. The large variety is needed because of the large number of surfaces to be generated. For example, by the side cutting edge angle of the tool, it is possible to know the application of pockets for which the tool could be used. Similarly some tools would be required for facing applications while others are used for boring. The two types of form tools that are generally used. The circular form tool is held in a holder mounted on the cross slide. The centre of the tool should be mounted slightly above the centre of the work piece in order to get a clearance angle such that the tool will not rub the work. The circular form tool has a long life as it can be continuously sharpened over 270° of the tool. The straight or flat form tool is the simplest type. It is sharpened by grinding the top face that reduces the strength of the tool.

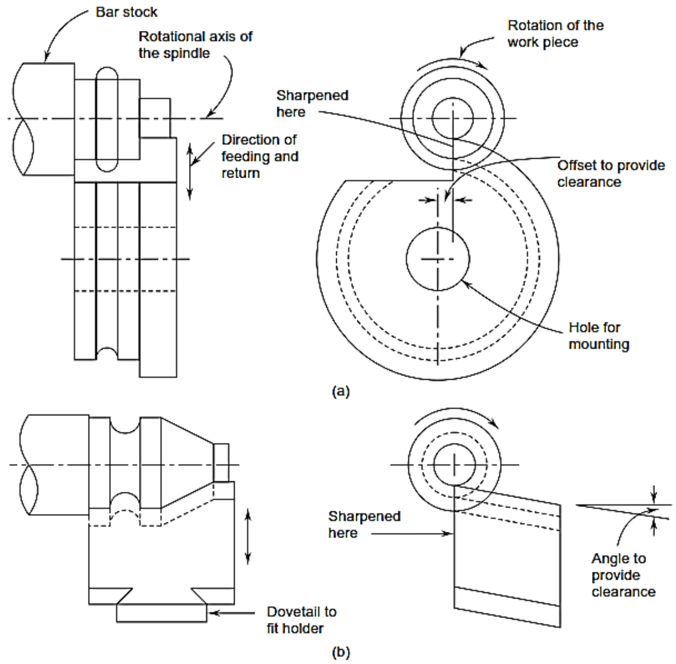


Fig.15 Form tool types used in centre lathe, (a) Circular form tool, (b) Straight form tool

Now a majority of the tools used are of the cemented carbide type with indexable insert type. It therefore becomes necessary to understand the ISO coding systems for these to be able to easily make the selection. The ISO coding system (as per ISO 1832–1991) for tungsten carbide inserts and external turning tools

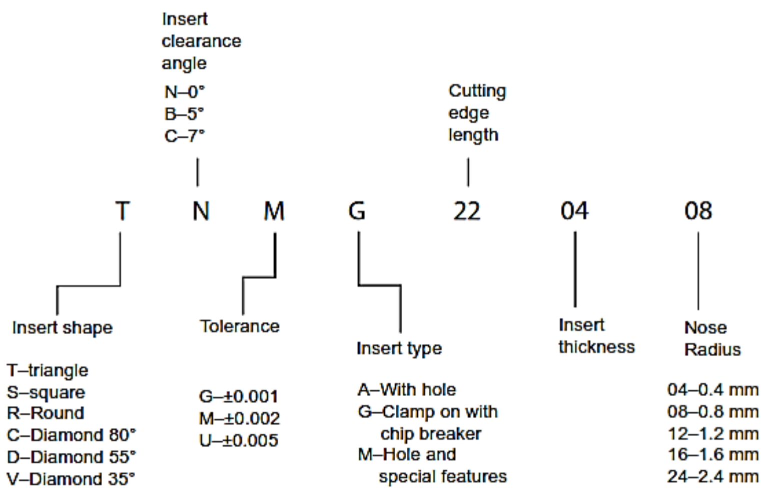


Fig.16 The ISO coding system for tungsten carbide inserts used in turning

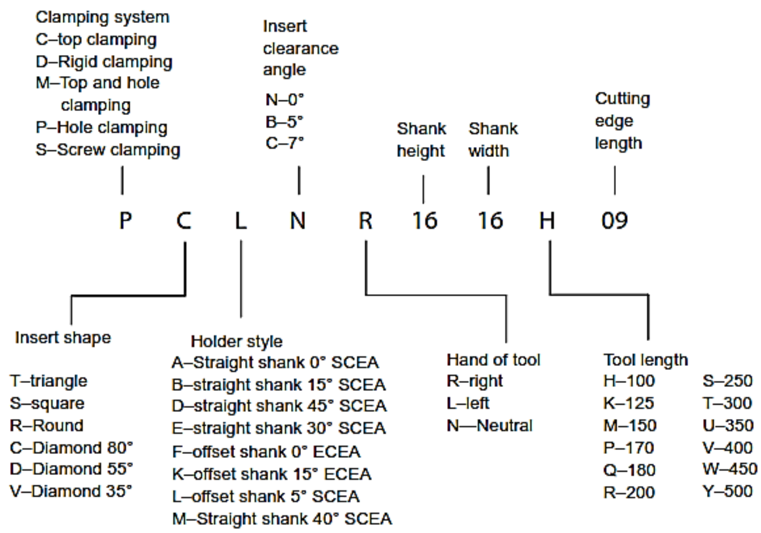


Fig.17 The ISO coding system for tungsten carbide turning tool holders used in external turning (SCEA - side cutting edge angle, ECEA - end cutting edge angle)

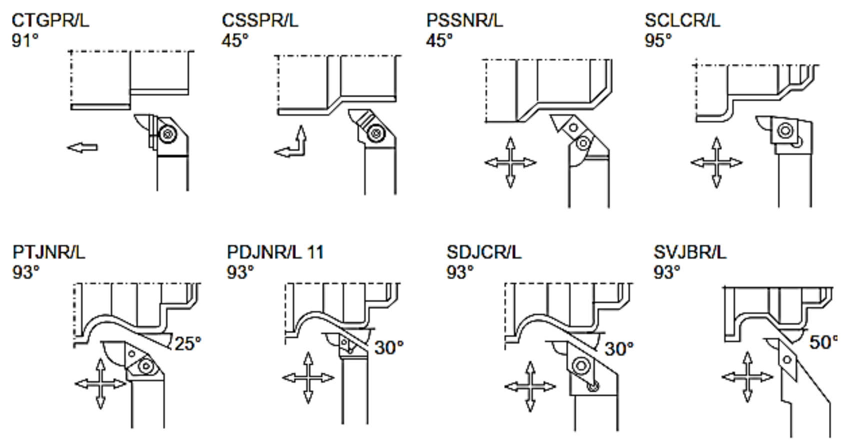


Fig.18 The ISO coding system for tungsten carbide turning tool holders used in external turning (SCEA - side cutting edge angle, ECEA - end cutting edge angle)

Errors in tool setting

The tool should be set exactly at the centre of the work piece for proper cutting. If the tool is kept below or above the work piece, the tool geometry gets affected

$$R = \sqrt{(r^2 - h^2)}$$

where R is the actual radius of the component produced, and

r is the radius set

$$\propto = \sin^{-1}\left(\frac{h}{r}\right)$$

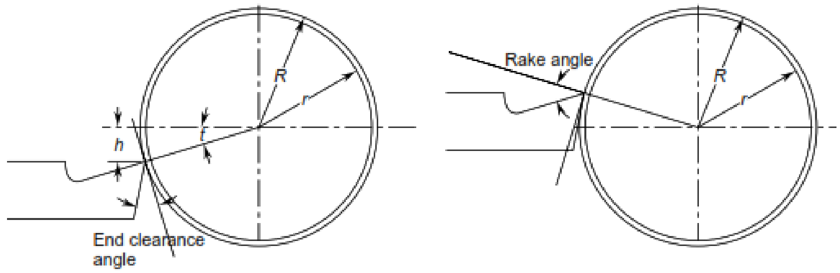


Fig.19 Tool setting errors

Types of Tool Posts

The turret tool post is a swivelling block that can hold many different tool bits or tool holders. Each cutting tool can quickly be swivelled into cutting position and clamped into place using a quick clamping handle. The turret tool post is used mainly for high-speed production operations. The heavy-duty or open-sided tool post is used for holding a single carbide-tipped tool bit or tool holder. It is used mainly for very heavy cuts that require a rigid tool holder.

Chip Control

Chips produced with some ductile materials become very long and often become hazardous to the operator as well as the surface finish of the part produced. Hence most of the tools need a mechanism by which these long chips can be broken. If the rake surface of the tool is flat, the chip will slide over the surface and will not have a chance to break

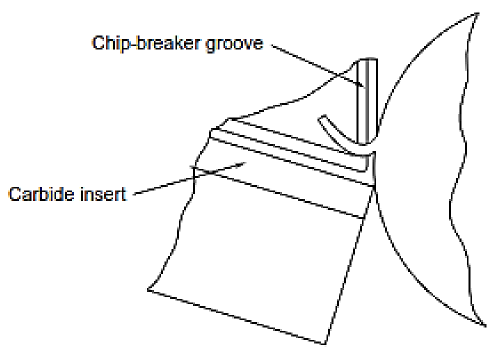


Fig.20 Chip breaker groove on a tungsten carbide tool bit

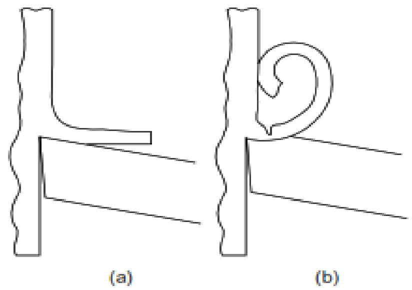


Fig.21 Chip breaking groove, (a) Cutting with a tool without chip breaking groove; (b) Cutting with chip breaker

Operations performed in a centre lathe

Turning is by far the most commonly used operation in a lathe. In this the work held in the spindle is rotated while the tool is fed past the work piece in a direction parallel to the axis of rotation. The surface thus generated is the cylindrical surface

Facing is an operation for generating flat surfaces in lathes. The feed in this case is given in a direction perpendicular to the axis of revolution. The tool used should have a suitable approach angle so that it would not interfere with the work piece during the tool feeding.

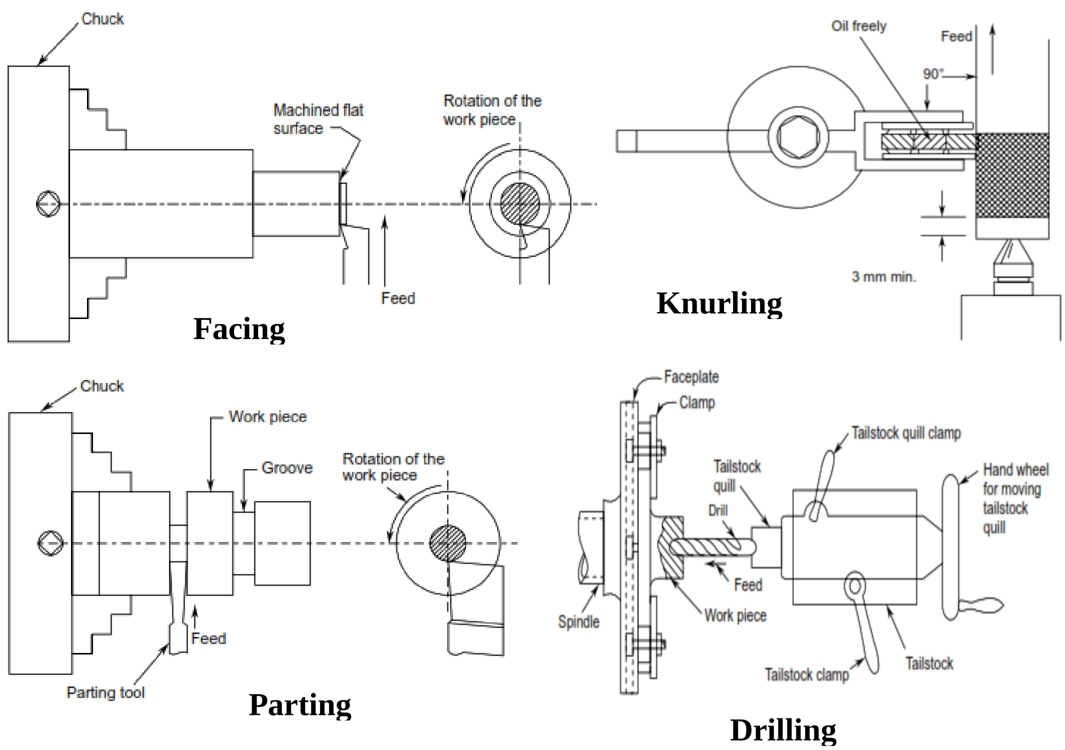


Fig.22 Operations performed in Lathe

Knurling is a metal working operation done in a lathe. In this a knurling tool having the requisite serrations is forced on to the work piece material, thus deforming the top layers

Parting and grooving are similar operations. In this a flat nosed tool would plunge cut the work piece with a feed in the direction perpendicular to the axis of revolution. This operation is generally carried out for cutting off the part from the parent material. When the tool goes beyond the centre, the part would be severed. Otherwise a rectangular groove would be obtained. It is also possible, in similar operation, to use a special form of tool to obtain the specific groove shape.

Drilling is the operation of making cylindrical holes into the solid material. A twist drill is held in the quill of the tailstock and is fed into the rotating work piece by feeding the tailstock quill. Since the work piece is rotating, the axis of the hole is very well maintained, even when the drill enters at an angle initially. The same operation can also be used for other hole making operations such as centre drilling, counter sinking and counter boring. This operation is limited to holes through the axis of rotation of the work piece and from any of the ends.

Boring is the operation of enlarging a hole already made by a single point boring tool termed as boring bar. The operation is somewhat similar to the external turning operation. However, in view of the internal operation, it is more restricted. The cutting forces experienced are somewhat more than the external operation. Also the tool used is less rigid compared to turning tool and as a result it cannot withstand the large cutting forces. Thus the process parameters used are somewhat lower than those used for turning.

Boring is used for generating an accurate hole with good surface finish.

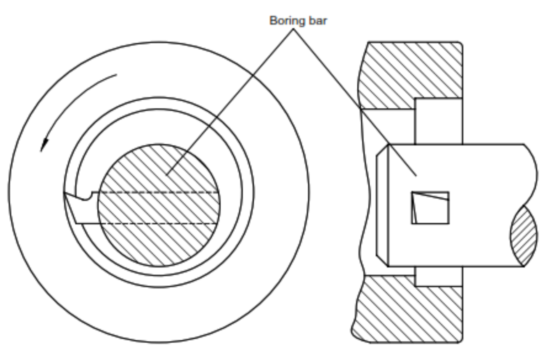
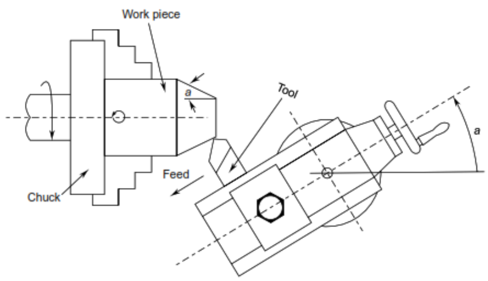


Fig.23 Boring operation in lathe

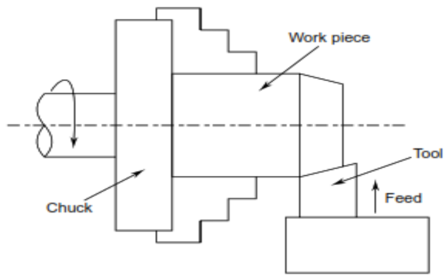
Taper turning

Cutting tapers on a lathe is one of the most common applications. A number of methods are available for cutting tapers in a lathe. They are:

- using a compound slide
- using form tools
- offsetting the tailstock
- using taper turning attachment



Compound slide method for taper turning



Taper turning using Form

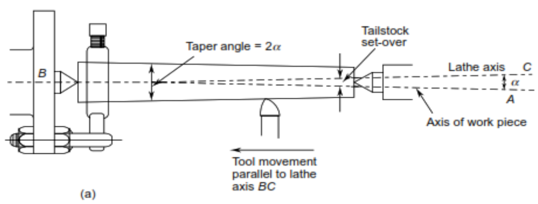


Fig.24 Tail stock offset

$$\sin \propto \;=\; \frac{BC}{AB}$$

$$S = AB\ \sin \propto$$

$$\sin \propto \;\approx\; \tan \propto \;=\; \frac{D - d}{2l}$$

$$S = L\left(\frac{D - d}{2l}\right)$$

If L = l, Offset, $S = \left(\frac{D - d}{2}\right)$

Example While turning a taper using taper turning attachment, the setting was done for 4°, but the tool is set 3 mm below the centre. If the work piece diameter at the small end is 40 mm, calculate the actual taper produced.

$$OC^2 = 3^2 + \left[100 \tan 4 + \sqrt{20^2 - 3^2}\right]^2 = 725.44$$

$$OC = \sqrt{725.44} = 26.933 \text{ mm}$$

The produced taper angle is therefore

$$\alpha = \tan^{-1}\left[\frac{26.933 - 20}{100}\right] = 3.9657°$$

The error produced is

$$\text{Error in half taper} = 4 - 3.9697 = 0.0303° = 2'$$

Thread cutting methods

Cutting screws is another of the most important tasks carried out in lathes. Thread cutting can be considered as another form of turning since the path to be travelled by the cutting tool is helical. However, there are some major differences between turning and thread cutting.

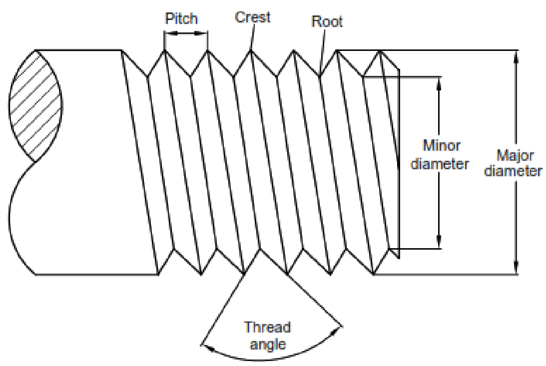
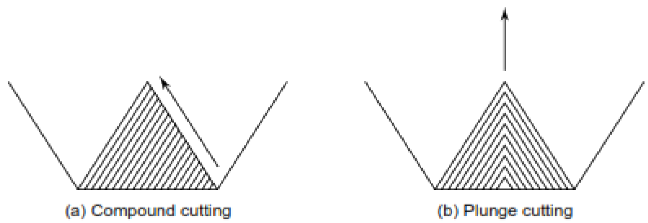


Fig.25 Simple thread

Table 4 Formulae for some common thread forms

Thread Form	Formulae for Calculating the Parameters
British Standard Whitworth (BSW)	Depth = 0.6403 × Pitch Angle = 55° in the plane of the axis Radius at the crest and root = 0.137329 × Pitch
British Association (BA)	Depth = 0.6 × Pitch Angle = 47.5° in the plane of the axis Radius at the crest and root = $\frac{2 \times \text{Pitch}}{11}$
International Standards Organisation (ISO) metric thread	Max. Depth = 0.7035 × Pitch Min. Depth = 0.6855 × Pitch Angle = 60° in the plane of the axis Root radius Maximum = 0.0633 × Pitch Minimum = 0.054 × Pitch
American Standard ACME	Height of thread = 0.5 × Pitch + 0.254 mm Angle = 29° in the plane of the axis Width at tip = 0.3707 × Pitch Width at root = 0.3707 × Pitch – 0.132 mm



Depth of cut in screw

Thread cutting using compound slide

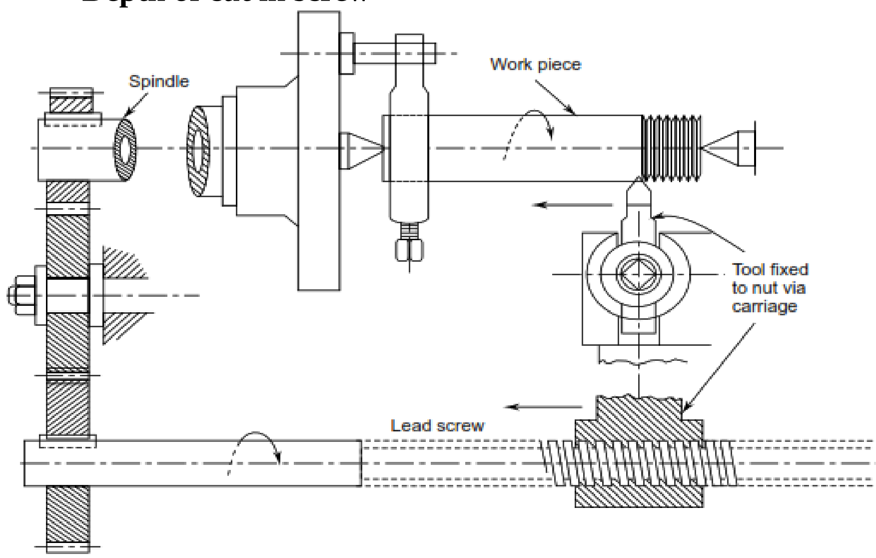


Fig.26 Thread cutting using compound slide

The compound slide is rotated by the half angle of the thread, and the cutting tool is adjusted to make it perpendicular to the work piece surface. For this purpose a thread setting gauge which contains the required form of the thread being cut is kept perpendicular to the surface of the work piece, and the tool tip is set.

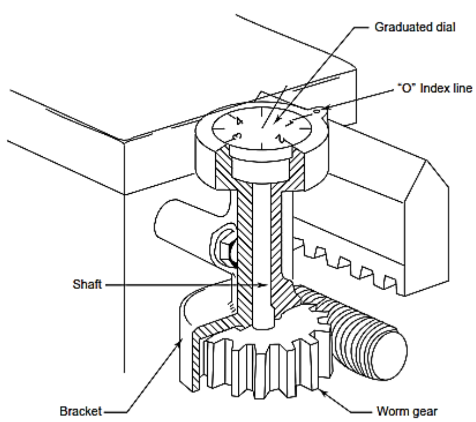


Fig.27 Chasing dial principle

Milling Attachment

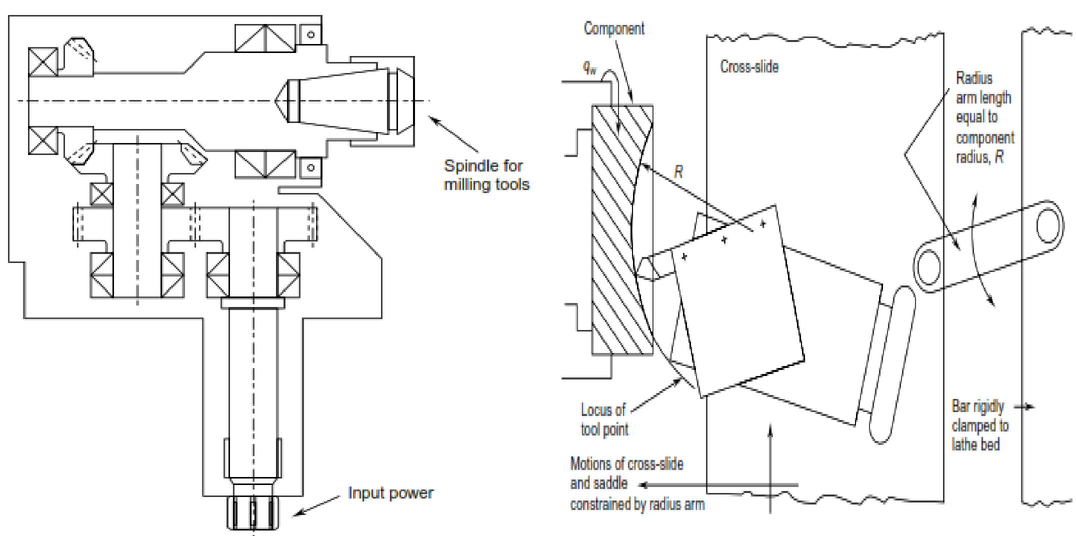
This is an attachment used in lathes to carry out milling operations. The attachment is provided with a separate spindle where the milling cutters could be located, and is attached to the cross slide, replacing the compound slide. The work is held between the lathe centres as in normal centre lathes. The milling cutter can normally be fed in all the three directions, thus permitting any type of milling operation.

Grinding Attachment

Similar to the milling attachment, a grinding attachment is used to finish a part in the lathe by completing the required grinding operations without disturbing the setup. It can be mounted in place of the tool post on the compound slide. These have two or three-axis movement and will be able perform a number of grinding operations.

Copy Turning Attachment

Many a times the need exists for machining complex contours, which require the feeding of the tool in two axes (X and Y) simultaneously, similar to taper turning. For such purposes copy turning is to be used. In this, the cross slide is directly driven by a stylus which can trace a master for the actual contour to be produced. The cross slide is made similar to the taper turning attachment.



Milling **Radius Attachment**

Fig.28 Special attachments

Machining time and power estimation

To estimate the machining times, it is necessary to select the proper process parameters. For this purpose it is necessary to know the work piece material and the cutting tool material combinations to arrive at the right combination of the process parameters, cutting speed, feed and depth of cut.

Table 5 Suggested cutting process parameters for turning

Work Material	Hardness	High Speed Steel Tool		Carbide Tool	
	BHN	Speed m/min	Feed mm/rev	Speed m/min	Feed mm/rev
Grey cast iron	150–180	30	0.25	140	0.30
Grey cast iron	220–260	20	0.25	90	0.30
Malleable Iron	160–220	33	0.25	50	0.25
Malleable Iron	240–270	—	—	45	0.30
Cast steel	140–180	40	0.25	150	0.30
Cast steel	190–240	26	0.25	125	0.30
C20 Steel	110–160	40	0.30	150	0.38
C40 Steel	120–185	30	0.30	145	0.38
C80 Steel	170–200	26	0.30	130	0.30
Alloy Steel	150–240	30	0.25	110	0.38
Alloy Steel	240–310	20	0.25	100	0.30
Alloy Steel	315–370	15	0.25	85	0.25
Alloy Steel	380–440	10	0.20	75	0.25
Alloy Steel	450–500	8	0.20	55	0.25
Tool Steel	150–200	18	0.25	70	0.25
Hot Work die steel	160–220	25	0.25	120	0.25
Hot Work die steel	340–375	15	0.25	75	0.25
Hot Work die steel	515–560	5	0.20	23	0.20
Stainless Steel	160–220	30	0.20	120	0.25
Stainless Steel	300–350	14	0.20	70	0.25
Stainless Steel	375–440	10	0.20	30	0.25
Aluminium Alloys	70–105	210	0.30	400	0.38
Copper Alloys	120–160	200	0.25	300	0.25
Copper Alloys	165–180	85	0.25	230	0.25

Table 6 Possible causes for the turning problems

Observed Problem	Possible Cause
Vibration	Work or chuck out of balance
Chatter	Work improperly supported Feed rate too high Tool overhang too large Tool is not properly ground or length of tool edge contact is high
Work piece not turned straight	Headstock and tailstock centres not aligned Work improperly supported Tool not in the centre
Work piece out of round	Work loose between centres Centres are excessively worn Centre out of round

The cutting speed in turning is the surface speed of the work piece.

$$V = \frac{\pi D N}{60}$$

where, V = cutting speed (surface), m/min

D = diameter of the work piece, mm

N = rotational speed of the work piece, rpm

The time, t for a single pass

$$t = \frac{L + L_o}{fN}$$

where L = length of the job, mm

Lo = over travel of the tool beyond the length of the job to help in the setting of the tool, mm

f = feed rate, mm/rev

The roughing passes, P_r

$$P_r = \frac{A - A_f}{d_r}$$

where A = Total machining allowance, mm

A_f = Finish machining allowance, mm

d_r = Depth of cut in roughing, mm

The finishing passes, P_f

$$P_f = \frac{A_f}{d_f}$$

where d_f = Depth of cut in finishing, mm

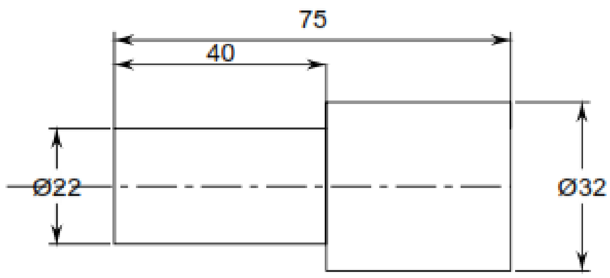
Example: A component is shown to be machined from a stock of CRS C40 steel, 40 mm in diameter and 75 mm long. Calculate the machining times required for completing the part with (a) HSS tool and (b) Carbide tool.

Pocket 1: HSS Tool

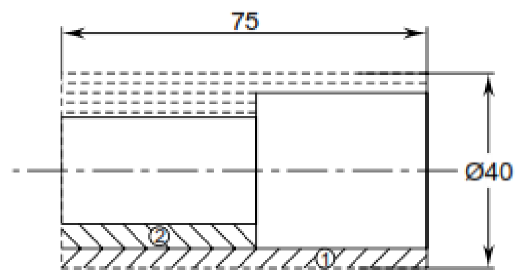
Assume Cutting speed, V = 30 m/min

Feed rate, f = 0.30 mm/rev.

Depth of cut = 2 mm



(a) Part drawing



(b) Machining plan

$$\text{Spindle speed} = \frac{1000 \times 30}{\pi \times 36} = 265.25 \text{ RPM} \approx 265 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{75 + 2}{0.30 \times 265} = 0.9686 \text{ minutes}$$

$$\text{Number of passes required} = \frac{40 - 32}{2 \times 2} = 2$$

Carbide Tool

Assume Cutting speed, $V = 145$ m/min

Feed rate, $f = 0.38$ mm/rev.

Depth of cut = 2 mm

$$\text{Spindle speed} = \frac{1000 \times 145}{\pi \times 36} = 1282.05 \text{ RPM} \approx 1280 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{75 + 2}{0.38 \times 1280} = 0.158 \text{ minutes}$$

Pocket 2: HSS Tool

$$\text{Number of passes required} = \frac{32 - 22}{2 \times 2} = 2.5 \approx 3$$

$$\text{Spindle speed} = \frac{1000 \times 30}{\pi \times 27} = 353.677 \text{ RPM} \approx 355 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{40 + 2}{0.30 \times 355} = 0.394 \text{ minutes}$$

Carbide Tool

$$\text{Spindle speed} = \frac{1000 \times 145}{\pi \times 27} = 1709.44 \text{ RPM} \approx 1710 \text{ RPM}$$

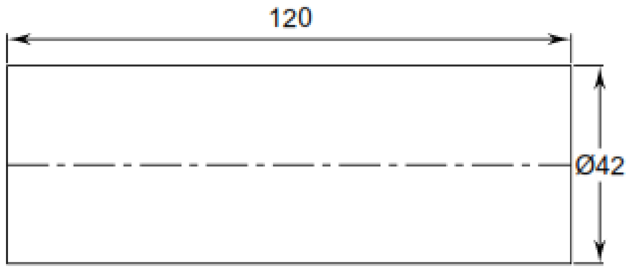
$$\text{Time for machining one pass} = \frac{40 + 2}{0.38 \times 1710} = 0.065 \text{ minutes}$$

Total machining time:

For HSS tool = $2 \times 0.9686 + 3 \times 0.394 = 3.1192$ minutes

For Carbide tool = $2 \times 0.158 + 3 \times 0.065 = 0.511$ minutes

Example: Estimate the actual machining time required for the component (C40 steel) shown in Fig. The available spindle speeds are, 70, 110, 176, 280, 440, 700, 1100, 1760 and 2800. Use a roughing speed of 30 m/min and finish speed of 60 m/min. The feed for roughing is 0.24 mm/rev while that for finishing is 0.10 mm/rev. The maximum depth of cut for roughing is 2 mm. Finish allowance may be taken as 0.75 mm. Blank to be used for machining is 50 mm in diameter. Calculate the power required for roughing and finishing passes.



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

Finish allowance = 0.75 mm

Roughing:

Roughing stock available = 4 – 0.75 = 3.25 mm

Since maximum depth of cut to be taken is 2 mm, there are 2 roughing passes.

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

$$\text{Average diameter} = \frac{50+42}{2} = 46 \text{ mm}$$

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

The nearest RPM available from the list is 176 RPM as 280 is very high compared to 207 as calculated.

$$\text{Machining time for one pass} = \frac{(120+2)}{0.24 \times 176} = 2.898 \text{ minutes}$$

Finishing:

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

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

The nearest RPM available from the list is 440 RPM.

$$\text{Machining time for one pass} = \frac{(120+2)}{0.10 \times 440} = 2.77 \text{ minutes}$$

Total machining time = $2 \times 2.888 + 2.77 = 8.546$ minutes

Roughing:

Given feed rate, $f = 0.24$ mm/rev

Depth of cut, $d = 2$ mm

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

The value of K from Table 4.7 = 1600 N/mm^2

$$\text{Power} = \frac{1600 \times 25.43 \times 0.24 \times 2}{60} = 325.5 \text{ W} = 0.326 \text{ kW}$$

Finishing:

Given feed rate, $f = 0.10$ mm/rev

Depth of cut, $d = 0.75$ mm

$$\text{Cutting speed, } V = \frac{\pi \times 440 \times 43.5}{1000} = 60.13 \text{ m/min}$$

$$\text{Power} = \frac{1600 \times 60.13 \times 0.10 \times 0.75}{60} = 120.26 \text{ W} = 0.120 \text{ kW}$$

Note:

Power = F x V

Cutting force = K x d x f

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

Special Purpose lathe

The main limitation of centre lathes areas follows:

- The setting time for the job in terms of holding the job is large.
- Only one tool can be used in the normal course. Sometimes the conventional tool post can be replaced by a square tool post with four tools.
- The idle times involved in the setting and movement of tools between the cuts is large.
- Precise movement of the tools to destined places is difficult to achieve, unless proper care is exercised by the operator.

All these difficulties mean that the centre lathe cannot be used for production work in view of the low production rate. Thus the centre lathe is modified to improve the production rate. The various modified lathes are:

- Turret and capstan lathes
- semi-automatics
- automatics

Capstan and turret Lathes

The main characteristic feature of the capstan and turret lathes are:

- The six sided (hexagonal) block mounted on one end of the bed replacing the normal tailstock.
- This allows for mounting six tool blocks each of which can contain one or more tools depending upon the requirement.
- The cross slide, two tool posts are mounted, one in the front and the other in the rear. Each one of them can hold up to four tools each.
- The total carrying capacity is a maximum of 14 tools when only one tool is mounted in each of the locations.

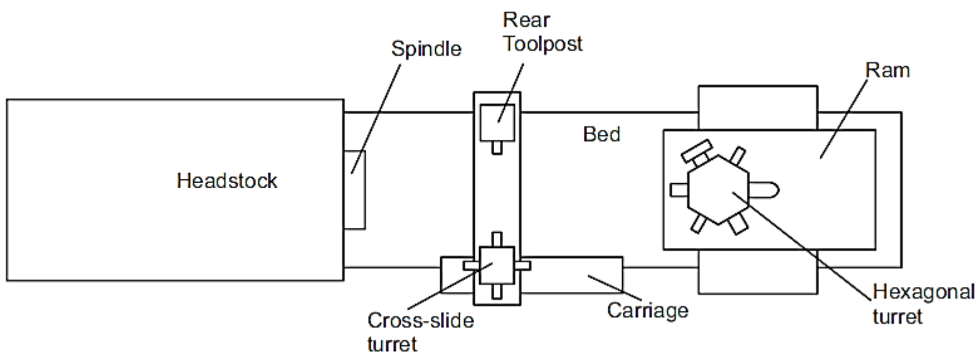


Fig.29 Turret lathe

The turret lathe consists of an all gear, heavy duty headstock with a greater range of spindle speeds. The turret is mounted on a saddle, which in turn is sliding on the bed. When the saddle moves on the bed during the return stroke it would automatically be indexed to the next tool position, thus reducing the idle time of the machine.

The tools in the turret lathe are provided with a system of stops and trips on the feed rod which can precisely control the actual distance moved by the tool. Thus it is possible to set and control the individual movements of the tools as required by the component.

The type of work holding devices that can be used with turret lathes is similar to the conventional lathes, but in view of the higher productivity demanded and greater repeatability required, generally automatic fixtures such as collets, self-centring chucks or pneumatic chucks are used. The collet chucks come in a variety of designs. The actual clamping is done by the movement of the collet tube along the axis of the spindle by either pushing or pulling.

Sometimes it is possible that the bar material will be either pushed or pulled back during the closing of the collet. This can be prevented by having an external tubular locking stop so that the axial movement is prevented.

Often a large variety of components on a turret lathe are machined from raw material which is in a bar form. For the purpose of continuous feeding of the bar special bar feeding arrangements are available which pushes the bar by a precise amount against a stop provided on the face of the hexagonal turret at the beginning of the cycle. The last operation in such cases is the parting off operation from the cross slide tool which separates the machined component from the bar stock.

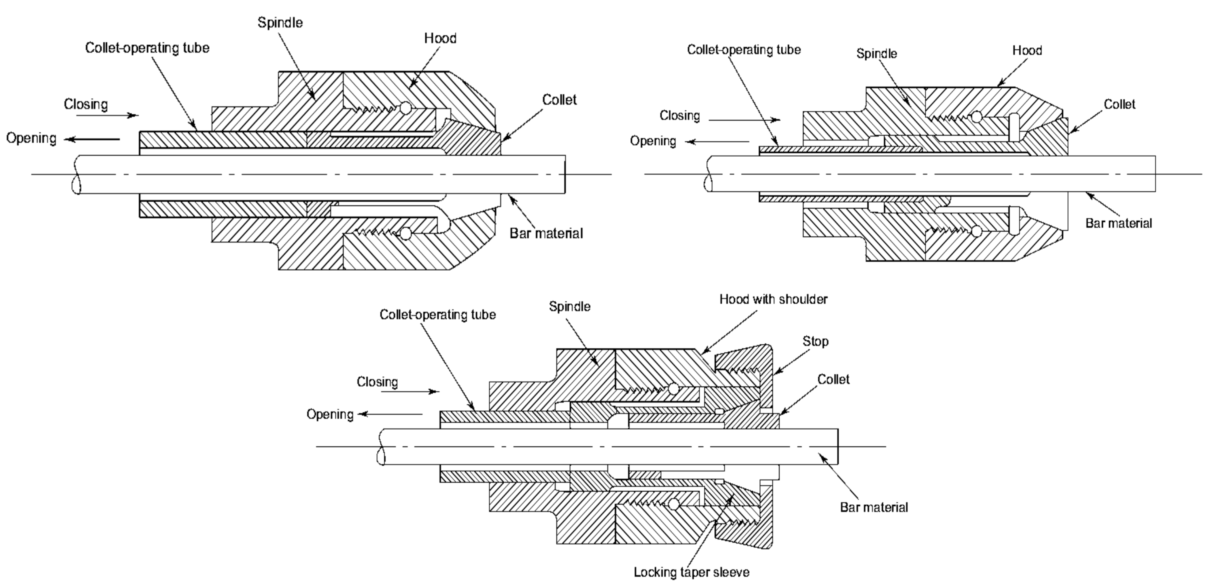


Fig.30 Collet chucks in turret lathe

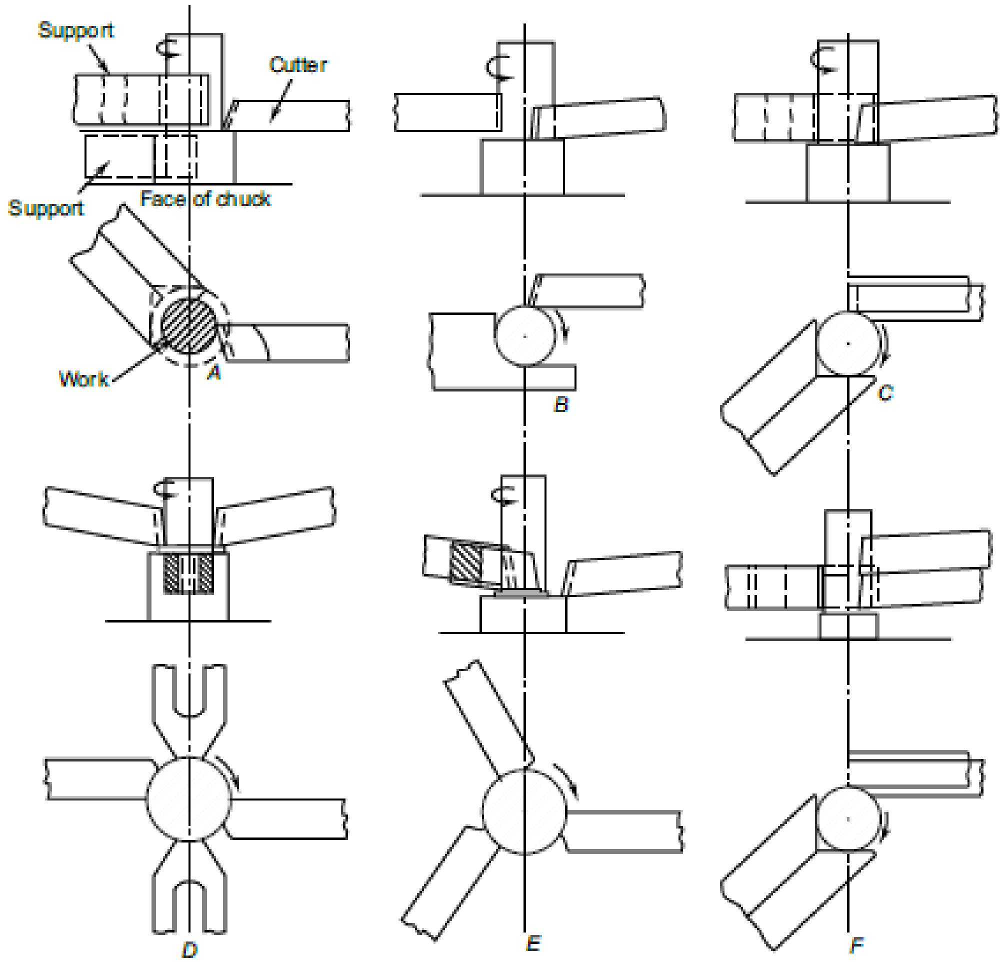


Fig.31 Different types of tool box in turret lathe

The various differences between capstan and turret lathes, and a general purpose centre lathe are:

1. Headstock has more range of speeds and is heavier to allow for higher rate of production.

2. Tool post is indexable (four tools). Any one tool can be brought into cutting position.

3. Tail stock is replaced by a tool turret with six tool positions.

4. Feed of each tool can be regulated by means of feed stops.

5. Two or more tools mounted on a single tool face can cut simultaneously.

6. Semi-skilled operators are required.

7. Used for production operations involving better repeatability

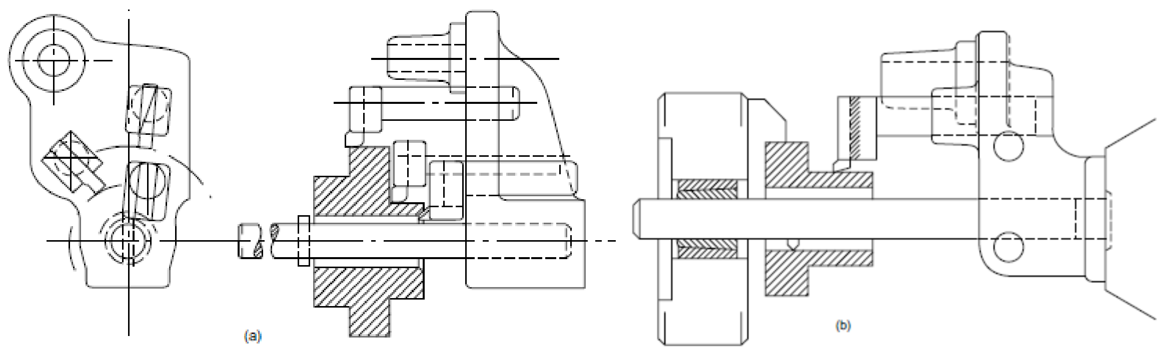


Fig.32 Special tools used in turret lathe

Thus the various differences between capstan and turret lathes, and a general purpose centre lathe are:

1. Headstock has more range of speeds and is heavier to allow for higher rate of production.

2. Tool post is indexable (four tools). Any one tool can be brought into cutting position.

3. Tail stock is replaced by a tool turret with six tool positions.

4. Feed of each tool can be regulated by means of feed stops.

5. Two or more tools mounted on a single tool face can cut simultaneously.

6. Semi-skilled operators are required.

7. Used for production operations involving better repeatability.

A variation of the turret lathe is the capstan lathe, in which the turret moves on the saddle while the saddle can itself be fixed at any position on the bed, depending upon the length of the job. Thus, the tool travel length is limited to the length of the saddle. This type of arrangement is normally used for small size machines.

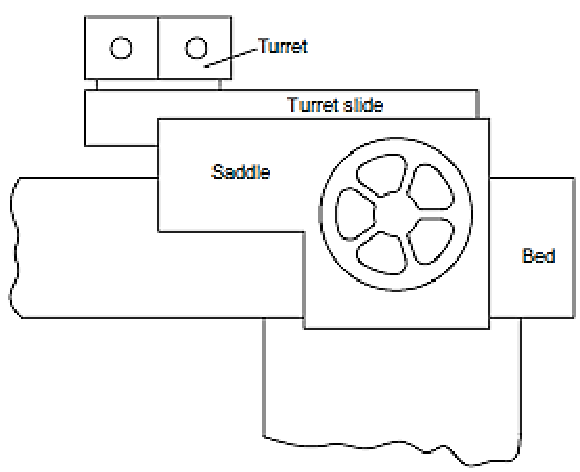


Fig.33 Intermediate slide arrangement in capstan lathe

Table 7 Difference between Casptan and Turret lathe

Capstan Lathe	Turret Lathe
Short slide since the saddle is clamped on the bed in position	Saddle moves along the bed, thus allowing the turret to be of large size.
Light duty machine, generally for components whose diameter is less than 50 mm	Heavy duty machine, generally for components with large diameters such as 200 mm
Too much overhang of the turret when it is nearing cut	Since the turret slides on the bed, there is no such overhang

Tool layout

A few rules that one should consider while planning the operations on turret lathes is given below:

1. For small batches use the standard tooling as far as possible and make the layout simple.

2. Cuts should be combined as far as possible. For example a tool from the hexagonal turret along with another from the square tool post on the cross slide can cut simultaneously. It would also be desirable to increase the number of tools operating simultaneously.

3. Similarly it is also necessary the handling operations be combined with the cutting operations such that total cycle time is reduced.

4. The planning for the finishing operations must be done till the end of the cycle. In between, there is a possibility of spoiling the finished surface. Also the combination of rough and finish operations in the same cycle should be done only when there is no detrimental effect on the quality of surface produced.

5. When multiple cutting tools are cutting at the same time, they should be so arranged that the cutting forces by the different tools get balanced.

6. If a given surface is achieved in a number of cuts, a finishing cut with a single tool is desirable in the interest of quality.

7. When concentricity is desired between two or more surfaces, all such surfaces should be machined in single setting only.

8. Contoured form surfaces are better obtained in two cuts rather than in single cut as far as possible.

9. While doing any heavy operation such as threading, care has to be taken to consider the rigidity of the work piece. Do not carry out any operation in the early stages, which reduces the rigidity of the component. Examples are deep grooves or large bores.

10. It is desirable to use centre drill before final drilling in case of small size drills. This would give rise to better drill axis location and smooth drilling.

11. Cored holes should normally be expanded and finished by boring and not by drilling.

12. In the case of stepped holes, make the large size hole first and follow with the small hole later. This would help in reducing the total drill travel and also reduce the machining time. The small drill would also not have to travel a long distance, which is always difficult.

13. To drill very long holes (e.g. length > 3 diameter) special care has to be taken. For example frequent withdrawal of the tool from the hole for flushing chips lodged in the flutes with cutting fluid is necessary in deep hole drilling, which is termed as peck drilling.

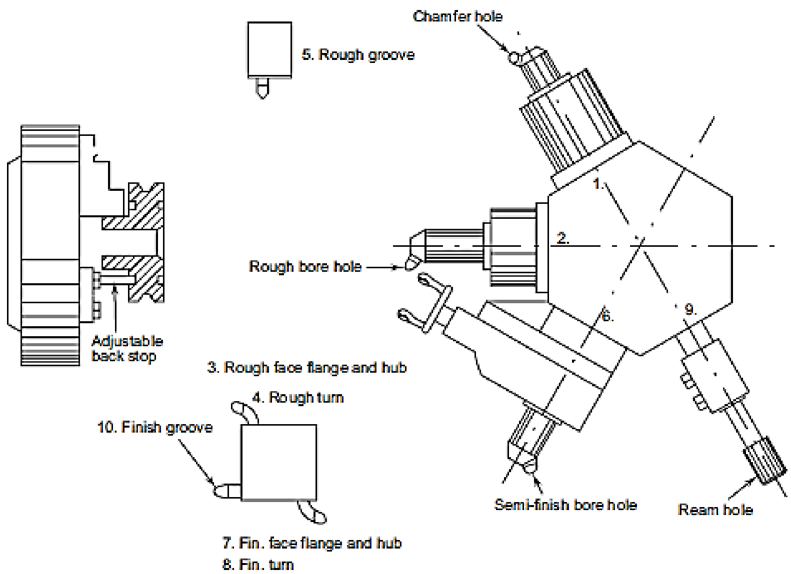


Fig.34 Tooling layout of cast iron V-belt pulley casting

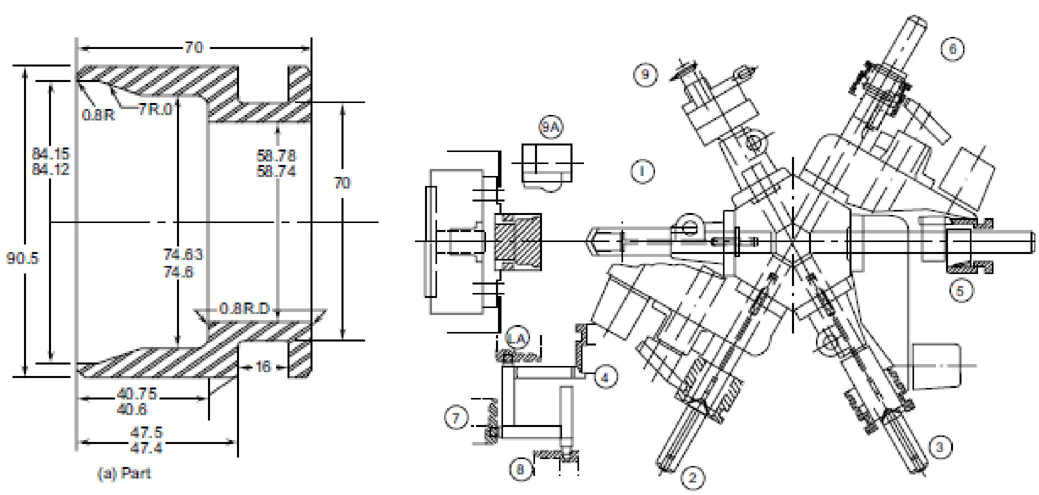


Fig.35 Tooling layout of closing sleeve of a collet chuck made of steel

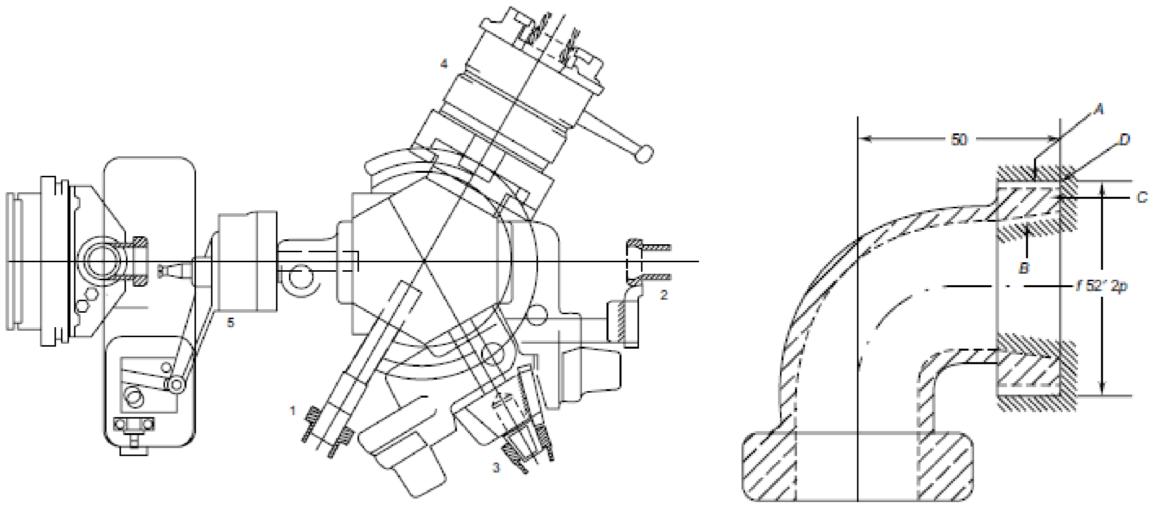


Fig.36 Tooling layout of brass pipe bend casting

Automatic lathe

- The term automatic is loosely applied, but is normally restricted to those machine tools capable of producing identical pieces without the attention of an operator after each piece is completed.
- All operations in machining a work piece on a metal cutting machine tool are classified as processing and handling operations.
    1. Processing operations are those in which the actual cutting process or chip removal takes place.
    2. The rest are handling operations and include loading and clamping the work, advancing and withdrawing the cutting tools, releasing and unloading the work, checking the size of the work, etc.
- The faster the working and handling operations are performed in a machine tool, the less time will be required to produce a work piece and more work pieces can be produced in the same period of time by a given machine tool, i.e. higher is the productivity of the machine tool.

Highly automated machine tools especially of the lathe family are ordinarily classified as automatics and semi-automatics.

Automatics as their name implies are machine tools with a fully automatic work cycle.

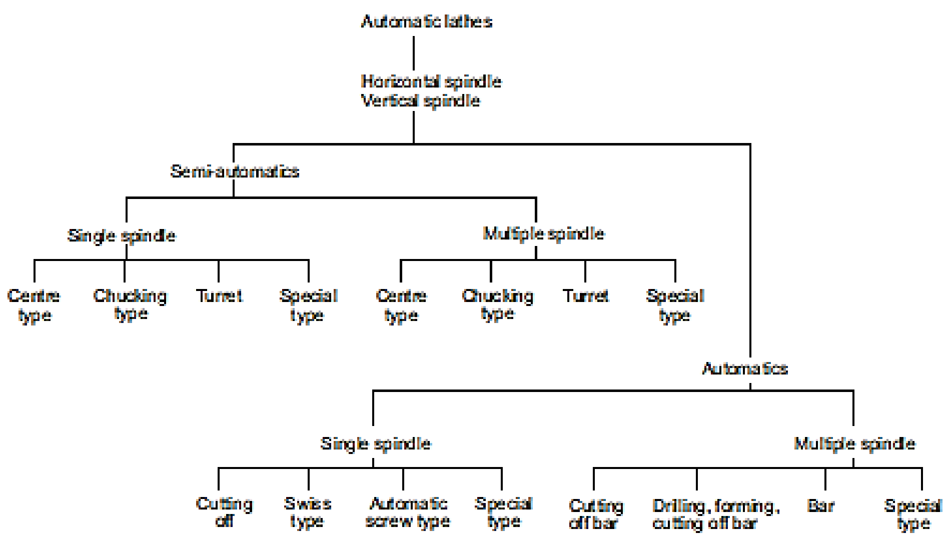
Semi-automatics are machine tools in which the actual machining operations are performed automatically in the same manner as on automatics

Classification

The automatic lathes may be classified based on their

- size
- type of blank machined
- processing capacity (operations performed)
- machining accuracy obtained
- principle of operation design features
- number of spindles and work positions
- type

The vertical machines are more rigid and more powerful than the horizontal models and are designed for machining large diameter work of comparatively short length. Vertical machines occupy less floor space in the shop but require higher bays than horizontal machines. Automatic bar machines are designed for producing work pieces of bar or pipe stock while magazine loaded automatic lathes process work from accurate separate blanks.



Chucking machines are employed for machining separate blanks (hammer or die forgings, castings or pieces of previously cut-off bar or pipe stock).

Automatic bar machines are employed for the manufacture of high quality fastenings (screws, nuts and studs), bushings, shafts, rings, rollers, handles, and other parts usually made of bar or pipe stock. The machining accuracy obtained by these automated machines depends on the type of machine and cutting tool employed.

Multiple spindle machines may have two to eight spindles. Their production capacity is higher than that of single spindle machines but their machining accuracy is somewhat lower. The rate of production of a multiple spindle automatic is less than that of the corresponding number of single spindle automatic machines. The production capacity of a four spindle machine, for example may be about 2 ½ to 3 times more than that of a single spindle machine.

The typical operations carried out on automatic lathes are:

- centring
- drilling
- reaming
- knurling
- facing
- turning cylindrical, tapered and formed surfaces
- boring
- spot facing
- thread cutting
- cutting off

Cutting off machine

Typical machining sequence of a cutting-off machine comprises of the following sequence:

1. Stock stop advances to the working position
2. Stock is fed till it meets the stock stop
3. Stock is chucked
4. Stock stop is withdrawn
5. Rapid approach of the tool slides
6. Working feed of the tool slides
7. Rapid return of the tool slides
8. Release of the stock

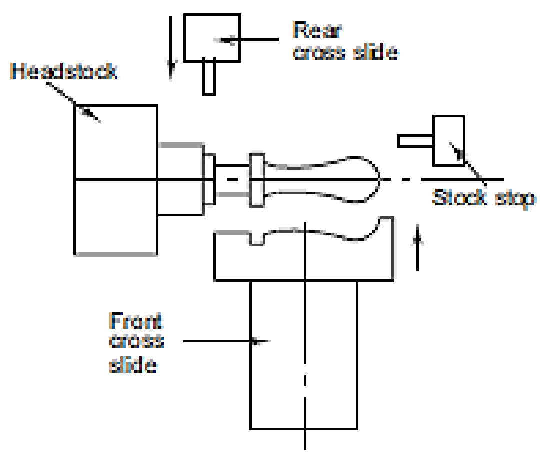


Fig.37 Tool arrangement in cutting off type automat

Swiss Type Automatics or Sliding Headstock Automatics

These are designed for machining long accurate parts of small diameter (4 to 25 mm). An exclusive feature of these machines is the longitudinal travel of the headstock or of a quill carrying the rotating work spindle. The end of the bar, projecting from the chuck, passes through a guide bushing (steady rest) beyond which the cross feeding tool slides are arranged. A wide variety of formed surfaces may be obtained on the work piece by co-ordinated alternating or simultaneous travel of the headstock (longitudinal feed) and the cross slide (approach to the depth of cut). Holes and threads are machined by attachments.

The bar stock used in these machines has to be highly accurate and is first ground on centre less grinding machines to ensure high accuracy. These consist of two rocker arm tool slides (front and rear) on which the turning tools are normally clamped. In addition three radial slides are arranged for additional tools.

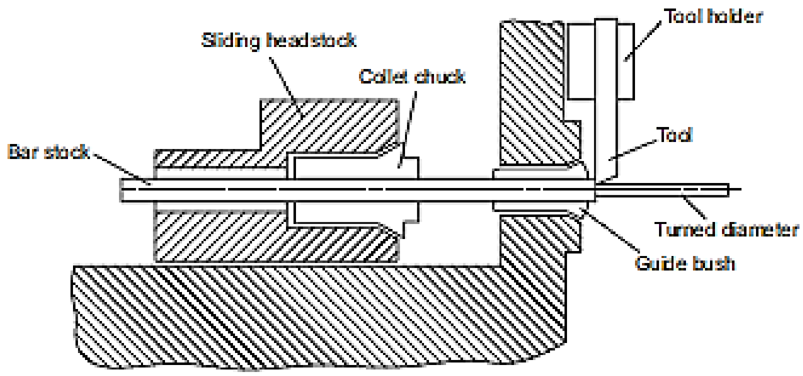


Fig.38 Schematic of a Swiss automatic lathe

Automatic Screw Machines

These are essentially wholly automatic bar type turret lathes. These are very similar to capstan and turret lathes with reference to tool layout, but all the tool movements are cam controlled such that full automation in manufacture is achieved. They are designed for machining complex external and internal surfaces on parts made of bar stock or of separate blanks. Up to ten different cutting tools may be employed at one time in the tooling of such a screw machine. The tools are clamped in the holes at the positions of the periodically indexing turret and in the cross slide. The stationary headstock, mounted on the left end of the bed, houses the spindle, which rotates in either direction. The turret slide is arranged at the right end of the bed and carries the turret having six tool holes. Two cross slides (front and rear) are provided for cross feeding tools.

A vertical slide attachment that provides an additional (third) cross slide may be employed. It is installed above the work spindle. All movements of the machine units are actuated by cams mounted on the cam shaft. Since these are fully automated, an operator can take care of more than two single spindle screw cutting or also called as bar automatic lathes, unlike a turret lathe for which an operator is required for each machine tool.

Multiple Spindle Automatics

In these machines there is more than one spindle where the work piece can be mounted. As a result, more than one work piece can be machined simultaneously in these machines. The number of spindles present could be four, five, six or eight. Each of the spindles is provided with its own set of tools for operation. The possible types of multi spindle machines are:

- Parallel action
- Progressive action

In parallel action machines the same operation on each spindle and a work piece is finished in one working cycle. This means that as many work pieces can be simultaneously machined as there are spindles. Such machines have a very high rate of production but may be applied for very simple work only, since the whole machining process takes place at one position.

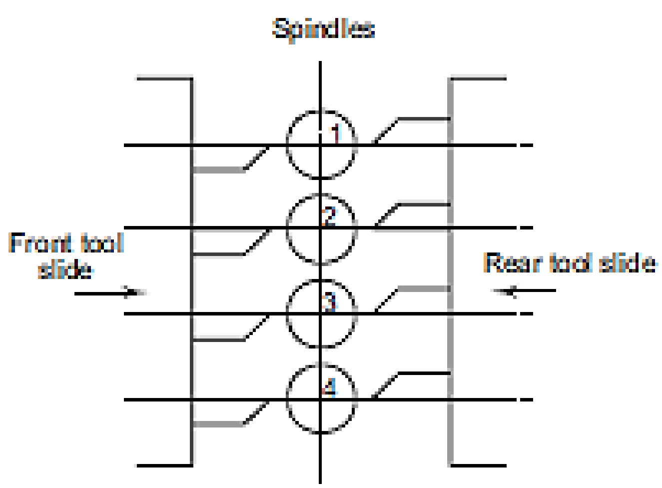


Fig.39 Parallel action multi spindle machine tool

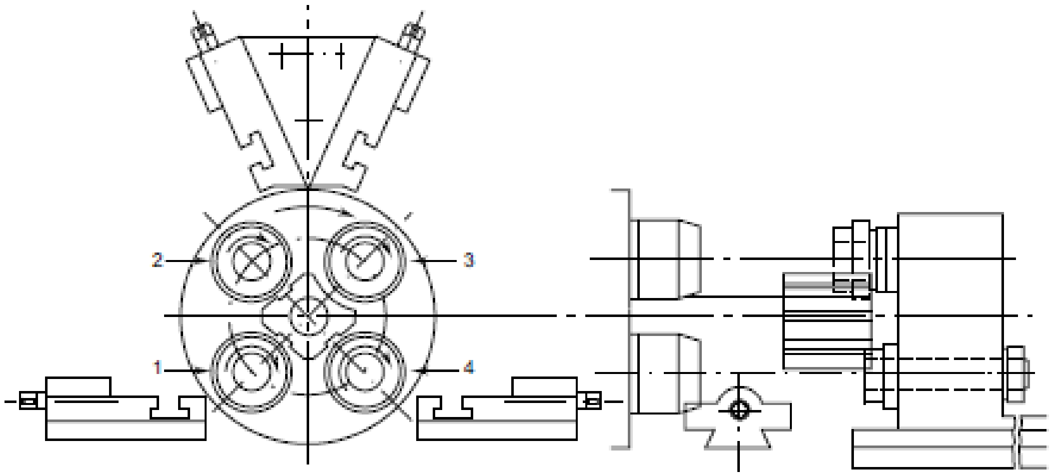


Fig.40 Progressive action multi-spindle machine tool

Table 8 Accuracy available in multi spindle machines

Types and Bar Capacities or Chucking Capacities, mm	Maximum Out of Roundness	Maximum Taper mm per Length mm	Maximum Diameter Variation in a Single Lot, mm
Swiss type automatic			
3 to 6.5	0.01	0.01 per 50	0.02
6.5 to 10	0.01	0.01 per 50	0.03
16 to 25	0.01	0.02 per 100	0.04
Automatic screw machines			
10 to 16	0.015	0.02 per 50	0.04
25 to 40	0.015	0.03 per 100	0.05
Single spindle semi-automatic			
80 to 100	0.015	0.03 per 100	0.08
120 to 200	0.020	0.03 per 150	0.10
250 to 300	0.030	0.03 per 200	0.15
400 to 500	0.040	0.03 per 250	0.20
Multiple spindle automatic bar machine			
25 to 40	0.015	0.03 per 100	0.08
65 to 100	0.020	0.03 per 150	0.10
Multiple spindle semi-automatic chucking machine			
80 to 100	0.015	0.03 per 100	0.10
120 to 200	0.020	0.03 per 150	0.12
250 to 300	0.030	0.03 per 200	0.20
400 to 500	0.040	0.03 per 250	0.25

Work holding and feeding

A chucking device must clamp the stock or blank reliably and rotate it with an accuracy that guarantees finished work of the specified form and size. Most of the time spring collet chucks are generally used with automats. They are very accurate besides the quick action, which can be achieved by the use of a simple lever.

Typical accuracy of spring collet is

12 mm dia 0.02 to 0.03 per 30 to 35 mm

40 mm dia 0.02 to 0.05 per 100 mm

Movement of the tool slides in case of automatic lathes is controlled by means of cams operated by a cam shaft which is linked to the main spindle drive. Mostly plate cams are used for this purpose while drum cams are used in multi-spindle automats. Cam controls the time, length of the tool stroke, as well as feed rate.

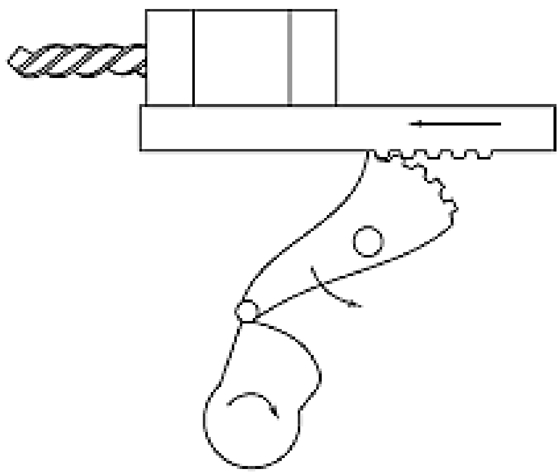


Fig.41 Cam controlled tool stroke in automate machine tool

Tooling Layout and Cam design for automatic Lathes

The tool layout and cam design for a job constitutes the predetermined plan for the order and method of the machining operations necessary to produce it. Accuracy and cost of manufacture are largely dependent on an efficient layout. The following steps are recommended while planning the layout for an automatic lathe.

1. Choose the best available machine taking into consideration the availability as well as price of the component.

2. Determine the sequence of operations.

3. Choose the available standard tooling as far as possible.

4. Decide on any possible design for special tooling if absolutely necessary.

5. Based on the machine capability and surface finish desired, decide the cutting process parameters for each of the tool to be used. In case of heavy jobs, the available spindle power of the machine may be
verified.

6. Check the movement of each of the tool in conjunction with the work piece for machining.

7. Arrange for any overlap of operations to reduce the total cycle time.

8. Compute the processing time including the idle time and from that the number of revolutions needed for each operation.

9. Calculate the spacing required on the cam periphery.

10. Draw the tool layout and cam details while verifying all the tool movements and clearances.

Tooling layout sequence

The sequence in which they are carried out.

1. The bar is fed against a stop located in the first turret position.

2. Index the turret and rough turn the large diameter using an overhang turning attachment. At the same time the drilling of the large hole is done to the full depth using a twist drill located in the second turret position.

3. Index the turret and knurl the external diameter using a knurling tool located in the third turret position.

4. Index the turret and drill the small hole to the full depth using a twist drill located in the fourth turret position. At the same time the overhang form tool is used to do the chamfering. In the same operation (1a) turn the external profile using circular form tool located in the front cross slide.

5. Index the turret and cut the external thread using a die located in the fifth turret position.

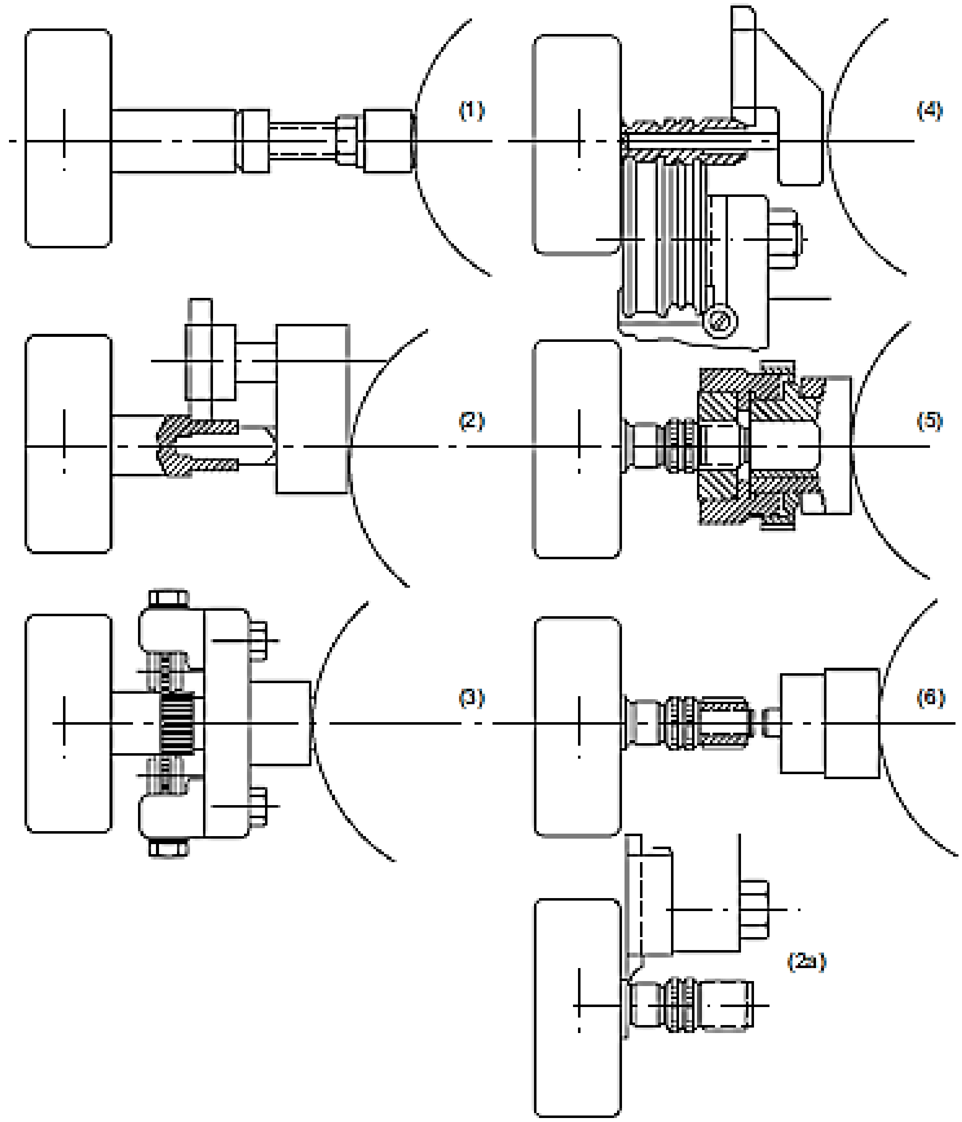


Fig.42 Cam controlling the tool stroke in automatic machine tool

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AUTOMOBILE ENGINEERING

SMEA 1505 _ MANUFACTURING TECHNOLOGY II

UNIT III SHAPER, MILLING, DRILLING, BORING, PLANER AND BROACHING

UNIT – 3 SHAPER, MILLING, DRILLING, BORING, PLANER AND BROACHING

The machines which use only reciprocating motion are discussed. The major machine tools that fall in this class are:

- Shaper
- Planer
- Slotter

The main characteristics of this class of machine tools are that they are simple in construction and as a result are very economical in initial cost as well as cost of operations.

Shaper

- The shaper is a relatively slow machine tool with very low metal removal capability.
- It is replaced by more versatile milling machine in many shops.
- This is a low cost machine tool and hence is used for initial rough machining of the blanks. It is rarely used in production operations.
- It uses a single point tool similar to a lathe which is clamped to a tool post mounted to a clapper box which in turn is mounted to a reciprocating ram.
- The ram while undertaking the cutting stroke pushes the cutting tool through the work piece to remove the material. When the ram returns, no cutting takes place.
- In between the return and cutting strokes, the table moves in the horizontal direction perpendicular to the cutting direction, this is termed as the feed direction.

Shaper Construction

The main constructional pieces of a mechanical shaper are:

- *The Base*
- *Column*
- *Ram*
- *Tool head (clapper box)*
- *Cross rail*
- *Table*

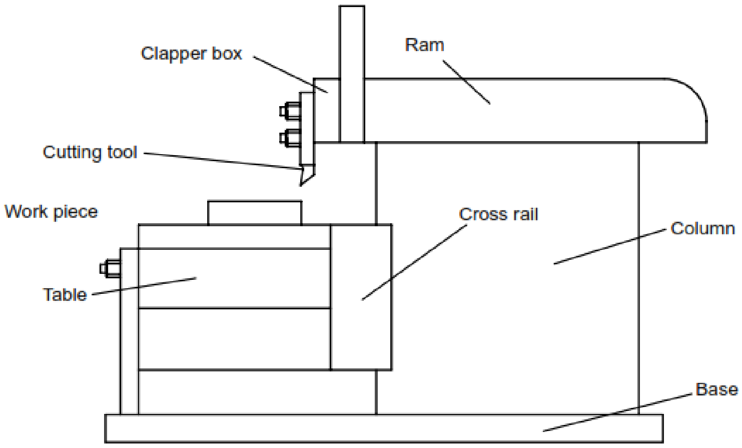


Fig.1 Shaper Layout

The Base The base provides the stability for the shaper as it supports all other equipment present as well as absorbs the forces coming due to the cutting. Generally it is made of grey cast iron and has the necessary arrangement of bolts so that it can be bolted to the factory floor with proper levelling.

Housing (column) The housing is a box like structure to provide the necessary rigidity and also houses all the motors and power transmission equipment. On the top of the housing the necessary guide ways are provided for the linear motion of the ram for the cutting stroke.

Ram It is the part of shaper that provides the reciprocating motion for the cutting tool. Ram gets the motions directly from the quick return mechanism (described later) present in the housing.

Tool head (clapper box) The single point cutting tool is clamped in the tool head. The tool head has the ability to swivel the cutting tool at any angle while clamping the tool with any overhang depending upon the requirement. The swivelling ability is important for the tool to machine surfaces that are not in horizontal plane. Further the tool should be firmly supported during the forward motion to carry out the material removal. During the return stroke, the cutting tool will not do the cutting and hence will be an idle stroke. If the tool is held firmly as in the cutting stroke, the tool will rub the already machined work piece and also the flank surface of the tool will wear out quickly. To reduce this, the tool is lifted during the return stroke by the clapper box arrangement

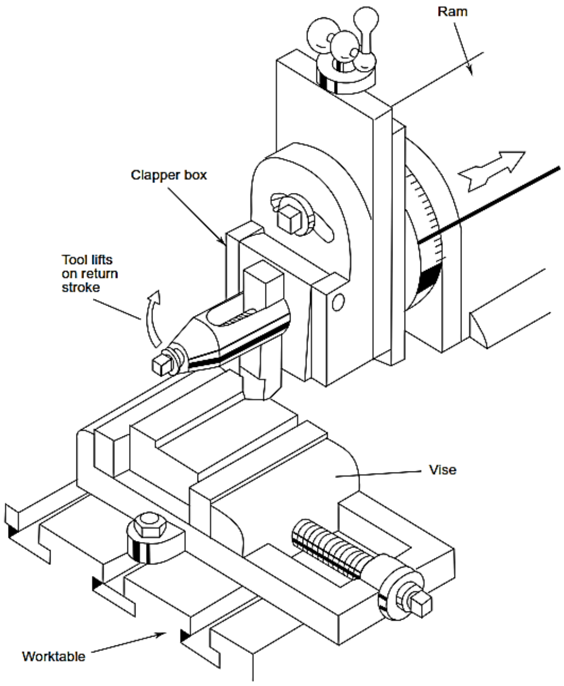


Fig.2 *Arrangement of tool and work material in shaper*

Table A heavy table is present at the front end of shaper. Table is provided with T-slots for mounting the work pieces or work holding fixtures. The table can be moved up and down along the guide ways provided on the cross rail attached to the housing.

A shaper is generally used for machining flat surfaces in horizontal, vertical and angular directions. It can also be used for machining convex and concave curved surfaces. The actual surface generated is by means of the linear motions of the cutting tool.

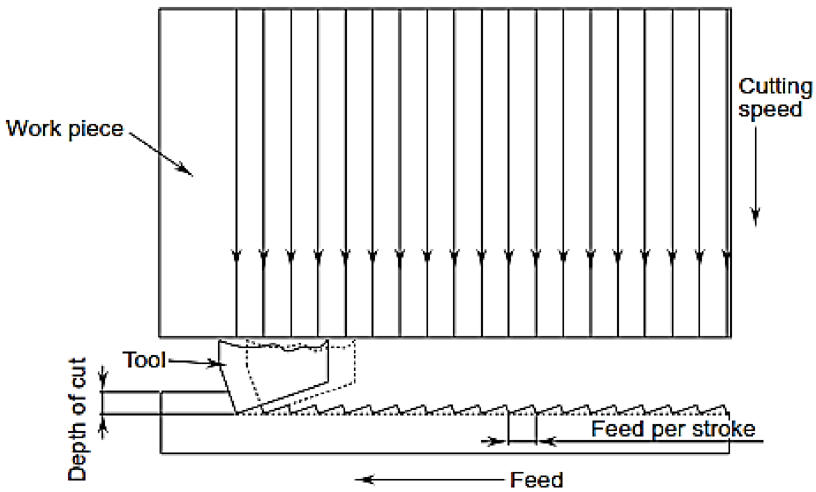


Fig.3 *Generation of a flat surface with a single point tool*

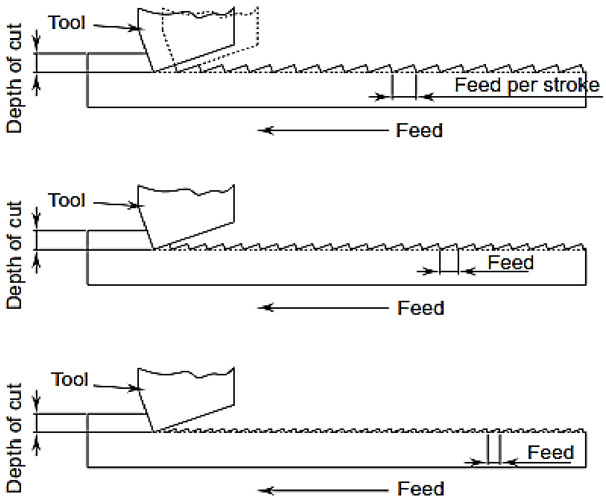


Fig.4 Effect *of feed per stroke on the flat surface generated in shaping*

Quick Return Motion

Cutting speed

The major motions required in a shaper are the reciprocating motion of the cutting tool and the auxiliary motion. During the forward motion of the tool cutting is performed while during the return stroke the tool will simply be sliding. Hence it is necessary to reduce the idle time of the machine to make the return stroke faster compared to the forward motion. This is generally accomplished by means of a quick return mechanism. One typical mechanism used in shapers is the quick return mechanism

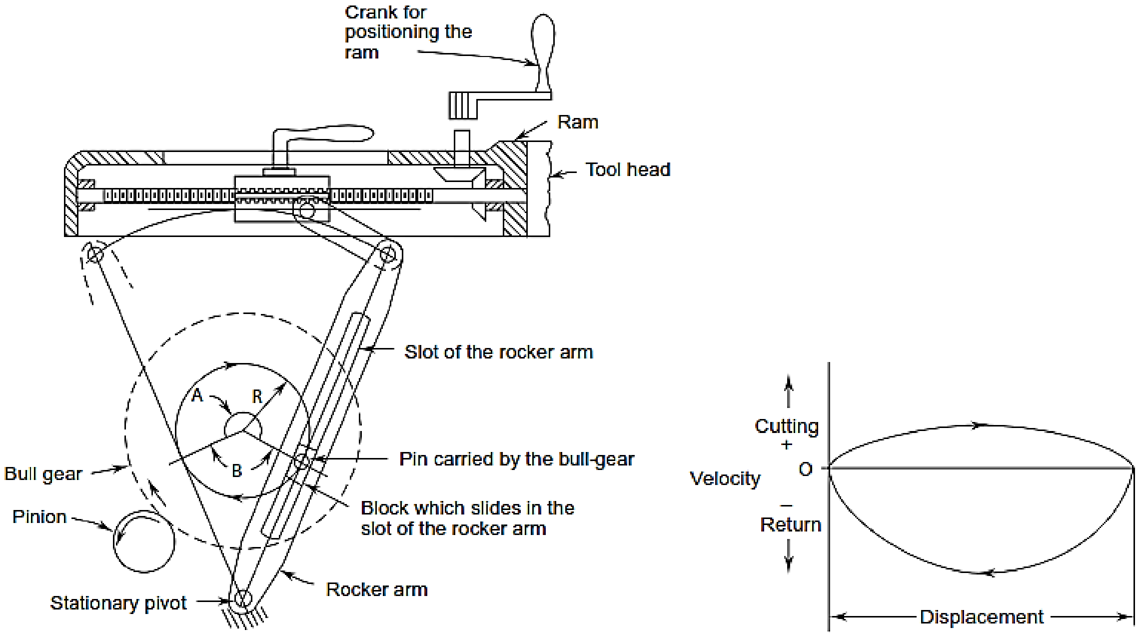


Fig.5 Quick return motion in a crank shaper

Shaper Specification

Shapers can be specified by means of a number of parameters as follows:

- Maximum length of stroke, mm
- Maximum table size, length, mm X width, mm X height, mm
- Maximum table travel, length, mm X width, mm
- Maximum power of the drive motor used in the machine, kW
- Range of cutting speeds, strokes/min
- Range of feeds, mm/stroke
- Maximum weight of the machine
- Maximum dimensions of the machine for installation (Floor space)

Types of shaper

- Universal shaper - similar to the mechanical shaper, but has a table that can be tilted about an axis parallel to the ram ways as well as an axis that is perpendicular to the ram ways.
- Draw cut shaper - Similar to the mechanical shaper, but built much heavier. The heavier construction helps it to take much deeper cuts without any chatter thereby increasing the material removal rate. Another difference is that the cutting stroke is when the ram moves towards the body rather than the away motion, as is most common in the mechanical shapers.
- Vertical shaper - similar to a mechanical shaper except the reciprocating axis here is vertical. This is similar to a slotter in action and often used interchangeably. The vertical shaper is a much smaller version of the slotter. The stroke of the ram is generally limited below 300 mm.

Hydraulic Shaper

The mechanical shaper has the problem of inertia of the main drive components, which require some time for reversal for every stroke and as a result, a large proportion of time is spent with the tool cutting air.

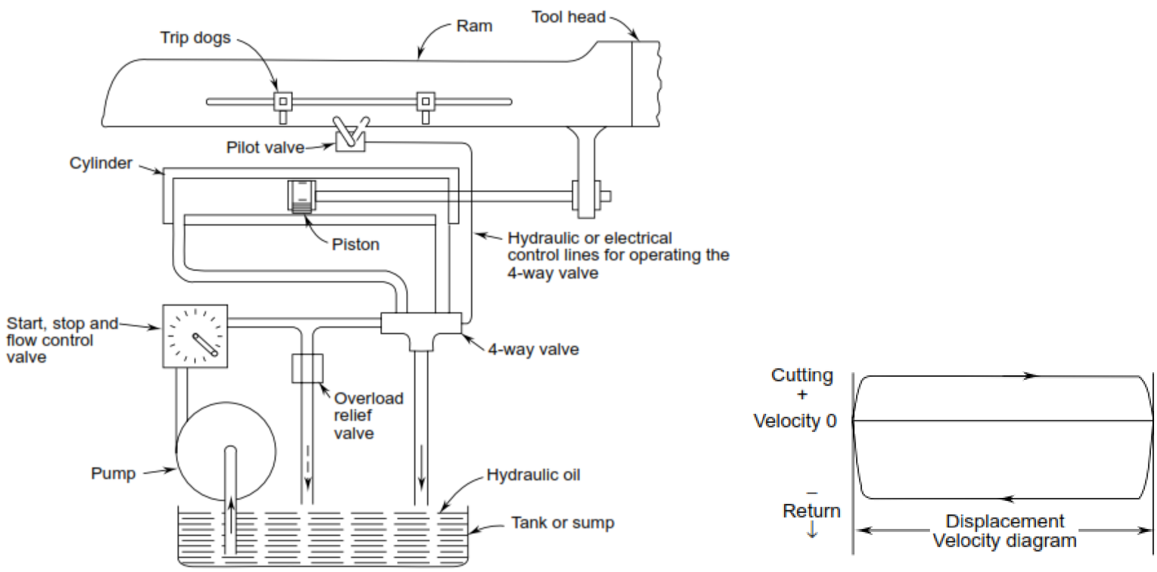


Fig.6 The functioning of a hydraulic shaper

Advantages of hydraulic shaping

1. Cutting speed remains constant throughout most of the cutting stroke unlike the crank shaper where the speed changes continuously.

2. Since the power available remains constant throughout, it is possible to utilise the full capacity of the cutting tool during the cutting stroke.

3. The ram reverses quickly without any shock due to the hydraulic cylinder utilised. The inertia of the moving parts is relatively small.

4. The range and number of cutting speeds possible are relatively large in hydraulic shaper.

5. More strokes per minute can be achieved by consuming less time for reversal and return strokes.

Disadvantages

1. It is more expensive compared to the mechanical shaper.

2. The stopping point of the cutting stroke in hydraulic shaper can vary depending upon the resistance offered to cutting by the work material.

Work Holding in Shaping

The work pieces which are small are normally held in a vice fixed to the shaper table. Some other work holding arrangements such as angle plates, strap clamps and support elements can also be used. Their use depends upon the work piece shape and the geometry to be generated.

Cutting Tools Used

A large variety of single point cutting tools with various approach and edge geometries can be used in the shaper. Single point cutting tools similar to the types used in lathe are used in shapers too. The main difference is that the clearance angle required will have to be properly ground into the tool as there is no adjustment possible in a shaper's clapper box arrangement. Both the right hand left hand tools are used in shapers. Since the cutting speeds used are relatively low, high speed steel is the most widely used cutting tool material for shaping tools.

Shaping Time and Power Estimation

The stroke of the mechanical shaper should be adjusted in such a way that the tool starts the cutting stroke a small distance before the work piece and completes the stroke after a further small distance. This distance is necessary to give sufficient time for the reversal of motion and also the cutting does not take place at a very low speed, which is the case at the beginning and end of the stroke. The approach distance and over travel each can be taken as 15 to 25 mm. The cutting speed in the case of shaping is the speed of the cutting tool in the forward direction during actual cutting. In the case of mechanical shaper it is the average speed. Let N be the rotational speed of the bull gear and L be the length of the stroke. The speed ratio indicates the proportion of time actual cutting is taking place and is defined as

$$\text{The speed ratio, } r = \frac{\text{Time for forward stroke}}{\text{Time for return stroke}} = \frac{N_f}{N_r} = \frac{3}{2} \text{ (normally)}$$

The value of r for typical mechanisms is about 1.5. This means that the time for completing the stroke is

$$\text{Time for completing the cutting stroke} = \frac{N_f}{N \times (N_f + N_r)}$$

The cutting speed is the speed of the ram in the cutting direction.

$$\text{Thus the cutting speed, } V = \frac{L \times N\,(N_r + N_f)}{N_f}$$

The time for completing one stroke, T is

$$T = \frac{L}{N} \text{ minutes}$$

Feed in shaping is the small lateral movement given to the tool in a direction perpendicular to the cutting speed direction. It is given before the beginning of each cutting stroke. This feed is specified as mm/stroke.

Number of strokes, S_N required for removing one layer of material from the surface of the work piece depends upon the width of the work piece, W and the feed rate, f employed.

$$\text{Number of strokes required, } S_N = \frac{W}{f}$$

$$\text{Total machining time} = T \times S_N$$

Example

A shaper is operated at 120 cutting strokes per minute and is used to machine a work piece of 250 mm in length and 120 mm wide. Use a feed of 0.6 mm per stroke and a depth of cut of 6 mm. Calculate the total machining time to for machining the component. If the forward stroke is completed in 230°, calculate the percentage of the time when the tool is not contacting the work piece.

Solution Let the approach distance = 25 mm

Length of stroke, $L = 250 + 25 = 275$ mm

Number of strokes required, $S_N = \frac{120}{0.6} = 200$

The time for completing one stroke, T is

$$T = \frac{275}{120} = 2.292 \text{ minutes}$$

Total machining time = $2.292 \times 200 = 458.33$ minutes

The forward stroke is during 230°.

Percentage of time when tool is not cutting = $\frac{360 - 230}{360} = 36.11\%$

The cutting speed, $V = \frac{275 \times 120 \times 360}{1000 \times 230} = 51.65$ m/min

Cutting tool Materials

- A large variety of single point cutting tools with various approach and edge geometries can be used in the shaper.
- Single point cutting tools similar to the types used in lathe are used in shapers too.
- The main difference is that the clearance angle required will have to be properly ground into the tool as there is no adjustment Possible in a Shapers clapper box arrangement.
- Both the right hand left hand tools are used in shapers.
- The cutting speeds used are relatively low, high speed steel is the most widely used cutting tool material for shaping tools.

Planing Machine (planer)

- Planing machine is very similar to the shaper and used for machining large work pieces
- In the shaper, the cutting tool reciprocates during the cutting motion, but in planer work table reciprocates.
- Feeding motion in the planer is given to the cutting tool, which remains stationary during the cutting motion.

The main constructional pieces of a planer are:

- Bed
- Table

- Column
- Cross Rail
- Tool head

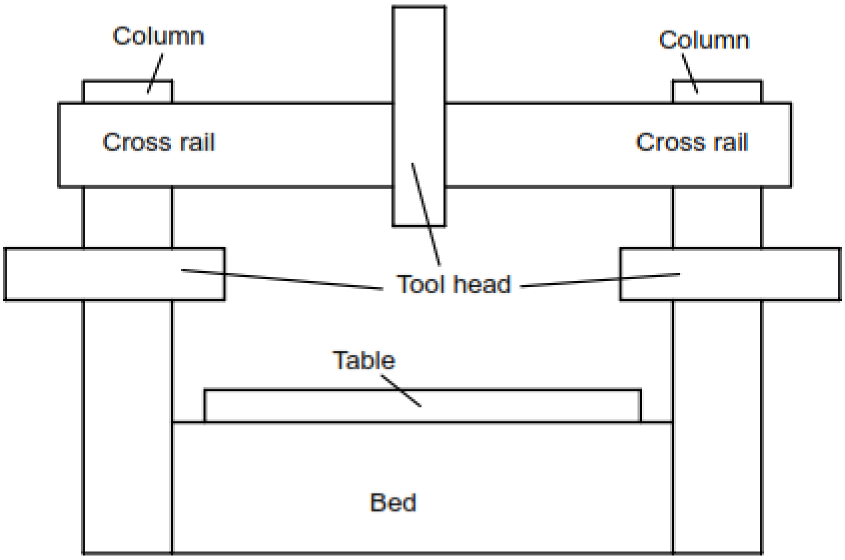


Fig.7 Planer construction

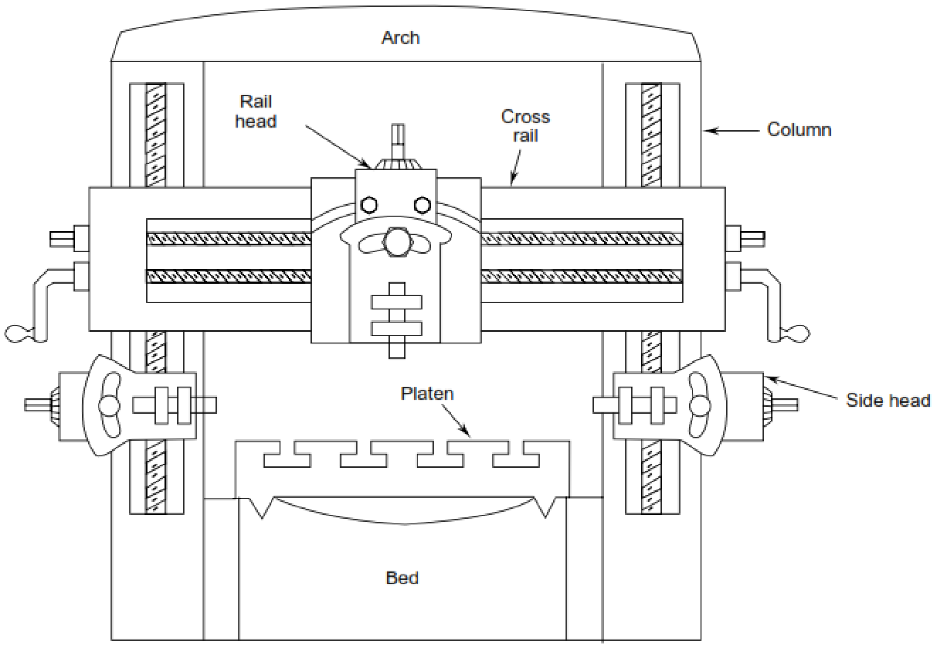


Fig.8 Planer construction

Bed The bed of a planer, generally made of cast iron, is large in size and consequently is heavier to support the heavy machine. Levelling jacks, or pads, are included on the bottom of the bed to provide a means of levelling it during installation. The bed is a little more than twice the length of the table to provide the full reciprocating motion for the entire part mounted on the table.

Guide ways both V and flat as described in chapter 3 are used depending upon the size of the planer are present along the entire length of the bed.

Table The table is the platform where the work will be setup for machining. The table supports and reciprocates the heavy work pieces during the cutting action. The table moves back and forth on the bed, carrying the work past the stationary tool T-slots are provided on the entire length of the table for securely fastening the work pieces to the table. In addition to the T-slots, a row of holes are provided on the table to accept stop pins that prevent the movement of the work under the heavy cuts common to the planer. To reverse the table automatically at the end of each stroke adjustable dogs are provided on the side of the table.

Tool feed is normally adjusted to occur during the return stroke of the work. Similar to a shaper, the planer also has a quick return mechanism to save on idle time during the return stroke of the table. Column On either side of the bed two heavily ribbed columns are located. Two tool heads can be mounted, one on each of these columns. Within the columns are the various mechanisms that transmit power to the upper parts of the machine from the main drive motor.

Cross Rail A cross rail is connected to the two columns to help in accommodating a third tool head. If necessary an additional tool head can be mounted on the cross rail. The rail can be clamped at any position on the columns by means of hand or power clamps. The tool heads can be fed manually or more generally by automatic feeding arrangement with the power coming from the motor and feed gearbox that is housed within the table.

Tool head The tool head in planer is similar in construction to the clapper box of a shaper and is mounted on the cross rail through a saddle. The tool head can be moved along the cross rail for the feeding action while the depth of cut can be controlled by moving the tool downwards. The tool head can be tilted for any given angle cut on the work piece. The tool heads will be provided with counterweights for force equalization and smooth movement. It is possible to mount more than one tool head on the cross rail as well as on the columns on both sides, so that multiple surfaces can be completed simultaneously.

This helps in reducing the total machining time since planing is a relatively slow operation like shaping.

The types of surfaces that are generally produced in planers are:

- Flat surfaces either in the horizontal plane or vertical plane
- Curved surfaces
- Any inclined plane
- Slots, grooves and dovetails

Typical examples of parts machined on a planer include the following: large castings, bases and tables for different types of machine tools, lathe beds, frames for printing presses, textile machines, forging-hammer die blocks, large fixtures and moulds, rolling-mill parts, and parts for large hydraulic presses. The work piece during the machining operation will have to remain fixed to the table. Since heavy cuts are made in planing, the cutting forces are also large and the work piece should not be shifted due to these forces. Most of the time complex work pieces will be machined, so the setting up process requires heavy duty T-bolts, clamps, angle plates, planer jacks, step blocks and stops to cater to the wide variety of geometries that can be handled. The cutting tools used in planers in general are single point cutting tools similar to shapers. However in view of the heavy cuts taken these normally have large cross section and are more rigid. If the entire tool is made from high speed steel, it becomes expensive. Tool holders made from less expensive heat-treated steel may be used to hold the high speed steel bits that are used for actual cutting.

Types of Planers

The type of planer that is common and described above is the double housing planer. Some other types of planers are as follows:

Open Side Planer Open side planer consists of only one vertical column or housing with the cross rail mounted as a cantilever. The column and the cross rail carry tool heads. This type of machine **Planer-miller** is suited for oversize work pieces.

Planer-miller This is a hybrid of a planer and a milling machine (Chapter 7), but is generally considered as a planer because of the size and construction. The construction is similar to a double housing planer, with a tool head which is different from the conventional planer tool head with a single point tool. The miller head holds a tool similar to a shell end mill driven by its own motor. The main advantage of this type of tool head is that more power can be provided to the tool so that higher material removal rates can be achieved.

Pit Planer A pit planer is used for work pieces that are larger than the ones taken by normal planers. In this the table and the work piece remain stationary while the cutting tool reciprocates across the work surface. The work piece is either mounted on a stationary table under the rail, or on the factory floor. No clamping of the work piece is required because of the massive sizes involved. One or two tool heads can be mounted on the cross rail as required. The tool heads will be travelling along the cross rails driving the cutting tool past the work surface during operations.

Divided Table Planer (Duplex Table Planner) The planer work in general calls for a lot of time in setting up the work piece because of the massive sizes involved. This type of planer has two work tables, such that when machining is taking place on one table the other work piece can be set on the other table, thereby increasing the productivity of the planer. It is also possible to join the two tables together for really large work pieces.

Types of Planers

The type of planer that is common and described above is the double housing planer. Some other types of planers are as follows:

Open Side Planer Open side planer consists of only one vertical column or housing with the cross rail mounted as a cantilever. The column and the cross rail carry tool heads. This type of machine is suited for oversize work pieces.

Planer-miller This is a hybrid of a planer and a milling machine, but is generally considered as a planer because of the size and construction. The construction is similar to a double housing planer, with a tool head which is different from the conventional planer tool head with a single point tool. The miller head holds a tool similar to a shell end mill driven by its own motor. The main advantage of this type of tool head is that more power can be provided to the tool so that higher material removal rates can be achieved.

Pit Planer A pit planer is used for work pieces that are larger than the ones taken by normal planers. In this the table and the work piece remain stationary while the cutting tool reciprocates across the work surface. The work piece is either mounted on a stationary table under the rail, or on the factory floor. No clamping of the work piece is required because of the massive sizes involved. One or two tool heads can be mounted on the cross rail as required. The tool heads will be travelling along the cross rails driving the cutting tool past the work surface during operations.

Divided Table Planer (Duplex Table Planner) The planer work in general calls for a lot of time in setting up the work piece because of the massive sizes involved. This type of planer has two work tables, such that when machining is taking place on one table the other work piece can be set on the other table, thereby increasing the productivity of the planer. It is also possible to join the two tables together for really large work pieces.

Planing Time Estimation

The time required to take one cut over a given surface in planing can be calculated based on the length of stroke, width of the surface to be cut, feed per stroke, cutting speed and table return speed. The approximate time can be calculated from the formula:

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right)$$

Where l = length of the planer stroke, m

W = width of surface to be cut, mm

f = feed rate, mm/stroke

V_c = cutting speed, m/ min

V_r = table return speed, m/ min

a = time for reversal of table, min which is about 0.015 to 0.040 minutes

Table 1 Starting speed and feed for planners

Work Material	HSS Tool	
	Speed, m/min	Feed, mm/Stroke
Aluminium	60–90	3.00
Brass (soft)	45–75	6.30
Bronze (medium)	20–40	1.90
Bronze (hard)	10–20	1.25
Cast iron (soft)	15–25	3.00
Cast iron (hard)	10–15	1.50
Malleable iron	15–30	2.25
Cast steel	10–20	1.25
Steel (soft)	20–30	1.25
Steel (medium)	15–20	1.50
Steel (hard)	5–10	0.90

Example

The flat surface of a large cast iron part measuring 2 m × 1 m × 300 mm is to be machined using a planer along its face (2 m × 1 m). Estimate the machining time taking approach as well as over travel as 20 mm each. Take cutting speed as 20 m/min, return speed is 40 m/min and a machining allowance on either side of plate width is 5 mm and feed is 1 mm/stroke.

Solution Given the approach and over travel distance = 20 mm

Length of stroke, $l = 20 + 2000 + 20 = 2040$ mm = 2.04 m

Width of the plate to be completed = 5 + 1000 + 5 = 1010 mm

Given Feed, $f = 1$ mm/stroke

Cutting speed, $V_c = 20$ m/min; $V_r = 40$ m/min

Take $a = 0.02$ minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1010}{1}\left(\frac{2.04}{20} + \frac{2.04}{40} + 0.02\right) = 174.73 \text{ minutes}$$

$$\text{Neglect the allowances, Planing time, } t = \frac{1000}{1}\left(\frac{2}{20} + \frac{2}{40} + 0.02\right) = 170 \text{ minutes}$$

Example

Estimate the time required to machine a grey cast iron part measuring 6 m long × 1.25 m wide on a double housing planer. Cutting tools used are made of carbide which can take a cutting speed of 80 m/min and return speed is 160 m/min. To finish the part it requires two rough cuts with a feed of 1.5 mm/stroke and two finish cuts with a feed of 0.5 mm/stroke. Assume the time for reversing the table is 0.02 minutes.

Solution Length of stroke, $L = 6$ m

Width of the plate to be completed = 1.25 m = 1250 mm

Roughing Passes:

Given Feed = 1.5 mm/stroke

Cutting speed, v = 80 m/min; Return speed = 160 m/min

Take a = 0.02 minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1250}{1.5}\left(\frac{6}{80} + \frac{6}{160} + 0.02\right) = 110.417 \text{ minutes}$$

Finishing Passes:

Given Feed = 0.5 mm/stroke

Cutting speed, v = 80 m/min; Return speed = 160 m/min

Take a = 0.02 minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1250}{0.5}\left(\frac{6}{80} + \frac{6}{160} + 0.02\right) = 331.25 \text{ minutes}$$

Total machining time = 2 × 110.417 + 2 × 331.25 = 883.333 minutes = 14.72 hours

Slotting Machine (Slotter)

- Slotting machine is basically a vertical axis shaper.
- ram strokes up to 1800 mm long.
- Generally keyways, splines, serrations, rectangular grooves and similar shapes are machined in a slotting machine.
- The stroke of the ram is smaller in slotting machines than in shapers to account for the type of the work that is handled in them.

  The main constructional pieces of a slotter are:

- Base
- Table
- Column
- Ram

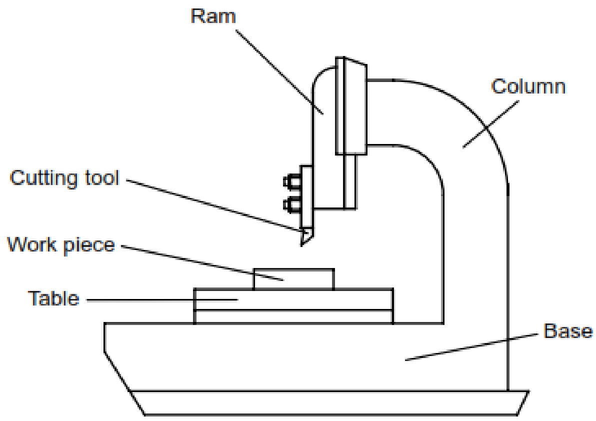


Fig.9 Schematic of a typical slotting machine

Base Similar to a shaper and planer the base of the slotter is a heavy structure to support all the weight of the machine tool and the accompanying cutting forces. Since the cutting force in slotting is directed against the table, the base of the machine is rigidly built. Precision guide ways are provided on top of the base for the cross-slide to move.

Table Table is generally a circular one similar to the rotary table of a milling machine. T-slots are cut on the table to facilitate the fixing of work pieces utilizing various fixturing elements such as T-bolts, clamps, etc. Tables on slotters can be rotated as well as moved longitudinally or transversely. With such flexibility in the feed direction, a slotter can cut any type of groove, slot, or keyway.

Column The column of a slotter is a support structure to the cutting tool and its reciprocating motion. It is also massive and houses the power and drive mechanism used for the reciprocation of the cutting tool.

Ram The ram holds and supports the tool head during the cutting action. Since gravity acts on the ram during its upward travel, a counterweight is added to equalize the power requirements on the upward and downward strokes. This will provide a smooth action to the machine. The actual cutting takes place during the downward motion of the tool. The stroke length can be adjusted suitably depending upon the part.

The types of tools used in slotter are very similar to that of a shaper except that the cutting actually takes place in the direction of cutting. However in view of the type of surfaces that are possible in the case of slotter, a large variety of boring bars or single point tools with long shanks are used.

Table 2 Comparison of shaper, planer and slotter

		Shaper	Planer	Slotter
1	Work-Tool motion	Tool reciprocates in horizontal axis and work feeds in intermittently	Work reciprocates in horizontal axis and tool feeds in intermittently	Tool reciprocates in vertical axis and work feeds in intermittently
2	Construction and rigidity	Lighter in construction and less rigid	Heavier in construction and more rigid	Lighter in construction and less rigid
3	Motor Power required	Relatively less power	Higher power compared to shaper	Relatively less power
4	Typical work size and setup time	Relatively small parts. Typical work envelope is: 450 × 450 × 600 mm. Quick setup time.	Bigger parts require lengthy setup time. Typical work envelope is: 3 × 3 × 15 m	Relatively small parts. Typical work envelope is: 450 × 450 × 300 mm. Quick setup time.
5	Number of surfaces that can be machined at a time	Only one surface at a time	Three surfaces can be machined at a time	Only one surface at a time
6	Material removal rate (MRR)	Low MRR	High since multiple tools can work at a time	Low MRR
7	Tool size	Regular size similar to lathe	Bigger size tools that can take higher depth of cut and feed	Regular size similar to lathe
8	Range of speeds and feeds	Smaller range and smaller number of speeds and feeds	Wide range and more number of speeds and feeds available	Smaller range and smaller number of speeds and feeds

Milling

Milling machines are the most widely used machine tools for manufacturing applications. In milling, the work piece is fed into a rotating milling cutter, which is a multi-point tool. It is unlike a lathe, which uses a single point cutting tool. The tool used in milling is called milling cutter.

The milling process is characterised by: Interrupted cutting each of the cutting edges Machined surface removes material for only part of the rotation of the milling cutter. As a result, the cutting edge has time to cool before it removes material again. Thus the milling operation is much cooler compared to the turning operations seen earlier. This allows for much larger material rates.

Small size of chips Though the size of the chips is small, in view of the multiple cutting edges in contact, a large amount of material is removed and as a result the component is generally completed in a single pass unlike the turning process which requires a large number of cuts for finishing. Variation in chip thickness this contributes to the non-steady state cyclic conditions of varying cutting forces during the contact of the cutting edge with the chip thickness varying from zero to maximum size or vice versa. This cyclic variation of the force can excite any of the natural frequencies of the machine tool system and would be harmful to the tool life and surface finish generated.

This is one of the most versatile machine tools. It is adaptable for quantity production as well as in job shops and tool rooms. Versatility of milling is because of the large variety of accessories and tools available with the milling machines. Typical tolerance expected from the process is about ± 0.050 mm.

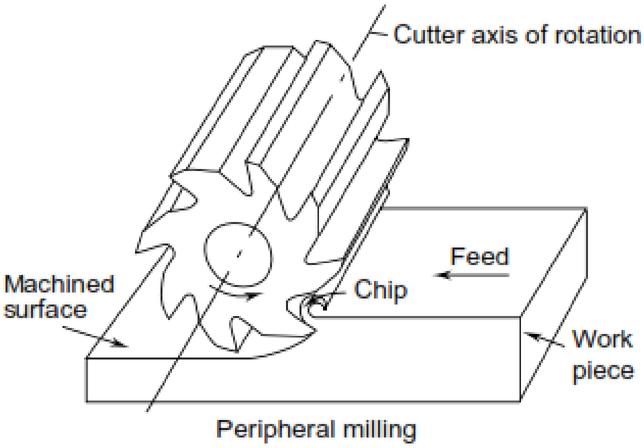


Fig.10 Milling operation

Types of milling machine

The large material removal rates the milling machines come with a very rigid spindle and large power. The varieties of milling machines available are:

(a) Knee and Column type

- Horizontal
- Vertical
- Universal
- Turret type

These are the general purpose milling machines, which have a high degree of flexibility and employed for all types of works including batch manufacturing. A large variety of attachments to improve the flexibility are available for this class of milling machines

(b) Production (Bed) type

- Simplex
- Duplex
- Triplex

These machines are generally meant for regular production involving large batch sizes. The flexibility is relatively less in these machines that suits productivity enhancement.

(c) Plano millers

These machines are used only for very large work pieces involving table travels in meters.

(d) Special type

- Rotary table
- Drum type
- Copy milling (Die sinking machines)
- Key way milling machines
- Spline shaft milling machines

These machines are special class to provide special facilities to suit specific applications that are not catered by the other classes of milling machines.

(a) **Knee and Column type** - commonly used m/c in view of its flexibility and easier setup

Horizontal

Vertical

Universal

Turret type

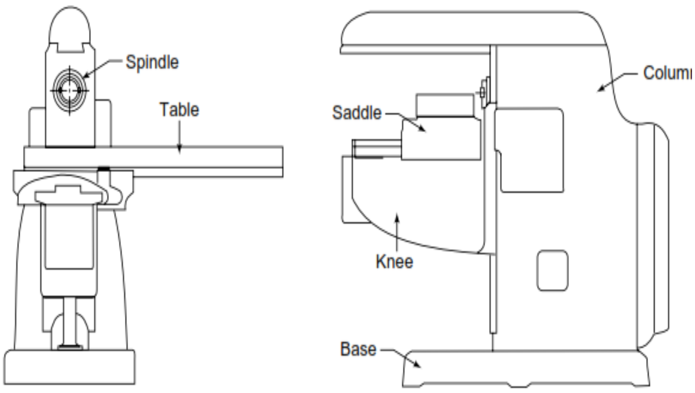


Fig.11 Horizontal knee column type milling machine

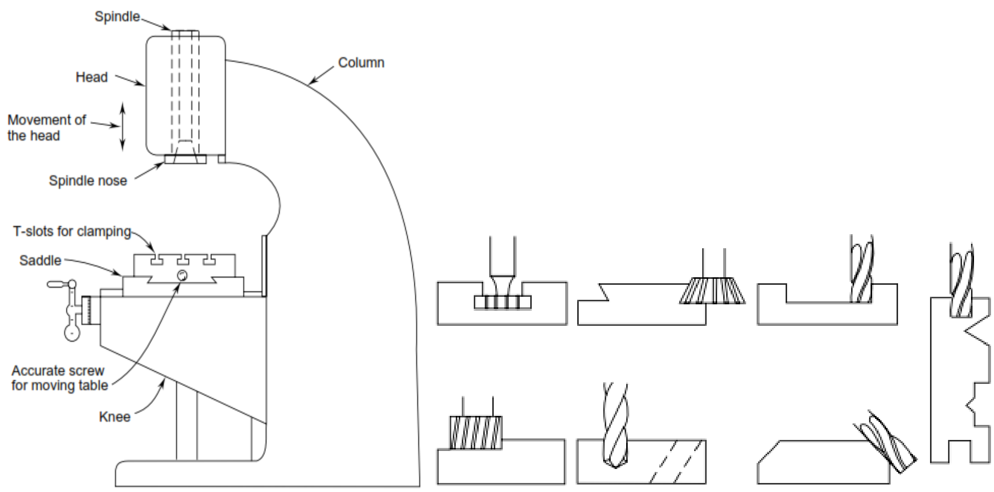


Fig.12 Vertical knee column type milling machine

Bed Type Milling Machine

Column

In production milling machines it is desirable to increase the metal removal rates. If it is done on conventional machines by increasing the depth of cut, chatter is likely to result. Hence another variety of milling machines named as the bed type machines, which are made more rugged and consequently are capable of removing more material. The ruggedness is obtained as a consequence of the reduction in the versatility. The table in the case of bed type machines is directly mounted on the bed and is provided with only the longitudinal motion.

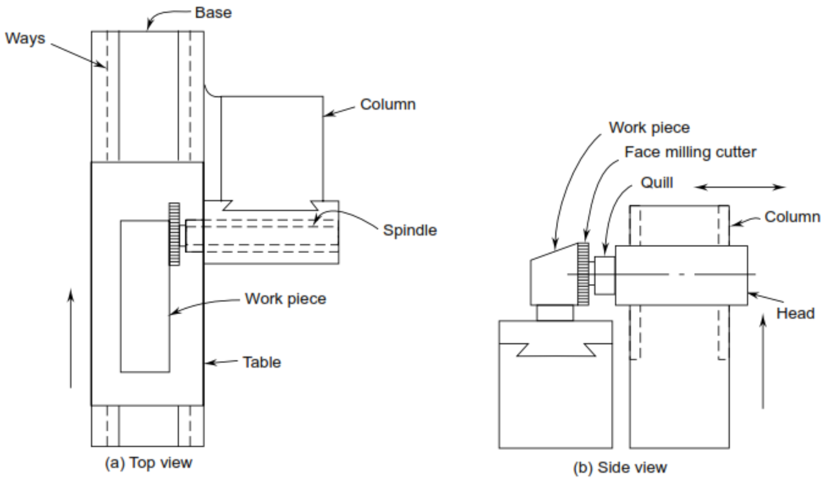


Fig.13 Simplex bed type milling machine

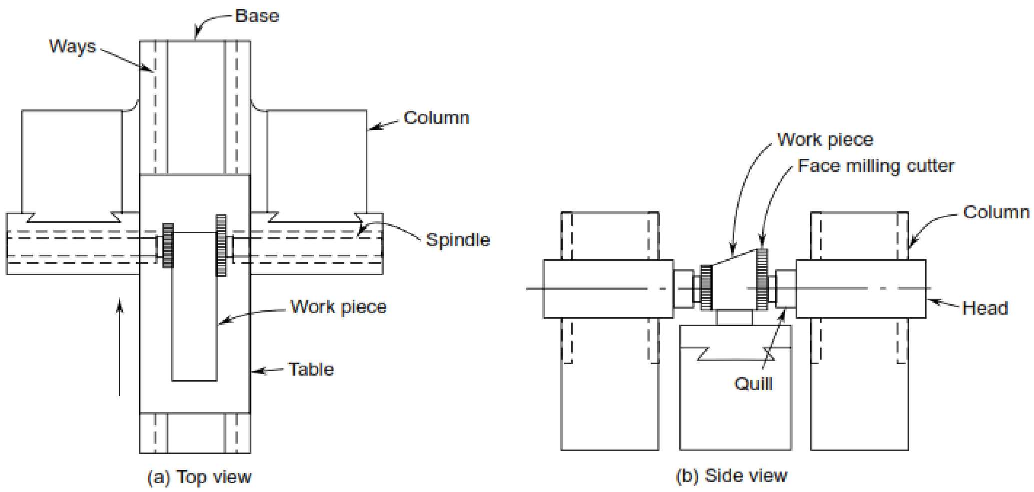


Fig.14 Duplex bed type milling machine

Milling cutters

There are a large variety of milling cutters available to suit specific requirements. The versatility of the milling machine is contributed to a great extent by the variety of milling cutters that are available.

Types

Milling cutters are classified into various types based on a variety of methods.

Based on construction

- Solid
- Inserted tooth type

Based on mounting

- Arbour mounted
- Shank mounted
- Nose mounted

Base on rotation

- Right hand rotation (Counter clockwise)
- Left hand rotation (Clockwise)

Based on helix

- Right hand helix
- Left hand helix

Milling cutters are generally made of high speed steel or cemented carbides. The cemented carbide cutters can be with brazed tip variety or more commonly with indexable tips. The indexable variety is more common since it is normally less expensive to replace the worn out cutting edges than to regrind them.

Plain Milling Cutters

These are also called slab milling cutters and are basically cylindrical with the cutting teeth on the periphery. These are generally used for machining flat surfaces.

- Light duty slab milling cutters generally have a face width
- The heavy duty slab milling cutters come with smaller number of teeth to allow for more chip space.
- Helical milling cutters have a very small number of teeth but a large helix angle.

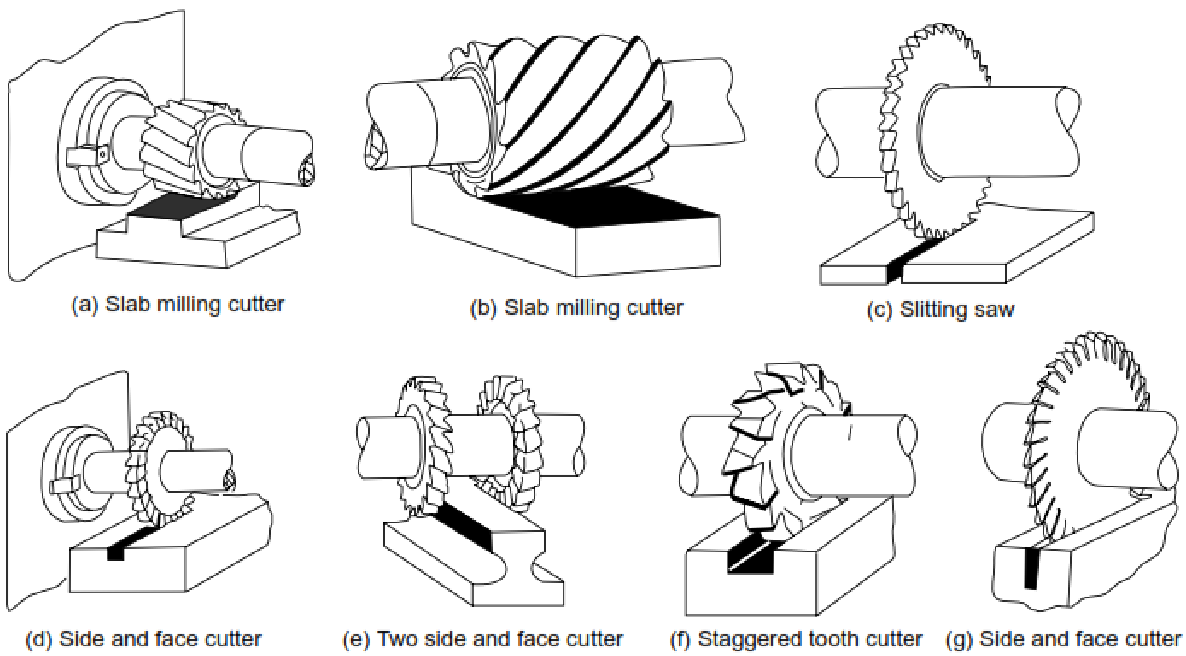


Fig.15 Arbor mounted milling cutters general purpose

Side and Face Milling Cutters

These have the cutting edges not only on the face like the slab milling cutters, but also on both the sides. These cutters become more versatile since they can be used for side milling as well as for slot milling.

Slitting Saws

The slitting saw is very similar to a saw blade. Most of these have teeth around the circumference while some have side teeth as well. The thickness of these cutters is very small and is used for cutting off operation or for deep slots.

Special Form Cutters

There are a large number of special form milling cutters available, which are used for machining specific profiles.

- Angular milling cutters are made in single or double angle cutters for milling any angle such as 30, 45 or 60°.
- Form relieved cutters are made of various shapes such as circular, corner rounding, convex or concave shapes.
- The central slot is to be milled first using an end mill before using the T-slot milling cutter.
- Woodruff key seat milling cutters as the name suggests are used for milling Woodruff key seats. Some other special form cutters are: Dovetail milling cutters, Gear milling cutters

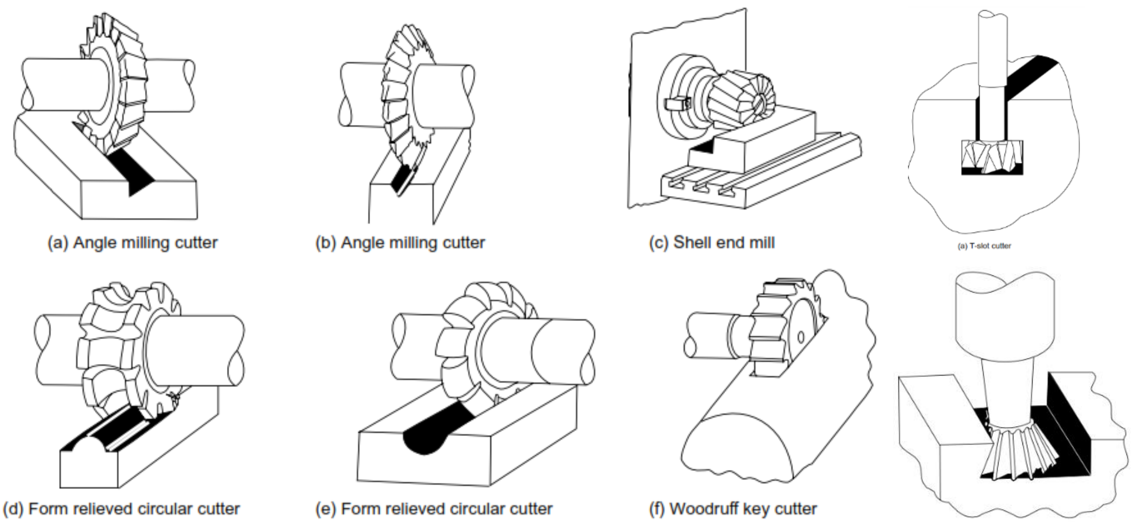


Fig.16 Arbor mounted milling cutters special forms

End Mills

These are shank mounted are generally used in vertical axis milling machines. They are used for milling slots, key ways and pockets where other type of milling cutters cannot be used. A depth of cut of almost half the diameter can be taken with the end mills.

Large size end mills are called shell end mills, which do not have any shank and can be mounted with the help of a central hole. Consequently these can be used in horizontal axis as well as vertical axis milling

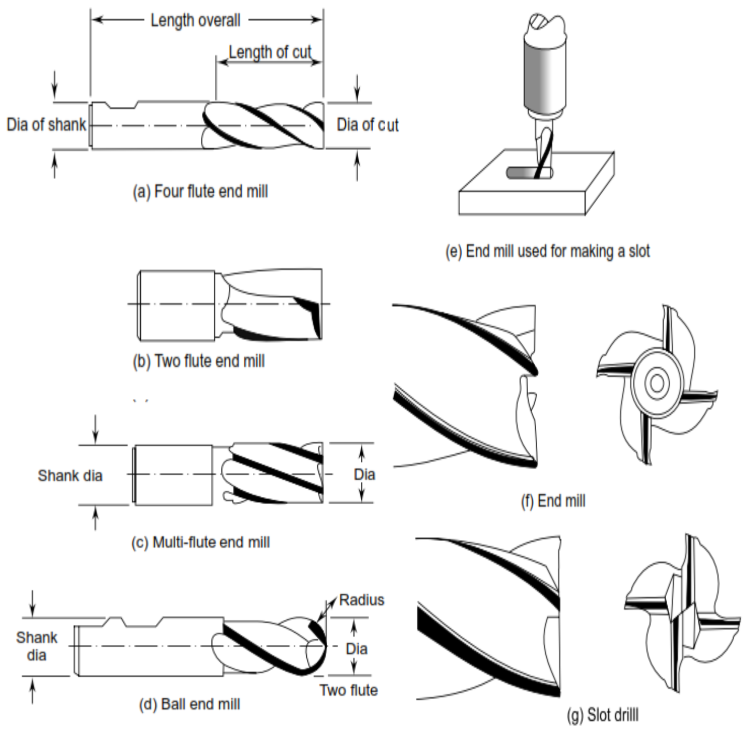


Fig.17 Shank mounted milling cutters, various types of end mills

The cutting edge along the side of an end mill is generally straight and some times can be tapered by grinding on a tool and cutter grinder such that the draft required for mould and die cavities can be automatically generated.

The end face can be square with the side as in the normal case or a ball end shape to be used for milling three dimensional contours such as in die cavities. It can also have a rounded corner for milling special round edged pockets.

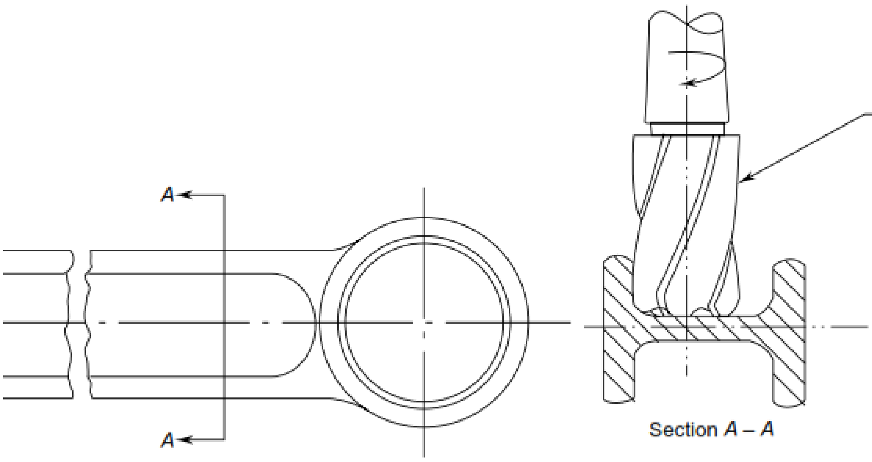


Fig.18 End milling using a corner radius

Face milling cutters are used for machining large, flat surfaces. They have the cutting edges on the face and periphery. It is generally mounted directly on the nose of the spindle

The teeth on the face do most of the machining while those on the side are used for cleaning the surface. These are generally made of carbide insert variety in view of the large material removal involved, though high speed steel types are also used.

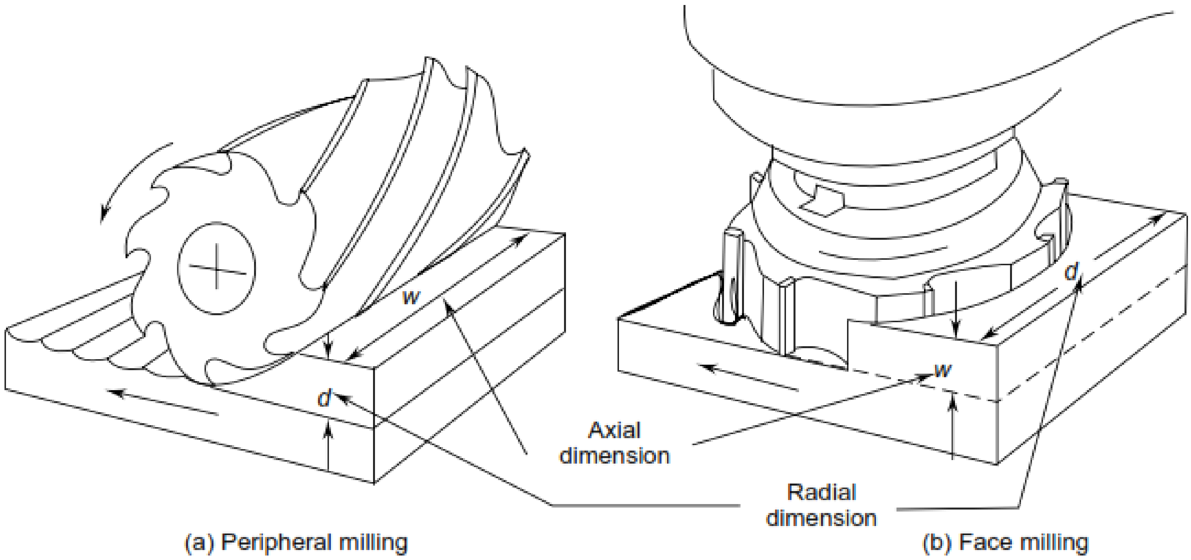


Fig.19 Generating plane surfaces in Milling using face milling

Hand of cut The direction in which the cutter is rotated. When viewed towards the spindle, if the cutter moves counter clockwise it is called right hand rotation while the opposite is called the left hand rotation.

Helical milling cutters, when viewed from the end if the flutes move in a clockwise direction it is called the right hand helix while the opposite is called the left hand helix. The axial cutting force direction depends upon the hand of the helix. If two milling cutters of different helices are arranged side by side in a gang milling operation.

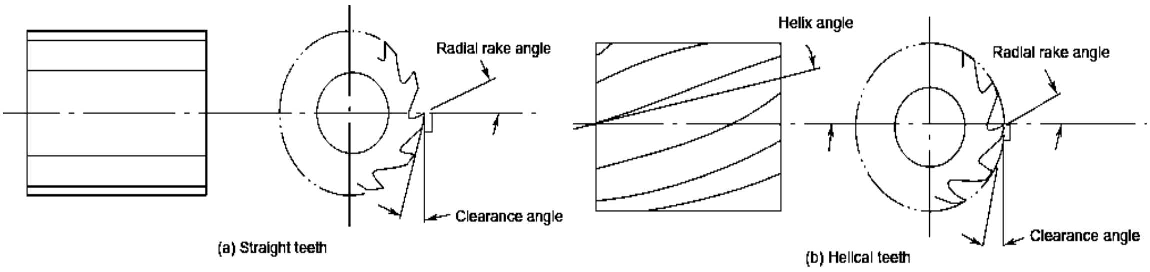


Fig.20 Slab milling cutter, straight and helical

The straight teeth will always have one tooth in contact with the work piece. When the cutting starts, the chip thickness is maximum, which gradually reduces to zero, before the next tooth comes into contact with the work piece. The cutting force rises to maximum value and then rapidly drops to zero before rising again. This force variation gives rise to impact loads on the milling cutter and may induce vibrations.

The helical milling cutter each tooth is longer than the straight tooth. At any given time more than one tooth will be in contact with the work piece each having a different chip thickness.

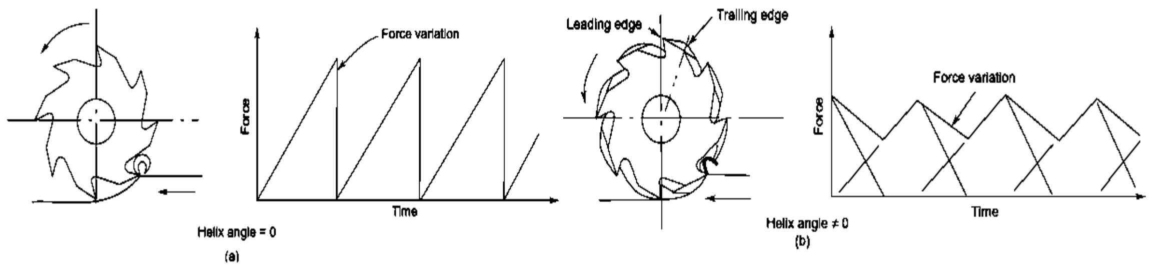


Fig.21 Slab milling cutter, effect of helix angle

Based on the directions of movement of the milling cutter and the feeding direction of the work piece, there are two possible types of milling:

Up milling (conventional milling) - cutting tool rotates in the opposite direction to the table movement. Chip starts at zero thickness and gradually increases to the maximum size

Down milling (Climb milling) - cutting tool rotates in the same direction as that of the table movement. chip starts at maximum thickness and goes to zero thickness gradually

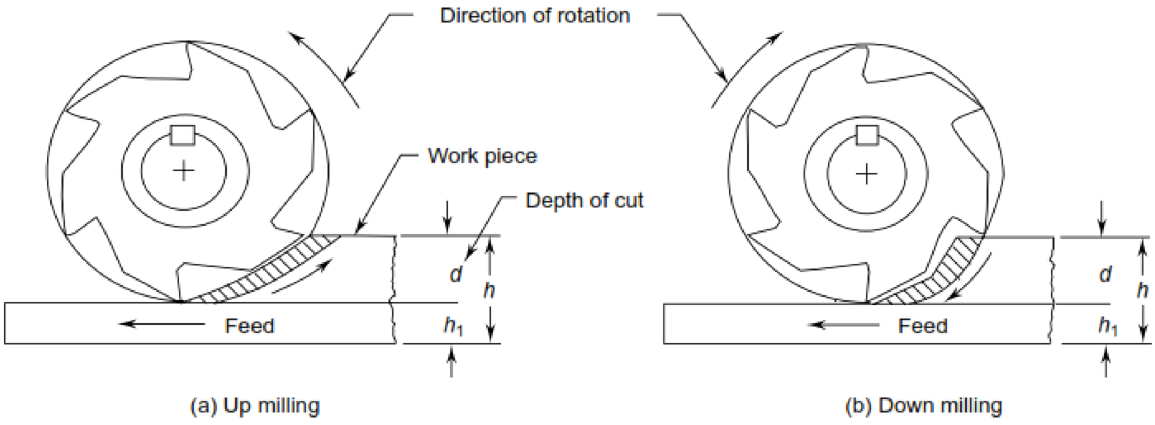


Fig.22 Up Milling and Down milling

Advantages

1. Suited for machine thin and hard-to-hold parts since the work piece is forced against the table or holding device by the cutter.
2. Work need not be clamped as tightly.
3. Consistent parallelism and size may be maintained, particularly on thin parts.
4. It may be used where breakout at the edge of the work piece could not be tolerated.
5. It requires upto 20% less power to cut by this method.
6. It may be used when cutting off stock or when milling deep, thin slots.

Disadvantages

1. It cannot be used unless the machine has a backlash eliminator and the table jibs have been tightened.
2. It cannot be used for machining castings or hot rolled steel, since the hard outer scale will damage the cutter.

Milling operations

Work Holding

- Milling machine table comes with precision parallel T-slots along the longitudinal axis. The work piece therefore can be mounted directly on the table using these T-slots. Alternatively a variety of work holding devices
- Vice is the most common form of work holding device used for holding small and regular work pieces.
- The vice is mounted on the table using the T-slots. A variety of vice jaws are available to suit different work piece geometries.
- Universal chuck is used for holding round work pieces for machining of end slots, splines, etc.

- Fixtures are the most common form of work holding devices used in production milling operations. These become almost a necessity to reduce the setup time and increase the locational accuracy and repeatability.

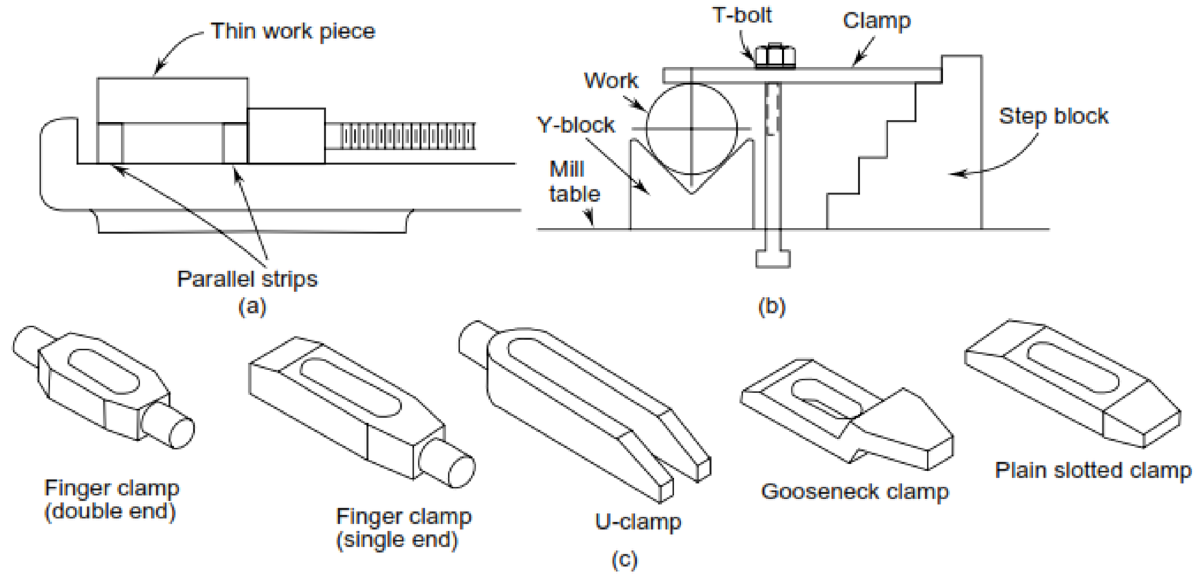


Fig.23 Common work holding methods in Milling

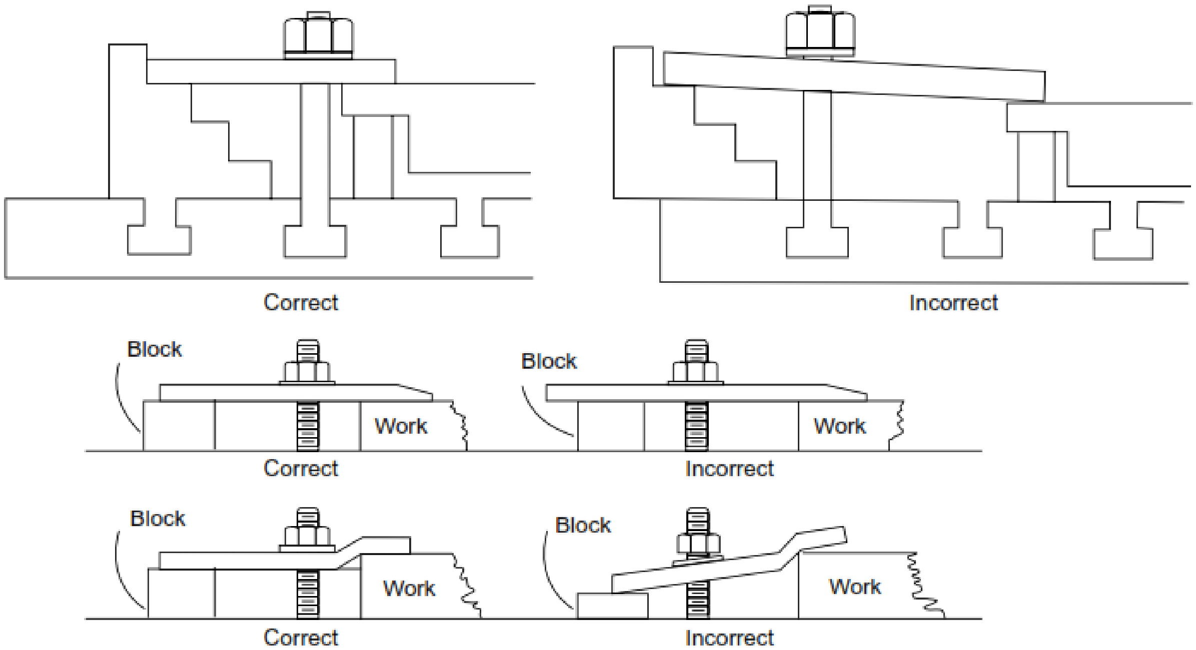


Fig.24 Work holding principles in Milling

Milling Setups

- **String Milling** - small work pieces which are to be milled are fed into the milling cutter one after the other. A number of the work pieces will be kept on the machine table in a line, hence are called string or line milling

- **Abreast or Reciprocal Milling** - is an operation done with special milling fixtures, which have a capability for indexing 180°. While one component is being machined at position 1, at the second position, which is at 180° to the first one, the second component will be loaded. When the machining is completed at location one, the fixtures indexes bring the already clamped component ready for machining.
- **Rotary or Circular Milling** - takes the reciprocal milling to a greater length. A number of fixtures depending upon their size are located on a rotary table such that a number of work pieces can be loaded simultaneously on the machine table.
- **Gang Milling** - a number of milling cutters are fastened to the arbor to suit the profile of the work piece to be machined.
- **Straddle Milling -** Straddle is a special form of gang milling where only side and face milling cutters are used. A typical sequence of processes used and the milling cutters required for a component machined in a milling machine

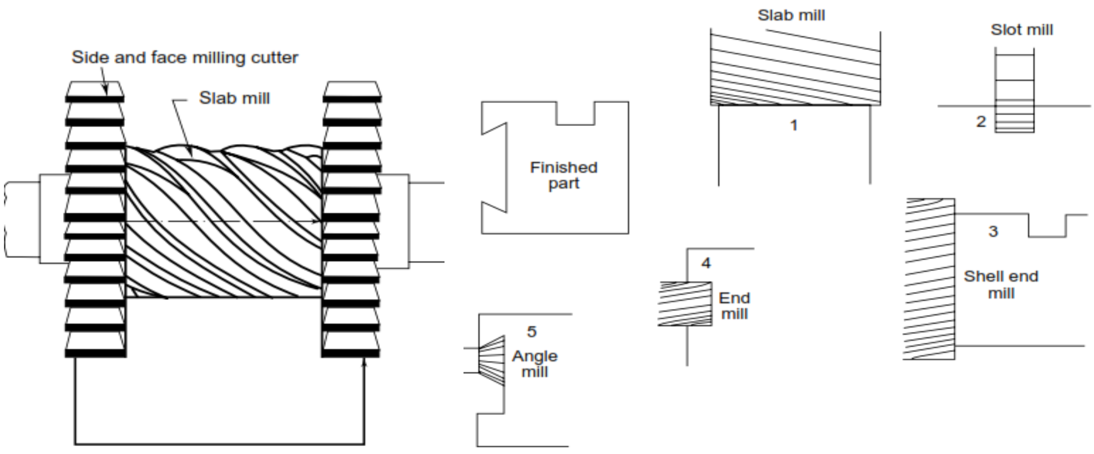


Fig.25 Typical process sequence in milling

Dividing head - is one of the most important attachments with the milling machine. The main spindle of the dividing head drives the work piece by means of a 3-jaw universal chuck or a dog and live centre similar to a lathe. The index plate of a dividing head consists of a number of holes with a crank and pin. The index crank drives the spindle and the live centre through a worm gear, which generally has 40 teeth. A full rotation of the work piece is produced by 40 full revolutions of the index crank.

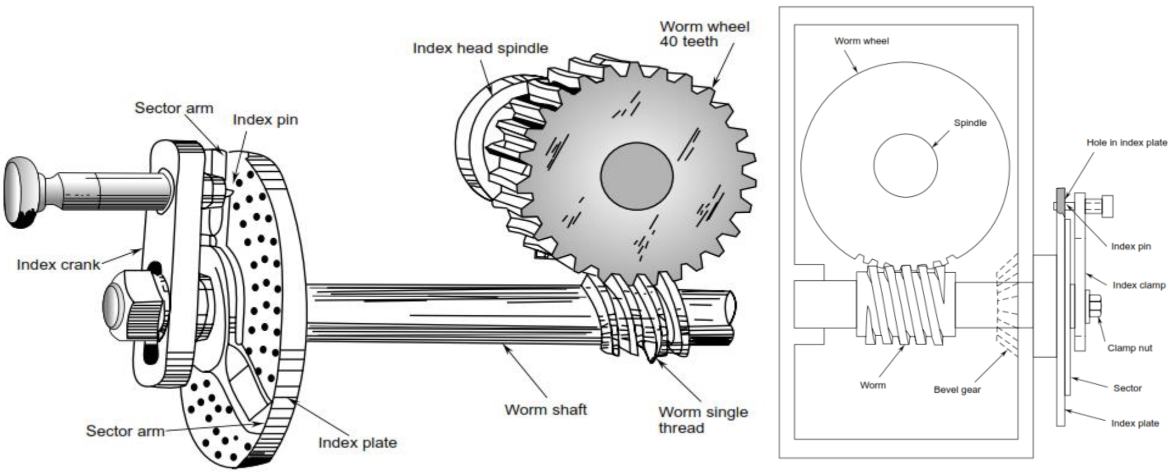


Fig.26 Indexing method of the Dividing head and Dividing head construction

Simple or Plain Indexing - the indexing method carried out using any of the indexing plates in conjunction with the worm. Let us consider a gear that is to be milled with 20 teeth. The gear blank held in the spindle of the dividing head is to be divided equally into 20 divisions. Since 40 revolutions of the index crank produce one full Revolution of the work piece, we need to rotate the index crank two full turns for cutting each tooth of the gear. Suppose we want to have 6 equal divisions to be made.

The rotation of the index crank = $\frac{40}{6} = 6\frac{2}{3}$ turns.

Plate no. 1:	10 holes in 15-hole circle
	12 holes in 18-hole circle
Plate no. 2:	14 holes in 21-hole circle
	18 holes in 27-hole circle
	22 holes in 33-hole circle
Plate no. 3:	26 holes in 39-hole circle

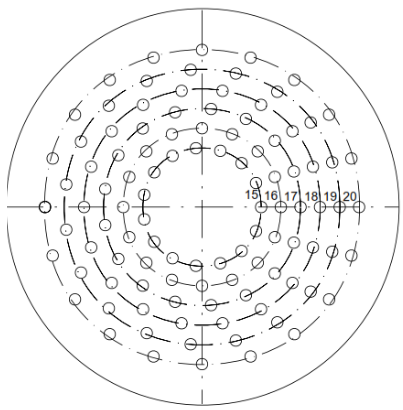


Fig.27 Index plate no. 1 of Brown and Sharpe Dividing head

The index plates available with the Brown and Sharpe milling machines are

Plate no. 1: 15, 16, 17, 18, 19, 20 holes

Plate no. 2: 21, 23, 27, 29, 31, 33 holes

Plate no. 3: 37, 39, 41, 43, 47, 49 holes

The index plate used on Cincinnati and Parkinson dividing heads is

Plate 1: Side 1 24, 25, 28, 30, 34, 37, 38, 39, 41, 42 and 43 holes

Side 2 46, 47, 49, 51, 53, 57, 58, 59, 62 and 66 holes

It is also possible to get additional plates from Cincinnati to increase the indexing capability as follows:

Plate 2: Side 1 34, 46, 79, 93, 109, 123, 139, 153, 167, 181, 197 holes

Side 2 32, 44, 77, 89, 107, 121, 137, 151, 163, 179, 193 holes

Plate 3: Side 1 26, 42, 73, 87, 103, 119, 133, 149, 161, 175, 191 holes

Side 2 28, 38, 71, 83, 101, 113, 131, 143, 159, 173, 187 holes

Simple or Plain Indexing

Plain indexing is the name given to the indexing method carried out using any of the indexing plates in conjunction with the worm. With this it is possible to obtain relatively simple divisions. To explain the procedure let us consider a gear that is to be milled with 20 teeth. This means that the gear blank held in the spindle of the dividing head is to be divided equally into 20 divisions. Since 40 revolutions of the index crank produces one full revolution of the work piece, we need to rotate the index crank two full turns for cutting each tooth of the gear.

Suppose we want to have 6 equal divisions to be made.

The rotation of the index crank $= \frac{40}{6} = 6\frac{2}{3}$ turns.

This means that the index crank should be rotated 6 full turns followed by two-thirds of a rotation. The fraction of a rotation required is to be obtained with the help of the index plates as given above. This can be done as follows, using any of the Brown & Sharpe plates.

Plate no. 1: 10 holes in 15-hole circle

12 holes in 18-hole circle

Plate no. 2: 14 holes in 21-hole circle

18 holes in 27-hole circle

22 holes in 33-hole circle

Plate no. 3: 26 holes in 39-hole circle

Compound Indexing - when the available capacity of the index plates is not sufficient to do a given indexing job, the compound indexing method could be used. First the crank is moved in the usual fashion in the forward direction. Then a further motion is added or subtracted by rotating the index plate after locking the plate with the plunger. This is termed as compound indexing.

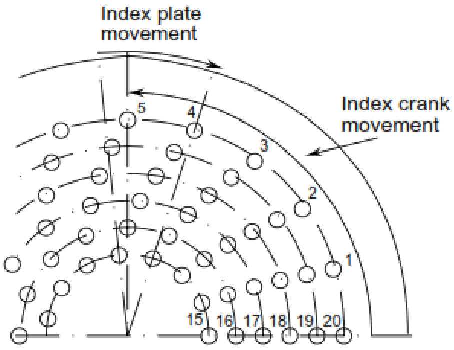


Fig.28 Compound indexing using the Index plate no. 1 of Brown and Sharpe Dividing head with 5 holes in 20-hole circle minus 1 hole in 15-hole circle

Example

Indexing 141 divisions.

Solution The indexing required is $\frac{141}{40}$ considering the worm.

It is necessary to convert this fraction into two fractions corresponding to the two hole circles in the same plate. Use trial and error method to obtain the same.

First step in compound indexing is to factorise the number into suitable hole circles available in a single plate.

$$141 = 47 \times 3$$

This can be achieved by using plate 3 with 39 and 47-hole circles.
The next step is to find the exact indexing required.

$$\frac{X}{39} \pm \frac{Y}{47} = \frac{13 \times 40}{39 \times 47} = \frac{40}{141}$$

$$47 \times X \pm 39 \times Y = 520$$

By trial and error, we get, $X = 26$, and $Y = 18$

$$47 \times 26 - 18 \times 39 = 520$$

$$\frac{26}{39} - \frac{18}{47} = \frac{40}{141}$$

Hence, the indexing required is 26 holes in the 39-hole circle subtracted by 18 holes in the 47-hole circle to get 141 divisions.

Angular Indexing - it is desirable to carry out indexing using the actual angles rather than equal numbers along the periphery, then angular indexing would be useful. The procedure remains the same as the previous cases, except that the angle will have to be first converted to equivalent divisions. Since the 40 revolutions of the crank equals to a full rotation of the workpiece, which means 360°, then one revolution of the crank is equivalent to 9°.

Example

Calculate the indexing for 41°.

Solution Indexing required = $\frac{41}{9} = 4\frac{5}{9}$

This is equivalent to 4 full rotations of the crank followed by 10 holes in the 18-hole circle in plate no. 1.

In **differential indexing**, the index plate is made free to rotate. A gear is connected to the back end of the dividing head spindle while another gear mounted on a shaft, is connected to the shaft of the index plate through bevel gears. When the index crank is rotated, the motion is transferred to the work piece spindle. Since the work piece spindle is connected to the index plate through the intermediate gearing. The index plate will have to be moved in the same direction as the movement of the crank, to add the additional motion

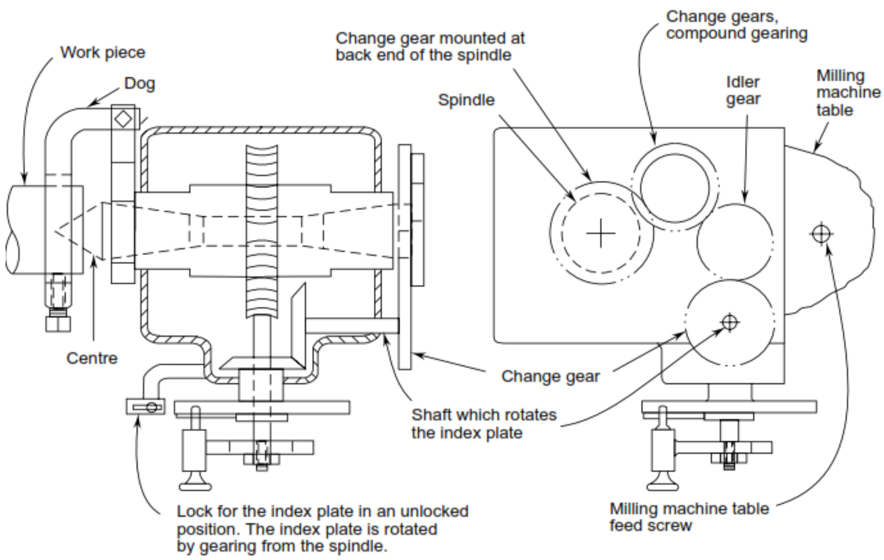


Fig.29 Differential Indexing

Milling time and power estimation

- The cutting speed in milling is the surface speed of the milling cutter

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

Where, V = cutting speed (surface), m/min
D = diameter of the milling cutter, mm
N = rotational speed of the milling cutter, rpm

$$\text{Approach distance, } A = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - d\right)^2} = \sqrt{d\,(D-d)}$$

Where D = diameter of the slab milling cutter
d = depth of cut

$$\text{Time for one pass} = \frac{1 + 2 \times A}{fZN} \text{ minutes}$$

Where Z = number of teeth in the milling cutter
f = feed per tooth, mm

Table 3 Data for milling

Work Material	Hardness	HSS		Carbide	
	BHN	Speed m/min	Feed mm/tooth	Speed m/min	Feed mm/Tooth
C20 Steel	110–160	20	0.13	90	0.18
C35 Steel	120–180	25	0.13	80	0.18
C50 Steel	160–200	20	0.13	60	0.18
Alloy Steel	180–220	30	0.10	60	0.18
Alloy Steel	220–300	18	0.08	90	0.18
Alloy Steel	220–300	14	0.08	60	0.15
Alloy Steel	300–400	14	0.05	60	0.13
Stainless steel	200–300	20	0.10	85	0.13
Cast iron	180–220	16	0.18	58	0.20
Malleable iron	160–240	27	0.15	85	0.18
Cast steel	140–200	16	0.15	50	0.18
Copper	120–160	38	0.15	180	0.15
Brass	120–180	75	0.28	240	0.25
Bronze	160–200	38	0.18	180	0.15
Aluminium	70–105	120	0.28	240	0.25
Magnesium	40– 60	210	0.28	380	0.25

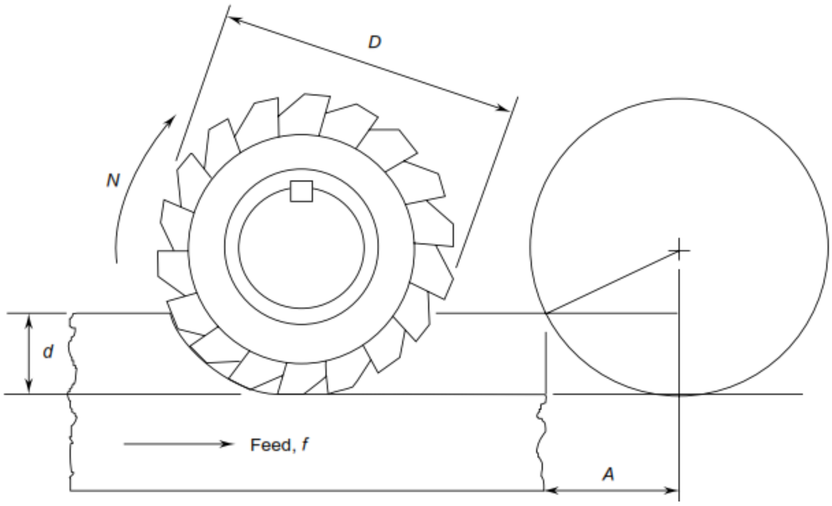


Fig.30 Slab milling operation

Example

A surface 115 mm wide and 250 mm long is to be rough milled with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill 150 mm in diameter. The work material is alloy steel (200 BHN). Estimate the cutting time.

Solution Given $Z = 16$

$D = 150$ mm

$d = 6$ mm

$W = 115$ mm

From the table,

Cutting speed, $V = 60$ m/min

Feed rate, $f = 0.18$ mm/tooth

Spindle speed, $N = \dfrac{1000 \times 60}{\pi \times 150} = 127.32 \approx 125$ rev/min

Since $W < \dfrac{D}{2}$

Approach distance, $A = \sqrt{115(150 - 115)} = 63.44 \approx 65$ mm

Time for machining $= \dfrac{250 + 2 \times 65}{0.18 \times 16 \times 125} = 1.06$ minutes

Milling Power Estimation

Milling power is proportional to the material removal rate. Material removal rate (Q) in milling is given by

$$Q = \frac{f_m \, w \, d}{60000} \text{ cm}^3\text{/s}$$

Where f_m = feed rate in mm/min = fZN

w = width of cut in mm

d = depth of cut in mm

f = feed rate in mm/tooth as normally given in cutting tables

Z = number of teeth in the milling cutter

N = rotational speed of the spindle in rpm

Milling power (P_m) in horse power units at the cutting tool is given by

$$P_m = K_p Q C W hp$$

Where P_m = milling power in hp

K_p = power constant as given in Table 7.2.

C = Feed factor given in Table 7.3

W = Tool wear factor given in Table 7.4

Table 4 Power constant for Milling

Work Material	Hardness BHN	Power Constant
Plain carbon Steel	100–120	1.80
	120–140	1.88
	140–160	2.02
	160–180	2.13
	180–200	2.24
	200–220	2.32
	220–240	2.43
Alloy Steel	180–200	1.88
	200–220	1.97
	220–240	2.07
	240–260	2.18
Cast iron	120–140	0.96
	140–160	1.04
	160–180	1.42
	180–200	1.64
	200–220	1.94
	220–240	2.48
Malleable iron	150–175	1.15
	175–200	1.56
	200–250	2.24
	250–300	3.22

Table 5 Feed factors for power calculation

Feed, mm/Tooth	Feed Factor	Feed, mm/Tooth	Feed Factor
0.02	1.70	0.22	1.06
0.05	1.40	0.25	1.04
0.07	1.30	0.28	1.01
0.10	1.25	0.30	1.00
0.12	1.20	0.33	0.98
0.15	1.15	0.35	0.97
0.18	1.11	0.38	0.95
0.20	1.08	0.40	0.94

Table 6 Tool wear factors for power calculation

Operation	Tool-wear Factor
Slab milling and end milling	1.10
Light and medium face milling	1.10 to 1.25
Heavy face milling	1.30 to 1.60

Example

Calculate the power required to rough mill a surface 115 mm wide and 250 mm long with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill that is 150 mm in diameter. The work material is alloy steel (200 BHN).

Solution Given $Z = 16$; $d = 6$ mm; $W = 115$ mm

From Table 7.1, Cutting speed, $V = 60$ m/min

Feed rate, $f = 0.18$ mm/tooth

$$\text{Spindle speed, } N = \frac{1000 \times 20}{\pi \times 100} = 63.66 \approx 65 \text{ rev/min}$$

Where f_m = feed rate in mm/min = $fZN = 0.18 \times 16 \times 65 = 187.2$ mm/min

Material removal rate (Q) is

$$Q = \frac{f_m wd}{60000} \text{ cm}^3/\text{s} = \frac{187.2 \times 115 \times 6}{60000} = 2.1528 \text{ cm}^3/\text{s}$$

From Table 7.2, $K_p = 1.88$

From Table 7.3, $C = 1.11$

From Table 7.4, $W = 1.30$

Milling power (P_m) in horse power units at the cutting tool is

$$P_m = 1.88 \times 2.1528 \times 1.11 \times 1.30 = 5.84 \text{ hp}$$

Hole Making Operations

Machining round holes in metal stock is one of the most common operations in the manufacturing industry. It is estimated that of all the machining operations carried out, about 20% are hole making operations.

The types of hole making operations performed on the holes are:

- Drilling
- Boring
- Reaming
- Counter sinking
- Counter boring and
- Tapping

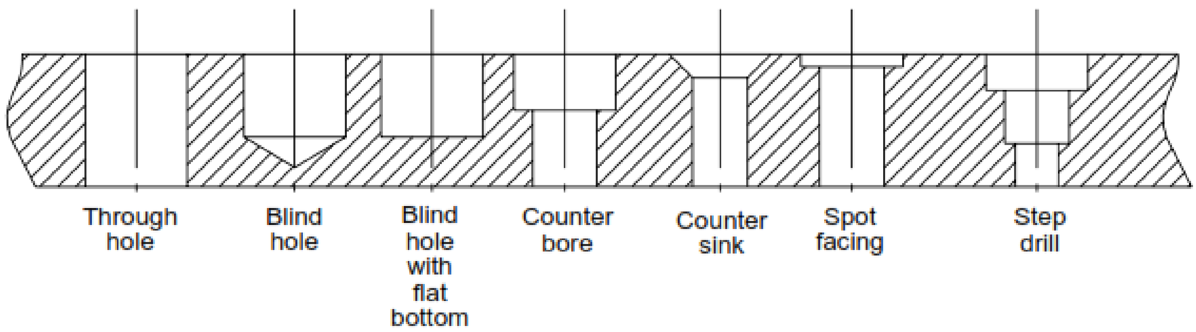


Fig.31 Various types of holes

Table 7 Comparative characteristics of hole making operations

Hole Parameter	Drilling	Reaming	Boring	Counter Boring
Smallest size, mm	1.6	1.6	9.5	6
Largest size, mm	50	100	250	75
Negative tol., mm	$0.896 \times D^{0.5}$	0.010	0.008	$0.512 \times D^{0.5} + 0.064$
Positive tol., mm	$0.896 \times D^{0.5} + 0.075$	0.010	0.008	$0.640 \times D^{0.5} + 0.075$
Straightness, mm	$0.013 \times (l/D)^3 + 0.050$	2.5	0.013	0.250
Roundness, mm	0.100	0.013	0.025	0.075
Parallelism, mm	$0.025 \times (l/D)^3 + 0.075$	0.25	0.025	0.250
Depth limit, mm	300	400	225	500
True position, mm	±0.200	±0.25	±0.003	±0.003
Surface finish, μm	2.54	0.41	0.20	1.25

Drilling

Twist Drill Geometry

The cutting tool used for making holes in solid material is called the twist drill. It basically consists of two parts; the body consisting of the cutting edges and the shank which is used for holding purpose. This has two cutting edges and two opposite spiral flutes cut into its surface. These flutes serve to provide clearance to the chips produced at the cutting edges. They also allow the cutting fluid to reach the cutting edges. The drill blanks are made by forging and then twisted to provide the torsional rigidity.

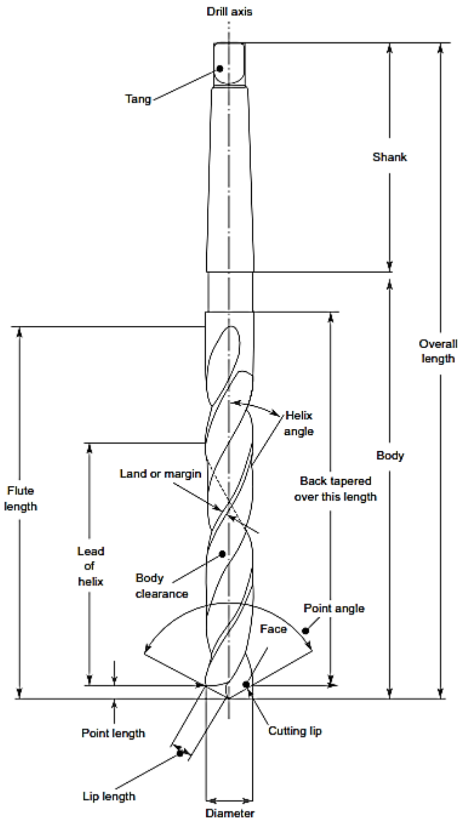


Fig.32 Twist drill geometry

- The surface on the drill, which extends behind the cutting lip to the following flute, is termed as **flank**.
- **Face** is the portion of the flute surface adjacent to the cutting lip on which the chip moves as it is cut from the work piece.
- The **cutting lip** is the edge formed by the intersection of the cutting edge or face and the flank face.
- **Land or margin** is the cylindrically ground body surface on the leading edge of the drill sometimes also termed as cylindrical land.
- The **cutting edge** is reduced in diameter after the margin to provide a body clearance.
- **Axial rake angle** is the angle between the face and the line parallel to the drill axis.
- **Helix angle** is the angle between the leading edge of the land and the axis of the drill

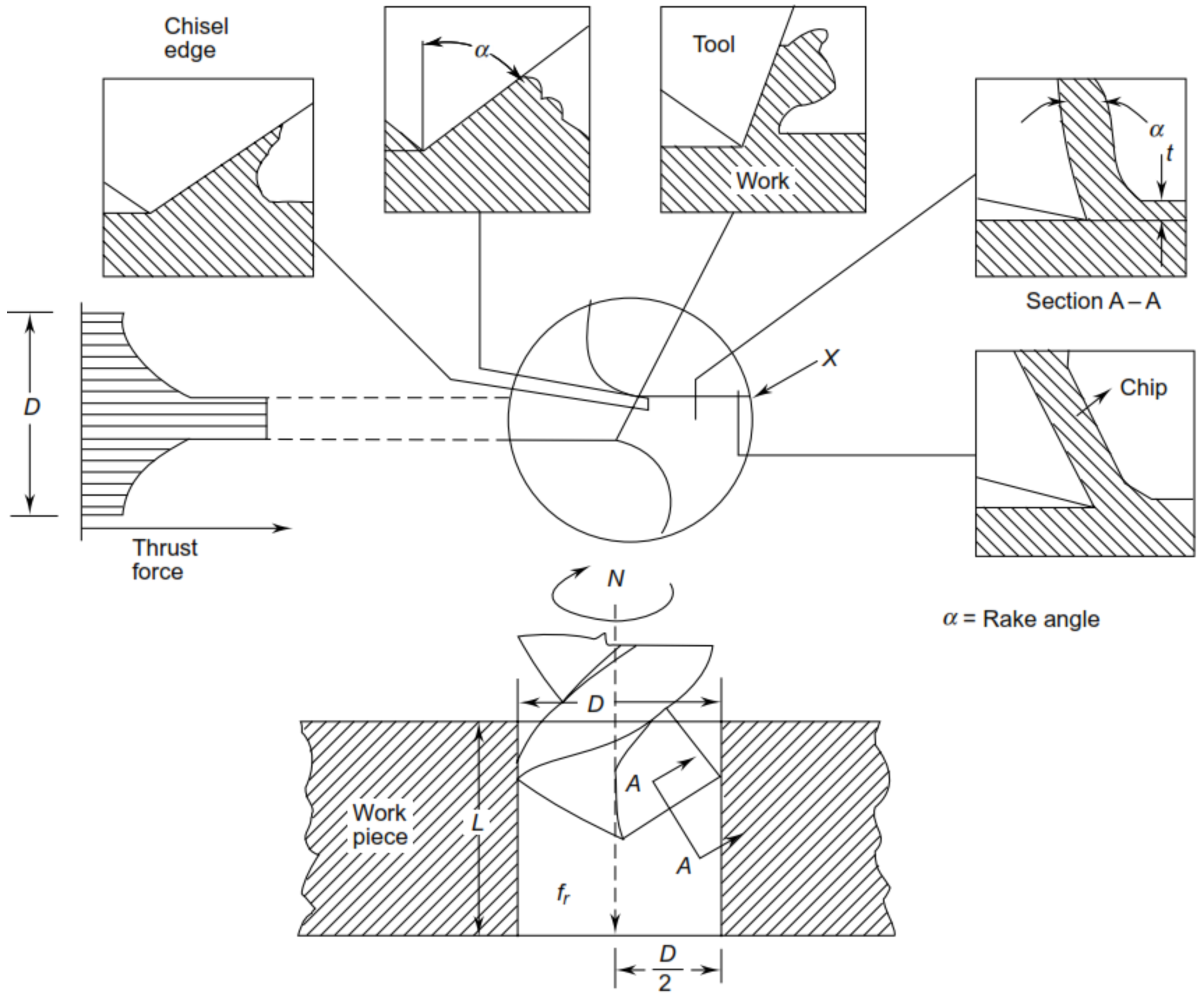


Fig.33 The rake angle in case of a twist drill

Types of Drills

- **Oil hole drills** These are most useful for deep hole drilling. These are provided with two internal holes extending through the length of the drill through which the cutting fluid can be pumped under pressure.
- **Step drills** A variety of step drills are developed for combining machining of operations such as multiple hole drilling, counter boring and counter sinking.
- **Core drills** These are special holes meant for enlarging already existing holes such as those in castings. These are either three-flute or four-flute type. The four-flute type is used for enlarging the drilled holes while the three-flute type is used for punched or cored holes.
- **Shell core drills** are similar to the core drills, but do not have a normal shank for the purpose of holding and are for the large diameters. This needs to be mounted using a stub arbour
- **Spade drills** are used to make smaller diameter holes with low cutting speeds and high feed rates. These have a long supporting bar with the cutting blade attached at the end. These are less expensive since the support structure can be made more rigid
- **Carbide tipped drills** The drills are made of high speed steel. The tungsten carbide tips of suitable geometry are clamped to the end of the tool to act as the cutting edges.

Drilling Machine Construction

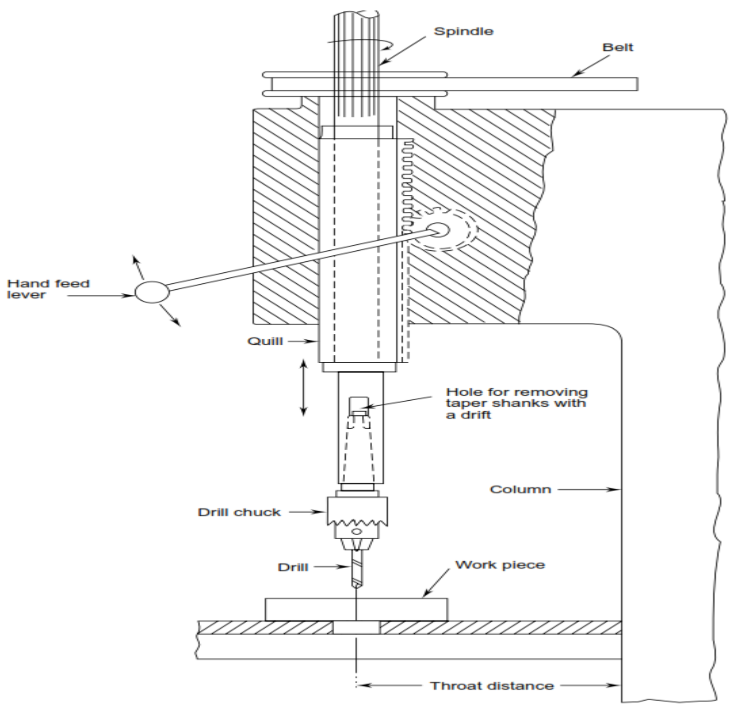


Fig.34 Drilling press

The cutting tool in this case called the drill bit is mounted into the spindle either with the help of the drill chuck for small sized drills that are straight shank type, or by means of the spindle taper. The spindle is located inside a quill, which can reciprocate by means of manual operation or by means of power feed. The work piece is normally placed on the table and clamped using a suitable work holding device. These are relatively simple and less expensive in operation. However, these are not suitable for mass production.

Radial Drilling

The radial drilling machine is more versatile than the drill press as described earlier. The drill head can move along the radial arm to any position while the radial arm itself can rotate on the column, thus reaching any position in the radial range of the machine. They are more convenient to be used for large work pieces, which cannot be moved easily because of their weight, such that the drill head itself will be moved to the actual location on the work piece, before carrying the drilling operation. In addition to the twist drills other hole making tools will also be used.

Multiple Spindle Drilling

For production operations, it is necessary to carry out a large number of operations simultaneously, which can be done through the multiple-spindle drilling machines. In the drilling heads of these machines more than one drill can be located with each of them getting the power from the same spindle motor. The use of these machines becomes more economical for large volume production of identical parts. These machines are capable of producing a large number of holes in a short time. Some machines have a fixed number of spindles in fixed locations while the others have the number fixed but their locations can be changed to suit the work piece geometry. The latter types of machines are more versatile.

Gang Drilling

Gang drilling machines are the equivalent of the progressive action type multiple spindle lathes. These machines have a number of spindles (often equal to four) laid out in parallel. Each of the spindles can have different drills or other hole making operation tools fixed in sequence. The work piece will move from one station to the other, with each completing the designated hole making operation. These are used for volume production with the work pieces located in a jig, with reasonable size to allow the operator to move the part with jig to the next station generally on a roller conveyor.

Work Holding

Work holding in drilling machines is similar to milling. Most of the small components are held in vices for drilling in job shops. However, for production operations, it is not only necessary to locate and clamp the work piece properly, but also to locate and guide the drill. Hence jigs are used to serve this function.

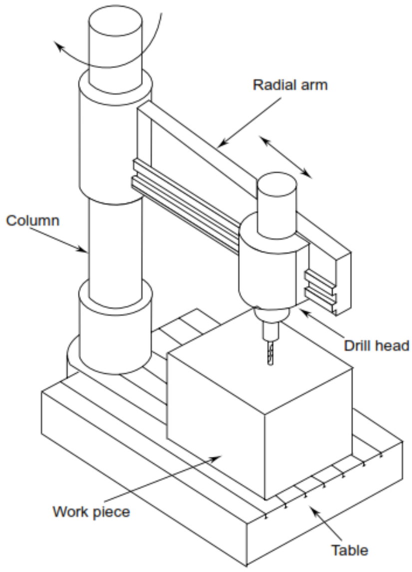


Fig.35 radial drilling

Work Holding in drilling machines is similar to milling. Most of the small components are held in vices for drilling in job shops. However, for production operations, it is not only necessary to locate and clamp the work piece properly, but also to locate and guide the drill. Hence jigs are used to serve this function.

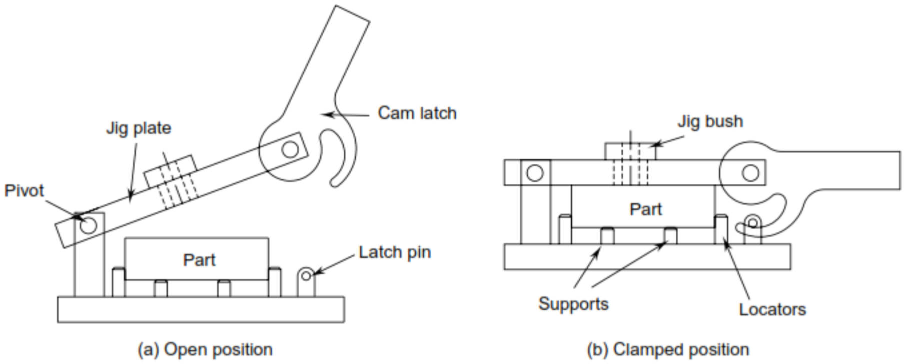


Fig.36 Drilling Jig for drilling a hole in a part

Drilling Time Estimation

The cutting speed in drilling is the surface speed of the twist

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

Where, V = cutting speed (surface), m/min

D = diameter of the twist drill, mm

N = rotational speed of the drill, rev/min

Table 8 Cutting process parameters for drilling

Work Material	Hardness	HSS	
	BHN	Speed m/min	Feed mm/rev
Cast iron	200	25–35	0.13–0.30
Cast steel	280–300	12–15	0.06–0.19
AISI 1020	110–160	35	0.20–0.50
AISI 1040	170–200	25	0.13–0.30
Manganese steel	185–215	5	0.06–0.19
Nickel steel	200–240	18	0.06–0.19
Stainless steel	150	15	0.13–0.30
Spring steel	400	6	0.06–0.19
Tool steel	150	23	0.20–0.50
Tool steel	200	18	0.13–0.30
Tool steel	215	15	0.13–0.30
Tool steel	300	12	0.06–0.19
Tool steel	400	5	0.06–0.19
Malleable iron	110–130	26	0.20–0.50
Aluminium	95	275	0.13–0.90
Aluminium alloys	170–190	18	0.13–0.30
Copper	80–85	21	0.06–0.19
Brass	190–200	70	0.20–0.50
Bronze	180–200	54	0.20–0.50
Zinc alloys	110–125	70	0.20–0.50
Glass		4.5	0.06–0.19

Table 9 Cutting speeds for drilling using tungsten carbide tip drills

Work Material	Cutting Speed m/min
Aluminium	50 to 150
Brass	50 to 100
Bronze	50 to 100
Cast iron soft	30 to 55
Cast iron chilled	10 to 15
Cast iron hard	30 to 45
Steel over 450 BHN	25 to 35

Table 10 Feed rates for drilling using tungsten carbide tip drills

Drill Size	Feed mm/rev
Up to 1.5 mm	0.010 to 0.025
1.5 to 3.0 mm	0.025 to 0.075
3 to 5 mm	0.05 to 0.10
6 to 8 mm	0.08 to 0.13
9 to 11 mm	0.10 to 0.20
12 to 18 mm	0.15 to 0.25
19 to 25 mm	0.20 to 0.30

Breakthrough distance, $A = \dfrac{D}{2 \tan \alpha}$

For the most common case of $\alpha = 59°$, it is given by

$$A = \frac{D}{3.3286}$$

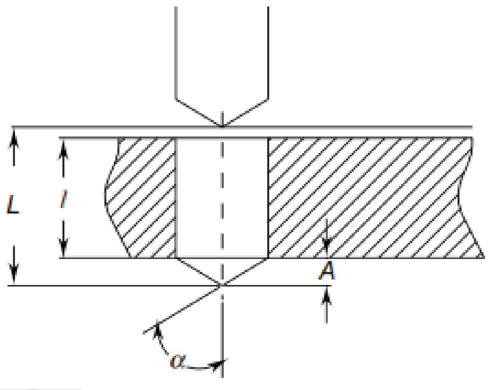
Total length of tool travel, $L = l + A + 2$ mm

Where l = length of the hole, mm

Time for drilling the hole = $\dfrac{L}{fN}$ minutes

Where f = feed rate, mm/rev

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



Example

A hole of 40 mm diameter and 50 mm deep is to be drilled in mild steel component. The cutting speed can be taken as 65 m/min and the feed rate as 0.25 mm/rev. Calculate the machining time and the material removal rate.

Solution Given, $V = 65$ m/min

$f = 0.25$ mm/rev

$D = 40$ mm

$L = 50$ mm

Spindle speed, $N = \dfrac{1000 \times 65}{\pi \times 40} = 517.25$ rev/min

$= 520$ rev/min

Breakthrough distance, $A = \dfrac{40}{2 \tan 59} = 12.02$ mm

Total length of drill travel, $L = 50 + 12 + 3 = 65$ mm

Time for drilling the hole = $\dfrac{65}{0.25 \times 520} = 0.50$ minutes

The material removal rate is MRR = $\dfrac{\pi \times 40^2 \times 0.25 \times 520}{4} = 163362.82$ mm^3/min

$= 163.363$ cm^3/min

Drilling force estimation

The two major forces acting on the drill during the machining operation are the torque and the thrust. The torque acting on a twist drill is

$$M = C\,d^{1.9} f^{0.8} \text{ N mm}$$

Where d is the diameter of the drill in mm

F is the feed rate of the drill in mm/rev

And C is a constant whose values are given in Table 8.6.

The thrust force is given by

$$T = Kdf0.7 \text{ Newtons}$$

The values of K are given as

Steel = 84.7

Cast iron = 60.5

Table 11 The constant C for torque calculations

Material	Hardness, BHN	Constant, *C*
Steel	200	616
	300	795
	400	872
Aluminium alloys		180
Magnesium alloys		103
Brasses		359

Deep Hole Drilling

The deep hole is normally classified as hole, which is longer than three times its diameter. Since the depth is longer, it creates some problems. The rigidity of the drill becomes low because of the extra length required. Further the chip space becomes small compared to the amount of chips generated. The cutting speed and feed used for deep hole drilling should be reduced. The cutting speed for deep holes

$$V_{\text{deep}} = V_{\text{drill}} \times \left[1 - \frac{\text{Depth of hole}}{40 \times \text{Dia. of hole}}\right]$$

Similarly the feed rate is also reduced as follows:

$$f_{\text{deep}} = f_{\text{drill}} \times \left[1 - \frac{\text{Depth of hole}}{50 \times \text{Dia. of hole}}\right]$$

Very small hole drilling such as those with diameters ranging from 0.4 to 1.0 mm will always be classified as deep hole drilling because of the length of the hole involved. To maintain the strength of these drills, the web is generally thicker than that for the normal size drills.

Gun drilling is the process of drilling extremely long or deep holes. This process is widely applied to produce precision long and short holes up to 200 mm dia and 30 m long. A typical gun drill consists of a hollow tube with a V shaped groove or flute along its length with a carbide cutting tip which also acts as its own guide bushing as it drills the hole.

Cutting fluid forces the chips formed by the cutting action up the flute and out of the hole. The high pressure (140MPa or higher) coolant system includes chip separation and filtration units to maintain clean coolant. If not done properly chips in the coolant directed to the tool will impair its function and clog small drills)

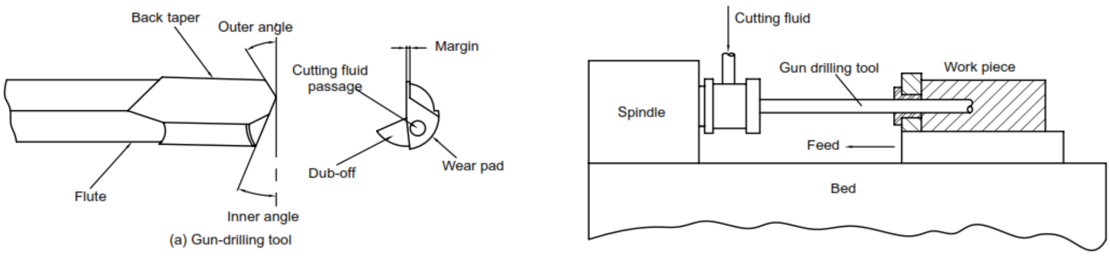


Fig.37 Gun drilling operation, (a) Gun drilling tool, (b) Schematic of a gun drilling machine

Trepanning produces a hole in solid material by directly cutting the circumference of the hole and not the full hole. After the trepanning operation the core at the centre of the hole which forms bulk of the material is left intact and can be re-used for other purpose. There are two forms of trepanning: one type used for thin sheets while the other is used on much heavier material. The tool used for thin sheet trepanning consists of a single point tool that is fixed to an adjustable arm that revolves around a pointed centre shank

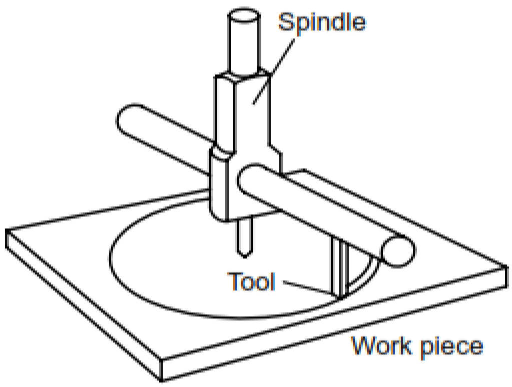


Fig.38 Thin sheet trepanning operation

Reamer is a multi-tooth cutter, which rotates and moves linearly into an already existing hole. The previous operation could be drilling or preferably boring. Reaming will provide smooth surface as well as close tolerance on the diameter of the hole. Generally the reamer follows the already existing hole A reamer is more like a form tool, since the cylindrical shape and size of the reamer is reproduced in the hole. In reaming very little material is removed.

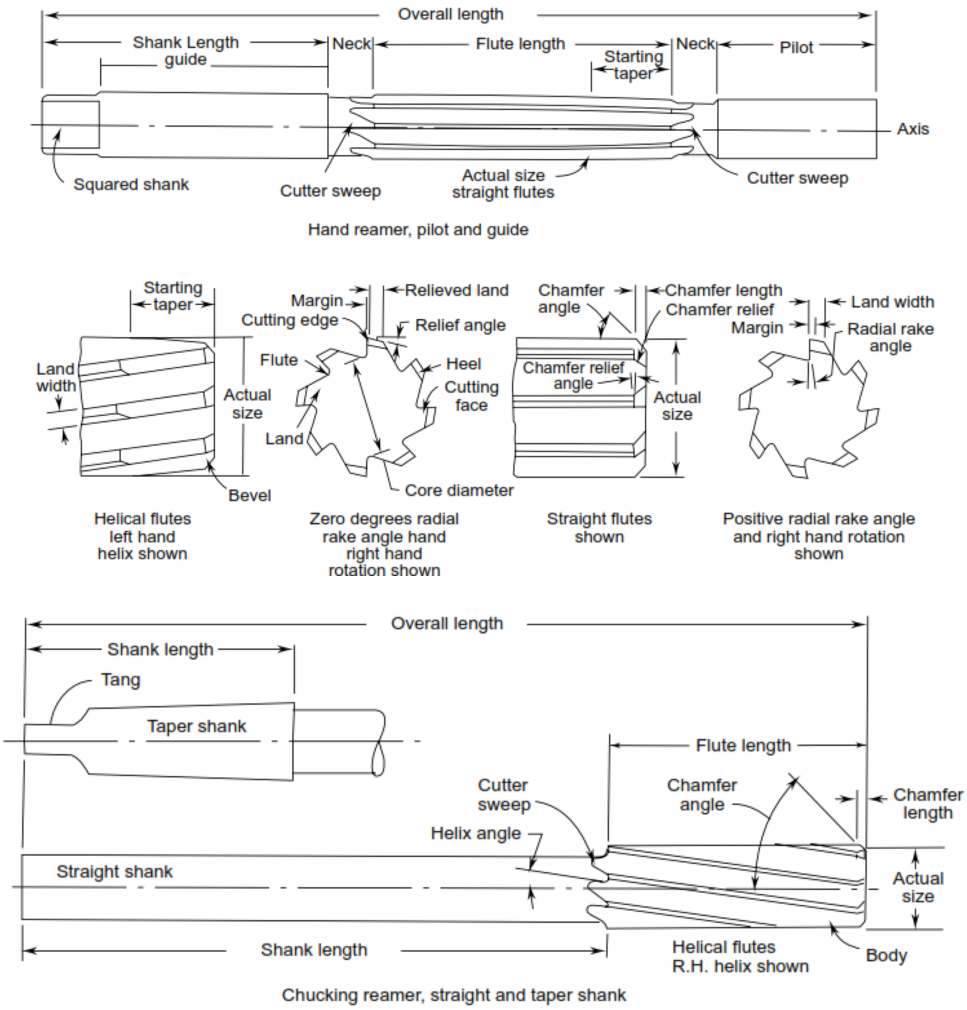


Fig.40 Typical geometry of a reamer and its nomenclature

Table 12 Cutting fluids for reaming

Work Material	Dry	Soluble Oil	Kerosene	Sulphurised Oil	Mineral Oil
Aluminium		X	X		
Brass	X	X			
Bronze	X	X			X
Cast iron	X				
Steel Low carbon		X		X	
Steel Alloy		X		X	
Stainless steel		X		X	

Boring

Boring is an operation of enlarging a hole. The single point cutting tool used for the boring operation. The single point tool bit is mounted in the boring bar of suitable diameter commensurate with the diameter to be bored.

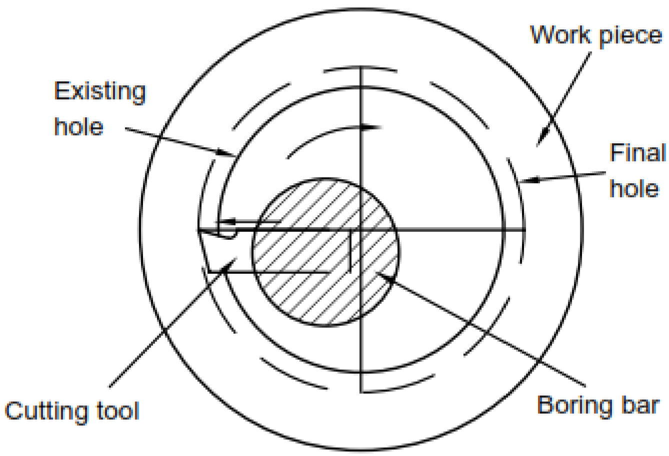


Fig.41 Boring operation

Horizontal Boring Machine

Horizontal boring machine or boring mill comes in a variety of configurations. The basic units that are present in any boring machine are:

● Headstock: It is the heart of the equipment that houses all the spindles with speed and feed arrangement. Different spindles are available depending upon the power and speed requirements of the different operations planned. The headstock also provides a station on which other attachments can be mounted. The spindle feed and hand feed controls are contained in the headstock.

● Column: The headstock is supported by the column and guides it up and down accurately by means of guideways. Columns are attached to bases. Some columns are stationary while others move with their bases.

● Column base: This base supports and secures the column. It houses the various gear and driving mechanisms.

● End-support column: An additional column is sometimes required for operations that involve the use of long boring bars or heavy tools. An outboard bearing in this column is utilized to support the end of the bar. This is also called backrest.

● Runways: These are used to carry the main column and end-support column in some machines.

● Table and saddle: It is provided for locating and clamping work piece. The table has accurate guideways to move the table in two perpendicular directions (X and Y in horizontal plane).

● Bed: Similar to the bed for heavy machine tools to support most of the machine tool parts.

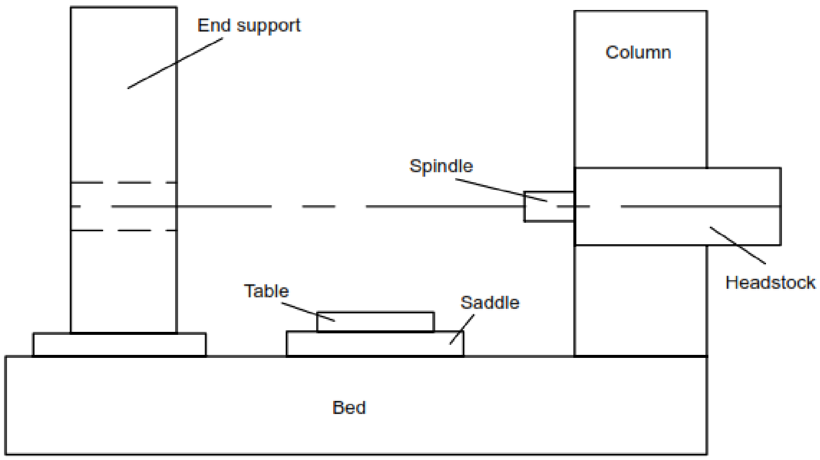


Fig.42 Schematic of a table type Horizontal Boring machine

Vertical boring machine

It has a number of tool heads (up to four), two columns, cross rail similar to a planer, table (rotary), and bed. It is used to machine internal and external diameters as well as to face large work pieces that are axi-symmetrical. A large rotary table is used to clamp the work piece and is rotated against a fixed tool, so only circular cuts can be made. The rotary table is mounted on the bed. Any size of the work piece that is within the size of the rotary table can be machined. Really large machines of this category are constructed with their tables flush with the floor. If necessary these tool heads can be arranged for angular cuts.

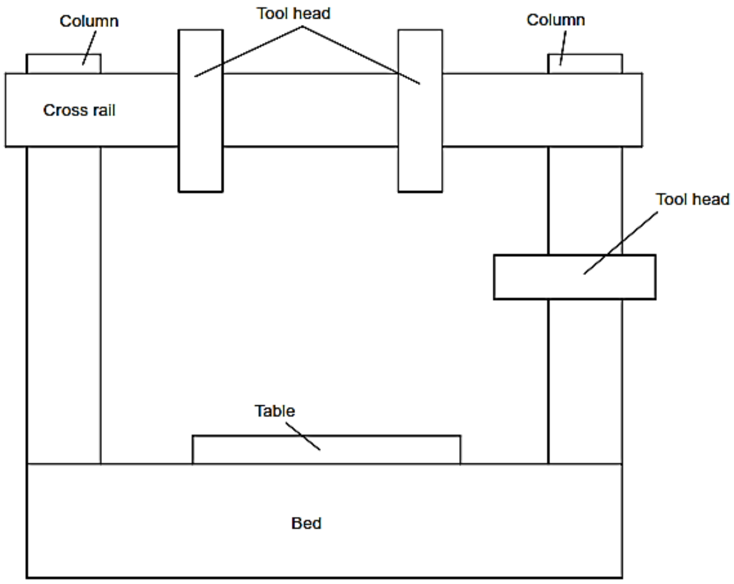


Fig.43 Schematic of a Vertical Boring machine

Vertical Turret Lathe

The vertical turret lathe is similar to vertical boring machine except it is much smaller in size. The types of jobs that can be done in a vertical turret lathe are similar to those of its larger cousin but much smaller in size. It looks almost like a turret lathe rotated 90° from the horizontal, in such a way that its headstock is sitting on the floor and its axis is vertical. The work piece is clamped to the table with jaws and is normally called a chuck. The tool holder is a five- sided turret mounted on the cross rail. It has five tool positions all of which can be used to machine a work piece in a single setup.

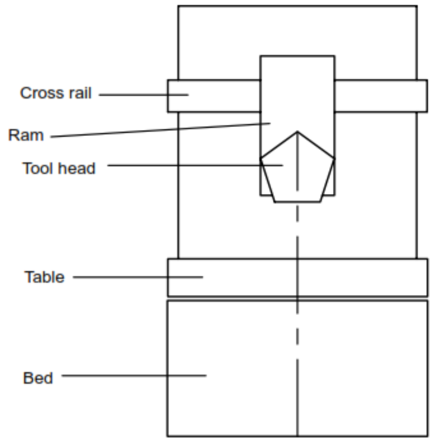


Fig.44 Vertical Turret

Jig Boring Machine

Hole Making Operations

A jig boring machine is similar to a vertical axis milling machine with the table closer to the floor and built with heavy, rigid and accurate construction. Table and saddle are mounted on the rigid bed that provide the necessary two axes (X and Y) movement for the work piece. A massive column supports all the necessary drives to provide power to the spindle. To provide the Z-axis movement the spindle moves inside a quill which is rigidly supported by the spindle head. The quill can also move vertically thereby providing a telescoping movement with added rigidity to extend the range of Z-axis movements for the tool. All the mechanisms involved in the three axes movement are manufactured under extremely careful and exacting conditions so that there will be no motion lost during tool movement. The materials for the housing are also carefully chosen so that thermal expansions due to the changes in temperature are minimized. The jig boring machine must be rugged enough for heavy cuts while remaining sensitive enough for light cuts. Accurate system of measurement is provided in jig boring machines for locating the tool positions precisely.

Tapping

A faster way of producing internal holes is by the use of tapping operation. A tap is a multi-fluted cutting tool with cutting edges on each blade resembling the shape of threads to be cut. A tap of the required size is to be used after carrying out the pre-drilling operations.

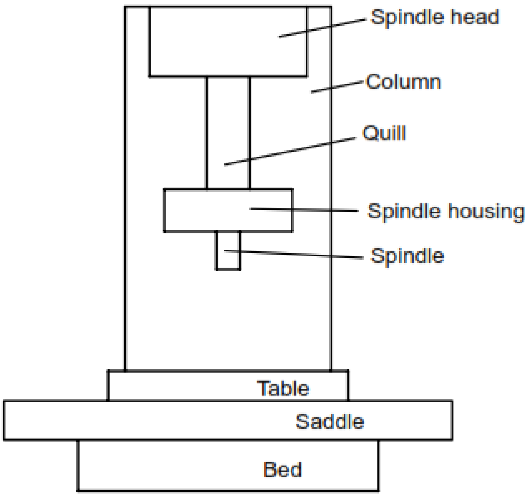


Fig.45 Jig Boring operation

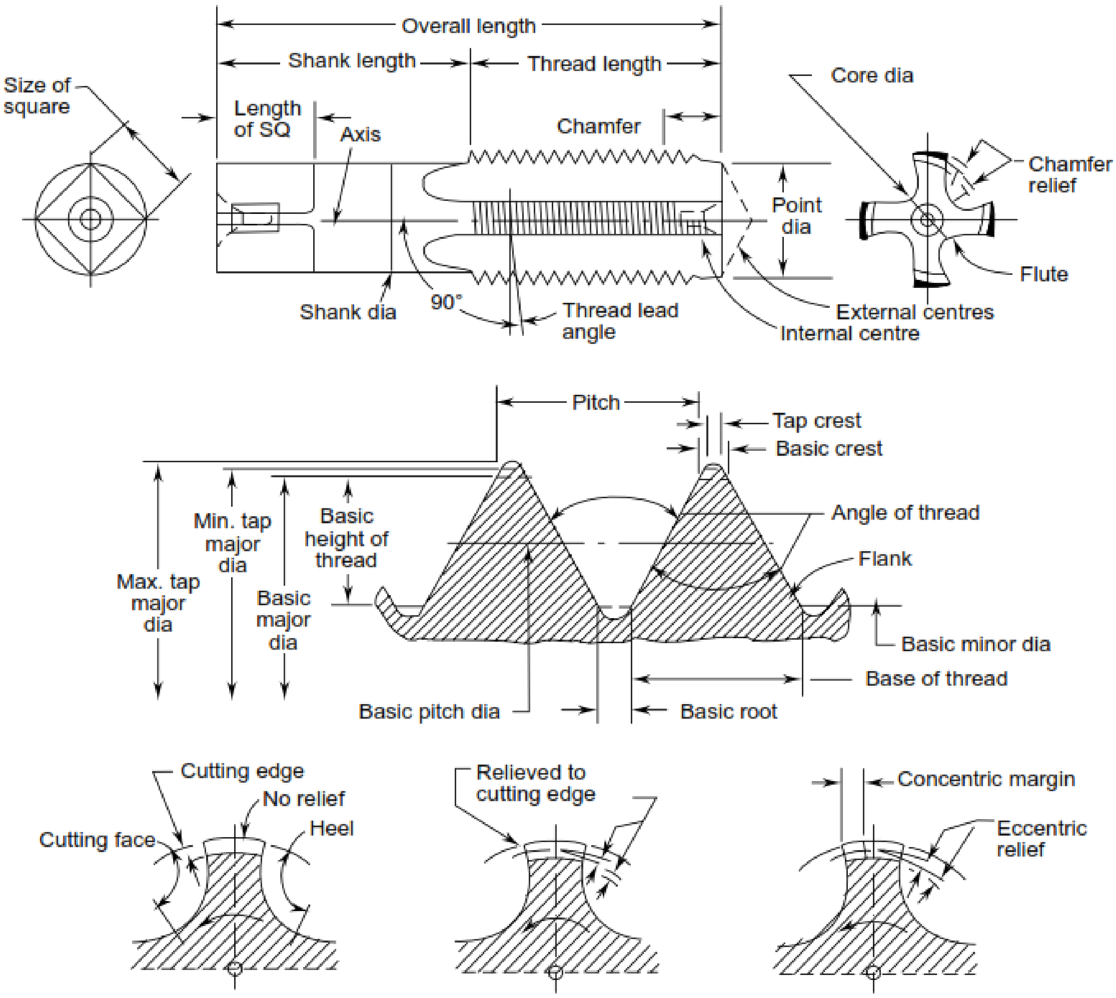


Fig.46 Important features of a tap

Counter Boring

Already existing holes in the components can be further machined for counter boring. Counter boring is done on surfaces that are rough, uneven and not evenly perpendicular to the axis of the hole. The counter boring can be done by a tool similar to the shell end mill with the cutting edges present along the side as well as the end, while a pilot portion is present for the tool to enter the already machined hole to provide the concentricity with the hole. The pilot should fit snugly in the hole and should have sufficient clearance facilitating the free movement of the tool.

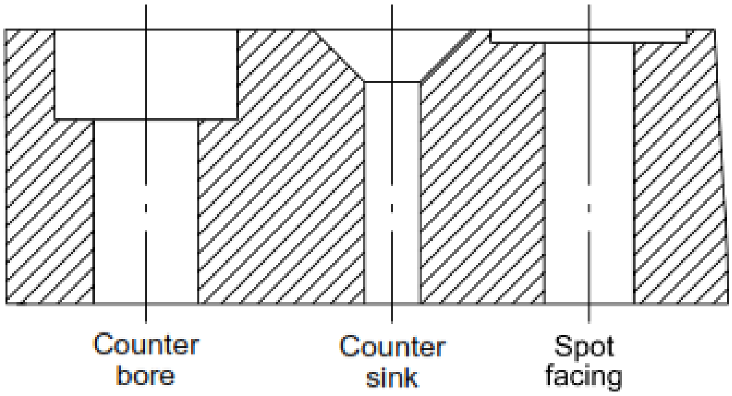


Fig.47 Other hole making operations

Sawing

- Sawing is narrow cutting zone through the successive removal of chips by the teeth on a saw blade.
- Each of the teeth removes a part of the chip, which is contained in the chip space of the saw blade, till the tooth comes out of the material.
- Sawing is economical processes because of the removal of a very small amount of material which consumes less power and at the same time is able to cut large sections

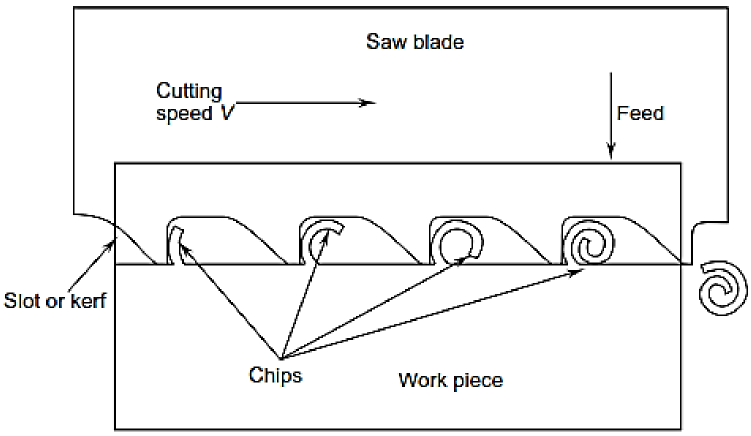


Fig.48 Sawing operation

Saw Blades

Saw blades are made either from carbon steel or high speed steel. The larger blades will have only the teeth portion made of high speed steel while the main portion of the blade is made of low cost alloy steel. The blade strip is electron beam welded to the main portion thereby reducing the overall cost.

Types of blade

Hack saw blade is in the form of a strip ranging in length from 250 to 600 mm and widths ranging from 12.5 to 50 mm. The teeth space is about 1.0 to 6.0 mm. Band saw blades These are long and continuous and generally formed into a closed band with teeth on one edge.

Circular saw This is a large circular blade similar to the slitting saw used in milling. The characteristics of cutting depend upon a number of geometric factors of the teeth.

Tooth form The saw teeth can be simple straight teeth or with an undercut-face- tooth with a positive rake angle. It is difficult to obtain with smaller teeth. The straight tooth form is used for high feeding while the undercut-face-tooth type is used for coarse feeding.

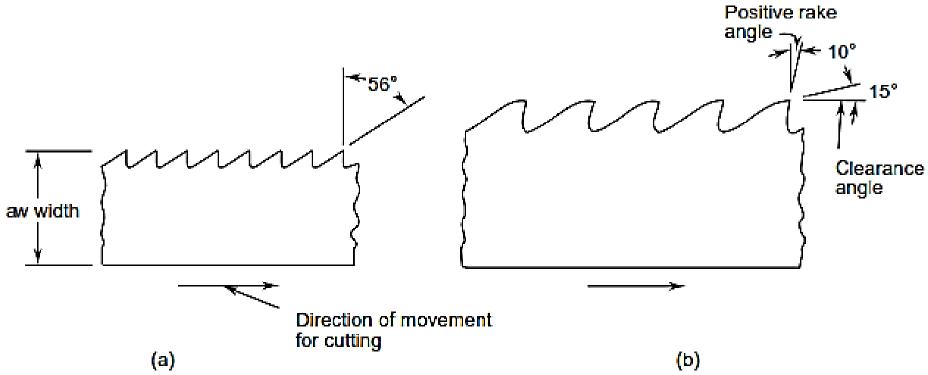


Fig.49 Commonly used saw tooth forms

Tooth spacing It is very important since it determines the size of teeth as well as the amount of chip space available. Large tooth spacing gives more space for the chips as well as strong teeth. Tooth spacing also determines How many teeth are in contact with the work piece at any given time.

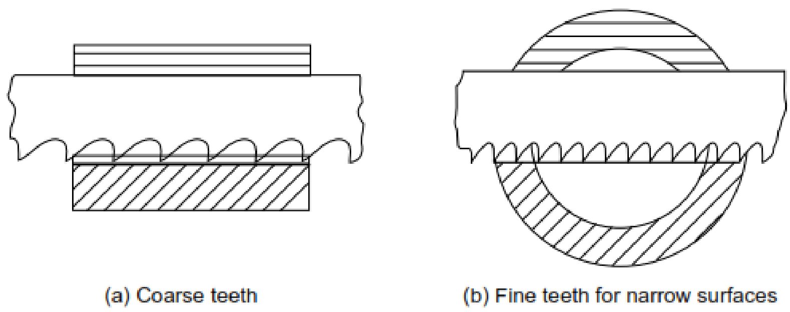


Fig.50 Effect of saw tooth pitch while sawing materials of different widths

Tooth set are offset with respect to the centre line of the blade width. The tooth set makes the saw cut, termed as kerf, wider to facilitate the easy movement of the saw blade for cutting. The straight set as is used for non-ferrous materials and plastics. The raker set is used for general purpose machining of ferrous materials. The wavy set is used in the fine pitch tooth blades for cutting thin sheets and tubes.

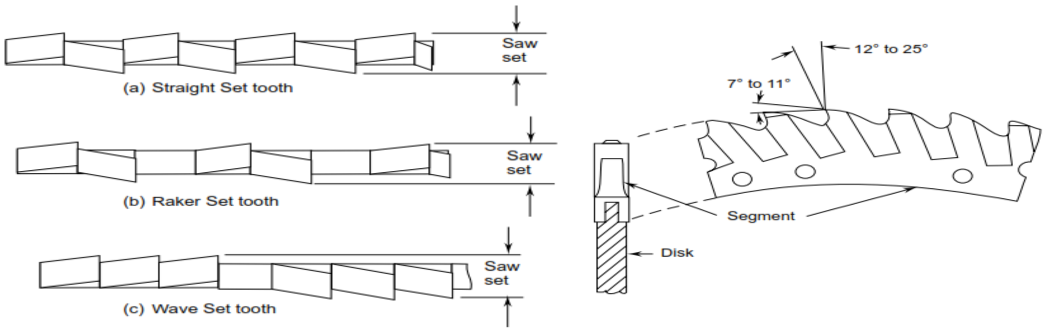


Fig.51 Different types of tooth sets used in sawing and Typical circular saw blade construction

Sawing Machines

The various types of sawing machines used are:

1.Hack saw	2.Band saw	3.Circular saw
Manual	Vertical	
Power	Horizontal	
	Contour	

Power Hack Saw

The power hack saw uses the hack saw blade. The blade is mounted in the hacksaw frame and reciprocated for the sawing operation. The reciprocating motion is inherently inefficient because no cutting takes place during the return stroke.

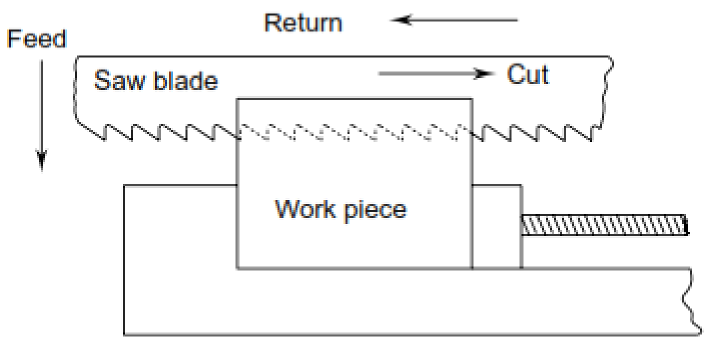


Fig.52 Power hack saw in action

Band Saw

These basically have a continuous band of saw blade rotated between two disks such that the cutting action will be continuous unlike the power hack saw. These are generally used for cutting off single stationary work pieces that can be held on to the table of the band saw. The saw blade can be tilted up to 45° to permit cutting at any angle.

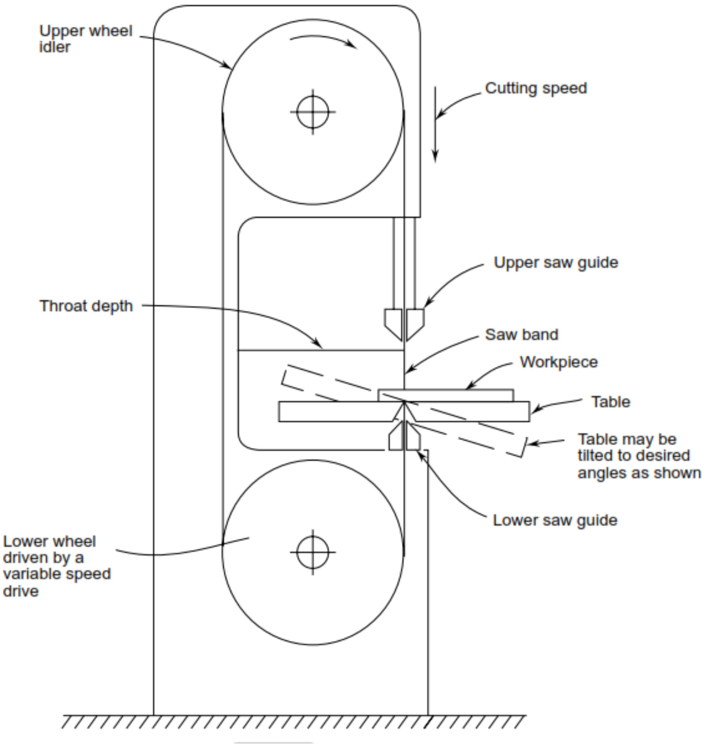


Fig.53 Vertical band saw

Circular Saw

These have the ability to run the saw at very high cutting speeds up to about 130 m/s and large feed rates. The stock can be cut very quickly and therefore care has to be taken for the selection of the parameters to maximise the productivity

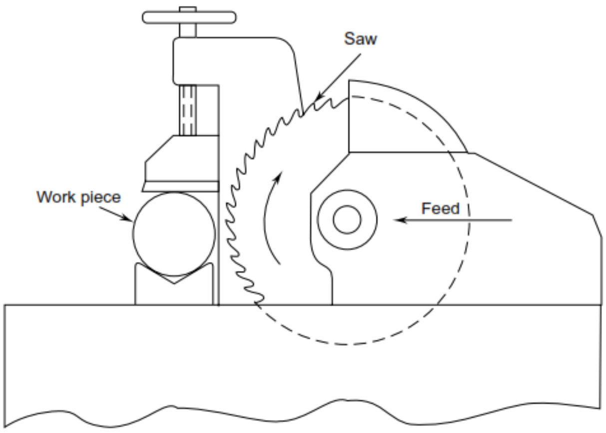


Fig.54 A circular sawing machine

Broaching

Broaching is a multiple-tooth cutting operation with the tool similar to the sawing operation. In broaching the machining operation is completed in a single stroke as the teeth on the cutting tool called broach, are at a gradually increasing height corresponding to the feed per tooth of a milling cutter. The dotted line indicates the amount of material removed by the successive individual teeth. Using broaching, Tolerances of the order of ±0.013 mm can be easily obtained Very high surface finish is not the aim of broaching, and as a result it is possible to get 0.80 µm with normal broaching. A better finish can be obtained by sacrificing tool life and employing expensive fixtures.

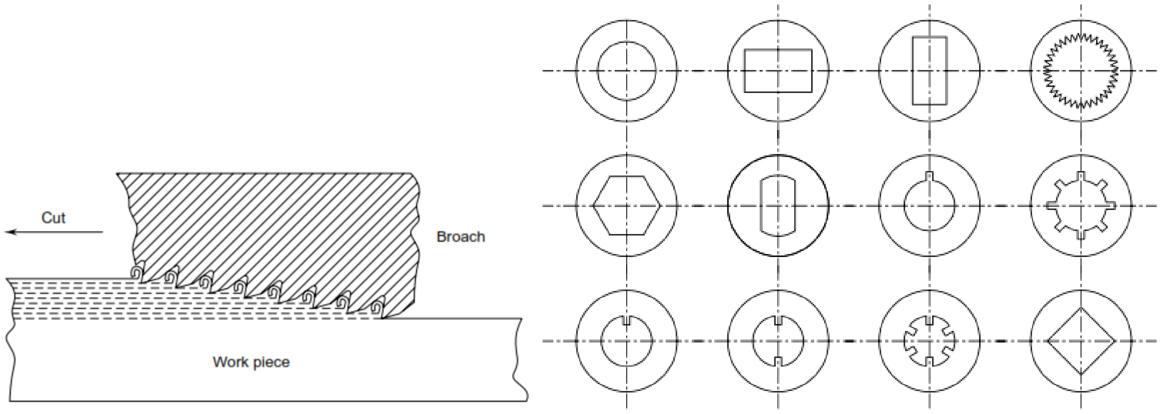


Fig.55 Some typical internal profiles that can be broached

Broaching construction

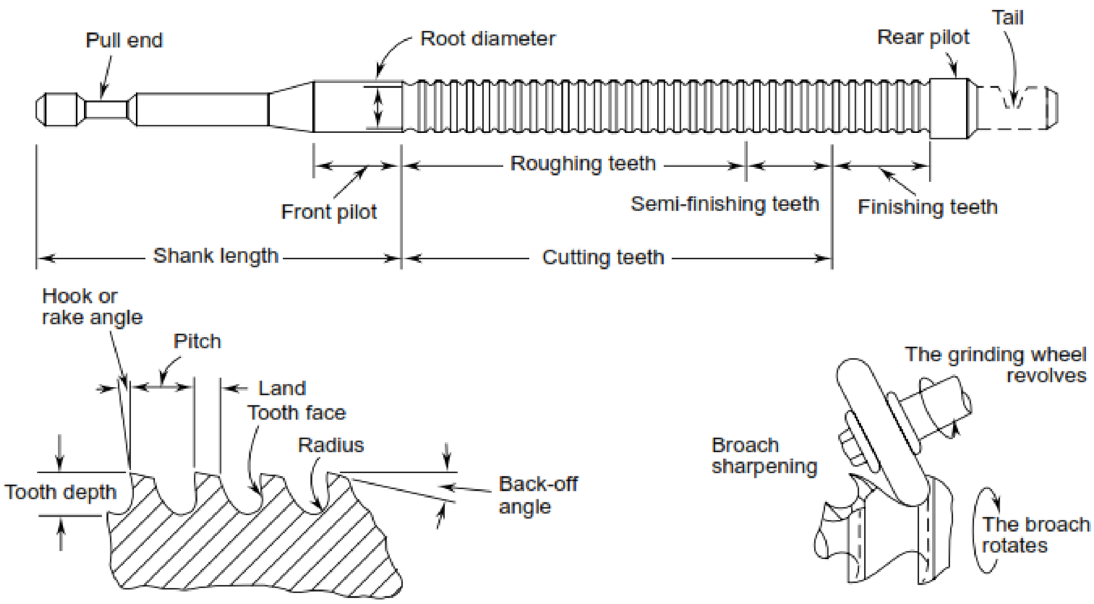


Fig.56 Typical construction of a pull broach

Pitch, $P = 0.35\sqrt{\text{Length of cut}}$

Length of broach, $L_B = \left(\dfrac{D_S}{D_T} + Z_f\right)P$

Where Z_f = the number of teeth required for finishing the operation and is assumed to be 4 or 5 teeth.

The broaching time, T_B in min = $\dfrac{L}{1000 \times V_f} + \dfrac{L}{1000 \times V_r}$

where L = Length of stroke, mm;

V_f = Cutting speed in the forward stroke, m/min;

V_r = Return speed, m/min.

Broaching Machines

There are basically four different types of broaching machines as follows:

- Push broaching machines,
- Pull broaching machines,
- Surface broaching machines, and
- Continuous surface broaching machines.

Advantages of broaching

1. It is the fastest way of finishing an operation with a single stroke.
2. Since all the machining parameters are built into the broach, very little skill is required from the operator.
3. Broaching machine is simple since only a single reciprocating motion is required for cutting.
4. Final cost of the machining operation is one of the lowest for mass production.
5. Any type of surface, internal or external, can be generated with broaching.
6. Many surfaces which are very difficult or impossible by other means can be done by broaching. For example, square hole and internal splines can be easily produced by broaching.
7. Good surface finish and fine dimensional tolerances can be achieved by broaching, often better than boring or reaming.

Limitations of broaching

1. Custom made broaches are very expensive and can therefore be justified only for very large volume

production.

2. A broach has to be designed for a specific application and can be used only for that application. Hence the lead time for manufacture is more for custom designed broaches.

3. Broaching being a very heavy metal removal operation requires that the work piece is rigid and capable of withstanding the large forces.

Broaching machines are specified mainly using the main parameters:

- The force applied on the cutting tool by the machines in tonnes or kN, e.g. 1000 kN
- The maximum length of stroke of the ram, e.g. 450 mm which provides an indication of the maximum length of cut that can be taken.
- Various ranges of cutting speeds available
- Various ranges of feeds available
- The type of power source utilized
- Physical dimensions of the machine

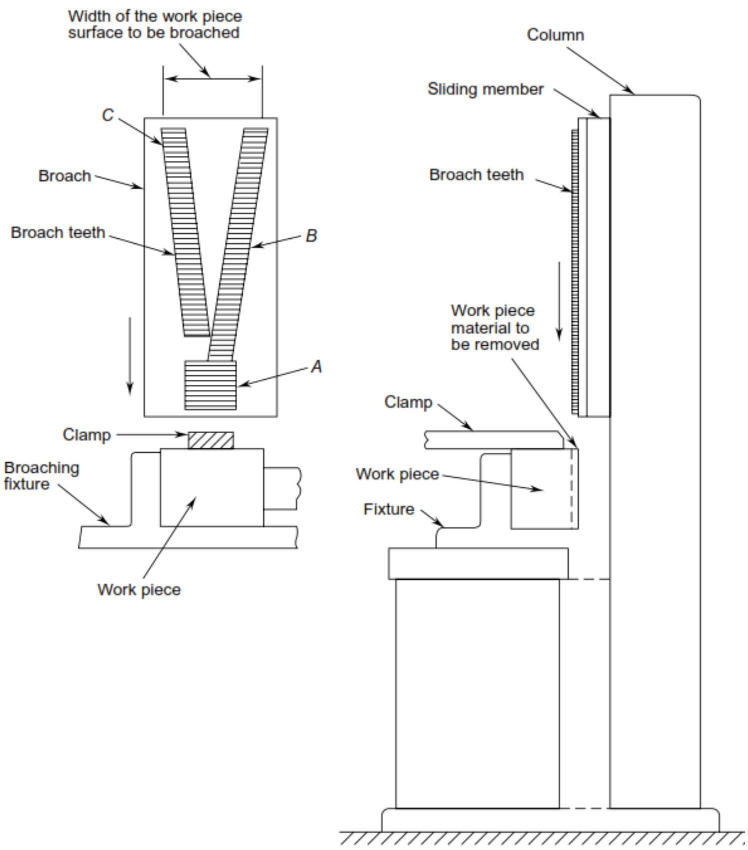


Fig.57 Surface Broaching machine

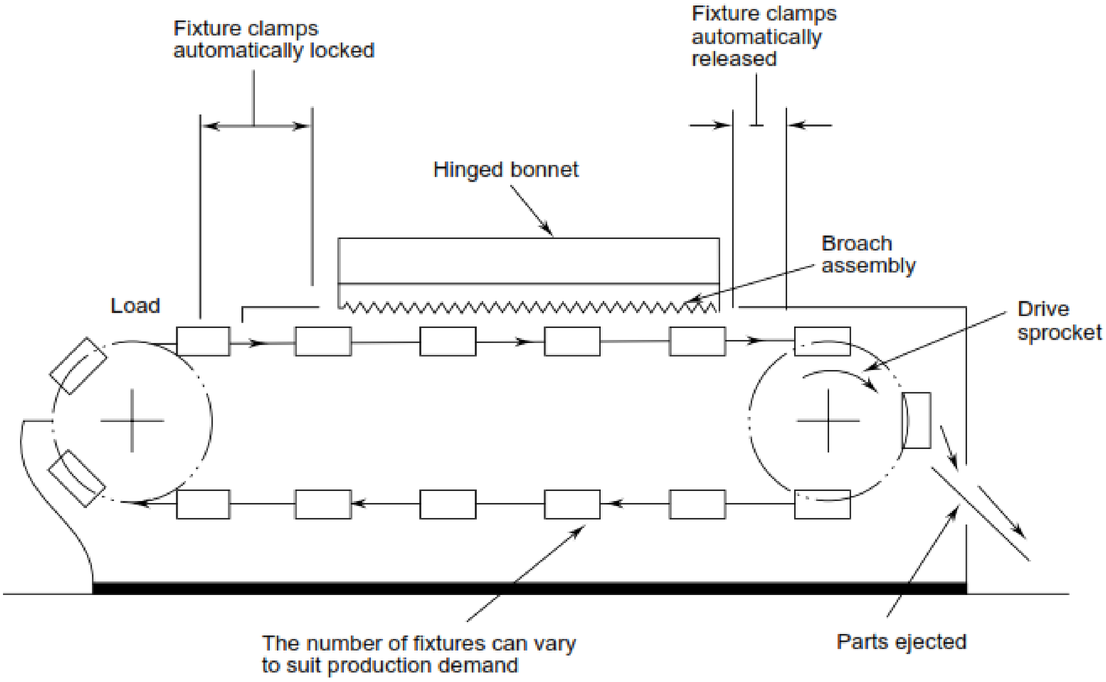


Fig.58 Continuous surface broaching machine

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AUTOMOBILE ENGINEERING

SMEA 1505 _ MANUFACTURING TECHNOLOGY II

UNIT IV ABRASIVE PROCESSES AND GEAR CUTTING

UNIT – 4 ABRASIVE PROCESSES AND GEAR CUTTING

Grinding is a process carried out with a grinding wheel made up of abrasive grains for removing very fine quantities of material from the work piece surface. The required size of abrasive grains are thoroughly mixed with the bonding material and then pressed into a disc shape of given diameter and thickness. This can be compared to a milling process with an infinite number of cutting edges.

Grinding is a process used for

- Machining materials which are too hard for other machining processes such as tool and die steels and hardened steel materials,
- Close dimensional accuracy of the order of 0.3 to 0.5 mm, and
- High degree of surface smoothness such as Ra = 0.15 to 1.25 µm

Abrasive grains are basically spherical in shape with large sharp points, which act as cutting edges. All the grains are of random orientations and as such the rake angle presented to the work material can vary from positive to a large negative value. Many grit also slide rather than cut because of its orientation.

Table 1 Characteristics of various abrasive processes

Process	Particle Mounting	Features
Grinding	Bonded	Wheels, generally for finishing. Low material removal rate
Creep feed grinding	Bonded Open soft	Wheels, slow feed and large depth of cut
Snagging	Bonded, Belted	High material removal rate, roughing to clean and deburr castings and forgings
Honing	Bonded	Stones contain fine abrasives for hole finishing
Lapping	Free	For super finishing

Grinding wheel designation and selection

The grinding wheels are produced by mixing the appropriate grain size of the abrasive with the required bond and pressed into shape. The characteristics of the grinding wheel depend upon a number of variables. They are described below:

Abrasive Types

These are the hard materials with adequate toughness so that they will be able to act as cutting edges for a sufficiently long time. They also have the ability to fracture into smaller pieces when the force increases, which is termed as friability. This property gives the abrasives the necessary self-sharpening capability. The abrasives that are generally used are:

- Aluminium oxide (Al2O3)
- Silicon Carbide (SiC)
- Cubic Boron Nitride (CBN)

- Diamond

Aluminium Oxide (Al_2O_3)

This is one of the natural abrasives found called corundum and emery. However the natural abrasives generally have impurities and as a result their performance is inconsistent. Hence the abrasive used in grinding wheels is generally manufactured from the aluminium ore, bauxite.

Silicon Carbide (SiC)

Silicon carbide is made from silica, sand, and coke with small amounts of common salt.

Cubic Boron Nitride (CBN)

Cubic Boron Nitride (CBN) next in hardness only to diamond (Knoop hardness ~ 4700 kg/mm^2). It is not a natural material but produced in the laboratory using a high temperature/ high pressure process similar to the making of artificial diamond. CBN is less reactive with materials like hardened steels, hard chill cast iron, and nickel base and cobalt based super alloys. CBN grains have 55 times higher thermal conductivity, four times higher the abrasive resistance and twice the hardness of the aluminum oxide abrasives. They can retain their strength above 10,000°C. CBN is very expensive, 10 to 20 times that of the conventional abrasive such as aluminium oxide.

Diamond

Diamond is the hardest known (Knoop hardness ~ 8000 kg/mm2) material that can be used as a cutting tool material. It has very high chemical resistance along with low coefficient of thermal expansion. Also it is inert towards iron.

Table 2 Characteristics of abrasives used in grinding wheels

Abrasive	Vickers Hardness Number	Knoop Hardness	Thermal Conductivity, W/m K	Uses
Aluminium oxide	2300	2000 to 3000	6	Softer and tougher than SiC used for steels and high strength materials
Silicon carbide	2800	2100 to 3000	85	Nonferrous, non-metallic materials, Hard and dense metals and good finish
Cubic Boron Nitride	5000	4000 to 5000	200	Hard and tough tool steels, stainless steel, aerospace alloys, hard coatings
Diamond (synthetic)	8600	7000 to 8000	1000 to 2000	Some die steels and tungsten carbide
Hardened steel		~700		

Grain Size

Compared to a normal cutting tool, the abrasives used in a grinding wheel are relatively small. The size of an abrasive grain, generally called grit, is identified by a number which is based on the sieve size used. These would vary from a very coarse size of 6 or 8 to a super fine size of 500 or 600. Sieve number is specified in terms of the number of openings per square inch. Thus larger the grain number finer is the grain size.

The surface finish generated would depend upon the grain size used as shown in Table 9.3. The fine grains would take a very small depth of cut and hence a better surface finish is produced. Also fine grains generate less heat and are good for faster material removal. Though each grain cuts less, there are more grains per unit surface area of the wheel in case of fine grain size. Fine grains are also used for making the form grinding wheels.

Table 3 Surface finish obtained with grain size

Grain Size	Surface Finish, μm
46	0.8
54	0.6 to 0.8
60	0.4 to 0.6
80	0.2 to 0.4

Coarse grains are good for higher material removal rates. These have better friability and as a result are not good for intermittent grinding where they are likely to chip easily.

Bond

The function of the bond is to keep the abrasive grains together under the action of the grinding forces. The commonly used bond materials are:

- Vitrified
- Silicate
- Synthetic resin
- Rubber
- Shellac
- Metal

Vitrified

This is the most commonly used bond. The bond is actually clay mixed with fluxes such as feldspar, which hardens to a glass like substance on firing to a temperature of about 1250°C and develops the strength. This bond is strong, rigid and porous, and not affected by fluids. However, this bond is brittle and hence sensitive to impacts. This bond is also called ceramic bond.

Silicate

This is sodium silicate ($NaSiO_3$) or water glass and hardens when heated. Not as strong as vitrified. This can be used in operations that generate less heat. It is affected by dampness but less sensitive to shocks. Relatively less used.

Synthetic Resin or Resinoid

These bonding materials are thermosetting resin such as phenol formaldehyde. This bond has good strength and is more elastic than the vitrified bond. However, this is not heat and chemical resistant. Generally used for rough grinding, parting off and high speed grinding (50 to 65 m/s). It can also be used for fine finishing of roll grinding.

Rubber

Of all the bonds used, this is the most flexible. The bond is made up of natural or synthetic rubber. The strength is developed with vulcanisation. This has high strength and is less porous. This bond is affected by dampness and alkaline solutions. Generally used for cutting off wheels, regulating wheels in centre less grinding and for polishing wheels.

Shellac

This is relatively less used bond. Used generally for getting very high finish. Typical applications are rolls, cutlery, and cam shaft finishing.

Metal

This is used in the manufacture of diamond and CBN wheels. The wheel can be made of any high thermal conductive metal such as copper alloys or aluminium alloys. The periphery of the wheel up to a small depth of the order of 5 mm or less contains the abrasive grit. The choice of the metal depends on the required strength, rigidity and dimensional stability. In view of the strong bond, the grit will not be knocked out till it is fully utilised. Powder metallurgy techniques are used to make the abrasive periphery.

Grade

It is also called the hardness of the wheel. This designates the force holding the grains. The grade of a wheel depends on the kind of bond, structure of wheel and amount of abrasive grains. Greater bond content and strong bond results in harder grinding wheel. Harder wheels hold the abrasive grains till the grinding force increases to a great extent. The grade is denoted by letter grades.

Soft wheels are generally used for hard materials and hard wheels are used for soft materials. While grinding hard materials the grit is likely to become dull quickly thereby increasing the grinding force, which tend to knock of the dull abrasive grains. This keeps the grinding wheel in sharp condition. In contrast the hard grinding wheel while grinding soft materials will be able to retain the grit for longer periods thus improving the material removal.

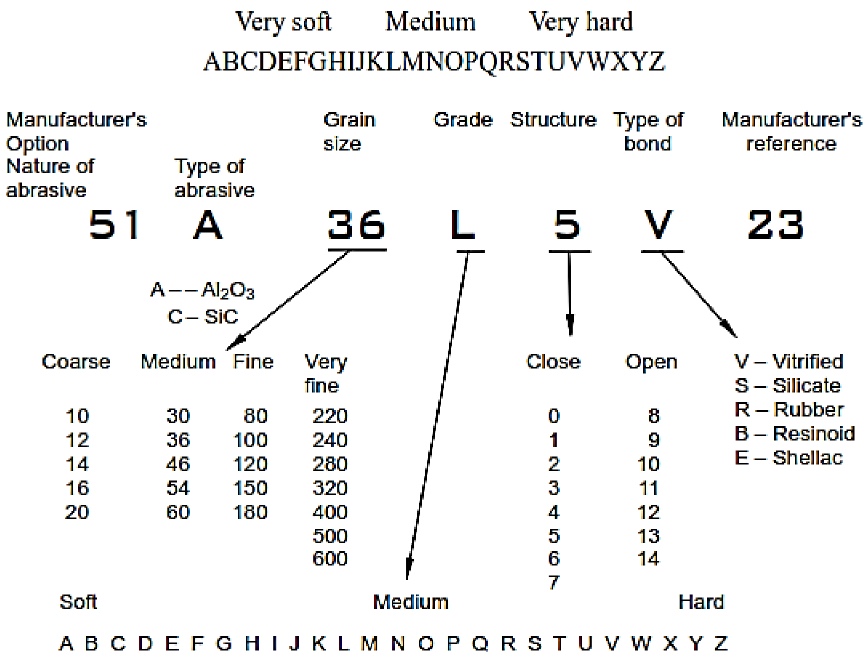


Fig.1 Grinding wheel standard marking system

Structure

The structure of a grinding wheel represents the grain spacing. It can be open or dense and is shown in fig.

Table 4 Grinding wheel hardness for different work materials

Work Piece Material	Wheel Hardness			
	Cylindrical Grinding	Surface Grinding	Internal Grinding	Deburring
Steel up to 80 kg/mm^2 Steel up to 140 kg/mm^2 Steel more than 140 kg/mm^2	L, M, N K J	K, L K, J I, J	K, L J I	O, P, Q, R
Light alloys Cast iron Bronze, brass and copper	J K L, M	I, K J J, K	I J J	

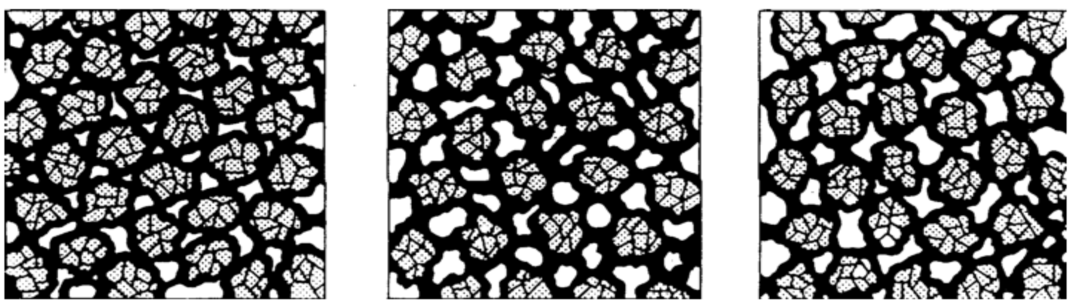


Fig. 2 Grinding wheel structure

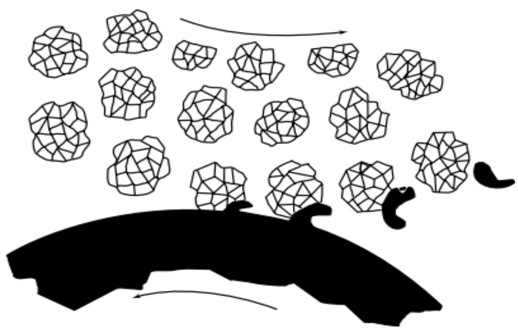


Fig.3 Illustration showing how the spaces between grit help in clearing the grinding chips

Table 5 Grinding wheel specifications for different grinding operations

Operation	Grinding Wheel Designation
Cylindrical grinding of hardened steel	A60L5V
Cylindrical grinding of soft steel	A54M5V
Cylindrical grinding of aluminium	C36K5V
Surface grinding of hardened steel	A60F12V
Surface grinding of soft steel	A46J5V
Surface grinding of grey cast iron	C36J8V
Tool grinding of high speed steel	A46K8V

Grinding Wheel Types

Grinding wheels come in a variety of shapes and standardised sizes as shown in Fig. These suit various work piece shapes and sizes, and are also used in different types of grinding machines. The most common is the straight shape, shown as type 1, which is used for a variety of cylindrical grinding applications. The type 1 wheel will have further modification of the end shape to suit specific applications. The cylinder shown as type 2 is used for grinding flat surfaces. Similarly the flaring cup is used for grinding the cutting tools.

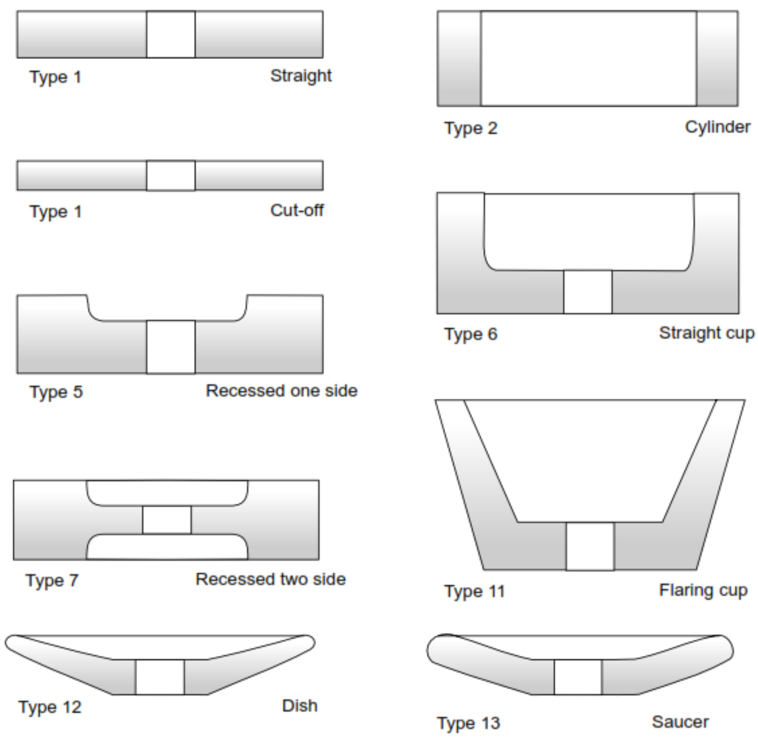


Fig.4 Grinding wheel shapes

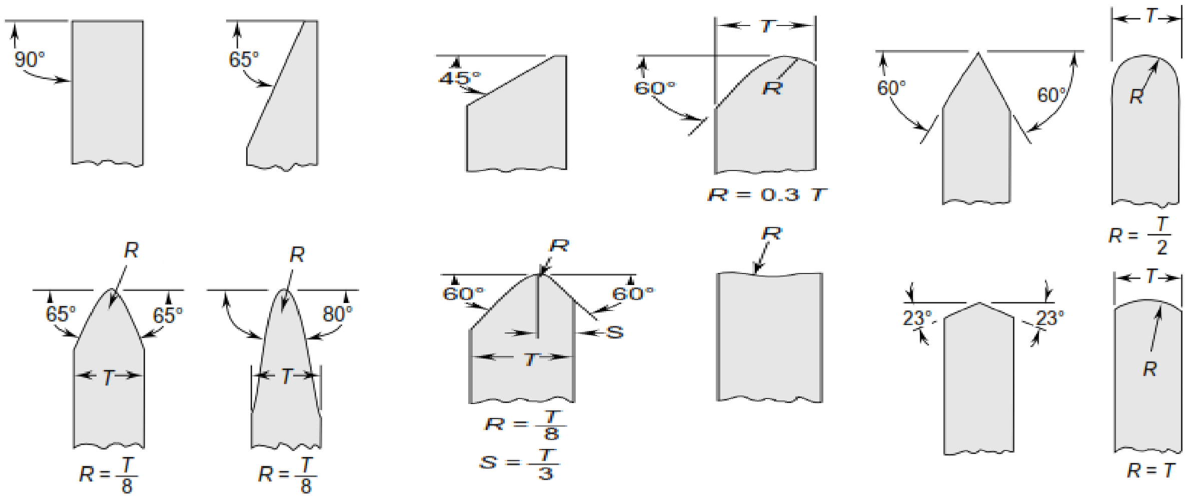


Fig.5 Various faces of grinding wheel form for the straight

The size of the grinding wheel is normally specified by the

- Diameter of the wheel
- Diameter of the spindle hole
- Face width of the wheel

Wheel Balancing

Balance of a grinding wheel also depends upon the machine spindle as well as the condition of tightening properly. In view of the high rotational speeds used, any residual unbalance left would be harmful for the machine part and also produce poor surface finish.

Such wheels are provided with movable balance weights for adjusting the balance mass location. The balancing operation can be carried in two ways:

- Static balancing
- Dynamic balancing

In static balancing the grinding wheel is rotated on an arbour and the balance weights adjusted until the wheel no longer stops its rotation in any one specific position. To do this the balance weights are removed and the wheel is kept on the balancing ways.

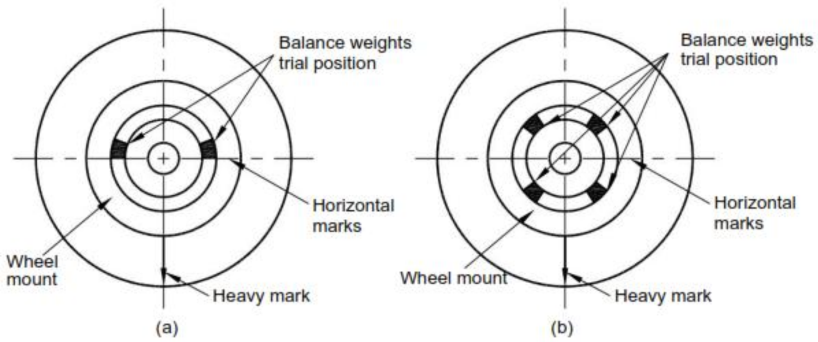


Fig.6 Static balancing of the Grinding wheel

Dressing and Truing

Balance weights trial position with continuous use a grinding wheel becomes dull with the sharp abrasive grains becoming rounded. This condition of dull grinding wheel with worn out grains is termed as glazing. Further, some grinding chips get lodged into the spaces between the grit with the resulting condition known as loaded wheel. Loading is generally caused during the Grinding of soft and ductile materials. A loaded grinding wheel cannot cut properly. Such a grinding wheel can be cleaned and sharpened by means of a process called dressing. A diamond used for truing is set in a closely fitting hole at the end of a short steel bar and is brazed.

Table 6 Cross feed for Diamond truing

Grain Size	Cross Feed per Wheel Revolution, mm
30	0.350 to 0.600
36	0.300 to 0.475
46	0.200 to 0.350
50	0.175 to 0.300
60	0.150 to 0.250
80	0.100 to 0.175

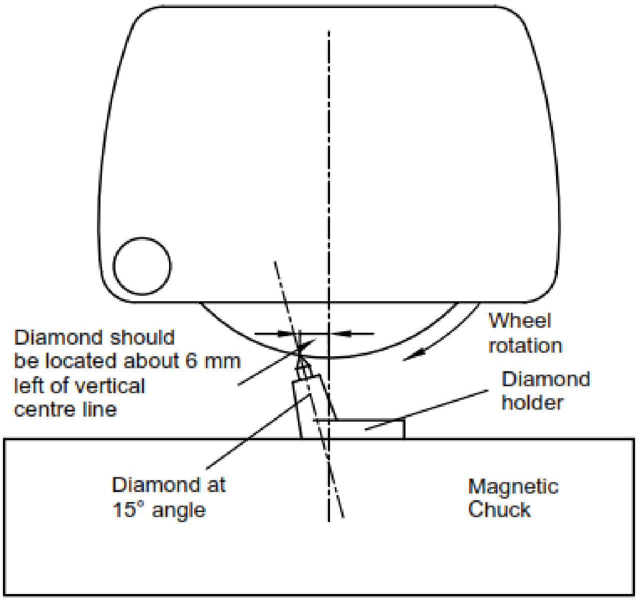


Fig.7 Truing of a Grinding wheel using a diamond dresser on a surface grinder

Types of Grinding Machines

Grinding operations are generally classified based on the type of surface produced. The grinding operations possible can be classified into

- Cylindrical grinding for generating cylindrical surfaces
- Surface grinding for generating flat surfaces, and
- Centre less grinding for generating axi-symmetric shapes.

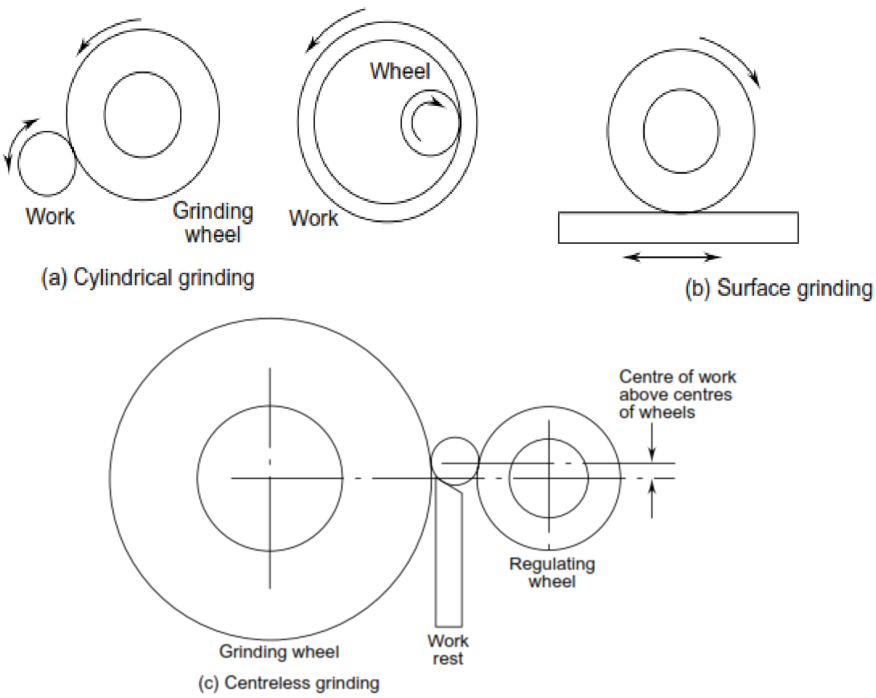


Fig.8 Typical grinding operations

Cylindrical grinding machine is used generally for producing external cylindrical surfaces. The machine is very similar to a centre lathe. The grinding wheel is located similar to the tool post, with an independent power driven at high speed suitable for grinding operation. Both the work and the grinding wheel rotate counter clockwise. The work that is normally held between centres is rotated at much lower speed compared to that of the grinding wheel.

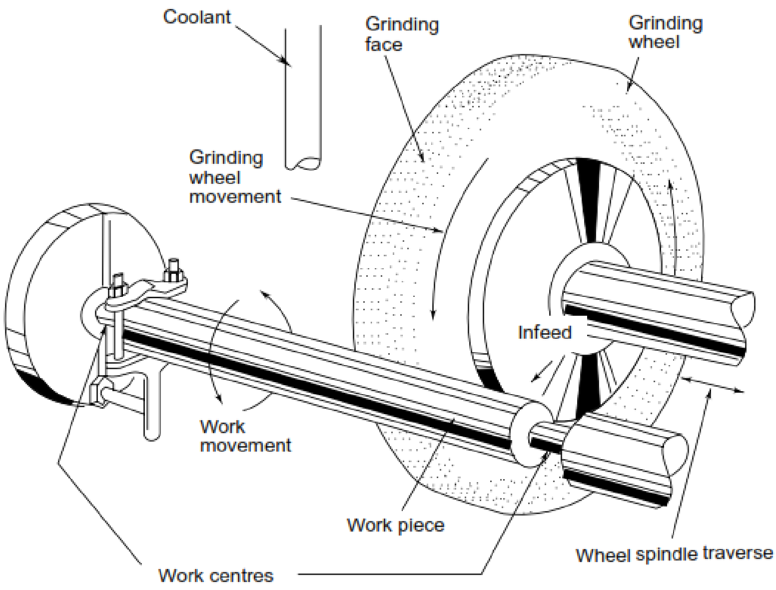


Fig.9 Cylindrical grinding operation

Surface grinding machines are generally used for generating flat surfaces. By far these are used for the largest amount of grinding work done in most of the machine shops. These machines are similar to milling machines in construction as well as motion. There are basically four types of machines depending upon the spindle direction and the table motion

Horizontal Spindle and Rotating Table

In this machine the grinding wheel cuts on its periphery, while the spindle traverses horizontally from the edge to the centre of the table. Feed is accomplished by moving the work mounted on the table up into the wheel with the table moving in a rotary fashion. Since the table and work rotate in a circle beneath the grinding wheel, the surface pattern is a series of intersecting arcs. This machine is used for round, flat parts because the wheel is in contact with the work at all times.

Vertical Spindle and Rotating Table

Vertical spindle machines are generally of bigger capacity. Complete machining surface is covered by the grinding wheel face. They are suitable for production grinding of large flat surfaces. In this machine both the work and the wheel rotate and feed into each other. By taking deep cuts this machine removes large amounts of material in a single pass. The side or the face of the wheel does the grinding. The wheel can be either complete solid or split into segments to save wheel material and in the process also provide cooler grinding action. In the case of small parts, the surface patterns created are a series of intersecting arcs if they are off centre around the table. It is a versatile machine and can be used to grind production parts and very large parts, as well as for grinding large batches of small parts

Horizontal Spindle and Reciprocating Table

The table in the case of reciprocating machines is generally moved by hydraulic power. The wheel head is given a cross feed motion at the end of each table motion. In this machine, the wheel should travel over the work piece at both the ends to prevent the grinding wheel removing the metal at the same work spot during the table reversal. This is the most common grinding machine found in the tool rooms. The tables for this type of machines are rectangular and usually 150 mm wide by either 300 mm or 450 mm long. The high production types have tables as big as 2m by 5 m. The grinding wheels cut on their peripheries and vary in sizes from 175 mm in diameter and 12.5 mm in width to 500 mm in diameter and 200 mm in width. This type of surface grinder is the most commonly used because of its high accuracy and the fine surface finishes that it imparts. The grinding wheel traverses in straight patterns that result in superior finish and high precision.

Vertical Spindle and Reciprocating Table

The grinding wheel in this machine is cylindrical and cuts on its side rather than on its periphery. The work is fed by the reciprocating motion of the table. Generally, the diameter of the wheel is wider than the work piece and as a result no traverse feed is required. These are generally high-production machine tools removing large amounts (as much as 10 mm) in a single pass.

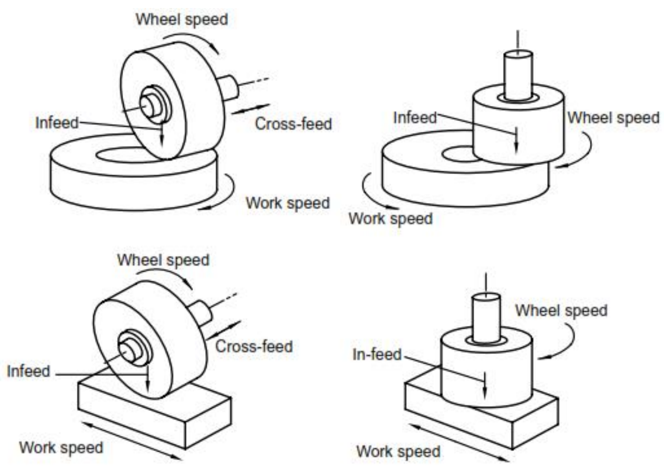


Fig.10 Different surface Grinding operations

Centre less grinding makes it possible to grind cylindrical work pieces without actually fixing the work piece using centres or a chuck. As a result no work rotation is separately provided.

Internal Centre-less Grinding is also possible to apply centre-less grinding for internal surfaces as well. However, in this case, the work piece needs to be supported by two support rolls

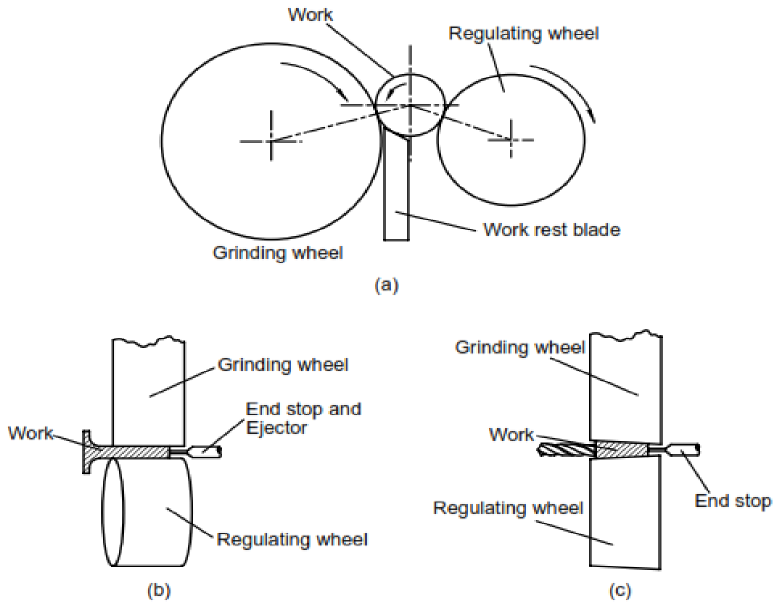


Fig.11 Centre less grinding operations; (a) Through feed, (b) in feed and (c) end feed

Advantages of centre less grinding

1. There is no need for having and maintaining centres and centre holes.
2. Work pieces can be loaded and unloaded from the machine rapidly. Grinding is almost continuous for through feed grinding.
3. Backing up the work piece by the regulating wheel and work rest blade practically eliminates any
4. Deflection of the work piece. This permits maximum material removal rates.
5. Minimum wear is observed in view of the large grinding wheels used. This minimises the adjustments needed for staying within dimensional tolerances and maximises the periods of time between wheel dressings.
6. Work pieces may often be loaded into the machine by the automatic feeding devices.
7. Less grinding allowance may be required, because the out-of-roundness is corrected across the
8. Diameter rather than the radius.

Limitations of centre less grinding

1. Setup time for a centre less grinding operation is usually large.
2. This process is useful only for large volume production. It may be necessary to have special equipment and additional setup time for special profiles.
3. This process is not suitable for large work piece sizes.

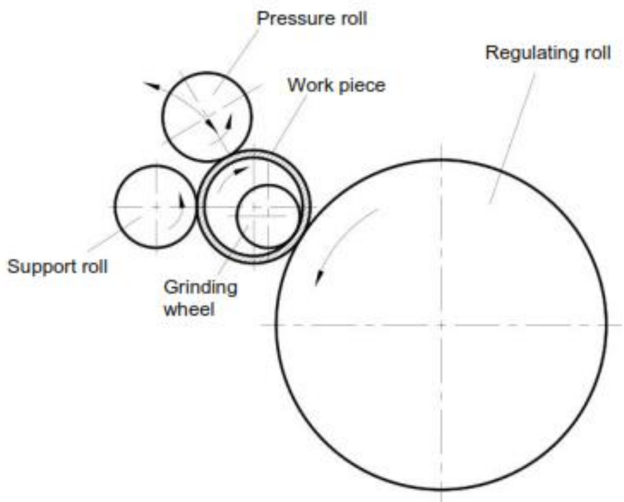


Fig.12 Internal Centre-less grinding operation

Grinding process

To understand the grinding process, it is convenient to study the metal removal process by the abrasive grain. For the sake of simplicity the surface grinding process is considered.

Surface Grinding - the removal of metal by a single abrasive grit in surface grinding operation

In surface grinding the stock removal rate, Q is given by

$$Q = bdv$$

In the case of cylindrical grinding it is

$$Q = 2\pi R_w df$$

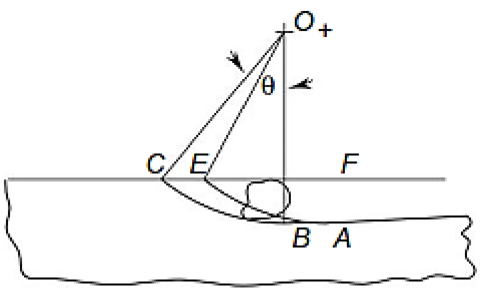
where d = depth of cut

v = work velocity for surface grinding

b = width of cut for surface grinding

f = wheel traverse velocity (feed rate)

R_w = Work radius



specific cutting energy, U_s is given by

$$U_s = \frac{F_h V}{vbd}$$

where F_h = the average horizontal force and V is the grinding wheel peripheral velocity.

Table 7 Grinding process parameters on performance

Variable	Grinding Ratio	Net Power	Surface Finish
Increase in wheel speed	Increased	Slightly increased	Improved
Increased depth of cut	Decreased	Increased	Deteriorates
Increase in work speed	Decreased	Increased	Deteriorates
Increase in work diameter	Increased	Slightly Increased	No significant change
Increase in metal removal rate	Decreased	Increased	Deteriorates
Increase in work material hardness	Optimum exhibited	Slightly Increased	Improves

Grinding process parameters

The operating parameters of a grinding process are:

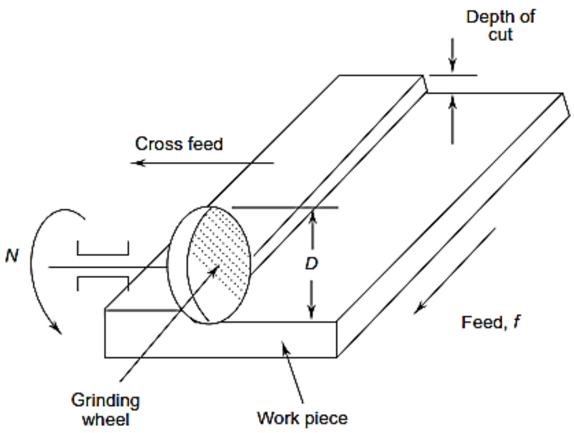
- Wheel speed
- Work speed
- Traverse speed
- In feed
- Area of contact

Table 8 Recommended grinding wheel speeds

Type of Grinding	Bond	Wheel Speed, m/s
Rough grinding	Vitrified	25
Rough grinding	Resinoid	45
Surface grinding	Vitrified	20–25
Internal grinding	Vitrified	20–35
Centre less grinding	Vitrified	30–80
Cylindrical grinding	Vitrified	30–35
Cutting off	Resinoid	45–80
Hand grinding of tools		20–25
Automatic grinding of tools		25–35
Hand grinding of carbide tools		18–25
Automatic grinding of carbide tools		4–20

Table 9 Recommended work speeds

Work Material	Cylindrical Grinding		Internal	Surface
	Roughing m/min	Finishing m/min	Grinding m/min	Grinding m/min
Soft steels	–15	6–8	15–20	
Hard steels	14–16	6–10	18–22	
Cast iron	12–15	6–10	18–22	8–15
Brass	18–20	14–16	28–32	
Aluminium	50–70	30–40	32–35	



Fig,13 Grinding operation

Approach

$$A = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - d\right)^2} = \sqrt{d\,(D - d)}$$

$$\text{Time for one pass} = \frac{\text{Length + Diameter}}{\text{Table feed rate}}$$

$$\text{Number of passes required} = \frac{\text{Width}}{\text{Infeed rate}}$$

Example

Using a horizontal axis surface grinder a flat surface of C65 steel of size 100 X 250 mm is to be ground. The grinding wheel used is 250 mm in diameter with a thickness of 20 mm. Calculate the grinding time required. Assume a table speed of 10 m/min and wheel speed of 20 m/s.

Solution The rpm of the grinding wheel $= \dfrac{1000 \times 20 \times 60}{\pi \times 250} = 1528$ rpm

Let approach distance = 125 mm

Time for one pass $= \dfrac{250 + 250}{10 \times 1000} = 0.05$ minutes

Assuming an in feed rate of 5 mm/pass

Number of passes required = 100/5 = 20

Total grinding time = $20 \times 0.05 = 1$ minute.

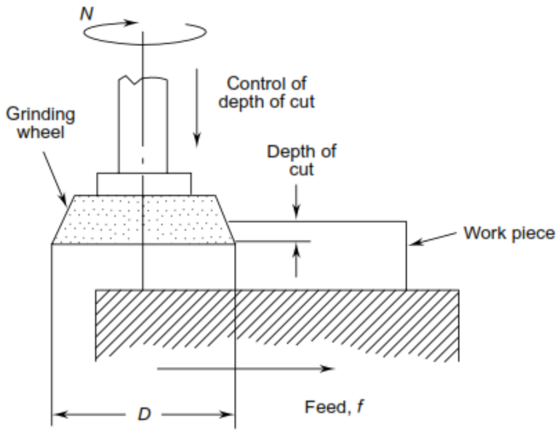
Fig. 9.17 shows the situation of a surface grinding operation using a vertical axis machine.

Approach distance for this case is given as

$$A = \frac{D}{2} \qquad \text{for } W = \frac{D}{2} \text{ up to } D$$

$$A = \sqrt{W\,(D - W)} \qquad \text{for } W < \frac{D}{2}$$

Where W = width of cut



Creep feed grinding is a new form of grinding operation, different from the conventional grinding process. In creep feed grinding the entire depth of cut is completed in one pass only using very small in feed rates. As shown in Fig. this process is characterised by high depth of cut of the order of 1 to 30 mm with low work speeds of the order of 1 to 0.025 m/min. The actual material removal rates calculated from these process parameters are generally in the same range as that of the conventional grinding.

The cutting forces and consequently the power required increases in the case of creep feed grinding, but has a favourable G-ratio. It is necessary to continuously dress the grinding wheel (to reduce the wheel dullness) for efficient operation. This however causes wheel wear and the necessity to adjust the wheel head. Use soft and open wheels to take care of the wheel dressing and accommodate large volume of chips generated in the process. The grinding wheel speeds used are also low of the order of 18 m/s compared to the 30 m/s used in the conventional grinding operations. Also the in feed rates used are low of the order of 0.005 mm/pass. The grinding fluids used are oil based in view of the low grinding speeds employed. However, the volume of grinding fluid is much more compared to the conventional grinding, in view of the high heats generated in the process. It is possible to achieve higher material removal rates by employing continuous dressing of the grinding wheel using a diamond dresser wheel.

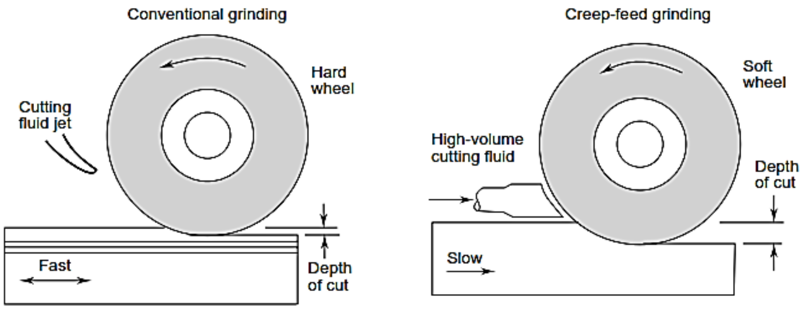


Fig.14 Creep Feed Grinding operation

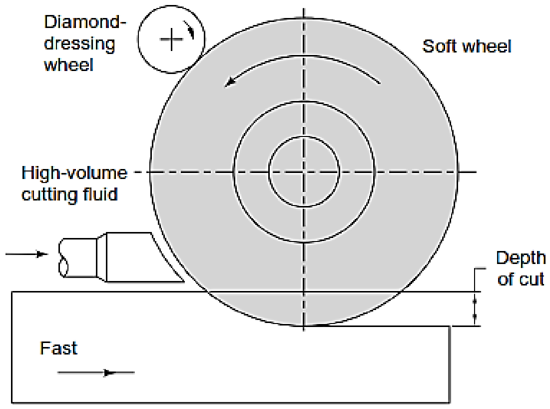


Fig.15 Creep Feed Grinding operation with continuous dressing

Honing is a low abrading process using bonded abrasive sticks for removing stock from metallic and non-metallic surfaces. However, it can also be used for external cylindrical surfaces as well as flat surfaces, for which it is rarely used. Commonly it is used for internal surfaces. This is an operation performed as the final operation to correct the errors that result from the previous machining operations. The characteristics that can be achieved by the honing process are:

- Correction of geometrical accuracy
    - Out of roundness
    - Taper
    - Axial distortion
- Dimensional accuracy
- A finish surface pattern is generated by the characteristic motion of the abrasive grains that provide the best possible surface to promote optimum lubricating conditions.

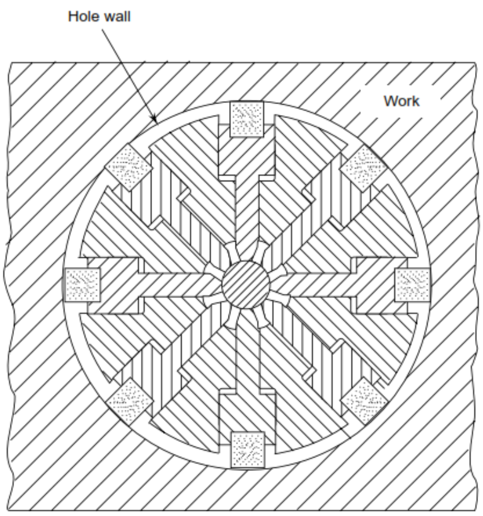


Fig.16 Honing

Abrasive grains are bonded in the form of sticks by a vitreous or resin material and the sticks are presented to the work so that their full cutting forces are in contact with the work surfaces. Since a large number of abrasive grains are presented to the work surface simultaneously, substantial material removal takes place. For cylindrical surfaces the abrasive grains are given a combination of two motions - rotation and reciprocation. The resultant motion of the grains is a cross hatch lay pattern as shown in Fig. with an included angle of 20 to 60°.

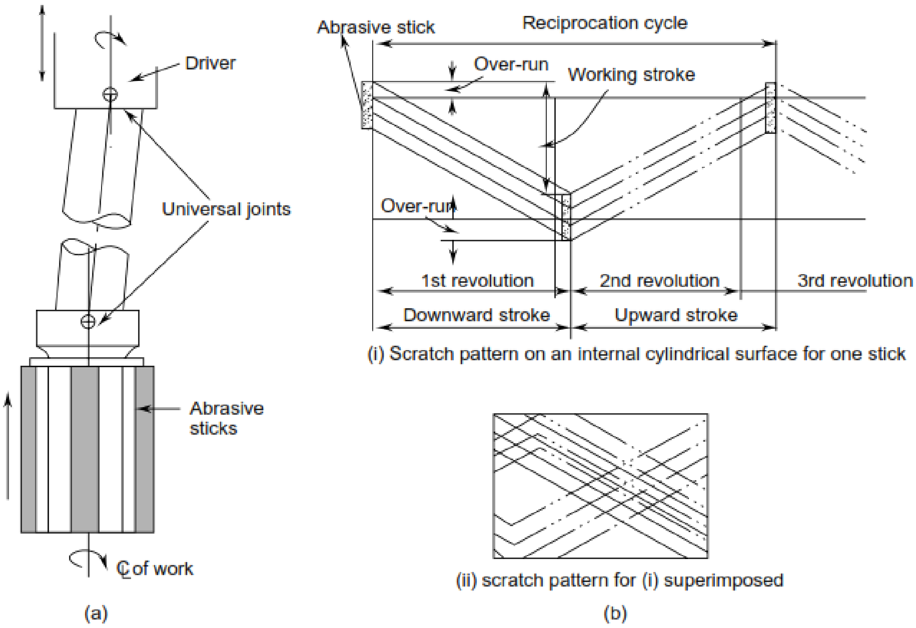


Fig.17 Honing operation

Honing Conditions

All materials can be honed. However, the material removal rate is affected by the hardness of the work material. The typical rates are:

- Soft material 1.15 mm/min on diameter
- Hard materials 0.30 mm/min on diameter

Maximum bore size that can be conveniently honed is about 1500 mm while the minimum size is 1.5 mm in diameter. Honing allowance should be small to be economical. However, the amount also depends upon the previous error to be corrected. The abrasive and the grain size to be selected depend upon the work material and the resultant finish desired.

Table 10 Selection of honing stone characteristics

Work Material	Hardness BHN	Abrasive	Grade	Grain Size for a Surface Finish, µm			
				0.01	0.025	0.3	0.4
Steel	200–300	Al_2O_3	R	600	500	400	320
	330–470		O	600	500	400	320
	50–65 R_C		J	500	400	320	280
Cast iron	200–470	SiC	Q	500	400	280	280
	50–65 R_C	SiC	J	400	280	220	150
Aluminium	120–140	SiC	R	600	500	400	320
Copper	180–200	SiC	R	600	500	400	320
		SiC	R	600	500	400	320

Table 11 Selection of honing process parameters

Material	Hardness R_C	Honing Speed, m/min			
		Rough Honing		Finish Honing	
		Rotary Speed	Reciprocating Speed	Rotary Speed	Reciprocating Speed
Cast iron	15–20	23–28	10–12	32	13.5
Steel	15–35	18–22	9–11	25	12
	35–60	14–21	12–15	28	17.5
Alloy steels	25–50	23–28	10–12	31	12
Bronze	8–15	21–26	12–26	30	17.5
Aluminium	–	21–26	12–26	31	17

Lapping

Lapping is generally the final finishing operation done with loose abrasive grains. The process is employed to get

- Extreme accuracy of dimension
- Correction of minor imperfection of shape
- Refinement of surface finish
- Close fit between mating surfaces

The service life of components which are in close contact during machining can be greatly increased by the lapping process which removes the valleys and hills present on the machined surfaces. Typical finishes obtained in the lapping process.

Table 12 Surface finish achieved in lapping process

Abrasive Used	Grain Size	Surface Finish, µm
Silicon carbide	220	0.75–1.00
	320	0.64–0.75
	400	0.46–0.64
	500	0.38–0.46
	600	0.25–0.38
	800	0.13–0.25
Aluminium oxide	400	0.08–0.13
	800	0.05–0.08
	900	0.03–0.08

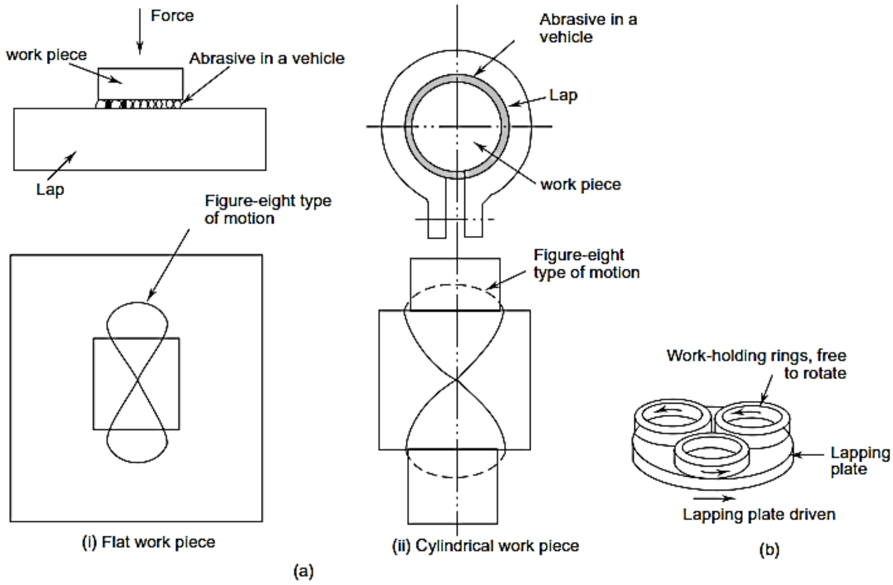


Fig.18 Lapping operation

Special lubricants generally called vehicles are used during the lapping process. The desirable properties of fluids used as vehicles are:

1. Abrasive should be held in uniform suspension during the operation.
2. It should not evaporate easily
3. It should be non-corrosive.
4. It can be easily removed by normal cleaning.

The materials which satisfy the above criteria are water soluble cutting fluids, vegetable oils, mineral oils and greases.

Lapping speed is 100 to 250 m/min. The material removed depends upon the lapping speed. Higher lapping allowances require higher lapping speeds. The lapping pressure applied is 0.01 to 0.03 MPa for soft materials and 0.07 MPa for hard materials. Higher pressures are likely to cause scouring of the work surface. Lapping allowance depends on the previous operation carried and the material hardness.

Table 13 Lapping allowances

Work Material	Lapping Allowance, mm
Cast iron	0.2
Aluminium	0.1
Soft Steel	0.01–0.02
Ductile Steel	0.05–0.10
Hardened Steel	0.005–0.020
Glass	0.03
Cemented Carbide	0.03–0.05
Bronze	0.03

Lapping can be carried out on flat surfaces as well as any other forms such as cylindrical or any form surfaces. The lap has to match the form surface required. A few points to be noted with lapping operation are:

- It is more difficult to lap soft metals than hard metals.
- Lap should be softer than the work material.
- The hardness of the abrasive should be based on the hardness of the work material. Softer work materials require softer abrasives and vice versa.
- The lapping medium should preferably be a little viscous to hold the abrasive so that it resists the movement or rolling of abrasive granules to help with the removal of the material.
- The lapping medium should be used sparingly. The increase in lapping medium may increase the material removal rate but not its ability to correct the errors in the part.
- Laps with serrations or grooves are preferable for flat surfaces with large areas while laps with no serration or grooves are preferred for cylindrical lapping.
- To get higher accuracy in lapping use a hard lap that cuts slowly and wears faster with a duller finish.
- A soft lap on the other hand cuts faster, wears longer and gives a brighter surface.
- As a thumb rule an abrasive particle will produce a scratch that is approximately half its size. For example, a 10 micron abrasive particle will produce a scratch of 5 microns.

Super Finishing

Super finishing is another abrasive process utilising either a bonded abrasive like honing for cylindrical surfaces or a cup wheel for flat surfaces. It is generally used for:

- Removing surface fragmentation.
- Reduce surface stresses and burns and thus restore surface integrity.
- Correct inequalities in geometry.
- Produces high wear resistant surface on any object which is symmetrical. Typical surfaces that are surface finished are cylindrical, flat, conical and spherical.

Contact surface in super finishing is large and the tool maintains a rotary contact with the work piece while oscillating.

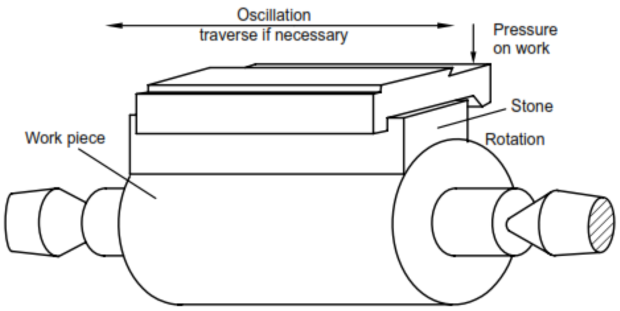


Fig.19 Super finishing operation

Polishing and buffing

Both these processes are used for making the surfaces smoother along with a glossy finish. Polishing and buffing wheels are made of cloth, felt or such material, which is soft and have a cushioning effect. Polishing is done with a very fine abrasive in loose form smeared on the polishing wheel with the work rubbing against the flexible wheel. A very small amount of material is removed in polishing. In buffing the abrasive grains in a suitable carrying medium such as grease are applied at suitable intervals to the buffing wheel. Negligible amount of material is removed in buffing while a very high lustre is generated on the buffed surface. The dimensional accuracy of the parts is not affected by the polishing and buffing operations.

Abrasive Belt Grinding

In this process a continuous moving belt with abrasive is used for grinding the surfaces. The abrasive belt is normally passed between two wheels with one being driven while the other is idling as shown in Fig. Use of abrasive belts results in much cooler cutting and rapid material removal rates compared to conventional grinding. The work piece is oscillated across the face of the abrasive belt to obtain a uniform belt wear and surface finish.

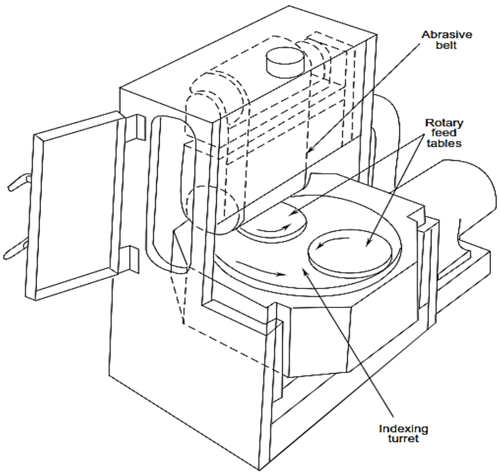


Fig. 20 Abrasive belt grinding

Gear cutting

Gears are important machine elements that transmit power and motion. There is a large variety of gears used in industrial equipment as well as a variety of other applications. The gear teeth have the contact surface as an involute curve along which they roll and slide while transmitting the motion. An involute curve is generated by unwinding a tautly held string from a base circle. The involute curve is chosen because it is simple, easy to manufacture and allows the centre lines to be varied between the mating gears. Pitch circle, which is an imaginary circle on the gear, corresponds to the diameter of the wheel. For proper operation, the two pitch circles must be tangential at the point where the centre line connecting the two centres of rotation intersects the pitch circle. To reduce the friction, gears are designed such that the teeth have rolling motion rather than the sliding motion. Gear tooth size is identified as module.

Pitch diameter = No. of teeth x module

Tooth thickness = 0.5 x p x module

Total depth = 2.25 x module

Spur gear is the most common and easy to make. It has straight teeth on the periphery of a cylinder. It transmits motion between two parallel shafts. A rack is a gear with infinite radius, having the teeth that lie on a straight line.

Gear Forming

This is a process of machining gears using a form milling cutter in a milling machine. Gears can be cut using a form milling cutter, which has the shape of the gear teeth. The form milling cutter called 'dp' (diametral pitch, used in inch systems, which is equivalent to inverse of module) cutters have the shape of the teeth similar to the tooth space with the involute form of the corresponding size gear. The commercial gear milling cutters are available as a set for a given module or diametral pitch. These can be used on either horizontal axis or vertical axis milling machines, though horizontal axis is more common.

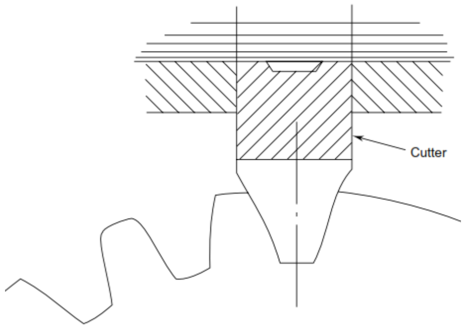


Fig.21 Form milling for spur gear

Table 14 DP cutter used for form milling

Cutter Number	Range of Teeth that can be Cut	Cutter Number	Range of Teeth that can be Cut
1	135 to Rack	1.5	80 to 134
2	55 to 134	2.5	42 to 54
3	35 to 54	3.5	30 to 34
4	26 to 34	4.5	23 to 25
5	21 to 25	5.5	19 to 20
6	17 to 20	6.5	15 to 16
7	14 to 16	7.5	13
8	12 to 13		

Gear generation

Gear Shaping and Planing

To understand the concept of gear shaping, imagine a gear blank, which has a periphery that is very soft and easily deformable. An ordinary involute gear is pressed into the rim of the gear blank until the two pitch circles are in contact. Then the gear blank is rolled together with the gear such that the pitch circles roll on each other without slipping. Since the rim of the gear blank is soft the gear teeth will be pressed and theoretically correct teeth will be formed on the gear blank. The teeth so formed will mesh with any other involute gear of the same module regardless of the number of teeth.

In actual gear shaping operations, the gear cutter is not actually pressed but removes the material by reciprocation similar to a vertical shaper. The gear shaper cutter, which is very similar to the gear, but the teeth are form relieved to act as the cutting edges.

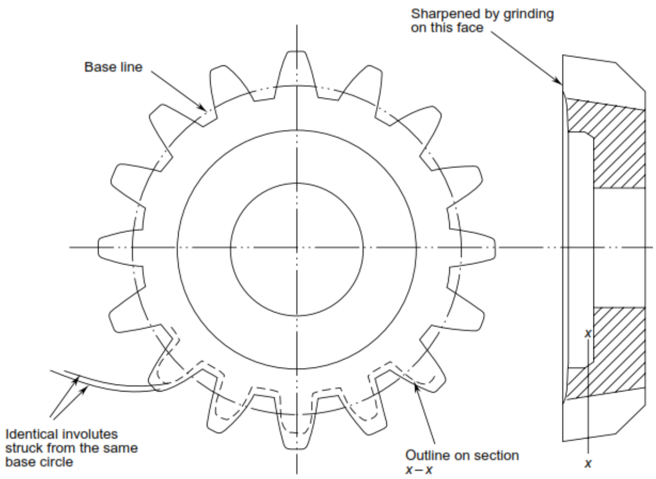


Fig.22 Gear shaping operation

The gear shaper cutter is mounted on a vertical ram and is rotated about its axis as it performs the reciprocating action. The work piece is also mounted on a vertical spindle as shown in Fig. 10.16 and rotates in mesh with the shaping cutter during the cutting operation. The relative rotary motions of the shaping cutter and the gear blank are calculated as per the requirement and incorporated with the change gears. The cutter will slowly move into the gear blank surface with incremental depths of cut, till it reaches the full depth. The cutter and gear blank are separated during the return (up) stroke and comes to the correct position during the cutting (down) stroke. Gear shaping can cut internal gears, splines and continuous herringbone gears that cannot be cut by the other processes. This process can also cut gears close to a shoulder with very small clearance.

Gear Hobbing

Gear hobbing is a continuous process eliminating the unproductive return motion of the gear shaping operation. To understand the concept of gear hobbing, an analogy similar to that used in gear shaping is given. Imagine an involute rack being pressed into the gear blank with a very soft rim up to the point when the pitch circles of the rack and the gear blank meet. During this process the rack is moved length-wise while the gear blank is rotated such that it rolls with the rack without slipping. Theoretically correct tooth profiles would be formed on the gear blank rim as it is very soft. The number of teeth formed depends upon the size of the gear blank used.

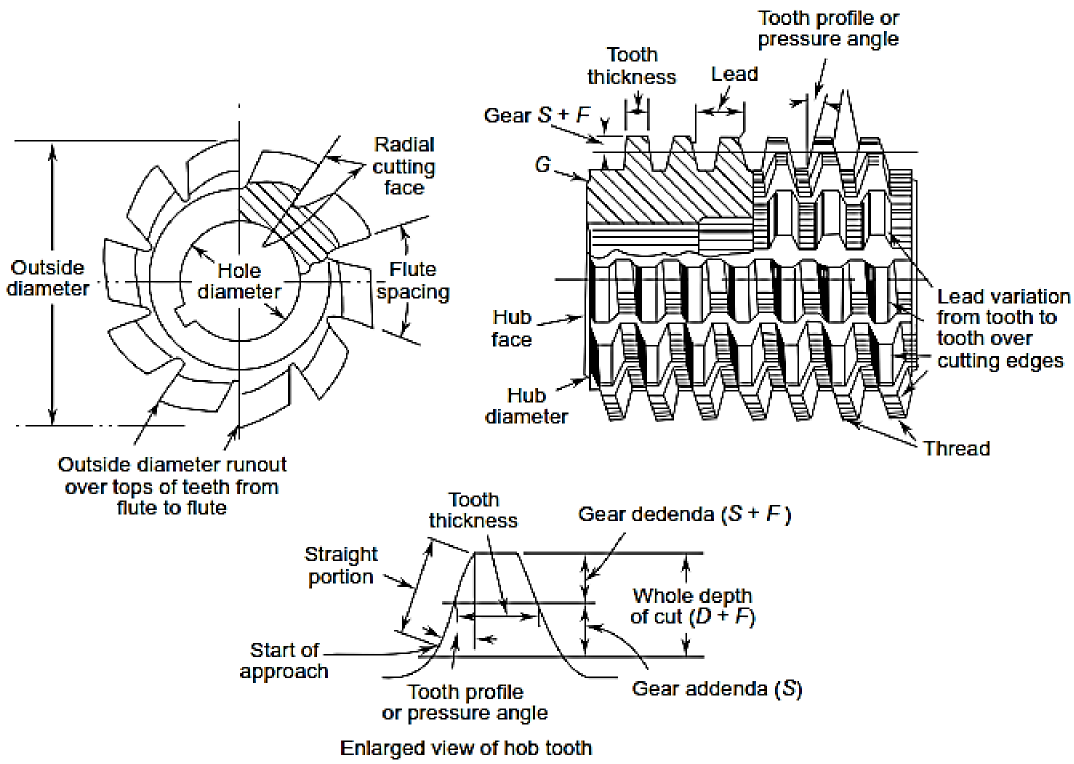


Fig.23 Gear Hobbing

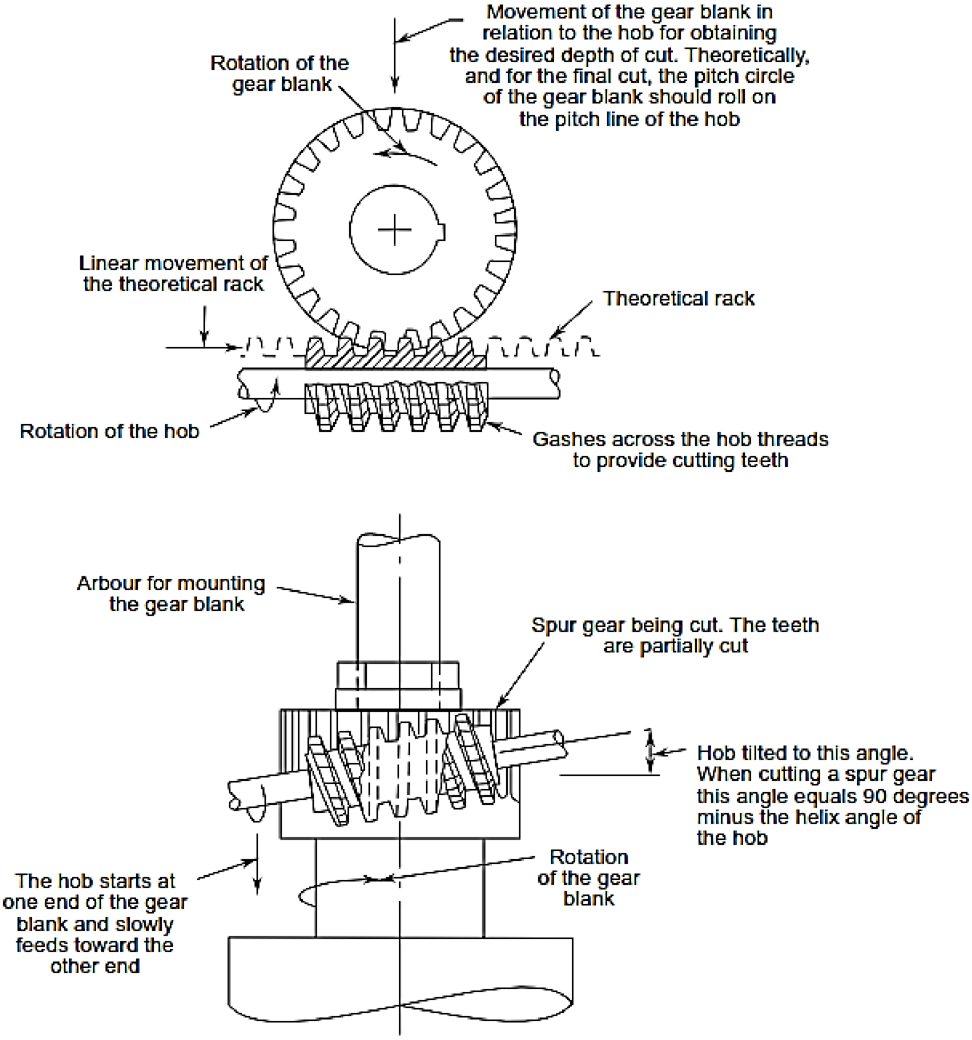


Fig.24 Gear hobbing operation

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AUTOMOBILE ENGINEERING

SMEA 1505 _ MANUFACTURING TECHNOLOGY II

UNIT V NUMERICAL CONTROL MACHINES AND PART PROGRAMMING

UNIT – V NUMERICAL CONTROL MACHINES AND PART PROGRAMMING

Small batch sizes will not be able to take advantage of the mass production techniques such as special purpose machines or transfer lines. Hence the need for flexible automation where you get the benefits of rigid automation but also be able to vary the products manufactured thus bringing in flexibility. Numerical control fits the bill perfectly, and we will see that future manufacturing would increasingly be dependent on'Numerical Control' or NC to be short.

Numerical Control (NC) or control by numbers is the concept which has revolutionised the manufacturing scene, which is partially due to the rapid advancement in microelectronics that has taken place since the late 1960's. The key factor responsible for the popularity of the numerical control is the flexibility it offers in manufacturing.

Numerical control

Numerical control of machine tools may be defined as a method of automation in which various functions of machine tools are controlled by letters, numbers and symbols. Basically a NC machine runs on a program fed to it. The program consists of precise instructions about the methodology of manufacture as well as the movements. For example what tool to be used, at what speed, at what feed and to move from which point to which point in what path. Since the program is the controlling point for product manufacture, the machine becomes versatile and can be used for any part. All the functions of a NC machine tool are therefore controlled electronically, hydraulically or pneumatically.

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

- starting and stopping of machine tool spindle
- controlling the spindle speed
- positioning the tool tip at desired locations and guiding it along desired paths by automatic control of the motion of slides
- controlling the rate of movement of the tool tip (i.e. feed rate)
- changing of tools in the spindle.

Initially the need of NC machines was felt for machining complex shaped small batch components as those belonging to an aircraft. However, currently this spectrum encompasses practically all activities of manufacturing, in particular the capital goods and white goods. Thus the range covered is very wide. Besides machining, with which we are concerned in this book, NC has been used in a variety of manufacturing situations. The majority of applications of NC are still in metal cutting machine tools such as milling machines, lathes, drilling machines, grinding machines and gear generating machines. Besides a number of metal forming machine tools such as presses, flame cutting machines, pipe bending and forming machines, folding and shearing machines also use NC for their programme control. The inspection machines called Co-ordinate Measuring Machines (CMM) are also based on NC. Lastly the robots basically may be material handling units, but their control principles are very close to the NC. Besides these applications listed for manufacturing, other applications such as filament winding or assembly machines based on the NC principles could also be widely seen in the industry.

NC machines have been found quite suitable in industries such as

1. For the parts having complex contours, that cannot be manufactured by conventional machine tools.
2. For small lot production, often for even single (one off) job production, such as for prototyping, tool manufacturing, etc.
3. For jobs requiring very high accuracy and repeatability.
4. For jobs requiring many set-ups and/or the set-ups are very expensive.
5. The parts that are subjected to frequent design changes and consequently require more expensive manufacturing methods.
6. The inspection cost is a significant portion of the total manufacturing cost.

One or more of the above considerations would justify the processing of a part by a NC machine tool. NC is superior to conventional manufacturing in a number of ways. The superiority comes because of the programmability. These are as follows:

1. Parts can be produced in less time and therefore are likely to be less expensive. The idle (non-cutting) time is reduced to absolute minimum. This of course depends on the way the program for the part is written. The endeavour of the machine tool builder is to provide facility where by the non-cutting time can be brought to the barest minimum possible. It is possible to reduce the non-productive time in NC machine tools in the following ways:
    - by reducing the number of set-ups,
    - by reducing set-up time,
    - by reducing work piece-handling time, and
    - by reducing tool-changing time.
    - These make NC machines highly productive.

2. Parts can be produced more accurately even for smaller batches. In the conventional machine tools, precision is largely determined by human skills. NC machines, because of automation and the absence of interrelated human factors, provide much higher precision and thereby promise a product of consistent quality for the whole of its batch.
3. The operator involvement in part manufacture is reduced to a minimum and as a result less scrap is generated due to operator errors. No operator skill is needed except in setting up of the tools and the work. Even here the setup has been simplified to a very great extent.
4. Since the part program takes care of the geometry generated, the need for expensive jigs and fixtures is reduced or eliminated, depending upon the part geometry. Even when the fixture is to be used, it would be very simple compared to a conventional machine tool. It is far easier to make and store part programs (tapes).
5. Inspection time is reduced, since all the parts in a batch would be identical, provided proper care is taken about the tool compensations and tool wear in part program preparation and operation. With the use of inspection probes in the case of some advanced CNC controllers, the measurement function also becomes part of the program.
6. The need for certain types of form tools is completely eliminated in NC machines. This is because the profile to be generated can be programmed, even if it involves 3 dimensions.
7. Lead times needed before the job can be put on the machine tool may be reduced to a great extent depending upon the complexity of the job. More complex jobs may require fixtures or templates if they are to be machined in the conventional machine tools, which can be reduced to a large extent.
8. CNC machining centres can perform a variety of machining operations that have to be carried out on several conventional machine tools, thus reducing the number of machine tools on the shop floor. This saves the floor space and also results in less lead-time in manufacture. This results in the overall reduction in production costs.
9. Set-up times are reduced in a number of situations, since the set-up involves simple location of the datum surface and position. Further the required number of setups can also be reduced. All this translates into lower processing times. Many a times, a component could be fully machined in a single machining centre or turning centre, each of which has wider machining capabilities. In conventional manufacture if the

part has to be processed through a number of machine tools which are located in different departments, the time involved in completion and the resultant process inventory, would be large. This would be greatly eliminated by the use of NC machine tools.

10. Machining times and costs are predictable to a greater accuracy, since all the elements involved in manufacturing would have to be thoroughly analysed before a part program is prepared.
11. Operator fatigue does not come into picture in the manufacturing of a part. The NC machine tool can be utilised continuously since these are more rigid than the conventional machine tools.
12. Tools can be utilised at optimum feeds and speeds that can be programmed.
13. The modification to part design can be very easily translated into manufacture by the simple changes in part programs without expensive and time consuming changes in jigs, fixtures and tooling. This adds to the flexibility of manufacture.
14. The capability (metal removal) of NC machines is generally high because of the very rigid construction employed in machine tool design compared to the conventional machine tools.

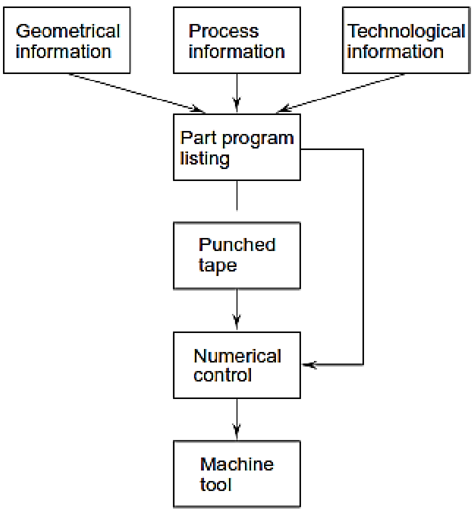
Though the NC machines have a range of advantages, there are certain limitations one should take care of while making a choice in favour of them.

1. The cost of NC machine tool is much high compared to an equivalent conventional machine tool. The cost is often 5 to 10 times higher. Also the cost of tooling is high. This is a very high initial investment. All this makes the machine hourly rate high. As a result, it is necessary to utilise the machine tool for a large percentage of time.
2. Cost and skill of the people required to operate a NC machine is generally high in view of the complex and sophisticated technology involved. The need is for part programmers, tool setters, punch operators and maintenance staff (electronics and hydraulics) who have to be more educated and trained compared to the conventional machine operators.
3. Special training is needed for the personnel manning the NC machine tools. NC manufacturing requires training of personnel both for software as well as hardware. Part programmers are trained to write instructions in desired languages for the machines on the shop floor. They also need to be acquainted with the manufacturing process. Similarly, machine operators have to be prepared for the new NC culture. These factors are important for the successful adoption and growth of NC technology.

4. As NC is a complex and sophisticated technology, it also requires higher investments for maintenance in terms of wages of highly skilled personnel and expensive spares. The need for maintenance engineers trained in all the sub systems present such as mechanical, hydraulic, pneumatic and electronics makes the job more difficult. Though the latest machines are equipped with a large number of diagnostic facilities, maintenance is still one of the major limitations.
5. The automatic operation of NC machines implies relatively higher running costs. Moreover, the requirement of a conditioned environment for operating NC technology adds further to the running costs.

NC machine tools

The CNC machining centre, at the moment appears to be the most capable and versatile automatic machine tool that can perform drilling, milling, boring, reaming and tapping operations. The general objective behind the development of NC machine tools continues to remain the reduction of cost of production by reducing the production time. This in turn is directed towards the avoidance of non-productive time which is mainly due to the number of set-ups, set-up time, work piece handling time, tool change time and lead time. The performance of a variety of machining operations on the same machining centre eliminates the non-productive waiting time that occurs if such operations are performed on several different machines. Provision of automatic tool changing, indexing of tables and several pallets further add to the productivity of the machining centres.



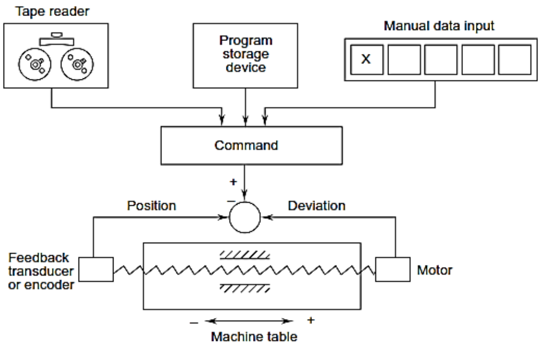


Fig.1 Data processing in a CNC machine tool in closed loop control

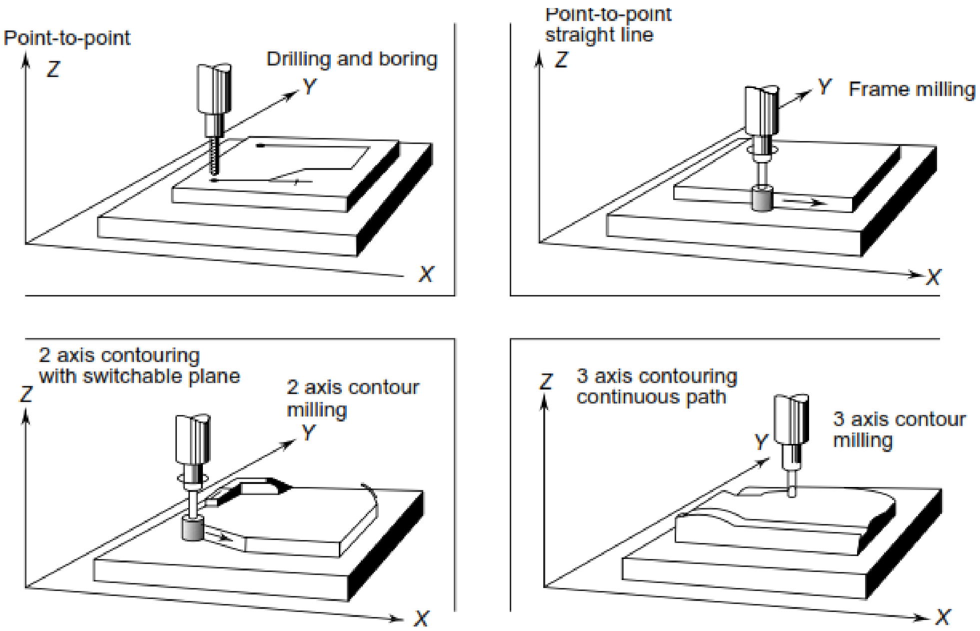


Fig. 2 Control system in CNC operation

Machine Control Unit

Every NC machine tool is fitted with a machine control unit (MCU) which performs the various controlling functions under the program control. The MCU may be generally housed in a separate cabinet-like body or may be mounted on the machine itself. When separately mounted, it may sometimes be like a pendant which could swing around for convenient handling by the operator. Appearance wise it looks like a computer with a display panel generally of small size (9 inches), and a number of buttons to control the machine tool along with a keyboard. This control unit controls the motion of the cutting tool, spindle speeds, feed rate, tool changes, cutting fluid application and several other functions of the machine tool.

Part Program

This is a very important software element in the NC manufacturing system. It looks like a computer program containing a number of lines/ statements/ instructions (called NC blocks). It is, therefore, the detailed plan of manufacturing instructions proposed for machining the part. It is written keeping in view the vocabulary understood by the MCU in terms of various standard words, codes and symbols. The format and the language are dependent on the machine tool hardware and the MCU. Some typical NC blocks written in the word address format as per ISO are shown below:

N30 G00 X120.0 Y 45.0 Z-85.0

N40 G90

N50 G03 X200.0 Y200.0 I-100.0 J0 F200

N60 G01 X120.0 Y110.0

The program can also be written in higher level languages such as APT, UNIAPT, COMPACT II, etc. These programs have to be converted into the earlier mentioned machine tool level program with the help of processors. It is similar to the practice by which computer programs written in high level languages such as FORTRAN are converted into the relevant computer machine language with the aid of a suitable compiler. Programming done in a language such as APT is mostly processed with the help of a computer and so is also known as computer aided part programming.

NC Tooling

The operator gathers, or is supplied with, the relevant tooling for the part to be machined. A distinctive deviation of the NC tooling from the conventional one is that each cutting tool is set in a different adapter the configuration suggested by ISO is now generally followed. A power-operated drawbar may be employed to pull the tooling at the retention knob. This helps eliminate any clearance between the mating surfaces of spindle and tooling shank. It is not uncommon to set apart an allocation of 20 to 30% of total budget for tooling during the buying of new NC machine tools.

A pre-set tool has adjustable locating faces. It enables the dimensions between the tool cutting edges and location faces to be pre-set to a close tolerance using a pre-setting device. The pre-set tool usually needs to be removed from the machine for adjustments required during batch production. The tools may be stored on a drum, which is operationally an integral part of the machine itself. In the latter case, the tools are automatically replaced or changed in the spindle. These inform the operator about the deviation of the tool tip the supplied tool has, with the one taken into account by the part programmer. The programmer gets the information from the tool files that are updated periodically.

Axes of NC Machines

The major component of a NC program involves the input of co-ordinates of the tool end point to produce any machining profile, it is necessary to follow a proper co-ordinate system.

Co-ordinate System

All the machine tools make use of the Cartesian co-ordinate system for the sake of simplicity. The guiding co-ordinate system followed for designating the axes is the familiar right hand co-ordinate system. The main axes to be designated are the rectangular axes and the rotary axes.

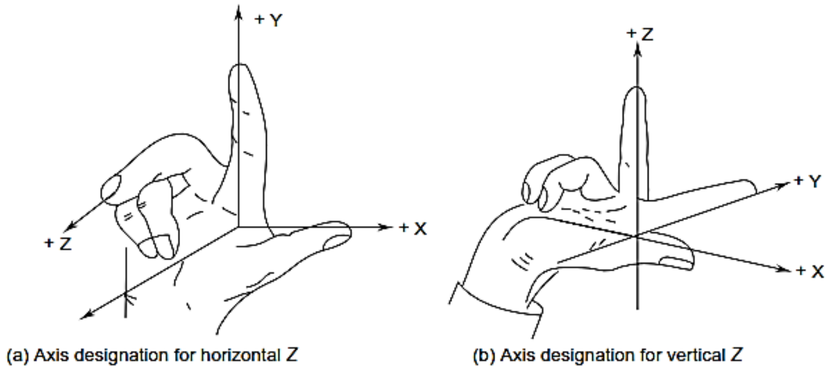


Fig.3 Finding directions in a Right Hand Co-ordinate System and also the positive directions for rotary motions

Designating the Axes

First axis to be identified is the Z-axis. This is followed by the X and Y axes respectively. Z-Axis and Motion Location the Z-axis motion is either along the spindle axis or parallel to the spindle axis. In the case of machine without a spindle such as shapers and planers, it is identified as the one perpendicular to the work-holding surface, which may or may not be passing through the controlled point (e.g. the cutting tool tip in case of shaper). Direction the tool moving away from the work holding surface towards the cutting tool is designated as the positive Z direction. This means in a drilling machine the drill moving into the work piece is the negative (–) Z direction. This helps in reducing the possible accidents because of wrong part program entry in the co-ordinate signs.

When there are several spindles and slide ways: In such cases, one of the spindles, preferably the one perpendicular to the work-holding surface may be chosen as the principal spindle. The primary Z motion is then close to the primary spindle. The tool motions of other spindle quills or other slides, which are termed as secondary and tertiary motions, may be designated as U, V, W and P, Q, R respectively. For other machines the positive (+) Z motion increases the clearance between the work surface and the tool holder.

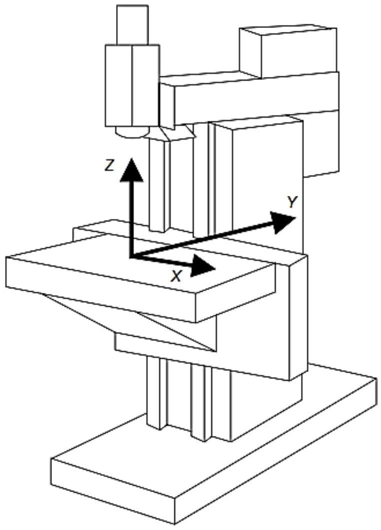


Fig. 4 Vertical axis Milling Machine or Machining centre

X-Axis

The X-axis is the principal motion direction in the positioning plane of the cutting tool or the work piece. Location it is perpendicular to the Z-axis and should be horizontal and parallel to the work- holding surface wherever possible. Direction when looking from the principal spindle to the column, the positive (+) X is to the RIGHT. For turning machines, it is radial and parallel to the cross slide. X is positive when the tool recedes from the axis of rotation of the work piece. For other machine tools, the X-axis is parallel to and positive along the principal direction of movement of the cutting or the guided point.

Y-Axis

It is perpendicular to both X- and Z- axes and the direction is identified by the right hand Cartesian co-ordinate system.

Rotary Motions

A, B and C define the primary rotary motions. Location these motions are located about the axis parallel to X, Y and Z respectively. In addition to the primary rotary motions, if there are secondary rotary motions, those should be designated as D or E regardless of whether they are parallel or not to A, B and C. Direction Positive A, B and C are in the directions which advance right hand screws in the positive X, Y, and Z directions respectively. In Fig. 17.7, the fingers of the right hand point towards the positive direction of the rotary motions. All the above-mentioned motions, viz. X, Y, Z; U, V, W; P, Q, R; A, B, C and D, E are with reference to a point, the movement of which is sought to be controlled. This point is generally the tip of the cutting tool.

CNC Machining Centres

CNC machine tools have grown from the conventional machine tools. For example, the successful tool room milling machines were converted to CNC by simply replacing the motion elements by automated devices. Later the CNC machine tools were redesigned with greater emphasis on the structural rigidity, power

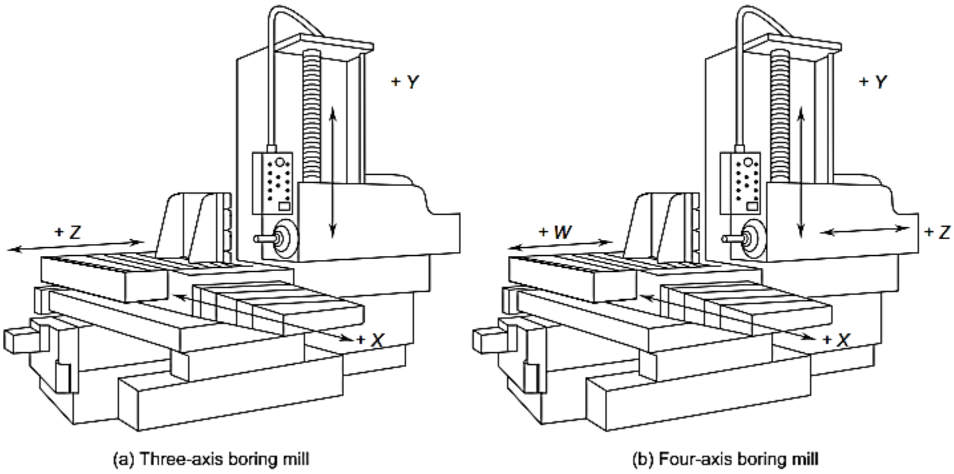


Fig. 5 CNC horizontal axes boring mill with 3 axes and 4 axes

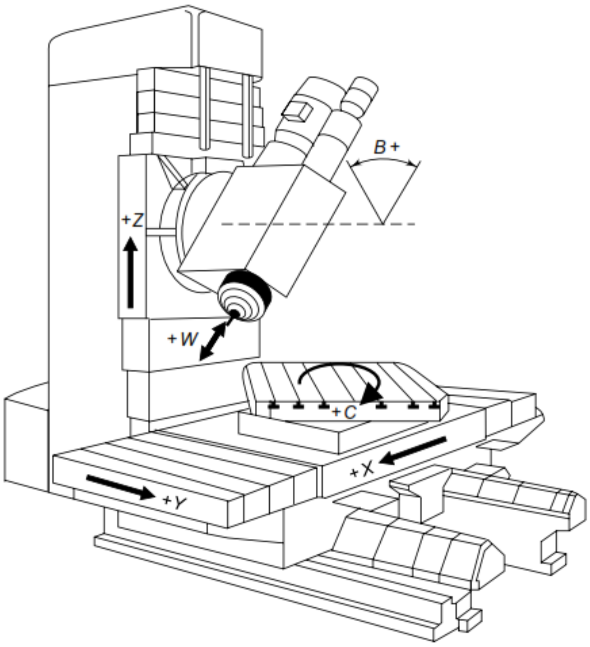


Fig. 6 CNC horizontal axes boring mill with 5 axes

The modern CNC milling machines have expanded machining capabilities by the addition of accessory devices, making them more versatile. That is why these are now called ‘machining centres’ rather than milling machines. The CNC machining centres can be broadly categorised into two varieties:

- Vertical axis machining centre, and
- Horizontal axis machining centre.

Vertical Axis Machining Centre

The vertical axis machining centres or VMC as is popularly abbreviated, are generally more versatile in terms of the tool being able to generate more complex surfaces compared to the horizontal axis. Most of the general machines come with 3 axes. Additional axes will be added to cater to the machining of more complex geometries. For example the spindle head can be swivelled in one or two axes (about X or Y axis) to provide A or B axis motion. These are required for machining sculptured surfaces.

Horizontal Axis Machining Centre

By its very configuration, the horizontal axis machining centre or popularly called HMC is sturdier than the vertical configuration and hence is used for heavier work pieces with large metal removal rates. Since these machines provide for heavier metal removal rates, the cutting tools used would normally be big. As a result, the tool magazine will have to provide larger place for each tool. This results in the tool magazines for HMC to become heavier. Also, they are normally provided with tool magazines having higher capacity.

CNC Turning Centres

Majority of the components machined in the industry are of the cylindrical shape. Hence the CNC lathes, more appropriately called turning centres, are also important machine tools. The evolution of the CNC turning centres follows the developments in the CNC machining centres closely.

Coordinate measuring machines (CMMs) are similar to any metal cutting machine tool except that the tool is replaced by means of a touch trigger probe. This makes it an extremely powerful metrological instrument. When the touch trigger probe touches the part surface it generates the coordinates of the point it contacted. Thus it can be used for measuring the geometrical characteristics of the part. The CMM can be manually operated by an operator using a joystick type control or can have full CNC control. Programs can be written using the programming language similar to the CNC part program, for the probe to contact the part surface to get the dimensions as required.

Part Programming Fundamentals

Part programs for simple components can be carried out manually. However, if the component has complex features which require too many repetitive and/or tedious calculations for preparing its program for cutter path description, then it is recommended that computer aided part programming be resorted machine shop experience is the prerequisite for a good programmer, as only careful process planning can lead to efficient and practical programs.

The following are the steps to be followed while developing the CNC part programs.

- Process planning
- Axes selection
- Tool selection
- Cutting process parameters planning
- Job and tool setup planning
- Machining path planning
- Part program writing
- Part program proving

Process Planning

Process plan is a detailed plan of the steps involved in manufacturing (machining) a given part. The following are the contents of a process plan:

- Machine tool used
- Fixture(s) required
- Sequence of operations
- For each of operation
- Cutting tools required
- Process parameters

A programmer is supposed to carry out a careful study of the part drawing to prepare the process plan. The choice of the machine tool used depends upon the operations required, accuracy requirements, machine tool capability and availability, cutting tool availability and the shop practices. A careful choice of various options at this stage would decide on the final cost of manufacture of the part.

Axes Selection

All the CNC machine tools rely on the axes system for describing the axes motion. To correctly describe the motion, it is therefore necessary to establish the axes system to be followed with the particular part. The ISO designation of axes was discussed earlier. In tune with that axes system, one has to choose the axes. However, it is also necessary for one to choose the axes system as appropriate to the machine tool co-ordinate system in question.

The axes system of all the CNC machine tools would generally have a fixed datum position as designated by the machine tool manufacturer. It may be called the reference position, fixed datum or home position. This absolute datum position of the CNC machine tool may not be very convenient for setting the job. Hence most of the CNC machine tools come with

'Floating Datum'. In this case the programmer can select the part datum anywhere in the machining limits of the machine tool, based on the geometry of the part being machined.

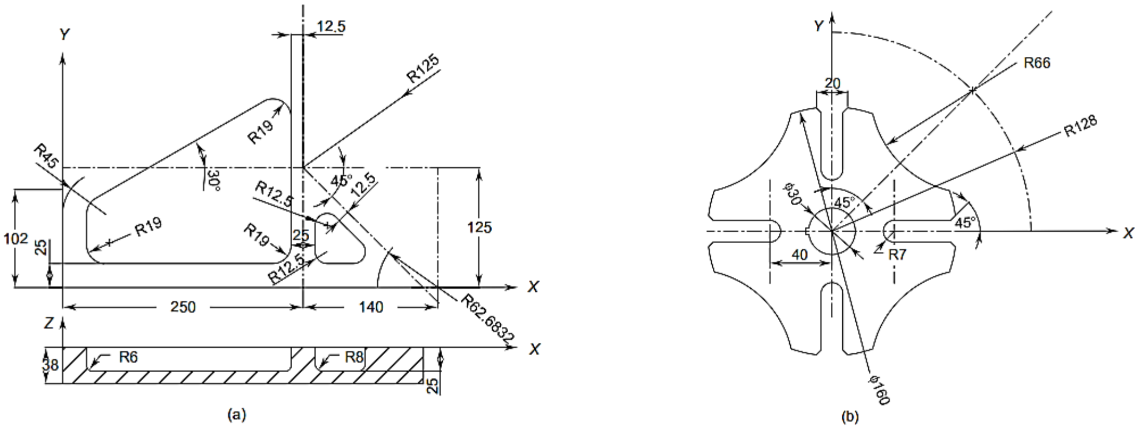


Fig.7 A typical component with axes system designated

The choice of cutting tools is a very important function. For a given operation many tools are feasible, but some of them are more economical than others. Therefore for the economy of manufacture, it is essential to choose the right tool for the job. As a rule, we will only select the regular cutting tools for use in CNC machine tools. No special tooling is generally suggested, since the geometry can very well be generated by the CNC control.

As an example when a contour is being milled, the choice can be an end mill or a slot drill (end cutting end mill). End mill is stronger and can take deeper cuts than a corresponding slot drill. However, slot drill can enter into a solid material, but an end mill cannot in view of the fact that the cutting edge in the bottom does not extend to the centre of the tool. As a result, an end mill should always approach the work piece from the side while the slot drill can approach from the side or from the top.

Cutting Process Parameters Planning

For a given tool and the operation selected, the appropriate process parameters are to be selected. These are to be generally taken from the handbooks supplied by the cutting tool manufacturers or based on the shop experience. It is important in the context of CNC manufacture that the feeds and speeds selected should be as high as possible to reduce the machining time consistent with the product quality achieved.

Job and Tool Setup Planning

This aspect would be covered in greater details later. This basically is aimed at setting the job on the machine tool and adjusting the cutting tool to the correct position. This is important since the accuracy of the geometry generated by the CNC machine tool is dependent on the initial position carefully defined.

Machining Path Planning

It is a very important aspect of programming wherein the knowledge of machining operations plays a vital role. A careful planning of the tool path ensures that the requisite manufacturing specifications are achieved at the lowest cost.

Part Program Writing

This aspect deal with the actual writing of the part programs, taking the format and syntax restrictions into account.

Part Program Proving

This is another aspect, which the programmer should very carefully do before the part program is released to the shop. Once the program is made, it should be verified before it can be loaded on the machine tool controller for the manufacture of the component. It is obvious that a faulty program can cause damage to the tool, work piece and the machine tool itself. Sometimes, these accidents can prove grave for the operator and others around. One of the preliminary ways of avoiding such possibilities is to carry out a visual check of the program manuscript and of the displayed program on the VDU of the controller.

Documentation for NC

It would now be amply clear that documentation is the most essential aspect of the CNC manufacturing practice. Therefore, it is worthwhile to list these as a checklist.

1. Component drawing.
2. Process planning sheet: As discussed earlier, this should contain details of the sequence of the operations, the machine tool used, the tools used with their numbers, speeds, feeds etc.
3. Tool cards: These should show each tool in assembled form with dimensions and identification numbers for each element (tool, collet, chuck etc.).
4. Setting card: This would show all tools, as in position on the machine tool, with their identification numbers, and the corresponding compensation values.
5. Programming sheets.
6. Punched paper tape, if this is the input form used.

The originals of these documents are kept in the programming room records cabinet while copies are sent to the shop floor as per the production planning. Whenever any changes are to be made, all issued copies are recalled and destroyed. The originals are updated (or made afresh) and copies are released accordingly.

Manual Part Programming methods

In the earlier days, a number of formats for NC part programs were used such as fixed sequence or tab sequential. These systems required giving a large number of unwanted or duplicate information in each block of a part program. Now these have been replaced by means of a system called 'Word Address Format' in which each of the information or data to be input in the form of numerical digits is preceded by a word address in the form of an English alphabet.

N115 G81 X120.5 Y55.0 Z-12.0 R2.0 F150 M3

ISO Standards for Coding

Character	Address For
A	Angular dimension around X axis
B	Angular dimension around Y axis
C	Angular dimension around Z axis
D	Angular dimension around special axis or third feed function*
E	Angular dimension around special axis or second feed function*
F	Feed function
G	Preparatory function
H	Unassigned
I	Distance to arc centre or thread lead parallel to X
J	Distance to arc centre or thread lead parallel to Y
K	Distance to arc centre or thread lead parallel to Z
L	Do not use
M	Miscellaneous function
N	Sequence number
O	Reference rewind stop
P	Third rapid traverse dimension or tertiary motion dimension parallel to X*
Q	Second rapid traverse dimension or tertiary motion dimension parallel to Y*
R	First rapid traverse dimension or tertiary motion dimension parallel to Z*
S	Spindle speed function
T	Tool function
U	Secondary motion dimension parallel to X*
V	Secondary motion dimension parallel to Y*
W	Secondary motion dimension parallel to Z*
X	Primary X motion dimension
Y	Primary Y motion dimension
Z	Primary Z motion dimension

* Where D, E, P, Q, R, U, V, and W are not used as indicated, they may be used elsewhere.

The complete part program for a given component consists of a beginning code of % which signifies the start of the tape (in case of paper tapes) or beginning of a program if direct computer communication is involved such as in DNC mode. A part program consists of large number of blocks (similar to sentences in a letter) each representing an operation to be carried out in the machining of a part.

Each block always starts with a block number used as identification and is programmed with a 'N' word address. This must be programmed at the beginning of every block. As per ISO 2539, it has a minimum of three digits, e.g. N009, N028. However, some control manufacturers, notably Fanuc dispense with this requirement. In their case, only those blocks which are to be specifically addressed as per the requirement of program flow would be given a block number. Other blocks can do away with this requirement. This saves valuable RAM space in the controller where the part programs are stored. As discussed above, the co-ordinates of the tool tip are programmed for generating a given component geometry. The co-ordinate values are specified using the word address such as X, Y, Z, U, V, W, I, J, K, etc.

X123.405 Y-34.450

Feed Function

Generally the feed is designated in velocity units using the 'F' word address. For example, F150 means that the feed rate is specified as 150 mm per minute. This is the actual speed with which the tool moves along the programmed path. However, depending upon the programmed path, there could be some deviations in the actual feed followed by the controller. Also the controller calculates the actual feed rate of each of the axis.

Once the feed rate is programmed in a block, it remains in force in all the subsequent blocks till it is replaced by another F value, i.e. it is modal. The feed rate programmed can be overridden by a setting on the controller console, in steps of 10% between 0 and 150%. However, in some situations this override will not work, for example, in case of thread cutting or thread milling. By using an appropriate 'G' code, it is also possible to change the feed rate units from mm per minute to mm per revolution or vice versa.

Speed Function

In some of the CNC machine tools, spindle speeds are set manually and so are not to be programmed. However, most of the CNC machines that are coming now would have the capability for the step less variation of spindle speeds. Hence they need to be programmed using spindle speed word 'S'. The speed can be set directly in the revolutions per minute or RPM mode using the S word address as follows:

S1500 means that spindle speed is to be set at 1500 rpm.

Tool Function

All NC machines are generally equipped with turrets or tool magazines with automatic tool changers (ATC) which enable the positioning of the pre-set tools in a few seconds. Thus the ratio of cutting time to total machine time is considerably increased. The tool function is normally indicated by the word address ‘T’. This may have 2 or more digits depending upon the tool magazine capacity. Most common is the 2 digits such as T15. This causes the tool magazine position 15 or tool number 15 to be brought into the spindle replacing the already present tool in the spindle. The tool replaced from the spindle will be brought back to the empty position created when the tool 15 was loaded.

Preparatory Functions

This is denoted by ‘G’. It is a pre-set function associated with the movement of machine axes and the associated geometry. As discusses earlier, it has two digits, e.g. G01, G42, and G90 as per ISO specifications.

Code	Function
G00	Point-to-point positioning, rapid traverse
G01	Line interpolation
G02	Circular interpolation, clockwise (WC)
G03	Circular interpolation, anti-clockwise (CCW)
G04	Dwell
G05	Hold/Delay
G06	Parabolic interpolation
G07	Unassigned
G08	Acceleration of feed rate
G09	Deceleration of feed rate
G10	Linear interpolation for “long dimensions” (10 inches-100 inches)
G11	Linear interpolation for “short dimensions” (up to 10 inches)
G12	Unassigned
G13-G16	Axis designation
G17	XY plane designation
G18	ZX plane designation
G19	YZ plane designation
G20	Circular interpolation, CW for “long dimensions”
G21	Circular interpolation, CW for “short dimensions”
G22-G29	Unassigned
G30	Circular interpolation, CCW for “long dimensions”
G31	Circular interpolation, CCW for “short dimensions”
G32	Unassigned
G33	Thread cutting, constant lead
G34	Thread cutting, linearly increasing lead
G35	Thread cutting, linearly decreasing lead
G36-G39	Unassigned
G40	Cutter compensation-cancels to zero
G41	Cutter radius compensation-offset left

G42	Cutter radius compensation-offset right
G43	Cutter compensation-positive
G44	Cutter compensation-negative
G45-G52	Unassigned
G53	Deletion of zero offset
G54-G59	Datum point/zero shift
G60	Target value, positioning tolerance 1
G61	Target value, positioning tolerance 2, or loop cycle
G62	Rapid traverse positioning
G63	Tapping cycle
G64	Change in feed rate or speed
G65-G69	Unassigned
G70	Dimensioning in inch units
G71	Dimensioning in metric units
G72-G79	Unassigned
G80	Canned cycle cancelled
G81-G89	Canned drilling and boring cycles
G90	Specifies absolute input dimensions
G91	Specifies incremental input dimensions
G92	Programmed reference point shift
G93	Unassigned
G94	Feed rate/min (inch units when combined with G70)
G95	Feed rate/rev (metric units when combined with G71)
G96	Spindle feed rate for constant surface feed
G97	Spindle speed in revolutions per minute
G98-G99	Unassigned

Motion group

*G00 Rapid Positioning

G01 Linear Interpolation

G02 Circular interpolation Clockwise

G03 Circular interpolation Counter clockwise

Dwell

G04 Dwell

Active plane selection group

*G17 XY Plane selection

G18 XZ Plane selection

G19 YZ Plane selection

Cutter compensation group

*G40 Cutter compensation, Cancel

G41 Cutter radius Compensation left

G42 Cutter radius Compensation right

Units group

*G70 Inch units

G71 Metric units

Hole making canned cycle group

*G80 Canned Cycle Cancel

G81-G89 Canned Cycles definition and ON

Co-ordinate system group

*G90 Absolute co-ordinate system

G91 Incremental co-ordinate system

Preset

G92 Absolute pre-set, Change the datum position

The * sign indicates the generally accepted default or turn on code in operation. However, some control manufacturers allow this to be modified to whatever suits them. The above is only a possible indication but not in any way standardised by ISO.

Co-ordinate System Group, G90 and G91

The input of dimensional information can be done either in the absolute or in the incremental system. The preparatory function G90 is used for absolute programming. In absolute system, the dimensions are given with respect to a common datum chosen by the programmer. It must be programmed and can be cancelled by function G91 (and also when the program statement has the word M02 or M30). OX and OY Units Group, G70, G71

Numerical Control of Machine Tools

This group of codes specifies the units in which the program is to be interpreted. G70 stands for programming in inch units while G71 stands for programming in mm units. Any one of these can be made as turn on code depending upon the default units likely to be used. Most of the controls destined for areas other than North America would generally have default G71.

Active Plane Selection Group, G17, G18, G19

Some of the functions in NC control can only work in a plane rather than in all the three possible co-ordinate axes. This therefore requires the selection of active plane. This can be done by using these codes.

Miscellaneous Functions, M

These functions actually operate some controls on the machine tool and thus affect the running of the machine. Generally only one M code is supposed to be given in a single block.

However, some controllers allow for two or more M codes to be given in a block, provided these are not mutually exclusive, e.g., coolant ON (M07) and OFF (M09) cannot be given in one block.

Code	Function
M00	Program stop, spindle and coolant off
M01	Optional programmable stop
M02	End of program-often interchangeable with M30
M03	Spindle on, CW
M04	Spindle on, CCW
M05	Spindle stop
M06	Tool change
M07	Coolant supply No. 1 on
M08	Coolant supply No. 2 on
M09	Coolant off
M10	Clamp
M11	Unclamp
M12	Unassigned
M13	Spindle on, CW + coolant on
M14	Spindle on, CCW + coolant on
M15	Rapid traverse in + direction
M16	Rapid traverse in – direction
M17–M18	Unassigned
M19	Spindle stop at specified angular position
M20–M29	Unassigned
M30	Program stop at end tape + tape rewind
M31	Interlock by-pass
M32–M35	Constant cutting velocity
M36–M39	Unassigned
M40–M45	Gear changes; otherwise unassigned
M46–M49	Unassigned
M50	Coolant supply No. 3 on
M51	Coolant supply No. 4 on
M52–M54	Unassigned
M55	Linear cutter offset No. 1 shift
M56	Linear cutter offset No. 2 shift
M57–M59	Unassigned
M60	Piece part change
M61	Linear piece part shift, location 1
M62	Linear piece part shift, location 2
M63–M67	Unassigned
M68	Clamp piece part
M69	Unclamp piece part
M70	Unassigned
M71	Angular piece part shift, location 1
M72	Angular piece part shift, location 2
M73–M77	Unassigned
M78	Clamp non-activated machine bed-ways
M79	Unclamp non-activated machine bed-ways
M80–M99	Unassigned

Example

The component to be machined is shown in Fig. 17.21. It is assumed that the pocket is through and hence only the outside is to be machined as a finish cut of the pocket. The tool to be used is a 20 mm diameter slot drill. If an end mill is to be used the program should be modified with a hole to be drilled at B first before the end mill is used. The setting is done with point A as reference (0, 0, 0) and the reference axes are along X and Y directions. A typical program, as per ISO (except the decimal point), for this would be:

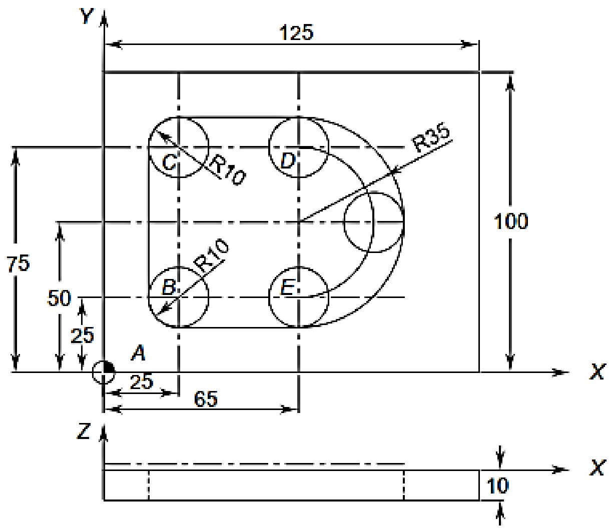


Fig. 17.21 *Example*

N001 G92 X0 Y0 Z0	absolute presetting at A
N002 G90	absolute programming
N003 G00 X25.0 Y25.0 Z2.0 T01 S3000 M03	tool brought rapidly at B, 2 mm above XY plane
N004 G01 Z-12.0 F120	tool goes down to full depth
N005 Y75.0	proceeds to C
N006 X65.0	proceeds towards right to D
N007 G02 Y25.0 I0 J-35.0	cuts curved profile till E
N008 X25.0	proceeds to B
N009 Z2.0	tool moves 2 mm above the XY plane
N010 G00 Z50.0 M05	spindle stops and rapidly moves up
N011 X0 Y0	rapid move to start position 0,0
N012 M30	end of program and tape rewind

Tool Length Compensation

Numerical Control of Machine Tools

In the programs discussed earlier, absolute pre-setting is per formed with a tool and the co-ordinates for reference are registered. However, if the tool were replaced by another tool, say 20 mm shorter in length, and then the tool movement would be 20 mm less along the Z axis if the same programmed values are used. So, the alternative is to modify the program every time the tool is changed.

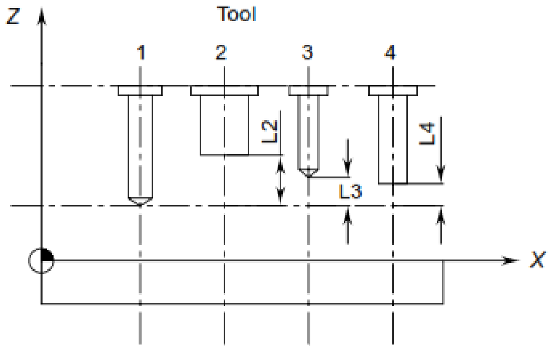


Fig.8 Tool length compensation

Cutter Radius Compensation

In profiling operations, one needs to calculate the tool path for preparing the program. This path refers to the spindle axis that is away from the profile required. Figure shows the component and the tool path. Apart from the problem of calculating, one should realise that whenever the cutter size changes, the program would need editing. However, if a compensation equal to the radius of the cutter is entered and stored in the control system, then the program could be written for the component profile and thus no change in program would be required.

The preparatory functions G40, G41 and G42 are used for radius compensation and form one group. These are modal and the one programmed in any block remains active till cancelled by the other.

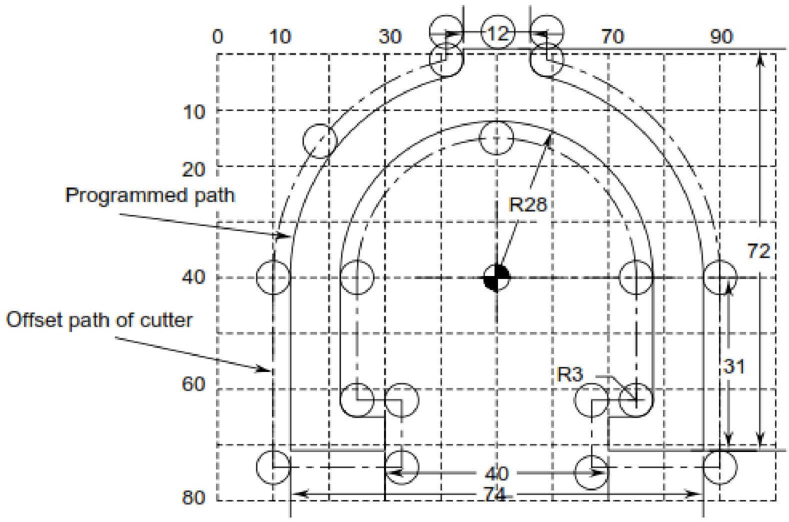


Fig.9 Cutter radius compensation

Canned Cycles

It is found many a times that a series of motions are to be repeated a number of times, many of which are fairly common to all the positions. For example, in the case of drilling operation the tool (twist drill) has to position a little above the hole in rapid position, then move to the required depth with the given feed rate and then the tool has to return to the top of the hole. The same actions are to be repeated for each of the holes. For each of the operation 3 NC blocks to be written, out of which two blocks need to be repeated without any change for each of the hole to be drilled in the same plane. It therefore is possible to define a canned cycle or fixed cycle which can repeat all these motions without having to repeat the same information for each of the hole. The most common cycles that would be useful are for the hole making operations such as drilling, reaming, tapping, etc.

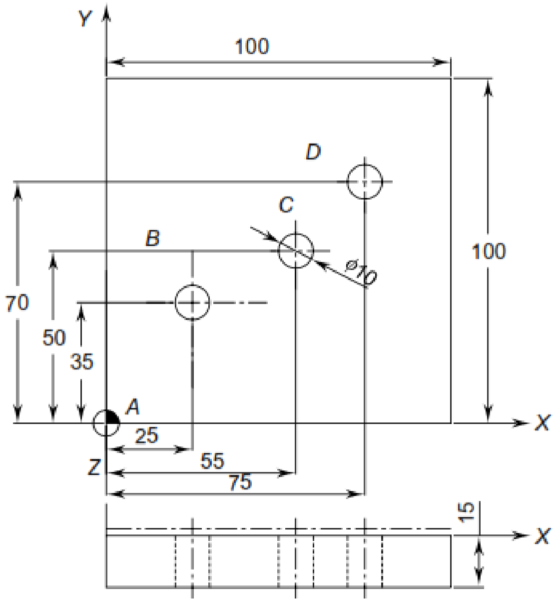


Fig.10 Canned cycle

Table 1 canned cycle motion

Canned Cycle Number	Feed from Surface	At Programmed Depth (End of Feed Point)			Used for
		Dwell	Spindle Speed	Spindle Return Motion	
G80	Off	—	Stop	—	Cancel canned cycle
G81	Constant	—	—	Rapid	Drilling, centre drilling
G82	Constant	Yes	—	Rapid	Counter sinking, Counter boring
G83	Intermittent	—	—	Rapid	Deep hole drilling
G84	Constant	—	Reverse	Feed	Tapping
G85	Constant	—	—	Feed	Reaming
G86	Constant	—	Stop	Rapid	Boring
G87	Constant	—	Stop	Manual	Multiple Boring
G88	Constant	Yes	Stop	Manual	Boring
G89	Constant	Yes	—	Feed	Boring

Computer aided Part Programming (CAP)

Preparing the part programs for CNC machine tools manually is a viable system for any kind of job. But the assistance of the computer would be desirable for part programming because of a variety of reasons. The first and foremost in this respect is the complexity of the work piece which makes manual part programming The APT (Automatically Programmed Tools) language system originated at the Servomechanism laboratory of Massachusetts Institute of Technology, as did the first NC machine tool in 1952. This was the pilot study sponsored by the US Air Marshal Command, which resulted in the prototype system being released for the whirlwind computer in 1955. Though this version was an important step towards the computer preparation of tapes, the user still had to calculate the end points of each straight line cut to be performed by the machine tool. MIT, under the sponsor ship of the Aerospace Industries Association, has released APT II for IBM 7040 wherein the complete job of part program preparation from the part drawing was undertaken by the computer. This version was continually developed until 1961, when the APT Long Range Program (ALRP) was created by the AIA and the job of keeping APT up-to-date was given to the Illinois Institute of Technology Research Institute, Chicago. In recognition of the role played by the computer in manufacture, over and above the simple guiding of the cutter tool along the work piece, the original sponsors have changed the ALRP in 1969 to Computer Aided Manufacture International or CAM-I. Now the work of CAM-I is done by IITRI as well as a large number of contractors all over the globe.

The APT NC reference language consists of a specially structured set of vocabulary, symbols, rules and conventions which are easily understood by the part programmer and would help him in faster preparation of control tapes. The vocabulary, which forms the mainstay of the reference language, is a carefully selected set of mnemonics chosen for their similarity in form and meaning with English. The computer translation of these English-like statements to the valid NC codes for any particular machine tool controller is generally carried out in two stages The first phase involves the conversion of input information into a generalised set of cutter location (CL) data and the relevant machine motions. At this stage, the output generated is the universally applicable cutter centre co-ordinates (called CLDATA, CLFILE, CLTAPE) which are independent of the machine tool on which the part is to be finally made. This set of programs is called processor and only one such processor is sufficient for any number of NC machine tools in the shop. The output of the processor contains information regarding the feed rates, spindle speeds, directions, coolant status, tool selection and other pieces of information which are machine tool/control unit-oriented, in addition to the cutter centre co-ordinates with respect to the work piece. The second set of programs, called post processor, converts the generalised cutter location data into the specific control codes of the machine tool. As a result, the post processor is no more general like the processor but one each would be needed for every machine tool/ control system combination. The post processor would convert the cutter location data along with the machine tool-oriented information into the appropriate NC codes employed or the particular machine tool/ control system combination in question.

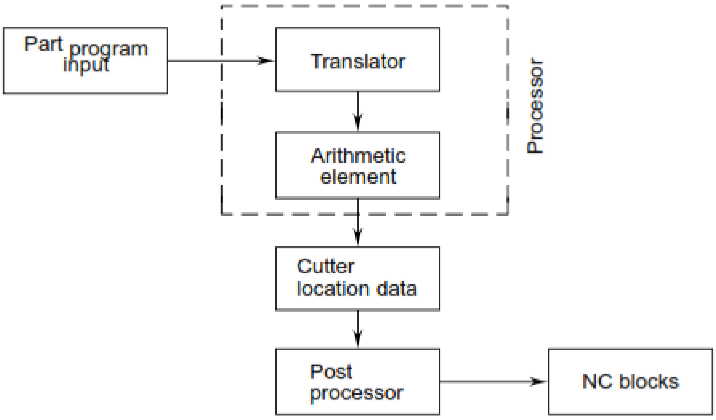


Fig.11 Computer part programming configuration

The two-pass preparation detailed above is most commonly used. The prime need for this is for making the part programming system more flexible. By taking out all the machine tool control unit-oriented information and making it a separate module, which would be far smaller, compared to the main tape preparation system, one is able to achieve the desired generality. Since the machine tool oriented information is embedded in the post processor, which happens to be a much smaller segment in the overall tape preparation system, it is far more economical to duplicate for the various other machine tools which one may be willing to operate.

The various functions that can be attributed to the postprocessor are:

1. Converting the CLDATA to the machine tool co-ordinate system.
2. Converting the CLDATA to the control unit understandable NC blocks taking care of the following machine tool functions:
    - Maximum table or spindle traverses,
    - Available feeds and speeds,
    - Available preparatory, miscellaneous and other functions,
    - Straight line and circular interpolations,

● Acceleration and decelerations of slides taking care of the overshoot of corners, and

● Other machine tool control unit system requirements such as tape reader time, servo setting time,etc.

3. Provide output

● Required control tape.

● Diagnostic listing on line printer, and

● Other operator/programmer instructions.

The modification of the main processor, which is very complex, will generally not be possible except by the people who had originally written it.

APT Language Structure

The APT language consists of many different types of statements made up of the following valid letters, numerals and punctuation marks.

Letters - ABCDEFGHIJKLMNOPQRSTUVWXYZ

Numerals - 0123456789

Punctuation marks

/ A slash divides a statement into two sections. To the left of the slash are the MAJOR words, and to the right are the words, symbols and/or scalars that modify the word on the left of the slash so as to give it a complete and precise meaning or definition, e.g. GO/PAST, LN, TO, CS.

, A comma is used as a separator between the elements in a statement generally to the right of the slash.

= An equals is used for assigning an entity to a symbolic name, e.g. CI = CIRCLE/25, 50, 30.

Words - The words to be used in the statements are built up from one to six letters or numerals with the first one being a letter. No special character is allowed in the words.

Key Words - There are certain reserved names called key words in the language, which have a fixed meaning. These words cannot be used for any other purpose. A key word may be replaced by another name using a SYN statement. All key words consist of between two and six letters, without any numerals. The key words are divided into two classes, the MAJOR key words, which define the type of the statement, and the MINOR key words, which give the required parameters and modifiers.

Symbols - Symbols are the words used as substitutes for geometrical definitions and numerical values, where the first character must be a letter. A symbol must be defined before it is referenced in a subsequent part program statement.

Labels - Label names are used to reference a statement so that control can be transferred to that statement changing the usual linear execution sequence. Labels are identical to the words with the difference that all the characters in a label can be numerals. A label must be terminated by a right parenthesis.

Numbers - Numbers have their usual meaning as in algebra and are often referred to as scalars. No distinction is made between integer and real numbers

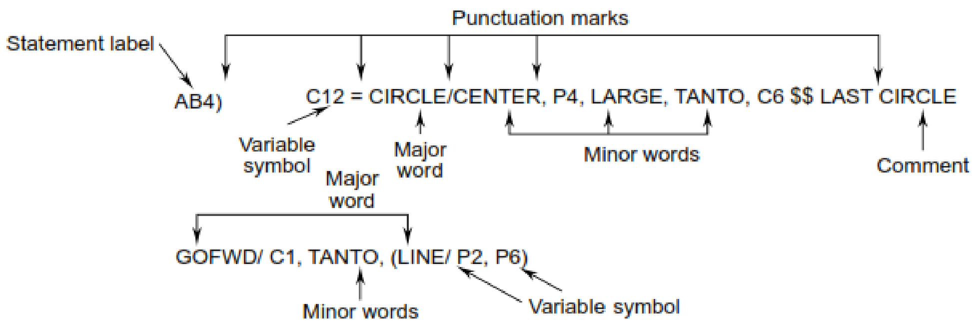


Fig.12 Structure of APT

APT Program

The complete APT part program consists of the following four types of statements:

- Geometry
- Motion
- Post processor
- Compilation control

Geometry Commands

There are many ways in which the part geometry in APT could be specified. The part geometry is normally broken into a number of surface elements that could be defined from the data given in a part print. These are POINT, LINE, CIRCLE, PLANE, VECTOR, PATERN, SPHERE, TABCYL, etc. For each of the type of surface that can be defined, a number of alternative ways are possible for definition to simplify the definition procedure. A few examples are shown below:

Point The point has three co-ordinates along X, Y and Z-axes. The Z co-ordinate when not specified is taken as either zero or the prevailing Z surface definition.

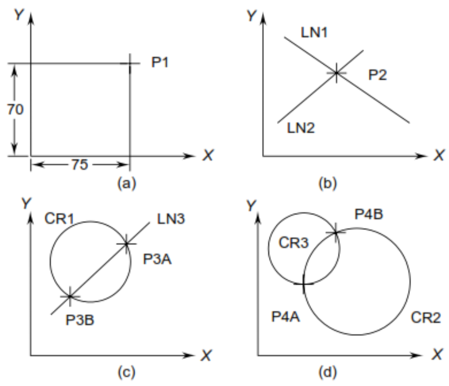


Fig.13 Point definition

P1 = POINT/75.0, 70.0

P2 = POINT/ INTOF, LN1, LN2

P3B = POINT/ XSMALL, INTOF, LN3, CR1

P3A = POINT/ XLARGE, INTOF, LN3, CR1

P4A = POINT/ XSMALL, INTOF, CR2, CR3

P4B = POINT/ XLARGE, INTOF, CR2, CR3

Line Lines are considered to be of infinite length and do not have a direction. Lines must not be perpendicular to the XY plane. Lines are considered planes perpendicular to the XY plane.

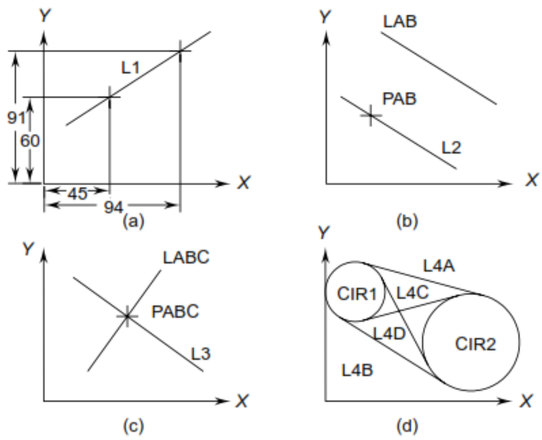


Fig.14 Line definition

L1 = LINE/ 45, 60, 94, 91

L2 = LINE/ PAB, PARLEL, LAB

L3 = LINE/ PABC, PERPTO, LABC

L4A = LINE/ LEFT, TANTO, CIR1, LEFT, TANTO, CIR2

L4B = LINE/ RIGHT, TANTO, CIR1, RIGHT, TANTO, CIR2

L4C = LINE/ LEFT, TANTO, CIR1, RIGHT, TANTO, CIR2

L4D = LINE/ RIGHT, TANTO, CIR1, LEFT, TANTO, CIR2

Right and left is established by looking from the first circle specified in the definition to the second circle.

Circle A circle is always considered as a circular cylinder perpendicular to the XY plane of infinite length. The radius value when specified must not be negative.

C1 = CIRCLE/ 61, 62, 37

C2 = CIRCLE/ CENTER, PT1, TANTO, LN1

C3 = CIRCLE/ PT4, PT2, PT3

C4A = CIRCLE/ YLARGE, LN2, YLARGE, LN3, RADIUS, 15

C4B = CIRCLE/ XSMALL, LN2, YLARGE, LN3, RADIUS, 15

C4C = CIRCLE/ YSMALL, LN2, YSMALL, LN3, RADIUS, 15

C4D = CIRCLE/ YSMALL, LN3, XLARGE, LN2, RADIUS, 15

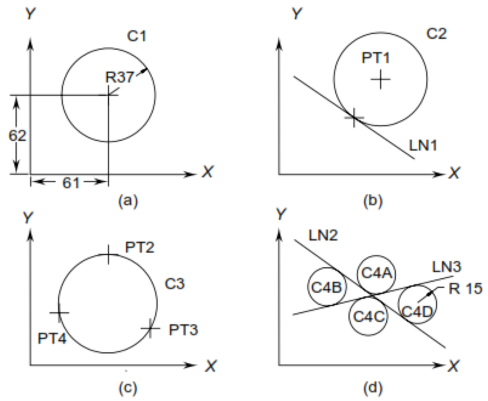


Fig.15 Circle definition

The following few examples of APT geometries of components (views shown in XY plane only) have been defined:

PARTNO/ EXAMPLE 1

P2 = POINT/ 0, 0

L1 = LINE/ 20, 20, 20, (20 + 80)

L2 = LINE/ (POINT/ 20, (20 + 80)), ATANGL, 45

P1 = POINT/ (20 + 30 + 40 + 20), 20

C2 = CIRCLE/ CENTER, P1, RADIUS, 20

L4 = LINE/ P1, PERPTO, (LINE/ XAXIS)

C1 = CIRCLE/ (20 + 30 + 40), (20 + 80 + 30 – 20), 20

L3 = LINE/(POINT/(20 + 30),(20 + 80 + 30)), PARLEL, (LINE/XAXIS)

PARTNO/ EXAMPLE 2

L4 = LINE/ XAXIS

L5 = LINE/ YAXIS

L3 = LINE/ PARLEL, L5, XLARGE, (20 – 5)

LM = LINE/ PARLEL, L5, XLARGE, (140/2)

C1 = CIRCLE/ (20 + 20), 22.5, 5

C2 = CIRCLE/ 20, (15 + 15), 5

C3 = CIRCLE/ 20, 15, 5

L1 = LINE/ LEFT, TANTO, C1, LEFT, TANTO, C3

L2 = LINE/ RIGHT, TANTO, C1, RIGHT, TANTO, C2

L3 = LINE/ PARLEL, L5, XLARGE, (20 – 5)

P1 = POINT/ CIRCLE, C1

P2 = POINT/ (20 + 20), (22.5 + 55)

P3 = POINT/ (20 + 100 – 20), (22.5 + 55)

P4 = POI NT/ (20 + 100 – 20), 22.5

PAT= PATERN/ RANDOM, P1, P2, P3, P4

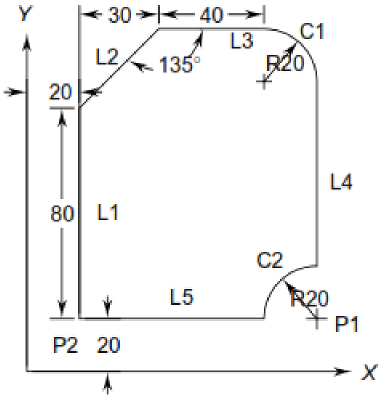


Fig.16 Example geomentry

Motion Commands

This section of commands is more complex compared to the rest of the part program. The main function of these commands is to describe the actual machining sequence making use of the geometry elements defined earlier. The processor assumes that the tool moves around the work piece for the purpose of machining. If this were not true for any machine tool control unit, the post-processor would take care of the necessary conversions.

The motion commands can be broadly divided into three groups:

- Set-up commands,
- Point-to-point motion commands, and
- Continuous path motion commands.

Set-up commands These commands are used to identify the initial conditions for the specified machining. They are:

- FROM/ point
- CUTTER/ dia, radius
- INTOL/ dsval, psval, cs1val, cs2val
- OUTTOL/ dsval, psval, cs1val, cs2val
- TOLER/ dsval, psval, cs1val, cs2val

Point-to-point motion commands These are used to specify the positioning commands used for point to point applications such as drilling operation. They are:

- GODLTA/ dx, dy, dz
- GOTO/ x, y, z
- GOTO/ point
- GOTO/ patern

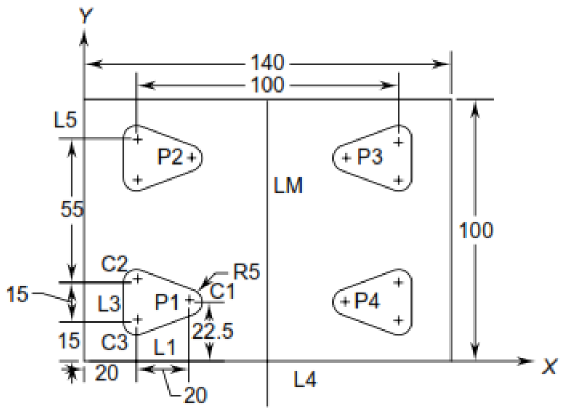


Fig.17 Motion Example

Continuous path motion commands These are used to specify the continuous path motion involving milling and turning operations to generate a variety of surfaces. The desired path where the cutting tool (cutter) is in continuous contact with the work piece surface is described by means of three intersecting surfaces. These surfaces are designated as drive surface (ds), part surface (ps) and check surface (cs) which are specified in that order in the motion statement. For any motion to be described, all the three surfaces are to be designated either explicitly or otherwise. The cutting tool is expected to move along the intersection of the part and drive surface till it is stopped by means of the check surface.

Part surface is in continual contact with the tool tip and helps in control of the depth of cut. Drive surface is the other surface with which the cutting tool is in continual contact during a given motion. The tool periphery or tool axis follows the drive surface. Check surface is the one which limits the given motion statement.

PSIS/ ps Part surface is

AUTOPS Automatic part surface (current Z level)

NOPS No part surface

GO/ TO, ds, TO, ps, TO, cs Start-up command for continuous path

The modifier TO can be replaced by either ON, PAST or TANTO depending upon the cutter relationship with the respective surface.

GOLFT/ ds, TO, cs Contour Motion command Go to left

GORGT/ ds, TO, cs Contour Motion command Go to right

GOFWD/ ds, TO, cs Contour Motion command Go forward

GOBACK/ ds, TO, cs Contour Motion command Go back

GOUP/ ds, TO, cs Contour Motion command Go up

GODOWN/ ds, TO, cs Contour Motion command Go down

Repetitive Programming

Just as DO loop, subroutines and macros are used in manual part programming, similarly facilities are available for repetitive programming in computer aided programming. These are described here. Looping normally, a part program is executed sequentially starting from a PARTNO statement to the FINI statement. But it would be possible to change this sequential execution by means of the transfer statements available in APT.

JUMPTO unconditional transfer er

IF conditional transfer

The usage is

JUMPTO/ lbl1

A better option for looping is the arithmetic IF statement which allows a conditional transfer to a segment of the program depending on the value of an arithmetical expression.
The general usage is

IF (< expression>) lb11, lb12, lb13

When the numerical value of the < expression > is negative, zero or positive, then control is transferred to the statement referenced by lb11, lb12, or lb13 respectively. It is always necessary to label the statement which immediately follows the IF statement in a part program. The <expression > could be a variable or an arithmetic expression.

X= 0

LB0) YVAL = 20

LB1) GOTO/X, YVAL, 0

GODLTA/-10

GODLTA/10

YVAL = YVAL+30

IF (500 -YVAL) LB2, LB1, LB1

LB2) X=X + 50

IF (500 -X) LB3, LB, LB0

LB3) ----------------Macro

The sequences of similar or identical statements which need to be referred more often in a part program are best referred by a MACRO statement such that the part program bulk is reduced. This statement is very similar to a FORTRAN SUBROU TINE statement. The syntax is

<name >= MACRO/<parameters>

TERMAC

All the statements that are enclosed between a MACRO statement and a TERMAC statement are to be executed whenever this macro is called by

CALL/name, <parameters>

Tracut

The result of TRACUT usage in motion statements is to TRAnspose the CUTter locations only without actually altering the original geometrical definitions. This is useful particularly for repetitive geometries. The usage is

TRACUT/matrix

TRACUT/NOMORE