3

THE LATHE

3.1 INTRODUCTION

The lathe is one of the oldest machine tools and came into existence from the early tree lathe which was then a novel device for rotating and machining a piece of work held between two adjacent trees. A rope wound round the work with its one end attached to a flexible branch of a tree and the other end being pulled by a man caused the job to rotate intermittently. Hand tools were then used. With its further development a strip of wood called "lath" was used to support the rope and that is how the machine came to be known as "lathe". This device continued to develop through centuries and in the year 1797 Henry Maudslay, an Englishman, designed the first screw cutting lathe which is the forerunner of the present day high speed, heavy duty production lathe, a machine tool which has practically given shape to our present day civilization by building machines and industries.

3.2 FUNCTION OF THE LATHE

The main function of a lathe is to remove metal from a piece of work to give it the required shape and size. This is accomplished by holding the work securely and rigidly on the machine and then turning it against cutting tool which will remove metal from the work in the form of chips. To cut the material properly the tool should be harder than the material of the workpiece, should be rigidly held on the machine and should be fed or progressed in a definite way relative to the work.

3.3 TYPES OF LATHE

Lathes of various designs and constructions have been developed to suit the various conditions of metal machining. But all of them employ the same fundamental principle of operation and perform the same function.

The types generally used are :

1. Speed lathe.
    (a) Wood working.
    (b) Centering.
    (c) Polishing.
    (d) Spinning.
2. Engine lathe.
    (a) Belt drive.
    (b) Individual motor drive.
    (c) Gear head lathe.
3. Bench lathe.
4. Tool room lathe.
5. Capstan and Turret lathe.
6. Special purpose.
    (a) Wheel lathe.
    (b) Gap bed lathe.
    (c) T-lathe.
    (d) Duplicating lathe.
7. Automatic lathe.

The Speed Lathe: The speed lathe, in construction and operation, is the simplest of all types of lathe. It consists of a bed, a headstock, a tailstock and a tool-post mounted on an adjustable slide. There is no feed box, leadscrew or conventional type of carriage. The tool is mounted on the adjustable slide and is fed into work purely by hand control. This characteristic of the lathe enables the designer to give high spindle speeds which usually range from 1200 to 3600 r.p.m. As the tool is controlled by hand, the depth of cut and the thickness of chip is very small.

The headstock construction is very simple and only two or three spindle speeds are available. Light cuts and high speeds necessitate the use of this type of machine where cutting force is minimum such as in *woodworking, spinning, centering, polishing*, etc. The "speed lathe" has been so named because of the very high speed of the headstock spindle.

The engine lathe or centre lathe : This lathe is the most important member of the lathe family and is the most widely used. The term "engine" is associated with the lathe owing to the fact that early lathes were driven by steam engines. Similar to the speed lathe, the engine lathe has got all the basic parts, e.g. bed, headstock, and tailstock. But the headstock of an engine lathe is much more robust in construction and it contains additional mechanism for driving the lathe spindle at multiple speeds. Unlike the speed lathe, the engine lathe can feed the cutting tool both in cross and longitudinal direction with reference to the lathe axis with the help of a carriage, feed rod and leadscrew. With these additional features an engine lathe has proved to be a versatile machine adapted for every type of lathe work.

Engine lathes are classified according to the various designs of the headstock and methods of transmitting power to the machine. A lathe that receives its power from an over-head line shaft is a *belt-driven lathe* and is equipped with a speed-cone and one or more back gears to get a wide range of spindle speeds. A lathe that receives its power from an individual motor integral with the machine is called a ***motor driven lathe***. A ***geared-head lathe*** gets its power from a constant speed motor, and all speed changes are obtained by shifting various gears located in the headstock. It has no cone pulley.

The bench lathe : This is a small lathe usually mounted on a bench. It has practically all the parts of an engine lathe or speed lathe and it performs almost all the operations, its only difference being in the size. This is used for small and precision work .

The tool room lathe : A tool room lathe having features similar to an engine lathe is much more accurately built and has a wide range of spindle speeds ranging from a very low to a quite high speed up to 2500 r.p.m. This is equipped, besides other things, with a chuck, taper turning attachment, draw in collet attachment, thread chasing dial, relieving attachment, steady and follower rest, pump for coolant, etc. This lathe is mainly used for precision work on tools, dies, gauges and in machining work where accuracy is needed. The machine is costlier than an engine lathe of the same size.

The capstan and turret lathe : These lathes are development of the engine lathe and are used for production work. The distinguishing feature of this type of lathe is that the tailstock of an engine lathe is replaced by a hexagonal turret, on the face of which multiple tools may be fitted and fed into the work in proper sequence. The advantage is that several different types of operations can be done on a workpiece without re-setting of work or tools, and a number of identical parts can be produced in the minimum time.

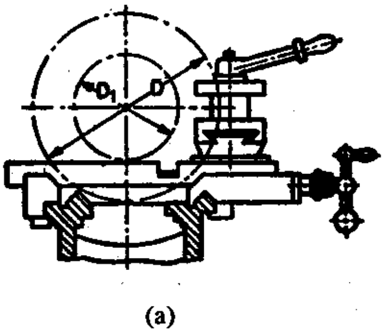
Special purpose lathe : As the name implies, they are used for special purposes and for jobs which cannot be accommodated or conveniently machined on a standard lathe. The *wheel lathe* is made for finishing the journals and turning the tread on railroad car and locomotive wheels. The *gap bed lathe*, in which a section of the bed adjacent to the headstock is recoverable, is used to swing extra-large diameter pieces. The *T-lathe*, a new member of the lathe family, is intended for machining of rotors for jet engines. The axis of the lathe bed is at right angles to the axis of the headstock spindle is the form of a T. The *duplicating lathe* is one for duplicating the shape of a flat or round template on to the workpiece. Mechanical, air, and hydraulic devices are all used to coordinate the movements of the tool to reproduce accurately the shape of the template.

The *missile lathe*, which has a very large swing for accommodating long missile component of very large diameter, is the most modern and latest in lathe design.

Automatic lathe : These are high speed, heavy duty, mass production lathes with complete automatic control. Once the tools are set and the machine is started it performs automatically all the operations to finish the job. The changing of tools, speeds, and feeds are also done automatically. After the job is complete, the machine will continue to repeat the cycles producing identical parts even without the attention of an operator. An operator who has to look after five or six automatic lathes at a time will simply look after the general maintenance of the machine and cutting tool, load up a bar stock and remove finished products from time to time.

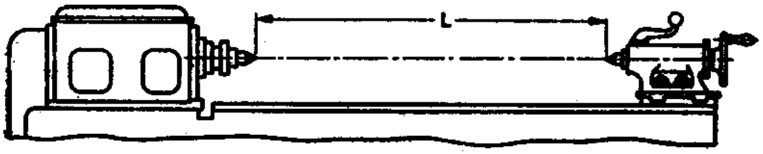
3.4 THE SIZE OF A LATHE

The size of a lathe is expressed or specified by the following items and illustrated in Fig.3.1 :



(a)

1. The *height of the centres* measured from the lathe bed.
2. The *swing diameter over bed*. This is the largest diameter of work that will revolve without touching the bed and is twice the height of the centre measured from the bed of the lathe.



(b)

Figure 3.1 Lathe size

(a) D. Swing diameter over bed, D_1. Swing diameter over carriage.

(b) L. Length between centres

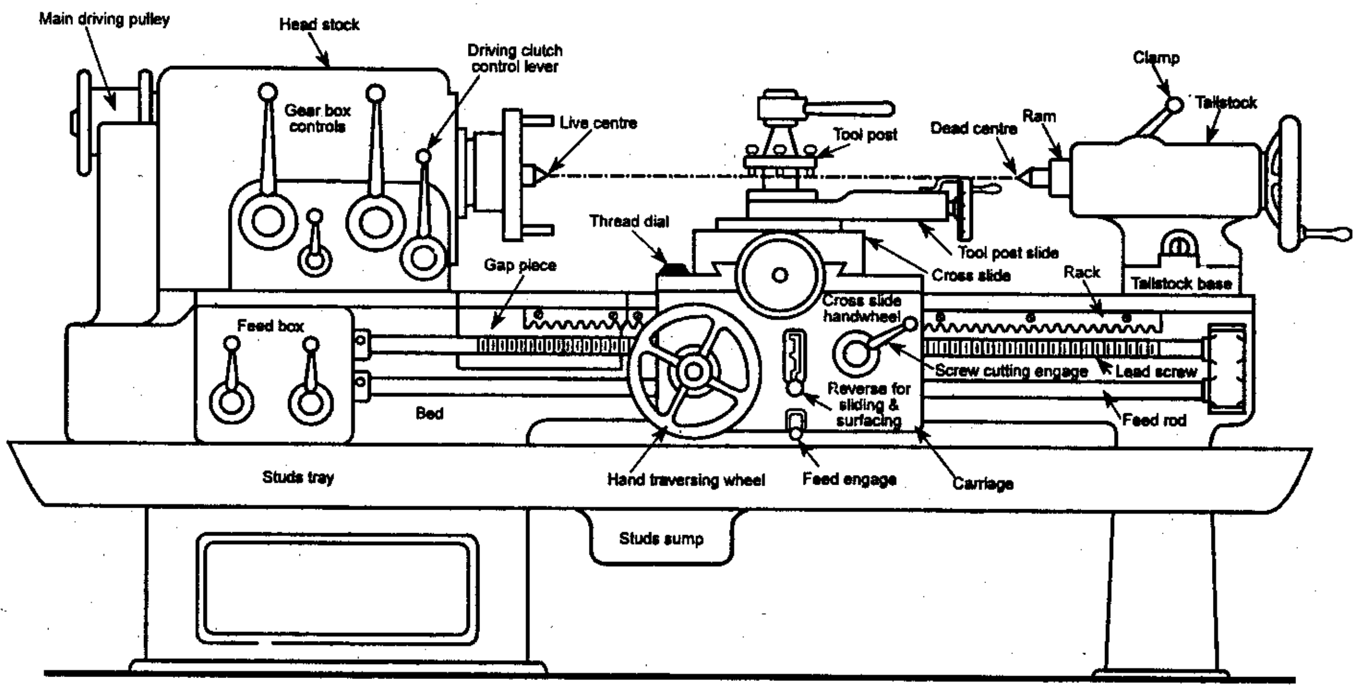


Figure 3.2 Lathe parts

3. The *length between centres*. This is the maximum length of work that can be mounted between the lathe centres.
4. The *swing diameter over carriage*. This is the largest diameter of work that will revolve over the lathe saddle, and is always less than the swing diameter over bed.
5. The *maximum bar diameter*. This is the maximum diameter of bar stock that will pass through hole of the headstock spindle.
6. The *length of bed*. This indicates the approximate floor space occupied by the lathe.

In ordering a lathe it is necessary to ask for certain other important particulars to specify the lathe correctly. These are : width of the bed, depth of the bed, depth and width of gap if it is a gap bed lathe, swing over gap, spindle nose diameter, centre taper Morse number and range of spindle speeds, number of feeds, number and range of metric and English threads that may be cut, pitch value of leadscrew, power input and floor are required.

3.5 DESCRIPTIONS AND FUNCTIONS OF LATHE PARTS

Fig.3.2 illustrates the basic parts of a geared head lathe. Following are the principal parts :

1. Bed.
2. Headstock.
3. Tailstock.
4. Carriage.
5. Feed mechanism.
6. Screw cutting mechanism.

3.6 THE BED

The lathe bed forms the base of the machine. The headstock and the tailstock are located at either end of the bed and the carriage rests over the lathe bed and slides on it. The lathe bed being the main guiding member of the tool, for accurate machining work, must satisfy the following conditions :

1. It should be sufficiently rigid to prevent deflection under tremendous cutting pressure transmitted through the tool-post and carriage to the lathe bed.
2. It must be massive with sufficient depth and width to absorb vibration.

3. It must resist the twisting stress set up due to the resultant of two forces– the downward cutting force on the tool and the force tending to move the tool away from the work in a horizontal direction. This is best done by diagonal ribbing or making box section casting shown in Fig.3.3 and Fig.3.4.
4. The bed should be seasoned naturally to avoid distortion or warp that may develop when it is cooled after the bed is cast. On the top of the bed there are two sets of *slides* or *guideways*– outerways and innerways. The outer guideways provide bearing and sliding surfaces for the carriage, and the innerways for the tailstock. The guiding surfaces are accurately machined to make them parallel to the lathe axis, absolutely horizontal, and sufficiently plain. The guiding surface should also be resistant to wear. Chilled castings are sometimes used to improve wear resisting qualities.

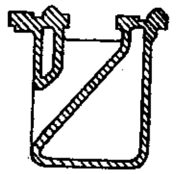


Figure 3.3 Box section lathe bed

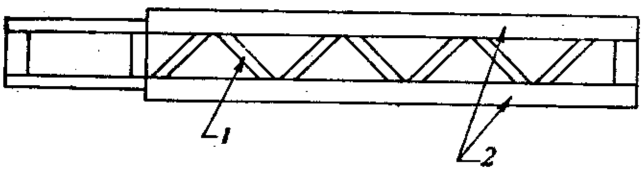


Figure 3.4 Lathe bed diagonal ribs
1. Diagonal rib, 2. Bedways.

The guideways of the lathe bed may be flat and inverted -V having an included angle of 90°. The wide flat guideways provide a large bearing surface with corresponding reduction in wear. Obviously the bearing surface requires particular care and attention to keep it always clean and perfectly smooth. In this type of guideways some adjustment of saddle keep-plates is necessary after wear. The inverted V type guideways, although expensive to machine, provide better guide for carriage and tailstock, ensure accurate alignment, and are unaffected by any wear. The shape of the V is such that the chips automatically fall through. But it has a small bearing surface which results increase in wear. This also weakens the saddle. Both V and flatways for each set of guideways are more commonly used to combine the advantages of both the types. Fig.3.5 illustrates different types of lathe bedways. Many lathes are made with a gap in the bed. This gap is used to swing extra large diameter pieces.

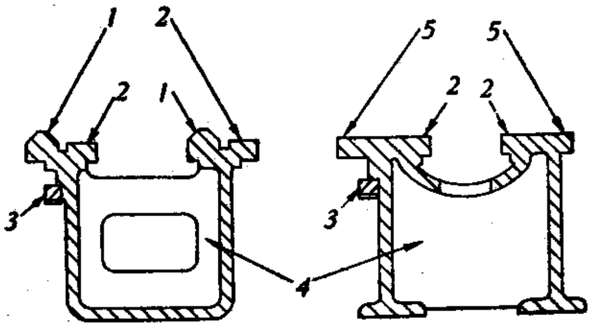


Figure 3.5 Types of lathe bedways
1. Inverted-V bedway, 2. Flat bedways, 3. Rack, 4. Box section, 5. Flat bedways for saddle.

The bed material should have high compressive strength, should be wear resistant and absorb vibration. Cast iron alloyed with nickel and chromium forms a good material suitable for lathe bed.

3.7 THE HEADSTOCK

The headstock is secured permanently on the innerways at the left hand end of the lathe bed, and it provides mechanical means of rotating the work at multiple speeds. It comprises essentially a hollow *spindle* and mechanism for driving and altering the spindle speed. All the parts are housed within the *headstock casting*.

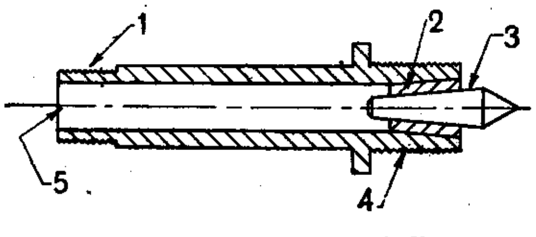


Figure 3.6 Headstock spindle
1. Threaded end, 2. Taper sleeve, 3. Live centre, 4. Threaded nose, 5. Spindle hole.

The *spindle* of the headstock, illustrated in Fig. 3.6, is made of carbon or nickel-chrome steel. This is usually of a large diameter to resist bending and it should be perfectly aligned with the lathe axis and accurately machined for producing true work surface. A hole extends through the spindle so that a long bar may be passed through the bore. The front end of the hole is appeared for holding centres and other tools having a standard Morse taper shank. A taper *sleeve* fits into the taper hole, and a *live centre* which supports the work and revolves with the work fits into the sleeve that acts as a bush. There are two common types of spindle noses : the *threaded design* which carries the chuck, driving plate and face plate, and the *flanged nose* which enable them to be directly attached. The lathe most commonly used has a threaded spindle nose.

The spindle revolves on two large bearings housed on the headstoc casting. The clearance between the spindle and the bearing should be minimum to prevent vibration. The bearing may be either bush, ball or roller type depending on whether it is a high speed, heavy duty or precision machine. Thrust bearings are provided to take up the end load owing to the feeding action of the tool. Provision is made for expansion of the spindle when it gets heated under high speed metal machining.

Speed changing : In a lathe it is necessary to vary the speed of the work to suit to different machining conditions. Tnese conditions are :

1. *The type of material to be cut.* Hard and tough materials like cast iron will require slower speed than soft materials like brass or aluminium.
2. *The type of cutting tool material used.* The spindle speed may be increased while using hard material like tungsten carbide.
3. *The type of finish desired.* The finishing cut requires small depth of cut and the work is rotated at a high speed . While rough turning the depth of cut is heavy and the work is rotated comparatively at a slower speed.
4. *The type of cutting fluid used.* Proper selection of coolant and lubricant permits high spindle speeds.
5. *The rigidity and condition of the machine.* A new and rigid machine can work at a speed higher than an old and worn out machine.
6. *The diameter of work.* The turning a work of large diameter requires slower spindle speed.
7. *The type of operation.* Operations, like turning, boring or drilling require higher spindle speeds than that required in thread cutting, tapping or reaming operations.

As the lathe may have to work under all the above conditions, it is necessary to provide arrangements for obtaining different speeds of the lathe spindle. The usual methods to vary the speed of a lathe spindle are :

1. By belt drive on cone pulley fitted on the headstock spindle with or without a back gear arrangement.
2. By all gear drive using sliding gears or clutches.
3. By variable speed motor.

Belt driven headstock : In a belt driven lathe fitted with back gear, there are usually two different methods of obtaining multiple speed of the lathe spindle. These are :

1. Direct speed or back gear out.
2. Indirect speed or back gear in.

Direct speeds : All belt driven lathes are provided with countershaft. The countershaft receives its power from the main shaft which is driven at constant speed. The countershaft is a short shaft having a set of fast and loose pulley and a stepped cone pulley for each machine. Fig.1.1 illustrates a typical countershaft drive. The step cone pulley on the countershaft is connected with the step cone pulley on the headstock spindle by a belt. A number of speeds can be obtained when the position of driving belt on the step pulley is changed. The spindle speed increases when the belt is shifted from a larger to a smaller step of the cone pulley. To stop the machine the belt is to be shifted from the fast to the loose pulley on the countershaft by means of a belt fork attached to a striking bar which is operated through a lever by simply pulling a cord.

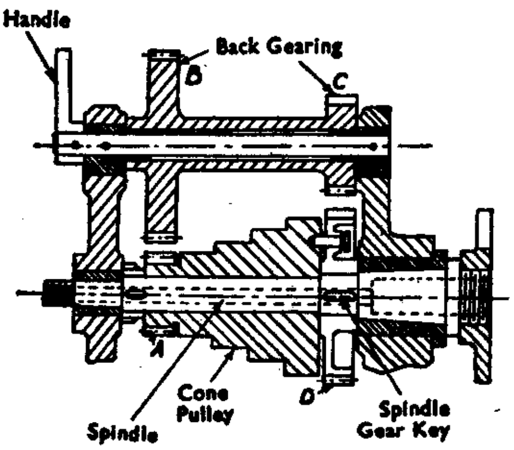


Figure 3.7 Backgeared headstock

A back geared headstock is illustrated in Fig.3.7 The cone pulley is not keyed to the spindle and revolves freely on it. The gear "*D*" called the "bull gear" is keyed to the spindle. In order to transmit motion from the step cone pulley to the spindle to obtain direct speed a lock pin is introduced into the hole provided on the face of the cone pulley. This lock pin engages the bull gear *D* with the cone pulley. The number of different speeds obtained by the lathe spindle depends on the number of steps on the cone pulley. A cone pulley with 4 steps will give 4 direct speeds.

The back gear : The back gear is an additional feature of a belt driven lathe and this is used to obtain wider range of spindle speeds, for the number of speeds obtained from "direct speeds" is limited to the number of steps only. When the backgear is engaged, the spindle speed reduces considerably. So it is also used when it is necessary to have a slow speed of the spindle that cannot otherwise be obtained by direct speed. A slow speed is necessary in the following cases :

1. In turning jobs of large diameter within the available cutting speed of the material.
2. In turning jobs of tough or hard material. When the material is hard it becomes necessary to apply greater cutting force by the tool to shear out the metal. This increase in cutting force will require greater turning torque necessitating slower spindle speed.
3. In operations like thread cutting, reaming, etc.
4. In taking deep cut as in rough turning.

Description of back gear : The back gears *B* and *C* in Fig.3.7 are both fastened to a quill. This is a hollow shaft that revolves on a fixed shaft which is housed on an eccentric bearing. This construction permits the changing of the position of the back gears putting them into engagement with the gears *A* and *D* by partly rotating the shaft by means of the back gear handle. The gear *A* is permanently connected with the cone pulley while the gear *D* is keyed to the spindle. The back gears are engaged when the lock pin connecting the bull gear *D* with the step cone pulley is out. In using back gears, the power is transmitted from the cone pulley and the gear *A* to the back gear *B* and *C*, from *C* to the gear *D*, and from the gear *D* to the spindle. If the gear *B* is three times as large as *A*, it will revolve one-third as fast as *B*. Gears *B* and *C* being both fastened to the quill will revolve at equal speed. If *D* is three times as large as *C*, it will revolve one-third as fast, with the result that *D* will revolve one-third of one-third, i.e. one-ninth as fast as *A* or cone pulley. For a particular speed of the cone pulley, the gear *D* or the spindle will rotate at a speed :

$$n_D = n_A \times \frac{Z_A}{Z_B} \times \frac{Z_C}{Z_D}$$

where, Z_A, Z_B, Z_C and Z_D are the numbers of teeth on gears *A, B, C* and *D,* and n_D and n_A are the speeds of the spindle and the speed of the cone pulley, respectively.

A lathe with four steps on the cone pulley and with backgears would thus have eight spindle speeds-- four direct and four indirect, the latter being slower than the former.

Example 3.1 : A lathe has four steps, the diameter of each being 90 mm, 130 mm, 170 mm and 210 mm. The countershaft pulley revolves at 100 r.p.m. The gears *A,B,C,* and *D* have 16,48,16,48 teeth respectively. Find the various speeds of the spindle.

Speeds without back gear :

As in Equ (1.1), $\frac{n_2}{n_1} = \frac{D_1}{D_2}$ or $n_2 = \frac{D_1}{D_2} \times n_1$

where, n_1 = r.p.m. of the countershaft,
n_2 = r.p.m. of the spindle,
D_1 = diameter of the countershaft pulley,
and D_2 = diameter of the spindle pulley.

Now,

1. $n_2 = \frac{210}{90} \times 100 = 233.3$ r.p.m.

2. $n_2 = \frac{170}{130} \times 100 = 130.7$ r.p.m.

3. $n_2 = \frac{130}{170} \times 100 = 76.5$ r.p.m.

4. $n_2 = \frac{90}{210} \times 100 = 42.8$ r.p.m.

Speed with back gears : $n_D = n_A \times \frac{Z_A}{Z_B} \times \frac{Z_C}{Z_D}$

So,

1. $n_D = 233.3 \times \frac{16}{48} \times \frac{16}{48} = 233.3 \times \frac{1}{9} = 25.9$ r.p.m.

2. $n_D = 130.7 \times \frac{16}{48} \times \frac{16}{48} = 130.7 \times \frac{1}{9} = 14.5$ r.p.m.

3. $n_D = 76.5 \times \frac{16}{48} \times \frac{16}{48} = 76.5 \times \frac{1}{9} = 8.5$ r.p.m.

4. $n_D = 42.8 \times \frac{16}{48} \times \frac{16}{48} = 42.8 \times \frac{1}{9} = 4.7$ r.p.m.

Available 8 speeds of the spindle are :
4.7, 8.5, 14.5, 25.9, 42.8, 76.5, 130.7, 233.3

All-gear drive : Modern lathes are equipped with all-geared headstock to obtain various spindle speeds. These lathes are driven by a constant–speed motor usually located in the base of the lathe, or they may be driven by belt on a single pulley. Speed changes are made through a series of gear combinations by shifting two or three levers in different positions. The gear drive is mostly used in heavy duty machines and where smooth running is required.

The different mechanisms that are commonly used in all-geared headstock are :

1. Sliding gear mechanism.
2. Sliding clutch mechanism.
3. Combination of the above two types.

A *two speed driving motor is* sometimes employed to extend the range of speeds available from gear mechanism.

In a lathe design, the standard practice is to arrange the gearing in a manner so that the spindle speed increases in geometrical progression, that is, each speed is multiplied by a constant to get the next higher speed. As for example, if L be the first sped and r be the constant number, the second speed will be $L \times r$, the third speed will be $L \times r \times r = Lr^2$ and so on. This method of computing spindle speeds is adopted to distribute the different speeds uniformly between the high and the low limits. If n be the number of speeds required and H and L be the highest and the lowest speed respectively, the constant r can be determined by the formula :

$$r = \sqrt[n-1]{\frac{H}{L}}$$

The international standard values of r are 1.12, 1.25, 1.4, 1.6 and 2. Please refer chapter 18 for further details.

The most simple and very common arrangement of obtaining multiple speeds in an all-geared headstock is the one by *sliding gear mechanism.* Various speed changes are obtained when a set of gears is made to slide on a splined shaft bringing them into mesh only one at a time with a cluster of gears mounted on a second shaft. A friction or claw clutch which ensures engagement or disengagement of a shaft is also employed in many headstocks to obtain multiple speeds.

Modern lathes are often equipped with both sliding gear and sliding clutch mechanism. Fig.3.8 illustrates a 9-speed all-geared headstock employing sliding gear mechanism. Gears 4, 5 and 6 are mounted on a splined shaft and receives power from the fast and the loose pulley. Gears 4, 5 and 6 may be made to mesh with gears 7, 8 and 9 respectively by shifting with the levers. Gears 7, 8 and 9 rotate freely on the intermediate shaft and cannot move axially. Similarly, gears 11, 12 and 13 may be made to slide by means of a second lever on the headstock spindle which is a splined one. Number of gear teeth corresponding a gear *i* is denoted by Z_i for example Z_1 is the number of teeth of gear 1.

The gear combinations for nine different speeds are :

1. $\frac{Z_4}{Z_7} \times \frac{Z_7}{Z_{11}}$
2. $\frac{Z_5}{Z_8} \times \frac{Z_7}{Z_{11}}$
3. $\frac{Z_6}{Z_9} \times \frac{Z_7}{Z_{11}}$
4. $\frac{Z_4}{Z_7} \times \frac{Z_8}{Z_{12}}$
5. $\frac{Z_5}{Z_8} \times \frac{Z_8}{Z_{12}}$
6. $\frac{Z_6}{Z_9} \times \frac{Z_8}{Z_{12}}$
7. $\frac{Z_4}{Z_7} \times \frac{Z_9}{Z_{13}}$
8. $\frac{Z_5}{Z_8} \times \frac{Z_9}{Z_{13}}$
9. $\frac{Z_6}{Z_9} \times \frac{Z_9}{Z_{13}}$

In gear selection the rule is : the total number of teeth between any one pair of gears mounted on two shafts must be equal to the total number of teeth on the other pair.

The advantages of an all-geared drive compared to a cone pulley drive are summarized below :

1. The design permits a totally enclosed compact unit giving better appearance and larger range of spindle speeds.
2. The initial high belt or motor speed of a geared drive ensures that practically the full power is available for all speeds, and the power input and that available at the tool edge are roughly constant at all spindle speeds. In a cone-pulley the power input to the machine spindle varies with the speed.
3. No belt shifting is necessary and the power supply to the headstock can be greatly increased by using a wide belt at high initial belt speed.
4. All changes in the spindle speed, being made by a simple movement of one or more levers, are obtained more quickly and with no chance of any accident.
5. No overhead shafting is needed, as the power is normally taken from an independent motor. Even if the power is not taken from an independent motor, the necessity of mounting a countershaft is eliminated. This makes the machine shop more spacious, clean, airy and lighted.

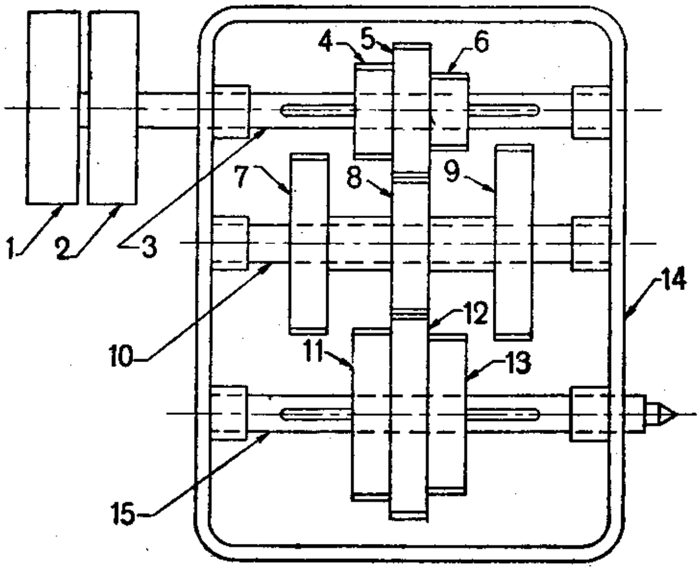


Figure 3.8 9-Speed all-geared headstock.

1. Fast pulley, 2. Loose pulley, 3. Splined shaft, 4, 5, 6, 7, 8, 9, 11, 12, 13. Gears having Z_4, Z_5, Z_6, Z_7, Z_8, Z_9, Z_{11}, Z_{12}, Z_{13} numbers of teeth respectively, 10. Intermediate shaft, 14. Headstock casting, 15. Splined headstock spindle.

6. The drive may be isolated from the headstock spindle by mounting the driving pulley on another shaft, and thus vibration of the spindle is reduced to a minimum.

The disadvantages are :

1. All -geared lathes are costlier than the belt driven lathes owing to more complicated gear and lever and lever mechanism.
2. Some power is lost due to friction of the gears.
3. In case of overloading the machine, for having no arrangement of belt slipping, there is little possibility to prevent damage to the parts.

3.8 TAILSTOCK OR LOOSE HEADSTOCK

The tailstock is located on the innerways at the right hand end of the bed. This has two main uses : (1) it supports the other end of the work when it is being machined between centres, and (2) it holds a tool for performing operations such as drilling, reaming, tapping, etc. A tailstock is illustrated in Fig.3.9.

To accommodate different lengths of work, the body of the tailstock can be adjusted along the ways chiefly by sliding it to the desired position where it can be clamped by bolts and plates. The upper casting of the body can be moved toward or away from the operator by means of the adjusting screws to offset the tailstock for taper turning and to realign the tailstock centre for straight turning. The body is bored to act as the *barrel* which carries the *tailstock spindle* that moves in and out of the barrel by means of a screw when the *tailstock handwheel* is turned. The front of the spindle has a taper hole into which the dead centre or other tools fit. The screw thread is left handed, so that clockwise rotation of the handwheel causes the spindle to advance, while anticlockwise rotation causes the spindle to be drawn inward and ultimately the end of the screw strikes the back of the dead centre or any tool that is fitted into the hole. To remove tools from the spindle, it is therefore, only necessary to back up on the handwheel until the spindle end is nearly inside the casting. The spindle has a key way in the underside which mates with a small key fitted on the barrel to prevent rotation. After the adjustment is made, the spindle is clamped in position by tightening the locking bolt on split lug.

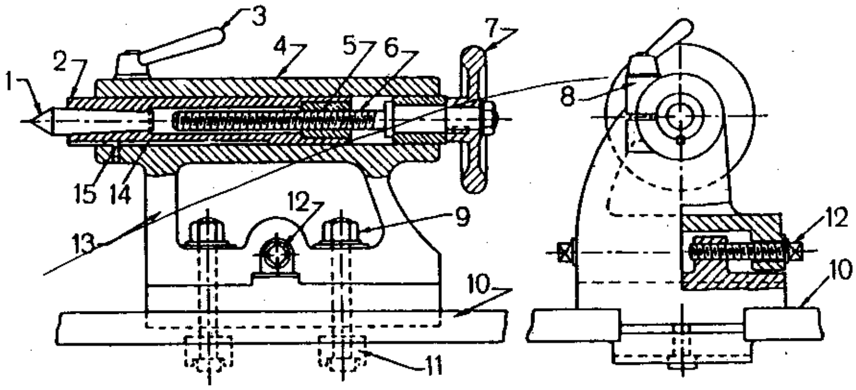


Figure 3.9 Tailstock

1. Dead centre, 2. Spindle, 3. Spindle clamp, 4. Barrel, 5. Bush, 6. Square threaded screw, 7. Hand wheel, 8. Split lug, 9, Tailstock clamping bolt, 10. Lathe bedways, 11. Clamping plate, 12. Setover screw, 13. Body, 14. Keyway, 15. Key.

Other features sometimes incorporated on tailstocks are graduated spindles and micrometer dials on the handwheels for accurate length setting, and felt wiper pads on the base and spindle.

3.9 CARRIAGE

The carriage of a lathe has several parts that serve to support, move and control the cutting tool. It consists of the following parts : (1) saddle, (2) cross-slide, (3) compound slide or compound rest, (4) tool post, and (5) apron. A sectional view of the carriage is shown in Fig.3.10.

Saddle : The saddle is an H-shaped casting that fits over the bed and slides along the ways. It carries the cross slide and tool post. Some means are generally provided for locking the saddle to prevent any movement when surfacing operations are carried out.

The cross-slide : The cross-slide comprises a casting, machined on the underside for attachment to the saddle and carries locations on the upper face for the tool post or compound rest. The cross-piece of the saddle is mechanized with a dovetail way, at right angles to the centre axis of the lathe, which serves to guide the cross-slide itself.

In order to move the cross-slide, the feed screw is turned by rotating the handwheel. Transverse movement is obtained when the nut mounted on the feed screw is engaged with the binder screw of the cross-slide. When a taper turning attachment is used the binder screw of the cross-slide. When a taper turning attachment is used the binder screw is opened to disconnect the cross-slide from the cross-feed screw, and the extension of the slide is attached with the guide block shown in Fig.3.49. Automatic movement of the cross-slide is obtained when the pinion keyed to the cross-feed screw is in mesh with the apron gearing.

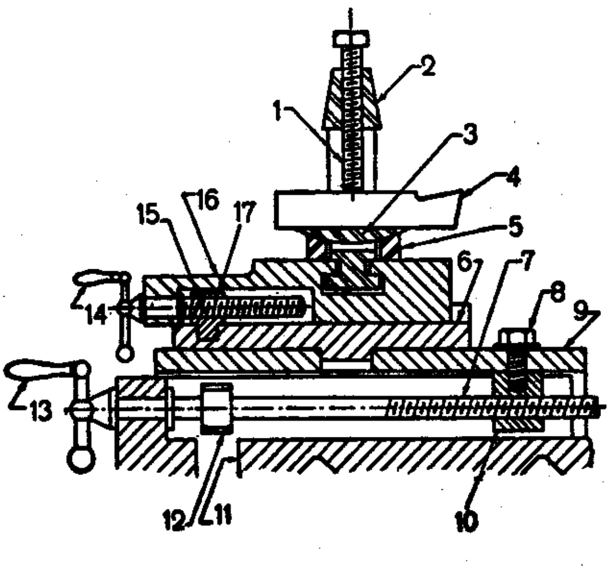


Figure 3.10 Carriage

1. Toolpost screw, 2. Tool post, 3. Rocker, 4. Tool, 5. Concave ring, 6. Compound rest swivel base, 7. Crossfeed screw, 8. Binder screw, 9. Cross slide, 10. Cross-slide nut, 11. saddle, 12. Pinion on Crossfeed screw for automatic feed, 13. Cross slide hand wheel, 14. Compound slide hand wheel, Compound slide feed screw, 16. Compound rest, 17. Compound slide nut.

Usually cross-slide hand wheels are graduated on their rims, or a separate micrometer dial may be fitted on them so that a known amount of feed can be applied. One small division of the dial is equal to 0.05 mm.

The compound rest : The compound rest or compound slide is mounted on the top of the cross-slide and has a circular base graduated in degrees. It is used for obtaining angular cuts and short tapers as well as convenient positioning of the tool to the work. By loosening two set-screws which fit in a V-groove around the compound-rest base, the rest or the slide may be swivelled to any angle within a circle. There is no power feed to the compound rest and it is hand operated. The compound-rest handle is also equipped with a micrometer dial to assist in determining the depth of the cut. After necessary setting the compound slide is locked solid with its base.

The tool post : This is located on the top of the compound rest to hold the tool and to enable it to be adjusted to a convenient working position. The type and mounting of the tool post depends upon the class of work for which it is to be used. The rigidity of the tool holder and effective method of securing are the essential factors in designing a toolpost. Following are the common types of tool post :

1. Single screw tool post
2. Four bolt tool post
3. Open side tool post
4. Four way tool post

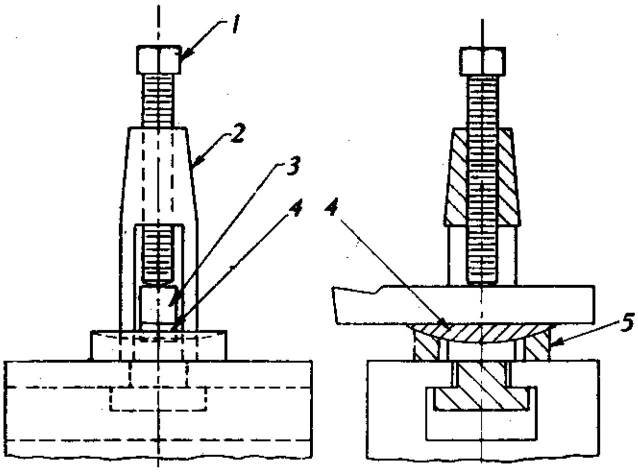


Figure 3.11 Single screw toolpost
1. Toolpost screw, 2. Toolpost body, 3. Tool, 4. Convex rocker, 5. Concave ring.

Single screw tool post : The single screw tool post is illustrated in Fig.3.11. This consists of a round bar with a slotted hole in the centre for fixing the tool by means of a set screw. The tool post with concave ring and convex rocker slides in a T-slot on the top of the compound rest. The height of the tool point can be adjusted by tilting the rocker and clamping it in position by the set screw. The tool post can be swivelled about its vertical axis.

The disadvantage with this type of tool post is that adjustment to height by tilting obviously alters all the cutting angles of the tool. The tool post is not also rigid enough for heavy work as only one clamping screw is used to clamp the tool.

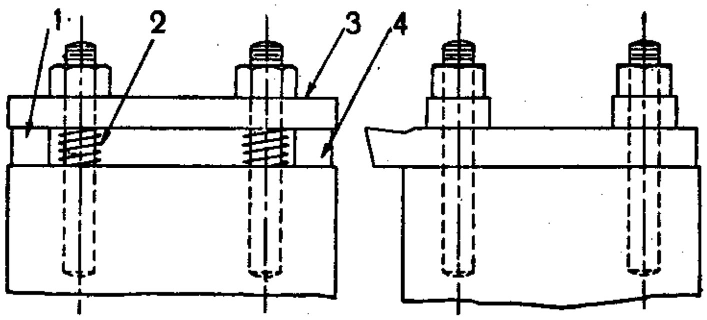


Figure 3.12 Four bolt toolpost

1. Tool, 2. Coil spring, 3. Strap, 4. Fulcrum block.

Four bolt tool post : The four bolt tool post is illustrated in Fig.3.12. The tool is held in position by two straps and four bolts. Loose coil springs are fitted to each bolt to keep the straps in place and greatly facilitate the setting up of the tools. Adjustment for tool height can be made by using parallel packing strips under the tools.

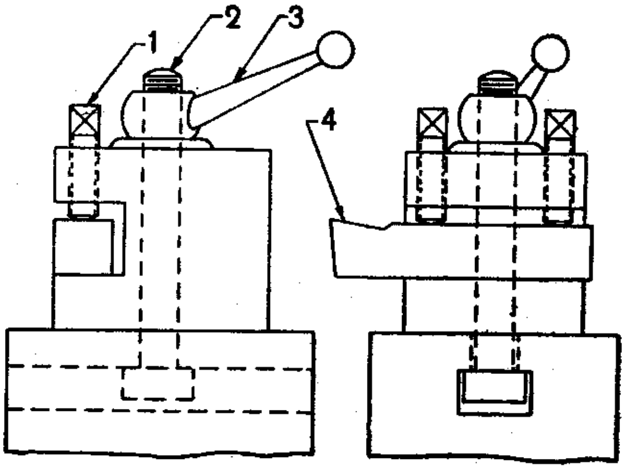


Figure 3.13 Open side toolpost

1. Setscrew, 2. Central clamping bolt, 3. Clamping bolt handle, 4. Tool.

This type forms a very firm support for either a single or double tool set-up. This is, therefore, often fitted to heavy duty lathes. This type does not swivel in itself, and setting of the tool in any desired angle is effected by the adjustment of the compound slide.

Open side tool post : The open side tool post is illustrated in Fig.3.13. The tool is held quite independent of the main fixing bolt and clamped in position by two set screws. The height of the cutting point can be adjusted by using parallel packing strips and the tool post slide can be swivelled to any desired position after loosening the central bolt which slides in a T-slot. This arrangement ensures quick replacement of the tool.

Four way tool post : The four way tool post is illustrated in Fig.3.14. In this type of tool post four sides are open to accommodate four tools at a time. The tool is held in position by separate screws and a locking bolt is located at the centre. The tools are fitted in proper sequence of operation and by indexing the tool post through 90° any one of the tools may be fed into the work. Indexing device may be incorporated to enable the post to be swivelled exactly through 90°. This type of tool post is used in moderately heavy lathes and is suitable for repetition work.

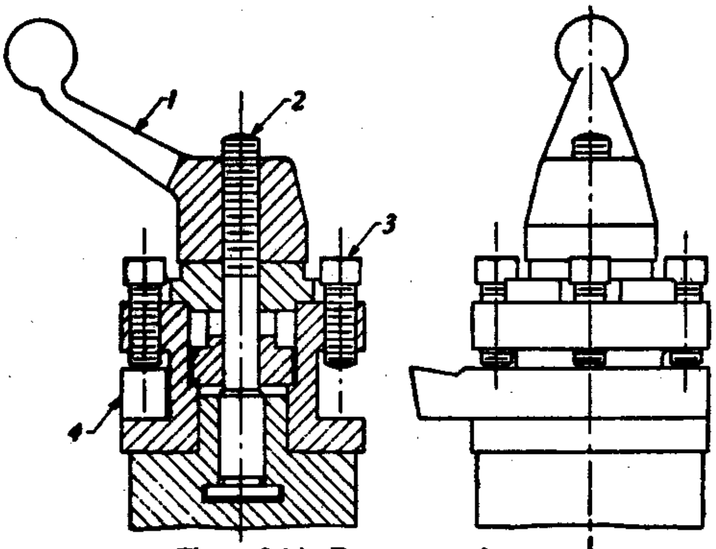


Figure 3.14 Four way toolpost
1. Clamping handle, 2. Central locking bolt, 3. Set screws, 4. Tool.

The apron : The apron is fastened to the saddle and hangs over the front of the bed. It contains gears, clutches, and levers for operating the carriage by hand and power feeds. The apron also contains friction clutches for automatic feeds. In addition, there is a *split nut* which engages, when required with the lead screw, when cutting either internal or external threads. The lay out of the apron includes an interlocking device which prevents the simultaneous engagement of the feed shaft and the lead screw. The apron hand wheel can be turned to move the carriage back and forth longitudinally by hand. The complementary motion to this is obtained by the cross-feed handle which moves the cross-slide back and forth across the saddle. The hand wheel is connected via a pinion to a series of spur gears, and then to pinion meshing with a rack fitted to the lathe bed. Usually a chasing dial or thread cutting dial is fitted either to the side or top of the apron and consists of a graduated dial. It has entirely independent drive provided by a worm wheel which is in constant mesh with the lead screw.

3.10 FEED MECHANISM

The movement of the tool relative to the work is termed as *"feed"*. A lathe tool may have three types of feed--longitudinal, cross, and angular. When the tool moves parallel to the lathe axis, the movement is termed as *longitudinal feed* and is effected by the movement of the carriage. When the tool moves at right angle to the lathe axis with the help of the cross slide the movement is termed as *cross feed*, while the movement of the tool by compound slide when it is swivelled at an angle to the lathe axis is termed as *angular feed*. Cross and longitudinal feed are both hand and power operated, but angular feed is only hand operated.

The feed mechanism has different units through which motion is transmitted from the headstock spindle to the carriage. Following are the units :

1. End of bed gearing.
2. Feed gear box.
3. Feed rod and lead screw.
4. Apron mechanism.

End of bed gearing : This gearing serves the purpose of transmitting the drive to the lead screw and feed shaft, either direct or through a gear box. In modern lathes, *tumbler gear mechanism or bevel gear feed reversing mechanism* is incorporated to reverse the direction of feed.

Tumbler gear mechanism : Fig.3.15 illustrates tumbler gear mechanism. Tumbler gears are used to give the desired direction of movement to the lathe carriage, via lead screw or the feed shaft. The tumbler gearing comprises of two pinions mounted on a bracket. The bracket is pivoted about the 1st stud shaft. The design provides three positions of the bracket : forward, neutral, and reverse. With the forward position, only one gear is in contact between the lathe spindle and the main gear train, and the lathe carriage is moved towards the headstock. With the reverse, the drive is through the two gears, the second gear being introduced only to reverse the direction of rotation, and the carriage is moved away from the headstock. If the tumbler gears are brought into the neutral position, the spindle is disengaged from the lead screw or feed shaft gear box.

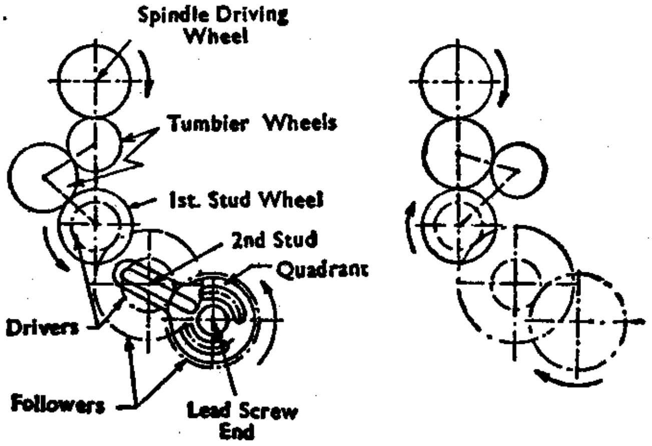


Figure 3.15 Tumbler gear feed reversing mechanism.

Bevel gear feed reversing mechanism : The tumbler gear mechanism being a non-rigid construction, cannot be used in a modern heavy duty lathe. The clutch operated bevel gear feed reversing mechanism incorporated below the headstock or in apron provides sufficient rigidity in construction. This is shown in Fig.3.16.

The motion is communicated from the spindle gear *2* to the gear on the stud shaft through the intermediate gear. The bevel gear *8* is attached to the gear on the stud shaft and both of them can freely rotate on shaft *7*. The bevel gear *8* meshes with bevel gear *12*. *12* meshes with *10*. *12*, *10*, and *8* are having equal number of teeth. The bevel gear *10* can also rotate freely on shaft *7*. A clutch *11* is keyed to the shaft *7* by a feather key and may be shifted to the left or right, by the lever *9* to be engaged with gear *8* or *10* or it remains in the neutral position. When the clutch engages with bevel gear *8*, gear *3* which is keyed to the shaft *7* and the lead screw, rotates in the same direction as the gear *2*. The direction of rotation is reversed when the clutch *11* engages with gear *10*.

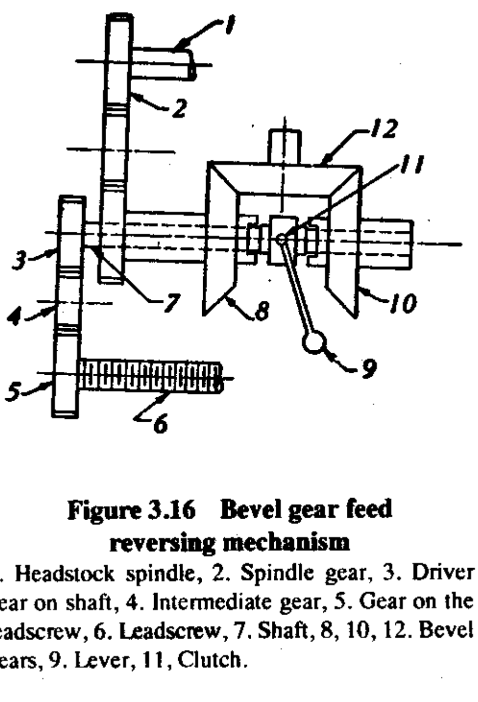


Figure 3.16 Bevel gear feed reversing mechanism

1. Headstock spindle, 2. Spindle gear, 3. Driver gear on shaft, 4. Intermediate gear, 5. Gear on the leadscrew, 6. Leadscrew, 7. Shaft, 8, 10, 12. Bevel gears, 9. Lever, 11, Clutch.

Change gears : The train of gears through which the motion is transmitted from the stud shaft to the lead screw is called change gears. The change gear train consists of the *gear on stud, the intermediate and the gear on the lead screw*. This is illustrated in Fig.3.15. The stud gear transmits motion through the intermediate gear to the driven gear. The intermediate gear is mounted on a 'quadrant', a slotted link pivoted on the lead screw and is arranged in any desired position. The intermediate gear can be adjusted to any position in a fairly long slot of the quadrant to engage both the stud gear and the lead screw gear. By changing the sizes of the gears on stud and screw various velocity ratio between the two may be obtained.

Feed gear box : The feed gear box or quick change gear box is fitted directly below the headstock assembly. Power from the lathe spindle is transmitted through gears to the quick change gear box. This gear box contains a number of different sizes of gears which provides a means to change the rate of feed, and the ratio between revolutions of the headstock spindle and the movement of the carriage for thread cutting by altering the speed of rotation of the feed rod or lead screw.

The arrangements which are employed in feed gear boxes to obtain multiple speeds and different rates of feeds are :

1. Sliding gear mechanism.
2. Sliding clutch mechanism.
3. Gear cone and tumbler gear mechanism.
4. Sliding key mechanism.
5. Combination of any two or more of the above.

The sliding gear and sliding clutch mechanism explained in Art 3.7 gives a smaller range of feed. The gear cone and tumbler gear mechanism is very common and is used in almost all modern lathes. When used in combination with sliding gear or sliding key mechanism this provides a wide range of feed.

Usually two or three levers must be moved to obtain the desired combination within a given range. An index chart or plate attached to the machine indicates the position of the lever for a given feed, or number of threads per inch or per so many millimeter pitch.

Gear cone and tumbler gear mechanism : Fig.3.17 illustrates the mechanism. A number of different sizes gears are keyed to the driving shaft in the form of a cone. A sliding gear *4*, keyed to the driven shaft, meshes with the idle gear *2* which is held in a bracket pivoted on the driven shaft. To change the gears the handle is pulled downwards to slide the whole assembly of gears *2* and *4* to the proper position and the idler is then engaged with the cone.

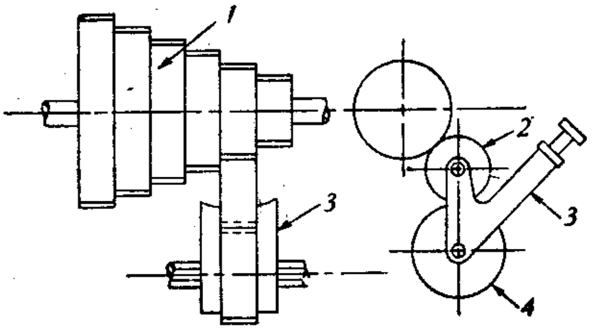


Figure 3.17 Tumbler gear mechanism
1. Gear cone, 2. Idle gear, 3. Bracket, 4. Sliding gear.

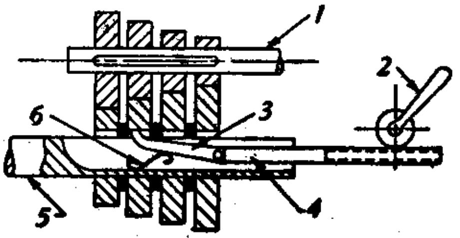


Figure 3.18 Sliding key mechanism
1. Driven shaft, 2. Sliding key operating handle, 3. Sliding key, 4. Sliding rod. 5. Driving' shaft, 6. Leaf spring.

Sliding key mechanism : The mechanism consists of a cone of gears mounted on a hollow shaft *5* and any one of the gears may be engaged with the shaft *1* by a sliding key *3* that slides within the hollow shaft. Steel washers are placed between the gears to prevent simultaneous engagement of adjacent gears by the sliding key. The cone of gears is permanently in mesh with the driven shaft gears transmits motion from the keyed gear to the driven shaft. Fig.3.18 illustrates a sliding key mechanism.

Drive of the feed rod and the lead screw : Fig.3.19 shows a complete driving arrangement of a feed rod and lead screw. The motion is transmitted from the spindle gear through the tumbler gears and change gears to the shaft *5* on which twelve gears are keyed. Twelve different speeds may be obtained by the shaft *7* by the sliding gear. With the use of the sliding key and four additional gears on shaft *7* and *12*, the shaft *12* can receive 12×4=48 speeds, i.e. 48 different feeds. The clutch enables the lead screw to be engaged or disengaged only one at a time.

Feed rod : The feed rod is a long shaft that has the keyway extending from the feed box across and in front of the bed. The power is transmitted from the lathe spindle to the apron gears through a feed rod via large number of gears. The feed rod is used to move the carriage or cross-slide for turning, boring, facing and all other operations except thread cutting.

Lead screw : The lead screw is a long threaded shaft used as a master screw, and is brought into operation only when threads have to be cut. In

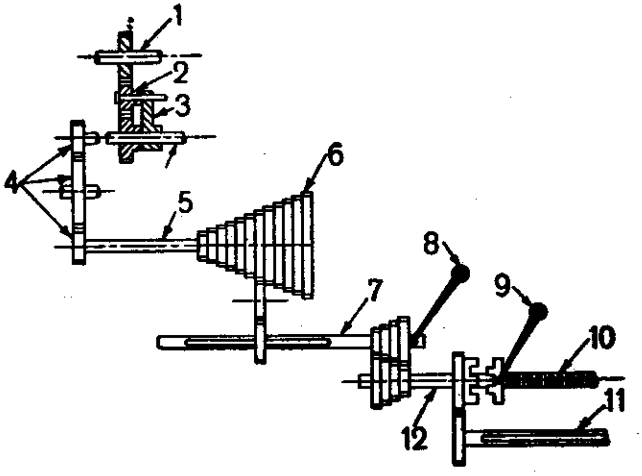


Figure 3.19 Layout diagram of feed drive

1. Headstock spindle, 2. Tumbler gear, 3. Tumbler bracket, 4. Change gears, 5. Cone gear shaft, 6. Cone gears, 7. Sliding gear shaft, 8. Sliding key operating handle, 9. Clutch handle, 10. Leadscrew, 11. Feed rod, 12. Driven shaft.

all other times the lead screw is disengaged from the gear box and remains stationary, but this may be used to provide motion for turning, boring, etc. in lathes that are not equipped with a feed rod.

Apron mechanism : Different designs of apron mechanism for transforming rotary motion of the feed rod and the lead screw into feed motion of the carriage are constructed by different makers of the lathe. Fig.3.20 illustrates a typical apron mechanism. A sliding gear *22* mounted on the feed rod drives the worm gear *8* through the gear *23* and worm *24*. To get an automatic longitudinal feed motion, a knob *17* on the apron is turned. This causes the cone clutch *9* mounted on the worm gear shaft to be engaged, and the motion is transmitted from the worm gear through the cone clutch and driving gears *10* and *11* to the pinion *13* mounted on the

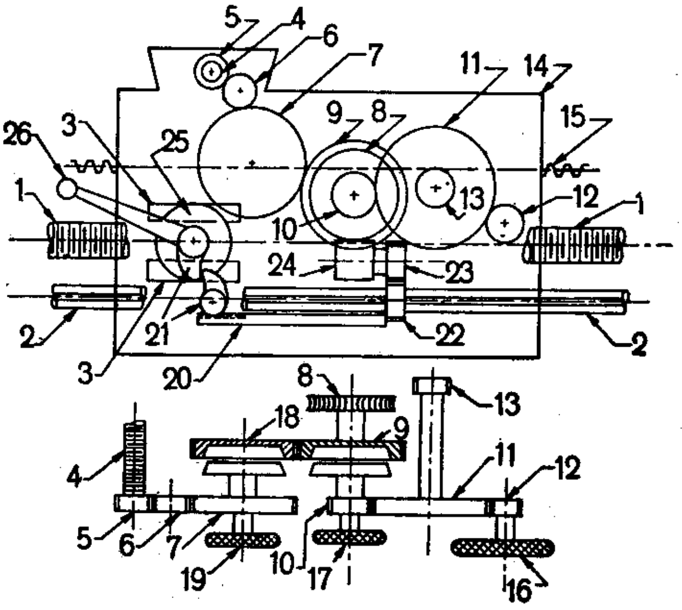


Figure 3.20 Apron mechanism

1. Leadscrew, 2. Feed rod, 3. Half nut, 4. Crossfeed screw, 5,6,7. Gear train for crossfeed, 8. Worm gear, 9. Cone clutch for longitudinal feed, 10, 11. Gear train for longitudinal feed, 12. Gear for hand feed, 13. Pinion, 14. Apron, 15. Rack, 16. Handwheel for longitudinal feed, 17. Knob for longitudinal feed, 18. Cone clutch for crossfeed, 19. Knob for cross-feed, 20. Lever, 21. Cam, 22. Sliding gear, 23. Gear on worm shaft, 24. Worm, 25. Cam plate, 26. Half nut operating handle.

driving gear shaft. The pinion meshes with the rack *15* and the carriage gets the automatic longitudinal feed motion. To obtain automatic cross-feed movement, a separate knob *19* on the apron is turned. This causes a second pair of cone clutch *18* to be engaged. Both the cone clutches have teeth cut on their periphery and they are in mesh with one another. So when the worm gear rotates, the motion is transmitted to the cross-feed screw *4* through the cone clutches *9* to *18* and through a set of gears *7*, *6* and *5*. The cross-feed movement is obtained as the screw is rotated within the cross slide nut. To disengage the feed, both cross and longitudinal, a lever *20* is operated to drop the worm out of engagement with the worm wheel. The mechanism also ensures that when the half nut is engaged with the lead screw the worm drops down disconnecting the feed motion. This interlocking device prevents simultaneous engagement of the carriage with the feed shaft and lead screw and saves the machine from any probable damage. This arrangement of the apron is called *fool-proof arrangement.*

3.11 THREAD CUTTING MECHANISM

The rotation of the lead screw is used to transverse the tool along the work to produce screw thread. The ***half-nut mechanism*** illustrated in Fig. 3.21 makes the carriage to engage or disengage with the lead screw. It comprises a pair of half nuts *7* capable of moving in or out of mesh with the lead screw. The two halves of the nut are connected in the cam slots *1* in a circular disc *6* by two pins *5*. When the disc is rotated by a hand lever *4* attached to it, the pins being guided in the cam slots serve to open or close

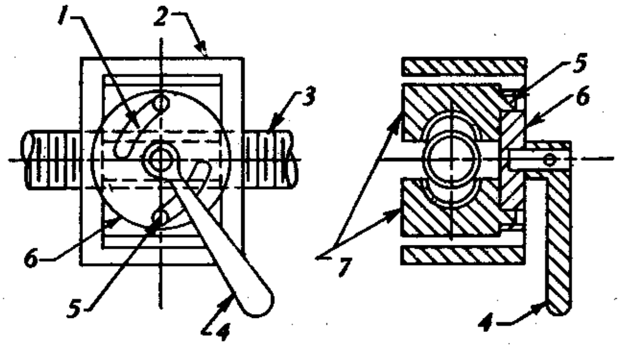


Figure 3.21 Half nut mechanism

1. Cam slot, 2. Guide or frame, 3. Lead screw, 4. Hand lever, 5. Pin, 6. Circular plate, 7. Half nuts.

the split nuts and thus engages or disengages with the lead screw. The half nuts slide within the guide or frame 2. Closing the half nuts causes the carriage to move a fixed distance for each revolution of the spindle. The direction in which it moves depends upon the position of the feed reverse lever on the headstock. The split nut is used only for thread cutting and never for any other operation.

3.12 LATHE ACCESSORIES AND ATTACHMENTS

Lathe accessories include centres, catch plates and carriers, chucks, collets, face plates, angle plates, mandrels, and rests. They are used either for holding and supporting the work or for holding the tool.

Attachments are additional equipment used for specific purposes. They include stops, ball turning rests, thread chasing dials, and taper turning, milling, grinding, gear cutting, turret, cutter, relieving and crank pin turning attachments.

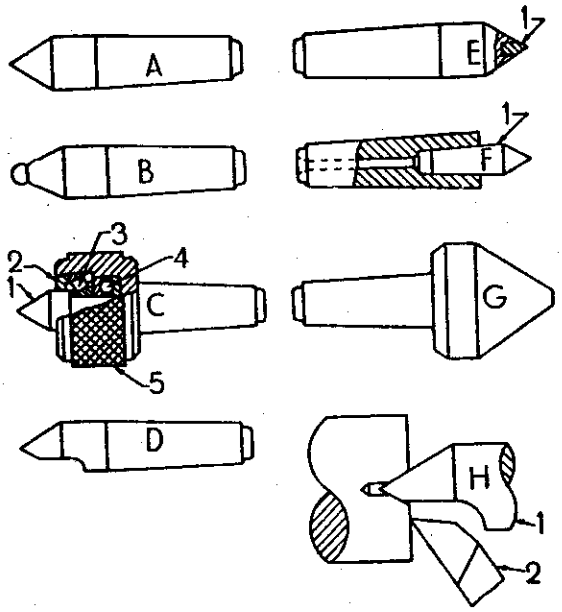


Figure 3.22 Lathe centre

A. Ordinary centre, B. Ball centre, C. Frictionless centre– 1. Insert type centre, 2. Nut, 3. Roller bearing, 4. Thrust bearing, 5. Housing, D. Half centre, E. Tipped centre–1. Brazed tip, F. Insert type centre–1. Insert, G. Pipe centre, H. Use of half centre–1. Half centre, 2. Facing tool.

Lathe centres : The most common method of holding the work in lathe is between the two centres–live centre and dead centre. These two centres take up the thrust due to metal cutting and the entire load of the workpiece on small bearing surface. So they are made of very hard materials to resist deflection and wear. The dead centre is subjected to wear due to friction. The included angle of the centre is usually 60° for general purpose work and 75° for heavy work. The shanks of all the centres are machined to the Morse (0 to 6) or Metric (4 and 6) standard tapers. Different types of centre for different types of work are shown in Fig.3.22.

The *ordinary centre* is the type used for most general work. In the *tipped centre,* the point consists of a hard alloy tip brazed into an ordinary steel shank. This is more expensive type of centre, one which will give excellent service against wear and strain. The *ball centre* is used to minimize wear and strain on the ordinary centre except that little less than half of the centre has been ground away. This construction facilitates facing of the bar ends without removal of the centre. Fig.3.22-H illustrates the use of a half-centre in a facing operation. The *insert type of centre* is used for reasons of economy as only the high-speed steel "insert" can be replaced instead of replacing the whole centre. The *rotating* or *Frictionless centre* is always used in tailstock for supporting heavy work revolving at a high speed. An ordinary insert type centre revolves freely on the ball and the roller bearings fitted in a housing having a taper shank corresponding to the

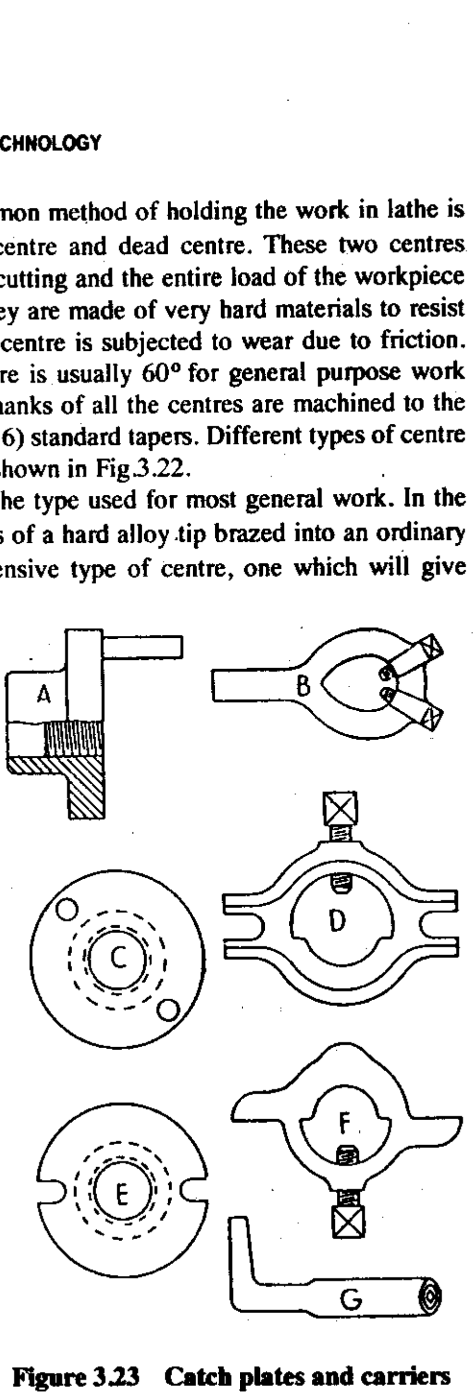


Figure 3.23 Catch plates and carriers
A. Single pin catch plate, B. Straight tail carrier, C. Double pin catch plate, D. Double slotted carrier, E. Double slotted catch plate, F. Double tail carrier, G. Bent tail carrier.

taper of the tailstock spindle. The ball and roller bearings reduce friction and take up end thrust and allow the centre to revolve with the work for a long period without developing any appreciable heat. The *pipe centre* is used for supporting the open end of pipes, shells, etc. for thread cutting or turning in the lathe.

To reduce friction at the dead centre point tallow, tallow and graphite or graphitized oil may be used.

Carriers and catch plates: Carriers and catch plates are used to drive a workpiece when it is held between two centres. Carriers or *driving dogs* are attached to the end of the workpiece by a set screw, and catch plates are either screwed or bolted to the nose of the headstock spindle. A projecting pin from the catch plate or carrier fits into the slot provided in either of them. This imparts a positive drive between the lathe spindle and workpiece. Fig.3.23 illustrates different types of catch plates and carriers. The projecting pin of a *single pin catch plate* drives the straight end or tail of a carrier attached to the workpiece. Two pins of a *double pin catch plate* engage with the *double tail or double slotted carrier* and provide uniform drive. The *bent tail* type is used in conjunction with a face plate or slotted catch plate.

Chucks : A chuck is one of the most important devices for holding and rotating a piece of work in a lathe. Workpieces of short length, and large diameter or of irregular shape which cannot be conveniently mounted between centres are held quickly and rigidly in a chuck. A chuck is attached to the lathe spindle by means of bolts with the back plate screwed on to the spindle nose. Accurate alignment of the chuck with the lathe axis is effected by spigotting. The different types of chucks are :

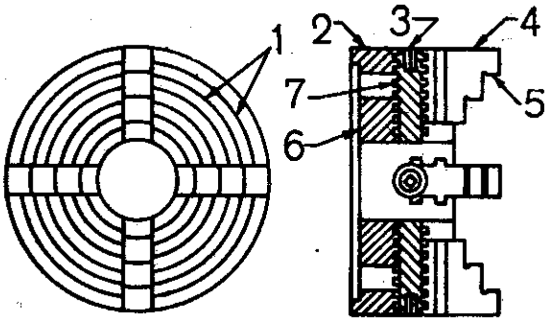


Figure 3.24 Four jaw independent chuck
1. Concentric circle, 2. Chuck body, 3. Jaw screw, 4. Jaw, 5. Gripping surface, 6. Recess for back plate.

1. Four jaw independent chuck
2. Three jaw universal chuck
3. Air or hydraulic operated chuck
4. Magnetic chuck
5. **Collet chuck**
6. **Combination chuck**
7. **Drill chuck**

Four jaw independent chuck : A four jaw independent chuck is illustrated in Fig.3.24. This chuck has four jaws which may be made to slide within the slots provided in the body of the chuck for gripping different sizes of work piece. Each jaw may be moved independently by rotating the screw which meshes with the teeth cut on the underside of the jaw. Each jaw made of tough steel has three inner and one outer gripping surfaces. The outer gripping surface is used for holding larger sizes of workpiece by reversing the jaw. Concentric circles inscribed on the face of the chuck facilitate quick centering of the workpiece. This type of chuck is particularly used in the setting up of heavy and irregular shaped articles. The diameter of the body specifies the size of the chuck.

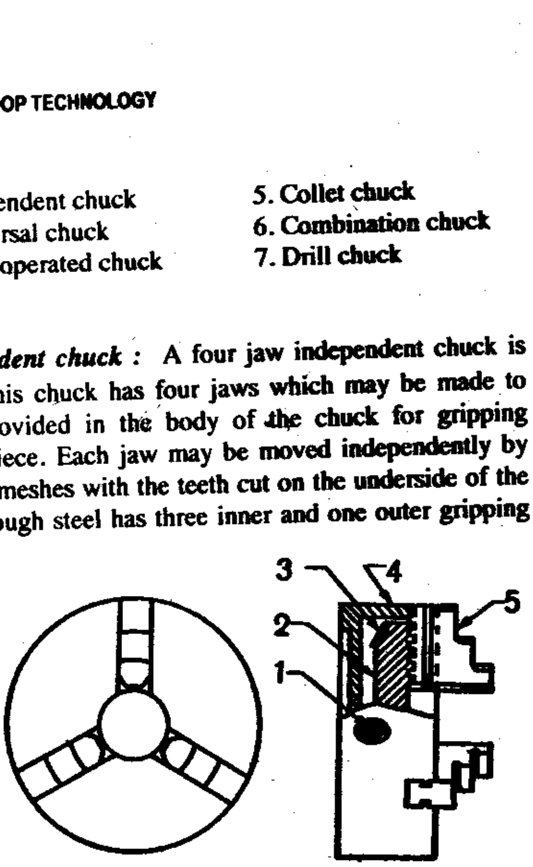


Figure 3.25 Universal chuck
1. Bevel pinion, 2. Scroll disc, 3. Bevel teeth on scroll disc, 4. Chuck body, 5. Jaw.

Universal or self centering chuck : In a three jaw universal chuck illustrated in Fig.3.25 all the jaws may be made to slide simultaneously by an equal amount within the slots provided on the body by rotating any one of the three pinions which meshes with the teeth cut on the underside of the scroll disc. The scroll disc having a spiral groove cut on the top face meshes with the teeth on the jaws. When the disc is made to rotate by any one of

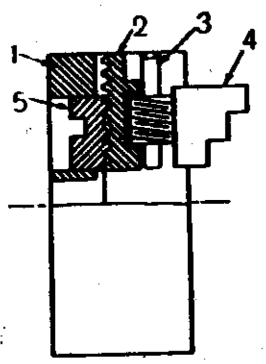


Figure 3.26 Combination chuck
1. Chuck body, 2. Frame, 3. Screw, 4. Jaw, 5. Scroll.

the pinions, all the three jaws move backward or forward by equal amount. The chuck is suitable for holding round, or hexagonal, and other similar shaped workpiece and the job is centered automatically and quickly. But it has less gripping capacity as only three jaws are used and centering accuracy is soon lost due to wear.

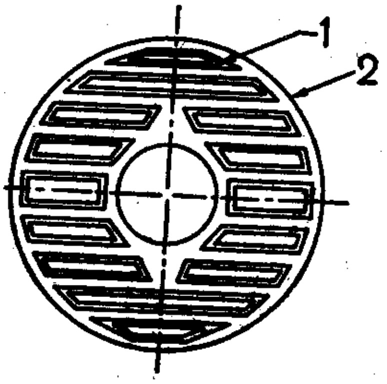


Figure 3.27 Magnetic chuck
1. Magnets, 2, Chuck body.

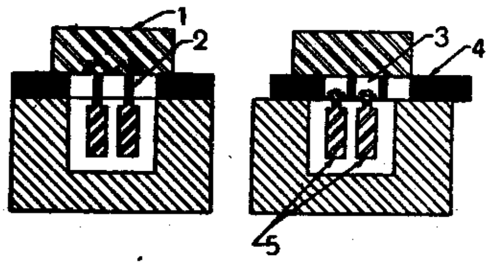


Figure 3.28 Principle of magnetic chuck
1. Work, 2. Non magnetic material, 3. Keepers, 4. Face plates, 5. Magnet.

Combination chuck : As the name implies, a combination chuck, shown in Fig.3.26, may be used both as a self centering and an independent chuck to take advantage of both the types. The jaws may be operated individually by separate screws or simultaneously by the scroll disc. The screws mounted on the frame have teeth cut on its underside which meshes with the scroll and all the jaws together with the screws move radially when the scroll is made to rotate by a pinion.

Magnetic chuck : The front view of a magnetic chuck is shown in Fig.3.27. The chuck is used for holding a very thin workpiece made of magnetic material which cannot be held in an ordinary chuck. It is also used where any distortion of the work piece due to the pressure of the jaws is undesirable. The working of a magnetic chuck is shown in Fig.3.28. The holding power of the chuck is obtained by the magnetic flux radiating either from the electro-magnets or from the permanent magnets introduced within the chuck. In the *ON* position the flux passes through the workpiece and grips it. In *OFF* position the magnets are set aside bringing them in contact with high permeable-"keepers" which short circuit the flux and prevent them from passing through the workpiece.

Collet chuck : Collet chucks are used for holding bar stock in production work where quick setting and accurate centering is needed. Fig.3.29 illustrates a collet chuck. The chuck attached to the spindle by a nut consists of a thin cylindrical bushing known as *collet* having a slot cut lengthwise on its periphery. The inside bore of the collet may be cylindrical, hexagonal, square, etc. depending on the shape of the work that will pass through it. The outside surface of the collet which is tapered fits in the taper hole on the body of the chuck, and the tail end which is threaded meshes with a key. When the key is turned from outside, the collet is drawn in resulting the split tapered end to be pushed inward due to the springy action and the workpiece is securely and accurately held in the chuck. Different sizes of collets are used for holding different sizes of the bar stock.

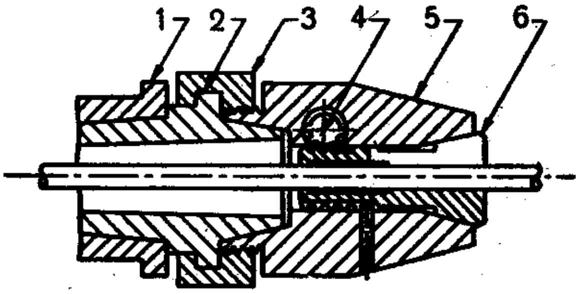


Figure 3.29 Collet chuck
1. Bearing, 2. Spindle, 3. Nut, 4. Key, 5. Chuck, 6. Collet.

Air or hydraulic operated chuck : This type of chuck shown in Fig.3.30 is used in mass production work for its fast and effective gripping

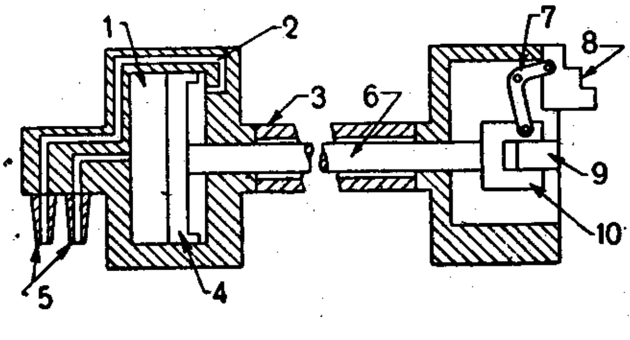


Figure 3.30 Air operated chuck
1. Cylinder, 2. Air passage, 3. Headstock spindle, 4. Piston, 5. Valves, 6. Piston rod, 7. Link, 8. Jaw, 9. Guide, 10. Sliding unit.

capacity. The mechanism incorporates a hydraulic or air cylinder mounted at the back end of the headstock spindle and rotates with it. Fluid pressure may be communicated to the cylinder by operating a valve with a lever and the piston will slide within the cylinder. The movement of the piston is transmitted to the jaws by a connecting rod and links and the jaws grip the workpiece securely.

Drill chuck : A drill chuck is sometimes used in a lathe for holding straight shank drill, reamer or tap for drilling, reaming or tapping operations. The chuck may be held either in headstock or tailstock spindle. It has self-centering jaws which may be operated by rotating a key. A drill chuck is explained in Art 5.7.

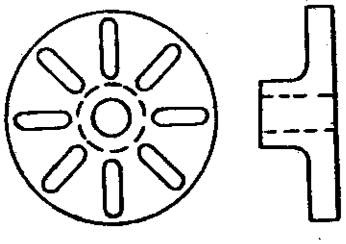


Figure 3.31 Faceplate

Face plates : A face plate consists of a circular disc bored out and threaded to fit the nose of the lathe spindle. This has the radial, plain and 'T' slots for holding work by bolts and clamps. Face plates are used for holding workpieces which cannot be conveniently held between centres or by chucks. This is shown in Fig.3.31

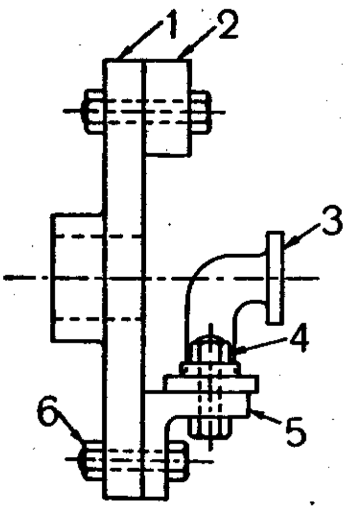


Figure 3.32 Angle plate
1. Faceplate, 2. Balance weight, 3. Elbow-pipe, 4. Clamping nut for elbow pipe, 5. Angle plate, 6. Clamping nut for angle plate.

Angle plates : This is a cast iron plate having two faces machined to make them absolutely at right angles to each other. Holes and slots are provided on both faces so that it may be clamped on a faceplate and can hold the workpiece on the other face by bolts and clamps. Angle plates are used in conjunction with a face plate when the holding surface of the workpiece should be kept horizontal, as for example, in machining a flange of a pipe elbow. When eccentric jobs are bolted to the face plate, a balance weight or counter weight must be added. Fig.3.32 illustrates the use of an angle plate.

Mandrels : A mandrel is a device for holding and rotating a hollow piece of work that has been previously drilled or bored. The work revolves with the mandrel which is mounted between two centres. The mandrel should be true with accurate centre holes for machining outer surface of the workpiece concentric with its bore. To avoid distortion and wear it is made of high carbon steel. The ends of a mandrel are slightly smaller in diameter and flattened to provide effective gripping surface of the lathe dog set screw. The mandrel is rotated by the lathe dog and the catch plate and it drives the work by friction. To secure the mandrel in the work, it is driven by a copper or lead hammer or pressed by a special mandrel press. Different types of mandrels are employed according to specific requirements.

Plain mandrel : The plain mandrel is illustrated in Fig.3.33. This type of mandrel is most commonly used in shops and finds wide application where a large number of identical pieces having standard size holes are required to be mounted on it. The body of the mandrel is slightly tapered, the difference in diameter being 1 to 2 mm per 100 mm length. The length varies from 55 to 430 mm. The taper is provided for proper gripping of the workpiece. This type of mandrel is suitable for only one size of bore. For different sizes of holes in workpieces different mandrels are used.

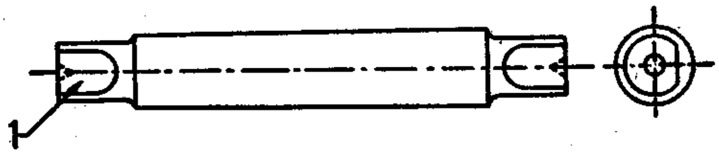


Figure 3.33 Plain mandrel
1. Flattened end

Step mandrel : A step mandrel having steps of different diameters may be employed to drive different workpieces having different sizes of holes without replacing the mandrel each time. This type of mandrel is suitable for turning collars, washers and odd sized jobs used in repairing workshops. A step mandrel is illustrated in Fig.3.34.

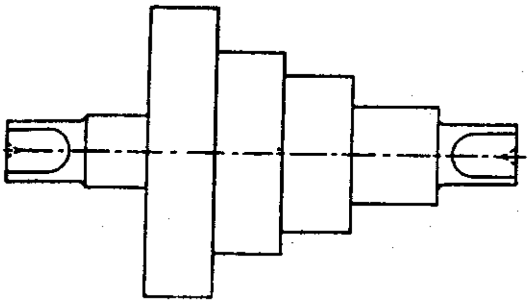


Figure 3.34 Step mandrel

Collar mandrel : A collar mandrel having solid collars is used for turning workpieces having holes of larger diameter, usually above 100 mm. This construction reduces weight and fits better than a solid mandrel of equal size. A collar mandrel is illustrated in Fig.3.35.

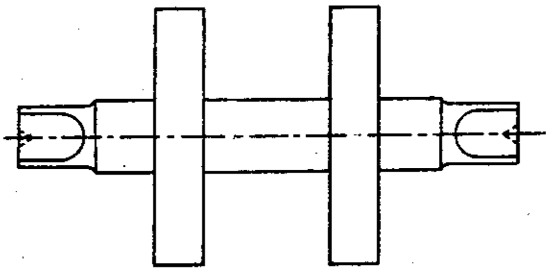


Figure 3.35 Collar mandrel

Screwed mandrel : A screwed mandrel illustrated in Fig.3.36 is threaded at one end with a collar. Workpieces having internal threads are screwed on to it against the collar for machining. The size and type of the thread used on the mandrel depends on the internal thread of the workpiece. It may be right or left handed, square, 'V' or any other type. External surface of screwed flanges, holding them on the screwed mandrel.

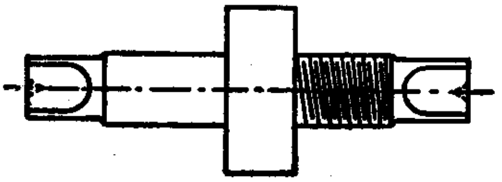


Figure 3.36 Screwed mandrel

Cone mandrel : A cone mandrel illustrated in Fig.3.37 consists of a solid attached to the one end of the body, and a sliding cone which can be adjusted by turning a nut at a threaded end. This type of mandrel is suitable for holding workpieces having different hole diameters by a placing the workpiece on two cones and tightening the nut. Forcing the cone too much tightly upon the workpiece may spoil its outer edge.

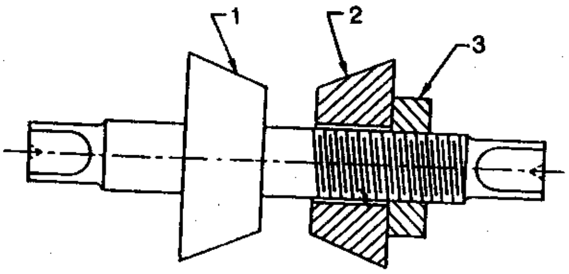


Figure 3.37 Cone mandrel
1. Solid cone, 2. Sliding cone, 3. Nut.

Gang mandrel : This has a fixed collar at one end and a movable collar at the threaded end which may be adjusted to this position by a nut. The mandrel is used to hole a set of hollow workpieces between two collars by tightening the nut. The friction between the sides of the work and the collar is sufficient to drive the work without slipping in the mandrel. A gang mandrel is illustrated in Fig.3.38.

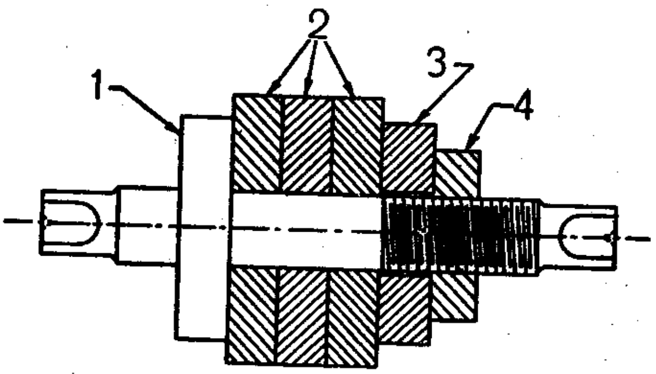


Figure 3.38 Gang mandrel
1. Fixed collar, 2. Hollow workpieces, 3. Movable collar, 4. Nut.

Expansion mandrel : There are different types of expansion mandrels. The mandrel as shown in Fig.3.39 consists of a tapered pin which is driven into a sleeve that is parallel outside and tapered inside. The sleeve has three longitudinal slots, two of which are cut nearly through, and the third splits it completely. This construction enables an expansion mandrel to grip various workpieces with different hole diameters within a limit that cannot otherwise be held in an ordinary mandrel. This has proved its use in repairing workshops. To use this mandrel, the sleeve is first placed within the work with the pin removed. The tapered pin is then pressed from the end into the sleeve and the sleeve expands, gripping the work securely and accurately. An expansion mandrel with a particular sleeve can hold workpieces of varying hole diameters ranging from 0.5 to 2 mm. This range am be increased with different sizes of the sleeve.

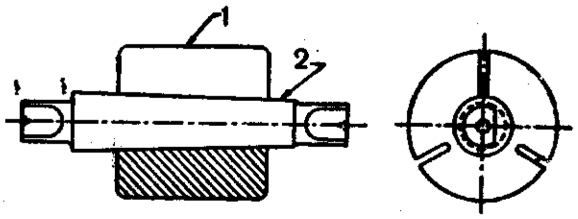


Figure 3.39 Expansion mandrel
1. Sleeve, 2. Tapered pin.

Rests : A rest is a mechanical device which supports a long slender workpiece, which is turned between centres or by a clutch, at some intermediate point to prevent bending of the workpiece due to its own weight and vibrations set up due to the cutting force that acts on it. A rest should always be used when the length is 10 to 12 times the diameter of workpiece. Rests when properly set provide greatest accuracy in machining and permit heavier depth of cut on the workpieces. The two types of rests used in an engine lathe are the steady or centre rest and the follower rest.

Steady rest : A steady rest shown in Fig.3.40 consists of a cast iron base, which may be made to slide on the lathe bedways and clamped at any desired position where a support is necessary. This is so designed that the upper portion is hinged at one end which facilitates

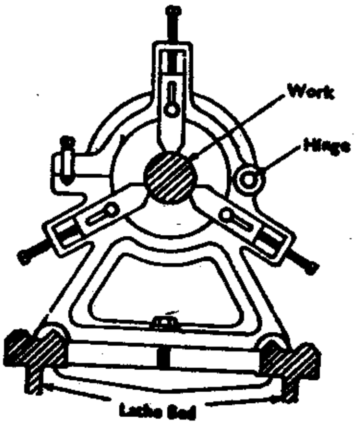


Figure 3.40 Steady rest

setting and removal of the workpiece without disturbing the position of the steady rest. The three jaws on the steady rest, two on the lower base and one on the upper frame, may be adjusted radially by rotating individual screws to accommodate workpieces of different diameters. The jaws which act as a bearing to the workpieces are clamped in position after the setting is properly made. They rest on a *spot* on the workpiece which has been previously turned to provide the true bearing surface. The main function of a steady rest is to provide support to a long slender work. For this purpose one or more steady rests may be used to support the free end of a long workpiece. It is also used to support the free end of a long workpiece for drilling, boring, tapping operations etc. when support from the tailstock end cannot be given. The carriage cannot be fed to the full length of the work when the steady rest is used.

Follower rest : A follower rest shown in Fig.3.41 consists of a "C" like casting having two adjustable jaws which support the workpiece. The rest is bolted to the back end of the carriage and moves with it. Before setting the follower rest, the end of the workpiece is machined slightly wider than the jaws to provide the true bearing surface. The tool is set slightly in advanced position than the jaws, and as the tool is fed longitudinally by the carriage, the jaws always follow the tool giving continuous support to the workpiece. The follower rest prevents the job from springing away when the cut is made and is used in finish turning operations or where the entire length of the workpiece is required to be turned without disturbing the setting.

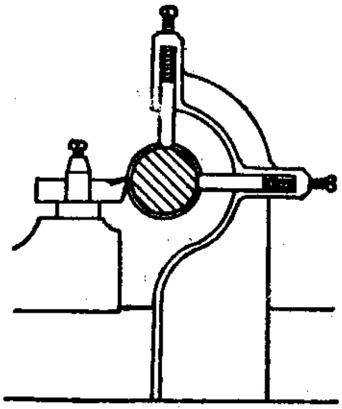


Figure 3.41 Follower rest

3.13 LATHE OPERATIONS

In order to perform different machining operations in a lathe, the workpiece may be supported and driven by any one of the following methods :

1. Held between centres and driven by carriers and catch plates.
2. Held on a mandrel which is supported between centres and driven by carriers and catch plates.

3. Held and driven by chuck with the other end supported on the tailstock centre.
4. Held and driven by a chuck or a faceplate or an angle plate.

The above methods of holding the work may be broadly classified under two headings :

1. Workpiece held between centres
2. Workpiece held by a chuck or any other fixture.

Operations which are performed in a lathe either by holding the workpiece between centres or by a chuck are :

1. Straight turning.
2. Shoulder turning.
3. Chamfering.
4. Thread cutting.
5. Facing.
6. Knurling.
7. Filing.
8. Taper turning.
9. Eccentric turning.
10. Polishing.
11. Grooving.
12. Spinning.
13. Spring winding.
14. Forming.

Operation which are performed by holding the work by a chuck or a faceplate or an angle plate are :

1. Drilling
2. Reaming
3. Boring
4. Counterboring
5. Taperboring
6. Internal thread cutting
7. Tapping
8. Undercutting
9. Parting-off

Operations which are performed by using special attachments are :

1. Grinding
2. Milling

3.14 CENTERING

Where the work is required to be turned between centres or between a chuck and a centre, conical shaped holes must be provided at the ends of the workpiece to provide bearing surface for lathe centres. Centering is the operation of producing conical holes in workpieces. To prepare a cylindrical workpiece for centering, it is first necessary to locate the centre hole by marking off. This is done by rubbing the end with a chalk and the centre may be located by any one of the following instruments : (1) using a centre head and steel rule of a combination set, (2) using a hermaphrodite caliper, (3) using a divider and surface plate, (4) using a surface gauge, and (5) using a bell centre punch. After the centre has been located, a centre punch and a hammer are used to make a deep indentation to produce the hole to hold and to revolve the work on lathe centres. Centre holes are produced by using a combined drill and countersink tool. This is held on a drill chuck and may be mounted on the headstock or on the tailstock spindle to produce a conical hole on the ends of the workpiece. The included angle of the hole should be exactly 60° to fit with the 60° point angle of the lathe centres. The straight hole projected beyond the conical hole serves as a small reservoir for lubricating oil and relieves the tip of the dead centre from rubbing with the workpiece.

3.15 TURNING

Turning in a lathe is to remove excess material from the workpiece to produce a cone-shaped or a cylindrical surface. The various types of turning made in lathe work for various purposes are described below.

Straight turning : The work is turned straight when it is made to rotate about the lathe axis, and the tool is fed parallel to the lathe axis The straight turning produces a cylindrical surface by removing excess metal from the workpiece.

After facing the ends and drilling the centre, the job is carefully mounted between the centres using a lathe dog attached to the workpiece, the bent tail of the dog fitting into the slot provided on the catch plate. If the workpiece is mounted on a chuck or a face plate, care should be taken to centre it accurately with the lathe axis. The trueness of the workpiece held on a chuck is tested by holding a chalk or a scriber or a dial indicator against the rotating workpiece.

A properly ground right hand turning tool selected for the purpose is clamped on the tool post with the minimum overhang and is set with its cutting edge approximately at the lathe axis or slightly above it. For light cuts the tool may be inclined towards the headstock, but for heavy cuts the tool must be inclined towards the tailstock to swing it clear out of the work, if there is any slip. The machine is started after the workpiece and the tool is properly set and the correct spindle speed and the amount of feed to be given is determined. The automatic feed is engaged to move the carriage to the desired length, then the feed is disengaged and the carriage is brought back to the starting position. The process is repeated until the job is finally finished after two or three similar cuts. There are two kinds of cuts in a machine shop work :

1. Roughing cut or rough turning.
2. Finishing cut or finish turning.

Rough turning : The rough turning is the process of removal of excess material from the workpiece in a minimum time by applying high rate of feed and heavy depth of cut. The roughing cut should be so made that the machine, the tool, and the workpiece can bear the load and it does not make too rough a surface and spoil the centres. The depth of cut for roughing operations in average machine shop work is form 2 to 5 mm and the rate of feed is from 0.3 to 1.5 mm per revolution of the work. In rough turning operations shown in Fig.3.42 a *rough turning* tool is used.

Finish turning : The finish turning operation requires high cutting speed, small feed, and a very small depth of cut to generate a smooth surface. A finish turning tool having sharp cutting edge is held securely on the tool post for this purpose. In finish turning operation shown in Fig.3.43 the depth of cut ranges from 0.5 to 1 mm and feed from 0.1 to 0.3 mm per revolution of the workpiece. The cross feed micrometer dial is used to set an accurate depth of cut. After measuring the diameter of rough turned surface, the depth of cut to be given is determined by subtracting the finished diameter from the measured value. The tool is then made to advance by half the above value by rotating the cross slide hand-wheel through required number of divisions on the dial. The machine is started and a trial cut is made from the end of the work to 5 or 6 mm by applying hand feed and the finished diameter is checked by a micrometer. Once the correct setting is made, the rest is finished by the automatic feed. Copious supply of coolant and lubricant should be used to produce a smooth surface.

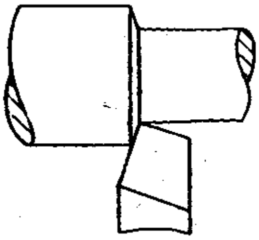


Figure 3.42 Rough turning operation

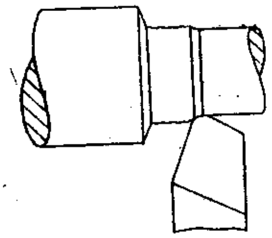


Figure 3.43 Finish turning operation

Shoulder turning : When a workpiece having different diameters is turned, the surface forming the step from one diameter to the other is called the shoulder, and machining this part of the workpiece is called shoulder turning. There are four kinds of the shoulder : (1). square shoulder, (2). angular or beveled shoulder, (3). radius shoulder, and (4). under cut shoulder, Fig.3.44 shows different types of shoulders. The location of the shoulder is first marked on the workpiece by a hermaphrodite caliper, measuring from one end by a steel rule. The first diameter is then turned to the finished size within 0.5 to 1 mm of the shoulder mark. For square or beveled shoulder a right cut facing tool is used to finish upto the shoulder mark. A round nose tool will produce a radius shoulder. A undercut shoulder may be machined by using a parting tool, being fed into the work below the first diameter surface to the desired depth.

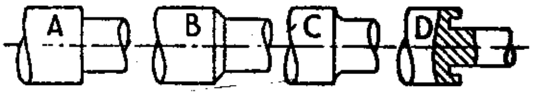


Figure 3.44 Types of shoulders
A. Square shoulder, B. Beveled shoulder, C. Radius shoulder, d. Undercut shoulder

3.16 TAPERS AND TAPER TURNING

A taper may be defined as a uniform increase or decrease in diameter of a piece of work measured along its length. In a lathe, *taper turning* means to produce a conical surface by gradual reduction in diameter from a cylindrical workpiece. This tapering of a part has wide applications in the construction of machines. Almost all machine spindles have taper holes which receive taper shank of various tools and work holding devices.

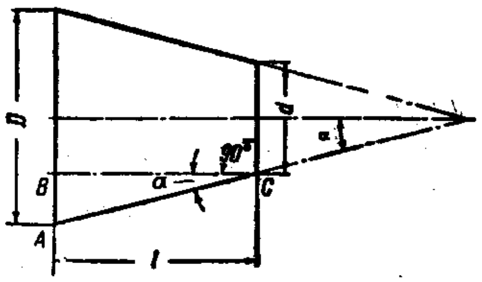


Figure 3.45 Taper elements
D. large diameter of taper, d. Small diameter of taper, l. Length of taper, a. Half angle of taper

Taper elements : A tapered piece shown in Fig.3.45 may be designated by the following symbols :

D = large diameter of taper in mm.
d = small diameter of taper in mm.
l = length of tapered part in mm.
2α = full taper angle.
α = angle of taper or half taper angle.

The amount of taper in a workpiece is usually specified by the ratio of the difference in diameters of the taper to its length. This is termed as the *conicity* and its designated by the letter K.

$$K = \frac{D-d}{l} \qquad 3.2$$

Example 3.2 : in fig. 3.45 lot D=90 mm, d=80 mm, and l=100 mm, find the value of K.

$$K = \frac{D-d}{l}$$

$$= \frac{90-80}{100} = \frac{1}{10}$$

This, 1/10 means that the amount of taper is 1:10, or in a length of 10 mm, the diameter of the taper is reduced by 1 mm. The amount of taper 1/10 may also be expressed as a decimal, i.e. 0.1.

The equation (3.2) may be rearranged in various ways to calculate any one of the unknown quantities.

$$l = \frac{D-d}{K} \qquad 3.3$$

If the large diameter, the small diameter and the conicity are known the length of the taper can be calculated.

Example 3.3 : In Fig. 3.45 let D=80 mm, d=70 mm, and the conicity or the amount of taper is 1/20, find the length of the taper.

$$l = \frac{D-d}{K} = \frac{80-70}{\frac{1}{20}} = 200 \text{ mm}$$

If the conicity, the length of taper, and the small diameter are known, the large diameter can be calculated from the equation :

$$D = Kl + d \qquad 3.4$$

Example 3.4 : In Fig.34.5 let the conicity is 1/30, the length of taper 300 mm the small diameter 80 mm, find the large diameter.

$$D = Kl + d$$

$$= \frac{1}{30} \times 300 + 80 = \mathbf{90\ mm.}$$

If conicity, the length of taper and the large diameter are known, the small diameter can be calculated from the equation :

$$d = D - Kl \qquad 3.5$$

Example 3.5 : In Fig.3.45 let the conicity is 1/50, the length of taper 250 mm, and the large diameter 55 mm, find the small diameter.

$$d = D - Kl$$

$$= 55 - \frac{1}{50} \times 250 = \mathbf{50\ mm.}$$

Referring to Fig.3.45, BC is drawn parallel to the axis and in the right angle triangle ABC.

$$AB = \frac{D-d}{2}, \qquad BC = l$$

Hence, $$\tan\alpha = \frac{D-d}{2l}, \qquad 3.6$$

The tangent of the half taper angle can be determined if the two diameters and the length of the taper are known.

From the equation (3.2)

$$K = \frac{D-d}{l},$$

$$\text{Therefore, } \tan\alpha = \frac{D-d}{2l} = \frac{K}{2}$$

$$\text{or} \quad K = 2\tan\alpha \qquad 3.7$$

The taper of a workpiece is sometimes expressed by the angle α the half taper angle or the angle of taper.

Example 3.6 : In Fig. 3.45 let $D=90$ mm, $d=80$ mm and $l=100$ mm. Find the angle of taper.

$$\text{Tan}\,\alpha = \frac{D-d}{2l} = \frac{90-80}{100\times 2} = \frac{10}{200} = \frac{1}{20}$$

$$\alpha = 1°25'56''$$

and the full taper angle, $\mathbf{2\alpha = 2°\,51'\,52''}$.

Taper in the British System is expressed in taper per foot or taper per inch. If D is the diameter of the large end, d the diameter of the small end, and l the length of the taper, all expressed in inches, then

$$\text{taper per inch} = \frac{D-d}{l} \qquad 3.8$$

When the taper is expressed in taper per foot, the length of the taper l is expressed in foot, but the diameters are expressed in inches.

Examples 3.7 : Find the taper per inch and the taper per foot, if the diameters of the taper are $\frac{5}{8}$ inch and $\frac{3}{8}$ inch ; and the length of the taper is $3\frac{1}{4}$ inch.

$$\text{Taper per inch} = \frac{\frac{5}{8}-\frac{3}{8}}{3\frac{1}{4}} = \frac{2}{8}\times\frac{4}{13} = \frac{1}{13}\ \textbf{in.}$$

$$\text{Taper per foot} = \frac{\frac{5}{8}-\frac{3}{8}}{\frac{3\frac{1}{4}}{12}} = \frac{2}{8}\times\frac{12}{13} = \frac{12}{13} = \mathbf{0.923\ in.}$$

Standard Tapers : Machine parts and tools having inside or outside taper are standardized to facilitate interchangeability of parts. Tapered surfaces which follow standard dimensions are called standard tapers. Standard tapers adapted by the Indian Standard Institution for various tools and machine parts like drills, reamers, milling cutter shanks, arbors, lathe centres, etc. are Morse tapers. *Morse tapers* are available in seven sizes numbered: 0, 1, 2, 3, 4, 5, and 6. The amount of taper varies from number to number. The No. 0 (zero) Morse taper is the smallest while No. 6 is the largest in size.

The non-uniformity of the angle of the taper for different Morse taper sizes is its greatest disadvantage.

Metric tapers are sometimes used as standard tapers. Metric tapers are made in seven sizes and designated by the number 4, 6, 80, 100, 120, 160 and 200. The taper number stands for the large diameter of the taper in mm. The advantage is that all metric tapers have the same angle of taper.

The amount of taper and taper angle for standard tapers are given in table 3.1.

TABLE 3.1 STANDARD TAPERS

Standard tapers	*Amount of taper or conicity*	*Half taper angle*	*Full taper angle*
Morse No. 0	1 : 19.212	1°29′27″	2°58′54″
Morse No. 1	1 : 20.047	1°25′43″	2°51′26″
Morse No. 2	1 : 20.020	1°25′50″	2°51′41″
Morse No. 3	1 : 19.922	1°26′16″	2°52′32″
Morse No. 4	1 : 19.254	1°29′15″	2°58′31″
Morse No. 5	1 : 19.002	1°30′26″	3°00′53″
Morse No. 6	1 : 19.180	1°29′36″	2°59′12″
Metric tapers : Nos. 4, 6, 80, 100, 120, 160, 200	1 : 20	1°25′26″	2°51′51″

In the British system, in addition to Morse standard tapers there are two other important standard tapers :

1. Brown and Sharpe standard taper.
2. Jarno standard taper.

The Brown and Sharpe standard taper is used mostly on milling machines. There are 18 sizes in the series, numbered from 1 to 18. In this standard, the taper is 0.500 in per ft in all of its numbers except the No. 10 which has a taper of 0.5161 in per ft.

The Jarno system of taper is the most sensible system. This was originally designed for use in lathes. There are 20 sizes in the series, each one being identified by a number ranging from 1 to 20, and the taper is 0.60 in per ft in each size.

Taper turning methods : A taper may be turned in a lathe by feeding the tool at an angle to the axis of rotation of the workpiece. The angle formed by the path of the tool with the axis of the workpiece should correspond to the half taper angle.

While turning taper, it is essential that the tool cutting edge should be set accurately on the centre line of the workpiece, otherwise correct taper will not be obtained. A taper may be turned by any one of the following methods :

1. By a broad nose form tool.
2. By setting over the tailstock centre.
3. By swivelling the compound rest.
4. By a taper turning attachment.
5. By combining longitudinal and cross feed in a special lathe.

Taper Turning by a form tool : Fig.3.46 illustrates the method of turning taper by a form tool. A broad nose tool having straight cutting edge is set on to the work at half taper angle, and is fed straight into the work to generate a tapered surface. The half angle of taper will correspond to 90 minus side cutting edge angle of the tool. In this method the tool angle should be properly checked before use. This method is limited to turn short length of taper only. This is due to the reason that the metal is removed by the entire cutting edge, and any increase in the length of the taper will necessitate the use of a wider cutting edge. This will require excessive cutting pressure, which may distort the work due to vibration and spoil the work surface.

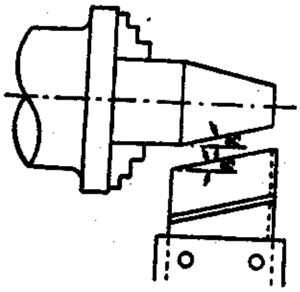


Figure 3.46 Taper turning by a form tool

α. Half angle of taper.

Taper turning by setting over the tailstock : The principle of turning taper by this method is to shift the axis of rotation of the workpiece, at an angle to the lathe axis, and feeding the tool parallel to the lathe axis. The angle at which the axis of rotation of the workpiece is shifted is equal to half angle of the taper. This is done when the body of the tailstock is made to slide on its base towards or away from the operator by a setover screw as illustrated in Fig.3.9. The amount of setover being limited, this method is suitable for turning small taper on long jobs. The main disadvantage of this method is that the live and dead centres are not equally stressed and the wear is not uniform. Moreover, the lathe carrier being set at an angle, the angular velocity of the work is not constant.

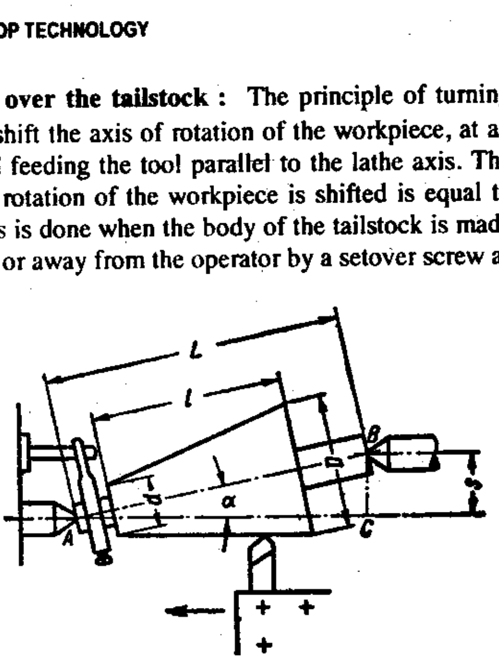


Figure 3.47 Taper angle by setover method
D. Large diameter of taper, d. Small diameter of taper, L. Length of the work, l. Length of the taper, α. Half taper angle, S. Setover.

The *amount* of setover required to machine a particular taper may be calculated as :

From the right angle triangle ABC in Fig.3.47

$BC = AB \sin\alpha$, where BC = setover

or setover $= L \sin\alpha$

If the angle α, the angle of taper, is very small, for all practical purposes, $\sin\alpha \approx \tan\alpha$

$$\text{or} \quad \text{setover} = L\tan\alpha = L \times \frac{D-d}{2l} \text{ in mm} \qquad 3.9$$

If the taper is turned on the entire length of the workpiece, then $l = L$, and the equation (3.9) becomes :

$$\text{Setover} = L \times \frac{D-d}{2L} = \frac{D-d}{2} \qquad 3.10$$

$\frac{D-d}{l}$ being termed as the conicity or amount of taper, the formula (3.9) may be written in the following form :

$$\text{Setover} = \frac{\text{entire length of the work} \times \text{conicity}}{2} \qquad 3.11$$

Example 3.8 : The length of a work is 200 mm, the amount of taper is 1 : 50. find the setover required.

Using equation (3.11)

$$\text{Setover} = \frac{200 \times \frac{1}{50}}{2} = \mathbf{2\ mm.}$$

Example 3.9 : Determine the amount of setover required to turn a taper on the entire length of a workpiece having diameter of the large end 30 mm and diameter of the small end 20 mm.

Using equation (3.10),

$$\text{Setover} = \frac{30-20}{2} = \mathbf{5\ mm.}$$

Example 3.10 : A shaft 1200 mm long has a taper of 1 : 200 for a length of 600 mm. The maximum diameter of the shaft is 75 mm. Determine the minimum diameter of the shaft and the amount setover.

Length of the taper = 600 mm

Using equation (3.2),

$$\frac{1}{200} = \frac{75-d}{600} \quad \text{or } d = 72 \text{ mm.}$$

$$\text{Setover} = L \times \frac{D-d}{2l} = 1200 \times \frac{75-72}{2 \times 600} = \mathbf{3\ mm.}$$

Example 3.11 : A shaft 900 mm long is to be turned taper for a length of 225 mm. The amount of taper is 1:100. Determine the setover required.

Using equation (3.11), $$\text{Setover} = \frac{900 \times \frac{1}{100}}{2} = \mathbf{4.5\ mm.}$$

in British system, $$\text{Setover} = L \times \frac{D-d}{2l}$$

But $\frac{D-d}{l}$ is taper per inch.

$$\text{Therefore, setover} = \frac{\text{entire length of the work in inch} \times \text{taper per inch}}{2}$$

Example 3.12 : A shaft 10 inch long has a taper of $\frac{3}{4}$ inch per foot for a distance of 4 inch. the maximum diameter of the shaft is $2\frac{7}{8}$ inch. How much must the dead centre be set out of line ?

$$\text{Taper per foot} = \tfrac{3}{4} \text{ inch}$$
$$\text{Taper per inch} = \frac{3}{4 \times 12} = \frac{3}{48} \text{ inch}$$

$$\text{Setover} = \frac{\text{entire length of the work in inch} \times \text{taper per inch}}{2}$$

$$= \frac{10 \times \frac{3}{48}}{2} = \frac{10 \times 3}{48 \times 2} = \frac{5}{16} \text{ inch}$$

Small diameter of the work can be determined from the formula :

$$\text{Taper per inch} = \frac{D-d}{l}$$

$$\therefore\ \frac{3}{48} = \frac{\frac{23}{8} - d}{4} \quad \text{and} \quad \therefore\ d = 2\frac{5}{8} \text{ in.}$$

Measuring the tailstock offset : Once the amount of setover required for taper turning is found out, the body of the tailstock is made to slide by the same amount accurately using a scale attached to the base of the tailstock. Where the tailstock is not equipped with the scale a double sided steel rule may be placed between two centres and the amount of offset is measured. But this is not a very accurate method.

The amount of the offset required may be more accurately set by allowing the tool post to touch the tailstock barrel in the normal and in the offset position. This is done by turning the crosslide screw when the offset is measured directly by the difference of readings on the micrometer dial. The dial indicator used in conjunction with the crosslide screw gives a more accurate reading.

Slip gauges are sometimes used for accurate setting of the tailstock.

Taper turning by swivelling the compound rest : This method employs the principle of turning taper by rotating the workpiece on the lathe axis and feeding the tool at an angle to the axis of rotation of the workpiece. The tool mounted on the compound rest is attached to a circular base, graduated in degree, which may be swivelled and clamped at any desired angle. This is illustrated in Fig.3.48. Once the compound rest is set at the desired half taper angle, rotation of the compound slide screw will cause the tool to be fed at that angle and generate a corresponding taper. This method is limited to turn a short taper owing to the limited movement of the compound rest. But a small taper may also be turned. The compound rest may be swivelled at 45° on either side of the lathe axis enabling it to turn a steep taper. The movement of the tool in this method being purely controlled by hand, this gives a low production capacity and poorer surface finish.

The *setting of the compound rest* is done by swivelling the rest at the half taper angle, if this is already known. If the diameter of the small and large end and length of taper are known, the half taper angle can be calculated from the equation (3.6).

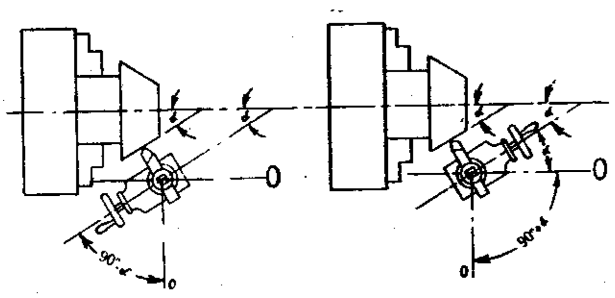


Figure 3.48 Taper turning by swivelling the compound rest.

α. Half angle of taper.

Example 3.13 : Determine the angle at which the compound rest will be switched when cutting a taper on a piece of work having the following dimensions : (i) outside diameter of the rod 60 mm, (ii) length of the tapered portion 80 mm, and (iii) smallest diameter on the tapered end of the rod 20 mm.

$$\tan\alpha = \frac{D-d}{2l} = \frac{60-20}{80\times 2} = \frac{1}{4} = 0.25$$

$$\text{or } \alpha = \mathbf{14^\circ 2'}$$

The rest should be switched at an angle of 14°2′.

Example 3.14 : The spindle end of a milling machine arbor has a taper of 7 : 24. Determine the setting of the compound rest.

$$\tan\alpha = \frac{K}{2} = \frac{7}{24\times 2} = \frac{7}{48} = 0.1458$$

$$\text{or, } \quad \alpha = \mathbf{8^\circ 18'}$$

The rest should be swivelled at an angle of 8°18′.

Taper turning by a taper attachment : The principle of turning taper by a taper attachment is to guide the tool in a straight path set at an angle to the axis of rotation of the workpiece, while the work is being revolved between centres or by a chuck aligned to the lathe axis. A taper turning attachment illustrated in Fig.3.49 consists essentially of a bracket or frame which is attached to the rear end of the lathe bed and supports a guide bar pivoted at the centre. The bar having graduations in degrees may be swivelled on either side of the zero graduation and is set at the desired angle with the lathe axis.

When the taper turning attachment is used, the crossslide is first made free from the lead screw by removing the binder screw. The rear end of the crossslide is then tightened with the guide block by means of a bolt. When the longitudinal feed is engaged, the tool mounted on the crossslide will follow the angular path, as the guide block will slide on the guide bar set at an angle to the lathe axis. The required depth of cut is given by the compound slide which is placed at right angles to the lathe axis. The guide bar must be set at half taper angle and the taper on the work must be converted in degrees. The maximum angle through which the guide bar may be swivelled is 10° to 12° on either side of the centre line.

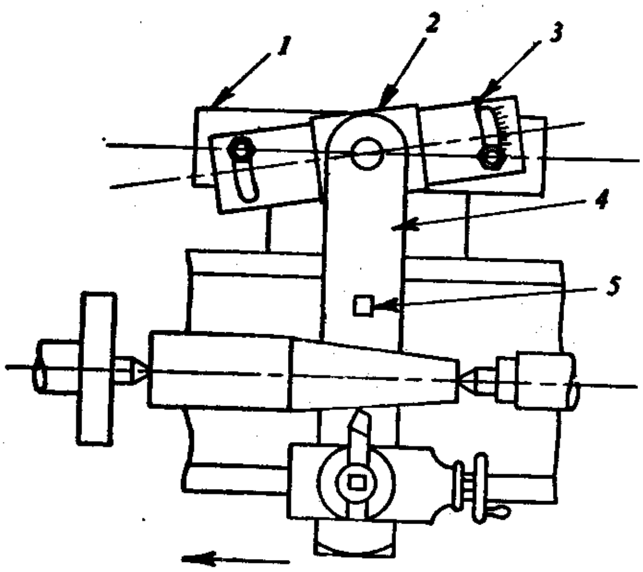


Figure 3.49 Taper turning attachment
1. Bracket or frame, 2. Guide block, 3. Guide bar, 4. Crossslide, 5. Binder Screw

If the diameters D and d and the length l of the taper are specified, the angle of swivelling the guide bar can be determined from equation:

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

The advantages of using a taper turning attachment are:

1. The alignment of live and dead centres being not disturbed, both straight and taper turning may be performed on a work piece in one setting without much loss of time.
2. Once the taper is set, any length of a piece of work may be turned taper within its limit.
3. Very steep taper on a long workpiece may be turned, which cannot be done by any other method.
4. Accurate taper on a large number of workpieces may be turned.
5. Internal tapers can be turned with ease.

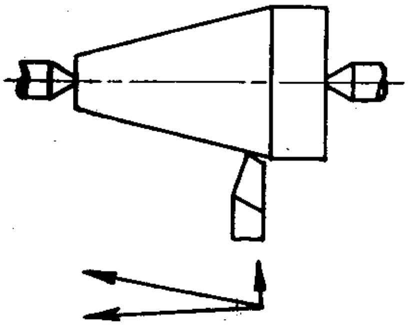


Figure 3.50 Taper turning by combining feed

Taper turning by combining feeds : Taper turning by combining feeds is illustrated in Fig.3.50. This is a more specialized method of turning taper. In certain lathes both longitudinal and cross feeds may be engaged simultaneously causing the tool to follow a diagonal path which is the resultant of the magnitude of the two feeds. The direction of the resultant may be changed by varying the rate of feeds by change gears provided inside the apron.

3.17 ECCENTRIC TURNING

If a cylindrical workpiece has two separate axis of rotation one being out of centre to the other, the workpiece is termed eccentric and turning of different surfaces of the workpiece is known as eccentric turning which is illustrated in Fig.3.51. Crank shaft turning in a lathe is a common example of eccentric turning. In turning a single throw overhang crank shaft, two sets of centre holes are drilled at the ends of the shaft spaced by an amount equal to one half the total throw of the crank. The shaft is first mounted on its true centre and the part forming the journal is turned. The job is then remounted on the offset centre and the eccentric surfaces are machined. In eccentric turning, counterbalance weights are mounted on the face plate to get uniform turning moment. If the throw of the crank is sufficiently high so that the offset centre cannot be accommodated on the face of the workpiece, special flanges are attached to the ends of the workpiece and centre holes are then drilled. For double or multithrow crank, centre holes are drilled to correspond with the different axis.

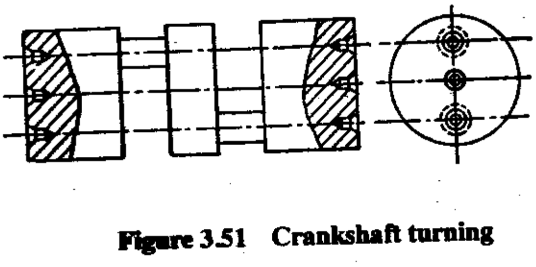


Figure 3.51 Crankshaft turning

3.18 CHAMFERING

Chamfering, illustrated in Fig.3.52, is the operation of bevelling the extreme end of a workpiece This is done to remove the burrs, to protect the end of the workpiece from being damaged and to have a better look. The operation may be performed after knurling, rough turning, boring, drilling or thread cutting. Chamfering is an essential operation after thread cutting so that the nut may pass freely on the threaded workpiece.

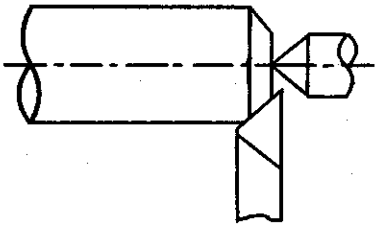


Figure 3.52 Chamfering operation

3.19 THREAD CUTTING

Thread cutting is one of the most important operations performed in a lathe. Different standard forms and proportions of screw threads have been described in Vol. I. under *Tapping and screwing.*

The principle of thread cutting is to produce a helical groove on a cylindrical or conical surface by feeding the tool longitudinally when the job is revolved between centres or by a chuck. The longitudinal feed should be equal to the pitch of the thread to be cut per revolution of the workpiece. The leadscrew of the lathe, through which the saddle receives its traversing motion, has a definite pitch. A definite ratio between the longitudinal feed and rotation of the headstock spindle should therefore be

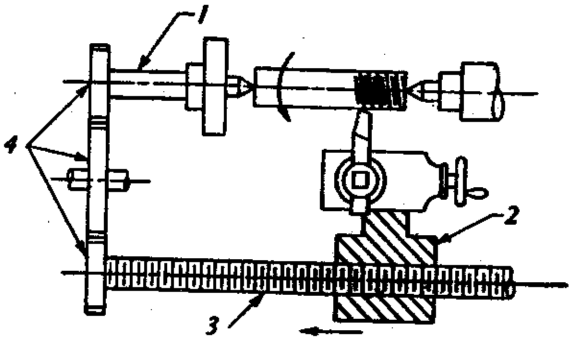


Figure 3.53 Principles of thread cutting
1. Headstock spindle, 2. Carriage, 3. Leadscrew, 4. Change gears

found out so that the relative speeds of rotation of the work and the leadscrew will result in the cutting of a screw of the desired pitch. This is effected by *change gears* arranged between the spindle and the leadscrew or by the *change gear mechanism* or *feed box* used in a modern lathe where it provides a wider range of feed and the speed ratio can be easily and quickly changed. Fig. 3.53 illustrates the principle of thread cutting.

Calculation for change-wheels : To calculate the wheels required for cutting a screw of certain pitch it is necessary to know how the ratio is obtained, and exactly where the driving and driven wheels are to be placed. Suppose the pitch of a lead screw is 12 mm and it is required to cut a screw of 3 mm pitch, then the lathe spindle must rotate 4 times the speed of the leadscrew, that is

$$\frac{\text{spindle turn}}{\text{leadscrew turn}} = \frac{4}{1}$$

But $\dfrac{\text{spindle turn mean}}{\text{leadscrew turn}} = \dfrac{4}{1}$ means that

$\dfrac{\text{driver teeth}}{\text{driven teeth}} = \dfrac{1}{4}$ since a small gear rotates faster than a larger one with which it is connected.

Hence, $$\frac{\text{driver teeth}}{\text{driven teeth}} = \frac{\text{leadscrew turn}}{\text{spindle turn}}$$

$$= \frac{\text{pitch of the screw to be cut}}{\text{pitch of the leadscrew}}$$

In English measurement,

$$\frac{\text{driver teeth}}{\text{driven teeth}} = \frac{\text{thdread per inch on leadscrew}}{\text{thdread per inch on work}}$$

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

The types of gear connections on a lathe may be *simple,* and *compound.* In a simple train, shown in Fig.3.54 the gear on the spindle drives direct through the *intermediate* gear to the gear on the leadscrew. This intermediate gear has no effect on the ratio between the driver and the driven, but merely acts as a connection between the two, and serves to keep the rotation of driver and driven in the same direction. In a compound train, shown in Fig.3.54 the stud carries two wheels which are keyed together so that they rotate as a unit.

The gear on the stud shaft acts as a driver, and in all calculations it is considered as the spindle gear, as usually it runs at the same spindle speed. In modern lathes using quick change gears, the correct gear ratio for cutting a particular thread is quickly obtained by simply shifting the levers in different positions which are given on the charts or instruction plates supplied with the machine.

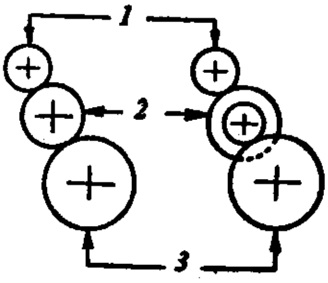


Figure 3.54 Simple and compound gear train
1. Gear on the spindle (Driver). 2. Intermediate gear. 3. Gear on the leadscrew (Driven).

Example 3.15 : The pitch of a leadscrew is 6 mm. and the pitch of the thread to be cut is 1 mm. Find change gears.

$$= \frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the work}}{\text{Pitch of the leadscrew}}$$

$$= \frac{1}{6} = \frac{1 \times 20}{6 \times 20} = \frac{20}{120} \quad \text{or} \quad \frac{\text{Driver teeth}}{\text{Driven teeth}}$$

In Fig. 3.54 the driver will have **20 T** and the driven gear on the lead screw **120 T.**

Example 3.16 : The pitch of a leadscrew is 6 mm. and the pitch of the thread to be cut is 1.25 mm. Find the change wheels.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the work}}{\text{Pitch of the leadscrew}}$$

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{1.25}{6} = \frac{1.25\times 4}{6\times 4} = \frac{5}{24} = \frac{5}{4}\times\frac{1}{6}$$

$$= \frac{5\times 10}{4\times 10}\times\frac{1\times 20}{6\times 20} = \frac{50}{40}\times\frac{20}{120}.$$

In Fig.3.54 the driving gears will have **50** and **20 T** and driven gears **40** and **120 T.**

Metric thread on English leadscrews : The cutting of metric threads on a lathe with an English pitch leadscrew may be carried out by introducing a translating gear of 127 teeth.

If the leadscrew has n threads per in. to cut p mm pitch then,

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

This is derived as follows :

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{pitch of the work } (p)}{\text{pitch of the leadscrew } \left(\frac{1}{n}\times\frac{127}{5}\right)} = \frac{5pn}{127}$$

$$\left(\text{since pitch } = \frac{1}{\text{No. of thread per inch}}\right).$$

The factor $\frac{127}{5}$ comes from the fact that 25.4 mm is equal to 1 in. So it is made whole number by multiplying and dividing by 5 as

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

Example 3.17 : It is required to cut a screw having 7 mm pitch on a lathe having leadscrew of 4 threads per inch. Calculate the gears.

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

or $$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{7\times5\times4}{127} = \frac{140}{127}$$

Since we have no gear of 140 teeth we must compound it thus :

$$\frac{140}{127} = \frac{70\times2}{127} = \frac{70\times2\times20}{127\times20} = \frac{70\times40}{127\times20}$$

Therefore, driver gears will have **70** and **40 T**, and driven gears **127** and **20 T.**

Example 3.18 : Find the gears for cutting screw of lead 1/28 inch on a lathe whose leadscrew has 4 threads per inch.

Lead screw has 4 t.p.i., so pitch = $\frac{1}{4}$ inch.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the work}}{\text{Pitch of the leadscrew}}$$

$$= \frac{\frac{1}{28}}{\frac{1}{4}} = \frac{4}{28} = \frac{2\times2}{4\times7} = \frac{2\times20}{4\times20}\times\frac{2\times10}{7\times10}$$

$$= \frac{40\times20}{80\times70}$$

The driving gears will have **40 & 20 T,** and the driven gears **80 & 70 T.**

Example 3.19 : Show the arrangement of gears for cutting a screw thread of 26 t.p.i. in a lathe with a leadscrew having 4 t.p.i., change wheels available are from 20 to 120 teeth with a progression of 5.

Pitch of the screw to be cut = $\frac{1}{26}$ in.

Pitch of the leadscrew = $\frac{1}{4}$ in.

$$\frac{\textit{Driver teeth}}{\textit{Driven teeth}} = \frac{\frac{1}{26}}{\frac{1}{4}} = \frac{4}{26} = \frac{4\times5}{26\times5} = \frac{4\times5}{13\times10} = \frac{4\times5}{13\times5}\times\frac{5\times10}{10\times10}$$

$$= \frac{20}{65}\times\frac{50}{100}$$

The driving gears will have **20** and **50 T** and the driven gears **65** and **100 T.**

Thread cutting operation : In a thread cutting operation the first step is to remove the excess material from the workpiece to make its diameter equal to the major diameter of the screw thread. Change gears of correct size are then fitted to the end of the bed between the spindle and the leadscrew. The shape or form of the thread depends on the shape of the cutting tool to be used. In a metric thread, the included angle of the cutting edge should be ground exactly 60°. The top of the tool nose should be set at the same height as the centre of the workpiece. A thread tool gauge is usually used against the turned surface to check the cutting tool so that each face of the tool may be equally inclined to the centre line of the workpiece. This is illustrated in Fig. 3.55.

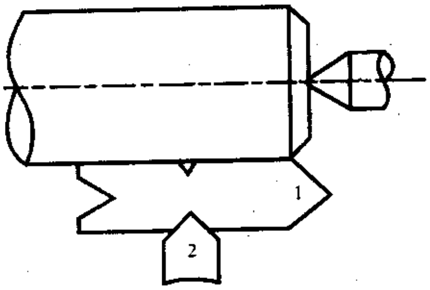


Figure 3.55 Thread tool gauge
1. Thread tool gauge, 2. Thread cutting tool.

The speed of the spindle is reduced by one half to one-fourth of the speed required for turning according to the type of the material being machined, and the half-nut is then engaged. The depth of cut which usually varies from 0.05 to 0.2 mm is applied by advancing the tool perpendicular to the axis of the work or at an angle equal to one-half of the angle of the thread, and 30° in the case of metric thread, by swivelling the compound rest. The different methods of applying depth of cuts are illustrated in Fig. 3.56 (a) & (b). Except when taking very light finishing cuts, the latter method is superior to the former as it

a. Permits the tool to have a top rake;
b. Permits cutting to take place on one edge of the tool only;
c. Allows the chips to slide easily across the face of the tool without crowding;
d. Reduces cutting strain that acts on the tool;
e. Reduces the tendency to cause the tool to "dig-in"

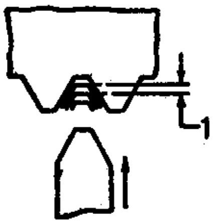


Figure 3.56(a) Applying perpendicular cut

1. Depth of cut

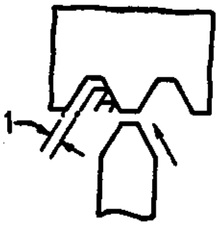


Figure 3.56(b) Applying angular cut

1. Depth of cut.

After the tool has produced a helical groove upto the end of the work this is quickly withdrawn by the use of the crossslide, the half-nut disengaged, and the tool is brought back to the starting position to give a fresh cut. Before re-engaging the half-nut it is necessary to ensure that the tool will follow the same path it has traversed in the previous cut, otherwise the job will be spoiled. Several cuts are necessary before the full depth of thread is reached. Arising from this comes the necessity to "pick-up" the thread. The different methods of picking up a thread are described below:

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

Marking the lathe parts : The general procedure is to mark the leadscrew and its bracket, the large gear and the headstock casting, and the starting position of the carriage on the lathe bed. The aim is to bring each of the markings on the leadscrew and on the gear opposite the markings on the stationary portions of the lathe, and have the carriage at the starting position before attempting to engage the split nut.

Using a chasing dial or thread indicator : The chasing dial illustrated in Fig.3.57 is a special attachment used in modern lathes for accurate "picking up" of the thread. This dial indicates when to close the split or half nuts. This is mounted on the right end of the apron. It consists of a vertical shaft with a worm gear engaged with the leadscrew. The top of the spindle has a revolving dial marked with lines and numbers. The dial turns with the leadscrew so long the half nut is not engaged. If the half nut is closed and the carriage moves along, the dial stands still. As the dial turns, the graduations pass a fixed reference line. The half-nut is closed for all *even* threads when *any line* on the dial coincides with the reference line. For all *odd* threads, the half-nut is closed at ***any numbered line*** on the dial determined from the charts. If the pitch of the thread to be cut is an exact multiple of the pitch of the leadscrew, the thread is called even thread, otherwise the thread is called odd thread.

In a chasing dial, the rule for determining the dial division is: In the case of metric threads, the product of the pitch of lead screw and the number of teeth on the worm wheel must be an exact multiple of the pitch of the threads to be cut. In the case of English threads, the product of the threads per inch to be cut and the number of teeth on the worm wheel must be an exact multiple of the number of threads per inch of the leadscrew. For example, if the pitch of a leadscrew is 6 mm and the worm wheel has 15 teeth, the product will be 90. So any pitch which is exactly divisible by 90, such as 1, 1.25, 1.5, 2, 2.25, 3, 3.75, 4.5, 6, 7.5, 9, 10, 15, 30, 45, 90, may be picked up when any line of the dial coincides with the reference line. For picking up threads of different pitches, a set of worm wheels is used to give the desired value.

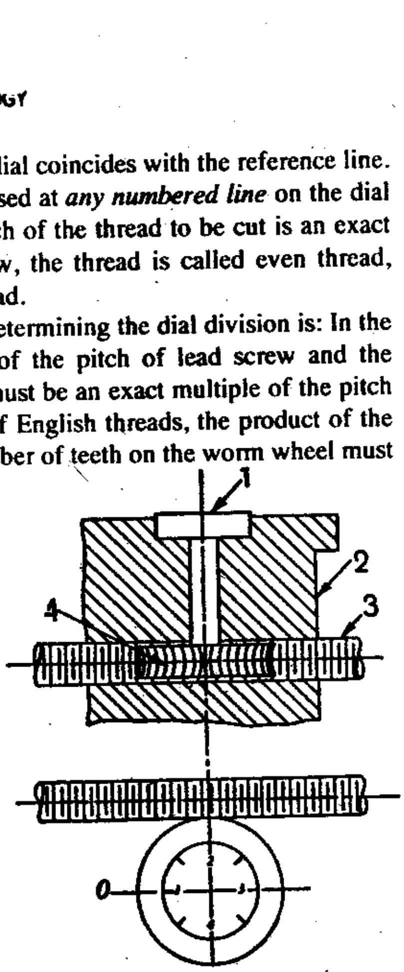


Figure 3.57 Thread chasing dial
1. Dial, 2. Carriage, 3. Leadscrew, 4. Worm gear.

Thread chaser : A chaser is a multipoint threading tool having the same form and pitch of the thread to be chased. An external thread chaser is shown in Fig. 3.58. A chaser is used to finish a partly cut thread to the size and shape required. Thread chasing is done at 1/3 to 1/2 of the speed of turning.

Cutting right-and left-hand thread : When cutting a right-hand thread the carriage must move towards the headstock, for a left hand thread the carriage moves away from the headstock and towards the tailstock. The job moves as always, in the anticlockwise direction when viewed from the tailstock end. As previously mentioned the direction at which the carriage moves in relation to lathe headstock is controlled by means of the tumbler gears or bevel gear feed reversing mechanism.

Cutting multiple threads : In a piece of work it is possible to have several separate and independent threads running along it. Accordingly, there may be single threaded screw and multiple or multi-start threaded screw. The independent threads are called *starts*. For one complete turn round the screw when there is a movement of one thread the screw is called single threaded screw, but when there is a movement of more than one thread the screw is called multiple or multi-start threaded screw. In the case of, say, a three-start thread, for one complete turn the thread advances three times as far as if it was a single thread. The distance a multiple screw thread advances along its axis in one turn is called *lead*.

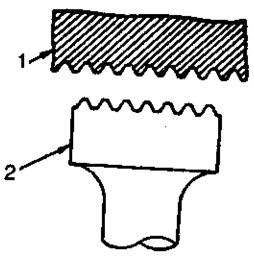


Figure 3.58 Thread chaser
1. Work, 2. Thread chaser.

The calculations for a multi-start thread are identical with those required for a single start thread. The ratio depends upon the relationship between the pitch of the lead screw of the machine, and the lead, but *not the pitch,* of the thread to be cut. This may be written as :

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Lead of the screw to be cut}}{\text{Pitch of the lead screw}}$$

Note: The lead screw of a lathe is always single-started.

Cutting procedure of multi-start threads is similar to that of single start threads. In multi-start threads, circumference of the job should be divided equally into as many parts as there are starts on the thread, and every part or division of the circumference of the job becomes the starting point for the new thread. There are three general methods of arranging the spacing of each start. They are listed as :

1. Marking the gearing, and indexing round after completing each start.
2. Moving the top slide, on which the tool is firmly clamped, the desired distance.
3. Using an index driving plate.

Indexing the gears : As regards the gear train it becomes necessary to arrange the layout so that the first driver is a multiple of the number of starts required; thus for a two start thread the gear teeth must be divisible by two, for a three-start by three, and so on.

Assuming that the gear train is correctly chosen, the driver is divided into the same number of equal teeth as there are starts and marked whilst the first driven wheel is marked at the tooth space which is mating with one of the marked driver teeth. For cutting a two-start thread, the gears are disengaged, those on the quadrant being drawn just clear of the driver on the top stud so that the top driver may, in this instance, be rotated half a revolution, thus permitting the second marked position to mesh with the marked tooth space in the first driven gear. Then the gears are locked in position so that the second start may be machined. This procedure is repeated where there are more than two starts. Fig. 3.59 illustrates marking on change gears.

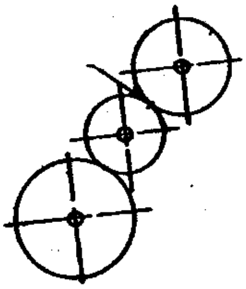


Figure 3.59 Marking on change gears
1. Chalk mark.

Moving the top slide. The top slide may be used for adjusting the tool to have the correct spacing while cutting multi-start threads. After one start of the thread has been cut the top slide is moved a distance equal to the pitch of the thread, whilst the tool is yet clamped in the position used when cutting the previous start. When adopting this procedure the top slide must be parallel to the axis of the workpiece, and hence cuts parallel; after such adjustment the slide should again be firmly clamped in position using a gib. Assuming that each is completed before adjusting the slide, the number of times the top slide is moved is equal to the number of starts less one.

Using an index driving plate : Special index plate may be used where a large number of multiple threads is cut on a lathe. On the plate means are provided to rotate the job through a given fractional part of a revolution.

Example 3.20 : Calculate the change gears to cut a 3-start thread having a pitch of 1.5 mm; the leadscrew has a pitch of 6 mm.

Lead of the work = pitch × no. of starts = $3 \times 1.5 = 4.5$ mm.

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{\text{Pitch of the work}}{\text{Pitch of the leadscrew}}$$

$$\frac{\text{Driver teeth}}{\text{Driven teeth}} = \frac{4.5}{6} = \frac{45}{60}$$

So the driver will have **45 T** and the driven **60 T.**

Cutting tapered thread : The surface is first turned taper to the required angle by any one of the taper turning methods described before. The thread cutting tool is then set perpendicular to the lathe axis and not to the tapered surface. To produce an accurate thread a taper turning attachment is used. This is swivelled to be the half taper angle. The thread is finished in the usual manner.

Checking a screw-cutting set-up : After setting a lathe for screw cutting operation a final examination should cover some or all of the following points:

1. The gear train; this must be correct for the thread to be cut.
2. The tumbler gears must give the carriage the movement in the right direction.
3. The slide must be so adjusted that vibration is avoided.
4. The tool and all portions of the machine should be clear of any rotating mass.
5. The spindle must be arranged to give the low cutting speed required.
6. The feed shaft must be disengaged.
7. The apron feed mechanism should be at neutral.

3.20 FACING

Facing is the operation of machining the ends of a piece of work to produce a flat surface square with the axis. This is also used to cut the work to the required length. The operation involves feeding the tool perpendicular to the axis of rotation of the workpiece. A properly ground facing tool is mounted in a toolholder in the tool post. A regular turning tool may also be used for facing a large workpiece. The cutting edge should be set at the same height as the centre of the workpiece.

A spindle speed is selected to give the proper surface speed at the outer edge of the face, and the lathe is started. The tool is brought in to clean stock from around the centre for the desired depth of cut and then is fed outward, generally by hand. The selection of hand-feed or power-feed depends upon the length of the cut. The surface is finished to the size by giving usual roughing and finishing cuts. For roughing the average value of the cross feed is from 0.3 to 0.7 mm per rev. and the depth of cut is from 2 to 5 mm, for finishing the feed is from 0.1 to 0.3 mm per rev. and the depth of cut is from 0.7 to 1 mm. The facing operation is illustrated in Fig.3.60.

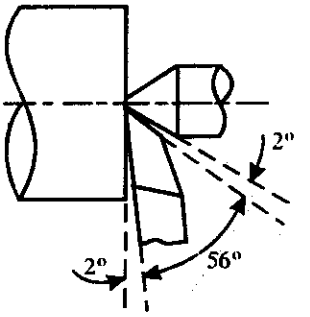


Figure 3.60 Facing operation

3.21 KNURLING

Knurling is the process of embossing a diamond shaped pattern on the surface of a workpiece. The purpose of knurling is to provide an effective gripping surface on a workpiece to prevent it from slipping when operated by hand. In some press fit work knurling is done to increase the diameter of a shaft. The operation is performed by a special knurling tool which consists of 1 set of hardened steel rollers in a holder with the teeth cut on their surface in a definite pattern. The tool is held rigidly on the tool post and the rollers are pressed against the revolving workpiece to squeeze the metal against the multiple cutting edges, producing depressions in a regular pattern on the surface of the workpiece. When a single roller is used to generate parallel grooves, the tool should be set at the centre height and perpendicular to the lathe axis. But when two rollers are used, one right hand and the other left hand, to generate crossed or diamond shaped pattern, the rollers are set at equal distance from the centre. Knurls are available in coarse, medium and fine pitches. Fig.3.61 illustrates a revolving holder with three sets of knurls. Any one set or pair may be brought into operation by revolving the unit. Knurling is done at the slowest speed available in a lathe. Usually the speed is reduced to 1/4th of that of turning, and plenty of oil is flowed on the tool and

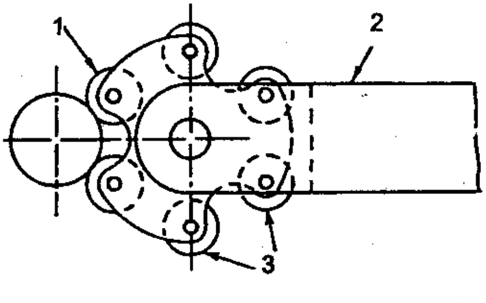


Figure 3.61 Revolving knurl holder
1. First set of Knurl, 2. Knurl holder, 3. Second set of knurl.

workpiece. If the surface to be knurled is wider than the rollers, automatic feed may be engaged. The feed varies from 1 to 2 mm per revolution. Two or three cuts may be necessary to give the full impression. At the end of the first cut, the tools is brought back to the starting position by reversing the machine, leaving the rollers engaged. A fresh cut is given and so on.

3.22 FILING

Filing is the finishing operation performed after turning. This is done in a lathe to remove burrs, sharp corners, and feed marks on a workpiece and also to bring it to the size by removing very small amount of metal. The operation consists of passing a flat single cut file over the workpiece which revolves at high speed. The speed is usually twice that of turning. The file should be slowly moved forward so that the work may pass 2 to 3 revolutions during the cutting stroke. During the return stroke the pressure is relieved but an endwise feeding movement is given, overlapping the previous cut. The file handle is gripped by the left hand and the tip of the file by the right hand to avoid accidents. Overfilling in a lathe damages the trueness of the workpiece.

3.23 POLISHING

Polishing is performed after filing to improve the surface quality of the workpiece. Polishing with successively finer grades of emery cloth after filing results in very smooth, bright surface. The lathe is run at high speeds from 1500 to 1800 m per min, and oil is used on the emery cloth.

3.24 GROOVING

Grooving is the process of reducing the diameter of a workpiece over a very narrow surface. It is often done at the end of a thread or adjacent to a shoulder to leave a small margin. The work is revolved at half the speed of turning and a grooving tool of required shape is fed straight into the work

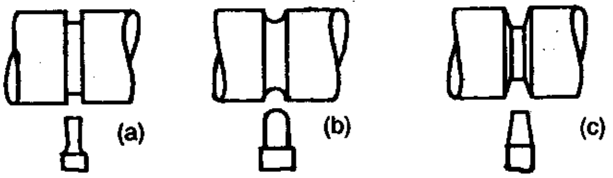


Figure 3.62 Grooving operation
(a). Square groove, (b). Round groove, (c). Bevelled groove.

by rotating the crossslide screw. A grooving tool is similar to a parting-off tool. Fig.3.62. illustrates a grooving operation.

3.25 SPINNING

Spinning is the process of forming a thin sheet of metal by revolving the job at high speed and pressing it against a "former" attached to the headstock spindle. A support is also given from the tailstock end. The pressure is gradually applied to the revolving sheet metal by a long round nose forming tool supported on the special tool rest when the piece slowly acquires the shape of the former. This is illustrated in Fig.3.63

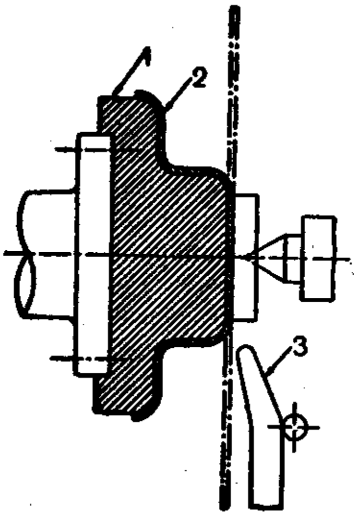


Figure 3.63 Spinning
1. Former, 2. Sheet of metal,
3. Forming tool.

3.26 SPRING WINDING

Spring winding is the process of making a coiled spring by passing a wire around a mandrel which is revolved on a chuck or between centres. A small hole is provided on a steel bar which is supported on the tool post and the wire is allowed to pass through it. The diameter of the mandrel should be less than the desired spring diameter as all springs expand in diameter after they are taken out of the mandrel. In order to wind the spring of the required pitch, the lathe is geared similar to the thread cutting operation.

3.27 FORMING

Forming is the process of turning a convex, concave or of any irregular shape. Form-turning may be accomplished by the following methods:

1. Using a forming tool.
2. Combining cross land longitudinal feed.
3. Tracing or copying a template.

For turning a small length of formed surface, a forming tool having cutting edges conforming to the shape required is fed straight into the work. Forming tools are not supposed to remove much of the material and is used mainly for finishing formed surfaces. Usually two types of forming tools are used -- straight and circular. Straight type is used for wider surfaces and the circular type for narrower surfaces. Fig.3.64 illustrates forming operations performed by straight or circular tools. The cross feed ranges from 0.01 to 0.08 mm per revolution and the cutting speed is slightly less than that of the straight turning.

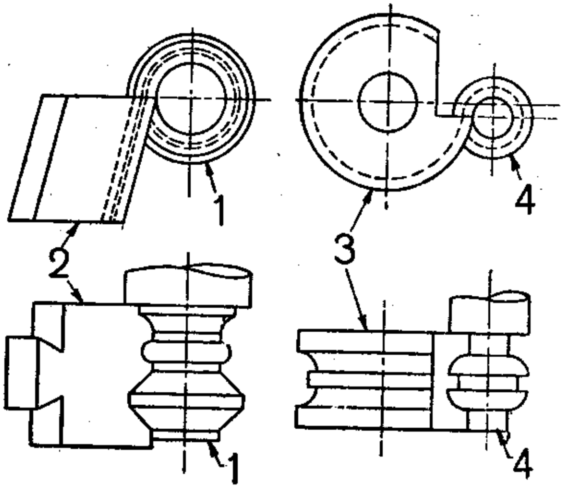


Figure 3.64 Forming operation
1. Work, 2. Straight forming tool, 3. Circular forming tool.

When the length of the formed surface is sufficiently great, the required shape may be obtained by using straight turning tool, which is fed into the work using both longitudinal and crossfeed simultaneously by hand. The process is tedious and requires much skill.

When a large number of wide, formed surfaces are to be turned, a template having the required shape is attached to the rear end of the lathe bed and the crossslide is attached to the guide block after disengaging the crossslide screw. With the longitudinal travel of the carriage, the tool will reproduce the contoured surface of the template as the guide block will trace the curved path.

3.28 DRILLING

Drilling is the operation of producing a cylindrical hole in a workpiece by the rotating cutting edge of a cutter known as the drill. Drilling in a lathe is performed by any one of the following methods:

1. The workpiece is revolved in a chuck or a faceplate and the drill is held in the tailstock drill holder or in a drill chuck. Feeding is effected by the movement of the tailstock spindle. This method is adopted for drilling regular shaped workplaces.
2. The drill is held and driven by a drill chuck attached to the headstock spindle, and the work is held against a pad or crotch supported by the tailstock spindle. Feeding is effected by the movement of the tailstock spindle. Workpieces of very irregular shape which cannot be accommodated on a chuck or faceplate are drilled by this method.

Taper shank drills are mounted on sockets or drill holders and the straight shank drills are fitted to the drill chucks. Speeds and feeds for drilling in a lathe are 25% lower than the corresponding figures for drilling in a drilling machine.

3.29 REAMING

Reaming is the operation of finishing and sizing a hole which has been previously drilled or bored. The tool used is called the reamer, which has multiple cutting edges. The reamer is held on the tailstock spindle, either direct or through a drill chuck and is held stationary while the work is revolved at a very slow speed. The feed varies from 0.5 to 2 mm per revolution.

3.30 BORING

Boring is the operation of enlarging and truing a hole produced by drilling, punching, casting or forging. Boring cannot originate a hole. Boring is similar to the external turning operation and can be performed in a lathe by the following two methods :

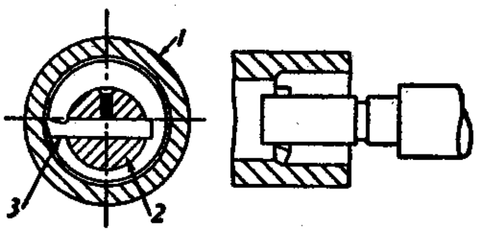


Figure 3.65 Boring operation
1. Work, 2. Boring bar, 3. Boring tool.

1. The work is revolved in a chuck or a face plate and the tool which is fitted to the tool post is fed into the work. This method is adapted for boring small sized works. One piece forged tool is used for boring small hole, whereas a boring bar with a tool bit attached to it is suitable for machining a large hole. The depth of cut is given by the crossslide screw and the feed is effected by the longitudinal travel of the carriage.
2. The work is clamped on the carriage and a boring bar holding the tool is supported between the centres and made to revolve. Longitudinal movement of the carriage provides feeding movement and the depth of cut is given by adjusting the position of the tool 'insert'. Fig.3.65. illustrates boring operation by a boring bar.

Counterboring : Counterboring is the operation of enlarging a hole through a certain distance from one end instead of enlarging the whole-drilled surface. It is similar to a shoulder work in external turning. The operation is similar to boring and a plain boring tool or a counterbore may be used.

Taper boring ; The principle of turning a taper hole is similar to the external taper turning operation and is accomplished by rotating the work on a chuck or a face plate, and feeding the tool at an angle to the axis of rotation of the workpiece. The taper boring may be done by any one of the following methods :

1. A boring tools is mounted on the tool post and by swivelling the compound slide to the desired angle, a short taper hole is machined by hand feeding.
2. The taper turning attachment may be used to guide the boring tool at an angle to the lathe axis by disengaging the crossslide from the crossslide screw. The operation is similar to a plain boring operation.
3. Standard small tapers may be bored by using taper reamers mounted on the tailstock spindle.

3.31 INTERNAL THREAD CUTTING

The principle of cutting internal threads shown in Fig.3.66 is similar to that of an external thread, the only difference being in the tool used. The tool is similar to a boring tool with cutting edges ground to the shape conforming to the type of the thread to be cut. The hole is first bored to the root diameter of the thread. For cutting metric thread, the compound slide is swiveled 30° towards the headstock. The tool is fixed on the tool post or on the boring bar after setting it at right angles to the lathe axis, using a thread gauge. The use of thread gauge is illustrated in Fig.3.67. The depth of cut is given by the compound slide and the thread is finished in the usual manner.

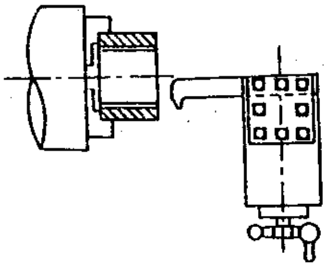


Figure 3.66 Internal thread cutting operation

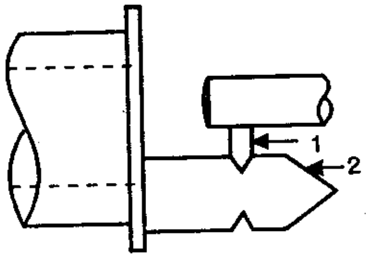


Figure 3.67 Use of thread tool gauge for internal thread cutting
1. Internal thread cutting tool,
2. Thread tool gauge.

3.32 TAPPING

Tapping is the operation of cutting internal threads of small diameter using a multipoint cutting tool called the tap. In a lathe, the work is mounted on a chuck or on a face plate and revolved at a very slow speed. A tap of required size held on a special fixture is mounted on the tailstock spindle. The axis of the tap should coincide exactly with the axis of the work. The tap will automatically feed into the work with the help of the special fixture.

3.33 UNDERCUTTING

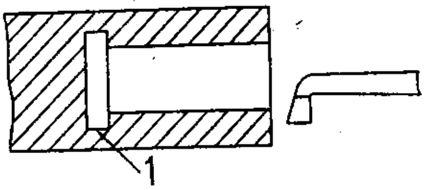


Figure 3.68 Undercutting operation
1. Undercut.

Undercutting shown in Fig.3.68 is similar to grooving operation when performed inside a hole. It is the process of boring a groove or a large hole at a fixed distance from the end of a hole this is similar to boring operation. except that a square nose parting tool is used. Undercutting is done at the end of an internal thread or a counterbore to provide clearance for the tool or any mating part.

3.34 PARTING-OFF

Parting-off is the operation of cutting a workpiece after it has been machined to the desired size and shape. A parting-off operation is shown in Fig.3.69. The process involves rotating the workpiece on a chuck or faceplate at half the speed that of turning and feeding by a narrow parting-off tool perpendicular to the lathe axis by rotating the crossslide screw by hand. Before the operation is started, the carriage is locked in position on the lathe bed and the cutting tool is held rigidly on the tool post with the compound slide set parallel to the lathe axis. The tool should be fed very slowly to prevent chatter. The feed varies from 0.07 to 0.15 mm per revolution and the depth of cut which is equal to the width of the tool ranges from 3 to 10 mm. In parting a work of very large diameter, cuts are made in stages. The parting tool is first fed through a certain depth it is then withdrawn, and two more cuts are made at the two sides of the central groove. The tool is next fed into the central groove until the work is cut off in two parts.

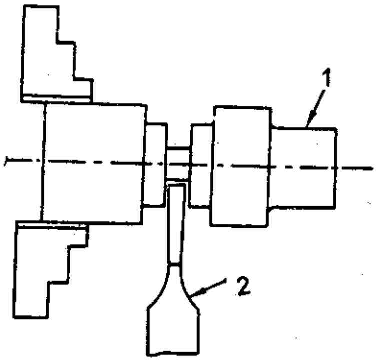


Figure 3.69 Parting off operation
1. Work, 2. Parting off tool.

3.35 MILLING

Milling is the operation of removing metal by feeding the work against a rotating cutter having multiple cutting edges. It is performed in a lathe by any one of the two methods :

1. For cutting keyways or grooves, the work is supported on the crossslide by a special attachment and fed against a rotating milling cutter held by a chuck. The depth of cut is given by vertical adjustment of the work provided by the attachment.
2. The work may be supported between centres and held stationary. The attachment mounted on the carriage drives the cutter from an individual motor. The feeding movement is provided by the carriage and the vertical movement of the cutter is arranged in the attachment. A number of grooves on the

periphery of the work may be cut by rotating the work by a fixed amount and machining it against the cutter. A gear wheel may be cut on a lathe by fixing a universal dividing head at the rear end of the headstock spindle. This permits dividing the periphery of the work by an equal amount.

3.36 GRINDING

Grinding is the operation of removing metal in the form of minute chips by feeding the work against a rotating abrasive wheel known as the grinding wheel. Both internal and external surfaces of a workpiece may be ground by using a special attachment mounted on the crosslide. For grinding external surface, the work may be revolved between centres or on a chuck. For internal grinding the work must be revolved on a chuck or faceplate. The feeding is done by the carriage and the depth of cut is provided by the crossslide. Grinding is performed in a lathe for finishing a job, sharpening a cutter, or sizing a workpiece after it has been hardened.

3.37 CUTTING TOOLS

For general purpose work, the tool used in a lathe is a single point tool, but for special ; operations multipoint tools may be used.

Classification : single point lathe tools are classified under the following groups :

1. *According to the method of manufacturing the tool :*
   (a) Forged tool.
   (b) Tipped tool brazed to the carbon steel shank.
   (c) Tipped tool fastened mechanically to the carbon steel shank.
2. *According to the method of holding the tool :*
   (a) Solid tool (b) Tool bit inserted in the tool holder.
3. *According to the method of using the tool :*
   (a) Turning (f) Forming
   (b) Chamfering (g) Boring
   (c) Thread cutting (h) Internal thread cutting
   (d) Facing (i) Parting-off
   (e) Grooving
4. *According to the method of applying feed :*
   (a) Right-hand (b) Left-hand
   (c) Round nose

3.38 FORGED TOOL

Forged tools are manufactured from high carbon steel or high speed steel. The required shape of the tool is given by forging the end of a solid tool steel shank. The cutting edges are then ground to the shape to provide necessary tool angles. Fig.3.70 shows a forged tool.

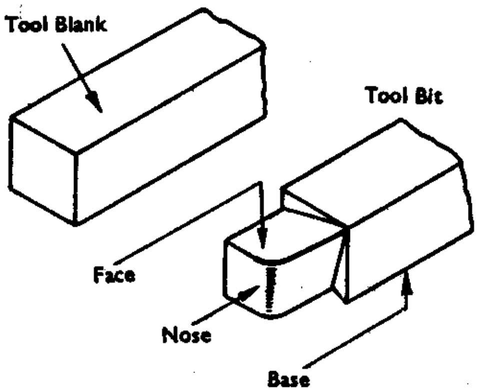


Figure 3.70 Forged tool

3.39 BRAZED TIPPED TOOL

Stellite and cemented carbide tool materials, in view of the very high cost, brittleness, and low tensile strength, are used in the form of small tips. They are made to the various shapes to form different types of tools and are attached permanently to the end of a carbon steel shank by a brazing operation. High speed steel due to its high cost is also sometimes used in the form of tips brazed on carbon steel shank.

Brazing : Brazing is a method of joining two or more metals by means of a fusible alloy or metal called "spelter" which fuses at some temperature above red heat, but below the melting temperature of the parts to be joined.

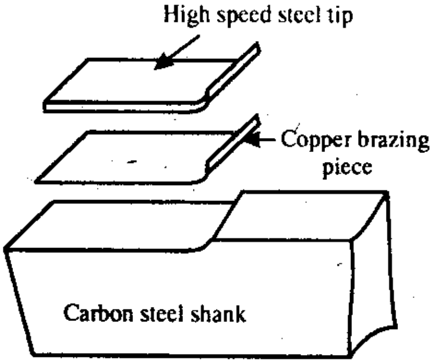


Figure 3.71 Brazing process

The brazing of a tip on the carbon steel shank is performed in four steps. This is illustrated in Fig.3.71. The first step is to prepare the shank and to form the recess at the end of the shank to accommodate the tip. The recess is formed either by grinding or milling and slant surfaces are machined approximately to the cutting angles. The second step is to clean the brazing surface either by mechanical or chemical means. Mechanical cleaning is done either by light grinding or sand blasting the surface. Chemical cleaning is done by dipping the edge in hydraulic acid and then in carbon tetrachloride solution for a few minutes. The cleaned surface is then coated with flux, usually borax, which prevents the surface from oxidization. The common brazing metals used are : copper which melts are 1080°C, tobinbronze which melts at 885°C and silver solder which melts at 700°C. The third step is to heat the shank and the tip as one piece. This is performed by any one of the following methods.

Torch brazing : When a small number of tools are brazed, a nonoxidizing flame from an oxy-acetylene torch is used to preheat the shank. The surface is then tinned with a brazing metal and the tip is placed in position. The whole assembly is next heated at the bottom and when it gets properly heated the tip is pressed from the top.

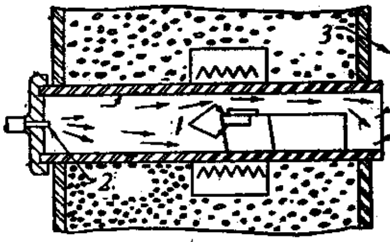


Figure 3.72 Furnace brazing
1. Tip on brazing metal, 2. Hydrogen inlet, 3. Hydrogen flame.

Furnace brazing : When a large number of tools are brazed the heating is done in an electric furnace in an atmosphere of hydrogen or in a gas furnace to prevent it from oxidization. The furnace brazing process is illustrated in Fig.3.72. The brazing surface is coated with a fluxing material and a thin sheet of brazing metal is placed on it. The tip is seated above the brazing metal and the whole assembly is then heated in the furnace. The tool is withdrawn at the correct temperature when the brazing metal melts and the tip is pressed on the recess.

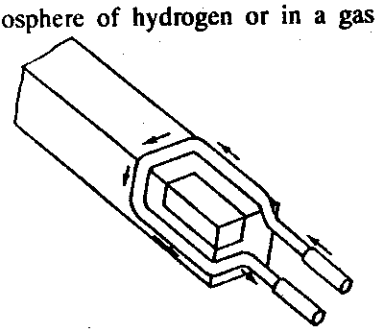


Figure 3.73 Induction brazing

High frequency induction brazing : To prevent accidental cracking of tips that may happen in any one of the previous methods, localized heating is effected by passing a high frequency alternating current through a coil surrounding the tip. The resistance offered by the tip to the flow of an electric current causes the heat to be generated and melts the brazing metal. the induction brazing process is illustrated in Fig.3.73.

After the tip has been securely brazed on the tool shank, the last and the final step is to grind the cutting edges to exact tool angles.

3.40 MECHANICALLY FASTENED TIPPED TOOL

To ensure rigidity that a brazed tool does not offer, tips are sometimes clamped at the end of a tool shank by means of a clamp and bolt. Ceramic tips which are difficult to braze are clamped at the end of a shank, Fig.3.74 shows a mechanically fastened tipped tool.

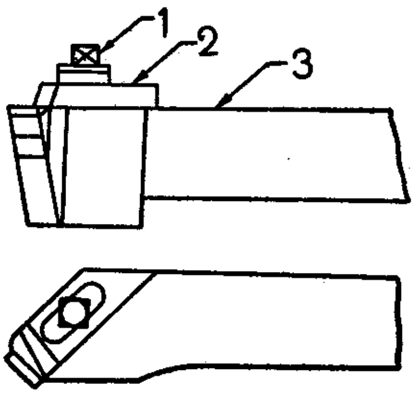


Figure 3.74 Mechanically fastened tipped tool
1. Clamping screw, 2. Clamp, 3. Shank.

3.41 SOLID TOOL

Solid tools are made of high carbon steel forged and ground to the required shape. They are mounted directly on the tool post of a lathe.

Shank section : The shank section of a tool may be round , square or rectangular. Round section ranges from 6 to 63 mm. In the Indian standard system this is denoted as 63 IS : 1983 if the diameter is 63 mm. Square section ranges from 6 × 6 to 63 × 63 all in mm and is denoted as, say, 20 × 20 IS : 1983. Rectangular section tools may have height to breadth ratio as 1.25 : 1 or 1.6 : 1 or 2:1. The height of the shank range from 6 to 63 mm in each case, and the breadth is calculated from the given ratio. For example, in the case of height to breadth ratio of 2 : 1 if the height is 6 mm, the breadth will be 3 mm and it will be denoted as 63 IS : 1983. the accepted numbers in each case which ranges from 6 to 63 mm are 6, 8, 10, 12, 16, 20, 25, 32, 40, 50 and 63. Severe machining conditions demand larger shank section of the tool.

The standard dimensions of carbide tool shanks are given in Table 3.2.

TABLE 3.2 INDIAN STANDARD DIMENSIONS OF CARBIDE TOOL SHANKS AND NOSE RADIUS

Shank section l x b mm × mm	*Tool length mm*	*Nose radius mm*
10×10	90	0.5
12×12	100	0.5
16×16	110	0.5
20×20	125	0.5
25×25	140	1.0
32×32	170	1.0
40×40	200	1.0
50×50	240	1.6

3.42 TOOL BIT AND TOOL HOLDERS

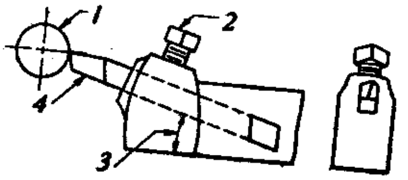


Figure 3.75 Tool bit and tool holder
1. Work, 2. Clamping screw,
3. Tool holder angle 15°, 4. Tool bit.

A tool bit is a small piece of cutting material having a very short shank which is inserted in a forged carbon steel tool holder and clamped in position by bolt or screw. A tool bit may be of solid type or tipped one according to the type of the cutting tool material. Tool holders are made of different designs according to the shape and purpose of the cutting tool. Fig.3.75 illustrates a common type of tool holder using h.s.s. tool bit. The tool bit is inserted in a slot set at an angle of 15° to the base. This inclination reduces the effective clearance angle and increases the top rake angle by 15°. So in grinding the tool this 15° is to be added to the actual clearance angle and deducted from the rake angle.

The advantages of using a tool bit over a solid tool are as follows :

1. The tool can be adjusted to the correct height easily by simply adjusting the position of the tool bit in the slot.
2. This is less expensive than a solid tool.
3. Regrinding of the tool is easier as only the end cutting edges are required to be ground.

4. A tool bit may be easily withdrawn and replaced in position without disturbing the setting

Inspite of all these advantages, solid tools are still preferred in cases where rigidity of the tool is of prime importance. Moreover, the solid tool being made of one material the heat is uniformly distributed throughout.

3.43 INFLUENCE OF TOOL ANGLES

Tool angles of a single point tool, which have been explained in Art. 2.14 and given in Fig.2.17 exert direct influence in all metal cutting operations in a lathe. These are summarized below :

Rake : The rake or slope of the tool face has the following functions :

1. It allows the chips to flow in a convenient direction.
2. It reduces the cutting force required to shear the metal and consequently helps to increase the tool life and reduces the power consumed.
3. It provides keenness to the cutting edge.
4. It improves the surface Finnish.

The amount of rake angle to be given in a tool, depends on the following factors :

(i) ***Type of material being cut*** : A harder material like cast iron may be machined with a smaller rake angle than that required by a soft metal like mild steel or aluminium. Maximum support to the cutting edge in machining hard metal is afforded to by increasing the lip angle and decreasing the rake angle.

(ii) ***Type of tool material being used*** : Tool material like cemented carbide permits turning at a very high cutting speed . It has been observed that in machining at a very high cutting speed rake angle has a little influence of cutting pressure. Under such condition, the rake angle is reduced to a minimum or even negative rake is provided to increase the tool strength . This increases the lip angle.

(iii) ***Depth of cut*** : In rough turning, high depth of cut is given to withstand severe cutting pressure. So the rake angle should be decreased to increase the lip angle that provides strength to the cutting edge. Tools may have larger rake angle where small depth of cut is necessary.

(iv) ***Rigidity of the toolholder and condition of machine :*** An improperly supported tool on an old and worn out machine cannot take up severe cutting pressure . So in machining under the above condition, the tool used should have larger rake angle than that at the normal condition to reduce the cutting pressure.

Front rake : The front rake influences the machining condition when the tool removes metal from its front cutting edge. A common example is a parting-off tool.

Side rake : The side rake influences the machining condition when it removes metal on its side cutting edge only . A knife edge turning tool may not have any front rake but it must have a definite amount or side rake. Side rake also allows the chips to flow by the side of the tool without getting entangled with the tool post.

True rake : As most of the standard tools remove metals both on its end and side cutting edges a slope on the face of the tool is given suitably combining the front and side rake together. This resultant slope is called true rake.

The rake or slope of the face may be : (a) positive, (b) zero or (c)negative. Fig. 3.76 shows positive, negative and zero rake of a single point cutting tool.

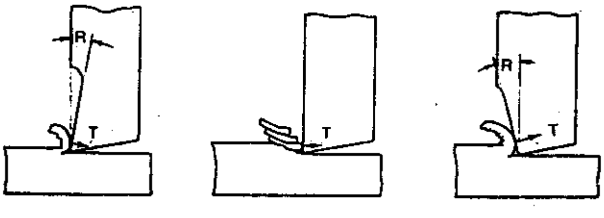


Figure 3.76 Positive, zero and negative rake
R. Rake, T. Thrust

Positive rake : A tool has a positive rake when the face of the tool slopes away from the cutting edges and slants towards the back or side of the tool. In most cases, tools are provided with a positive rake.

Zero rake : A tool has a zero rake when the face of the tool has no slope and is in the same plane or parallel to the upper surface of the shank. Turning tools for brass usually have zero rake as the metal is removed in short chips exerting little cutting pressure on the tool face. Zero rake increases the strength of the tool and prevents the cutting edge from digging into the work.

Negative rake : A tool has a negative rake when the face of the tool slopes away from the cutting edge and slants upward towards the back or side of the tool. Negative rake is used in turning metal with cemented carbide tipped tool in mass production work.

Turning with a negative rake has the following advantages :

1. The point of application of the cutting force is altered from the cutting edge where the tip is weak to a stronger section.
2. The thrust gives a compressive load on the tip. The cemented carbide tips which are capable of withstanding compressive loads 3 to 4 times than bending loads are very suitable for negative rake turning.
3. It can work against a very high cutting speed.
4. It decreases tool wear and consequently increases the tool life.
5. Negative rake increases the lip angle of the tool permitting it to take heavier depth of cut.

The conditions which limit the application of negative rake turning are as follows :

1. ***High speed :*** The machine must be operated at a very high speed to take full advantage of negative rake turning. This imposes a certain limit.
2. ***Rigidity of the machine :*** The machine and the tool holding device must be sufficiently rigid to resist vibration that may set up in the machine when it runs at a high speed. Cemented carbide tips being very brittle may fracture under vibration.
3. ***High heat dissipation :*** High heat generated in negative rake turning must be properly dissipated, otherwise the tool will fail very soon.
4. ***Increase power :*** The cutting force and the power required is increased by 10 to 15 per cent of that required in positive rake machining under similar condition.

Clearance angle : The main function of the clearance angle is to prevent the flank of the tool from rubbing against the surface of the work allowing the cutting edges of the tool only to come in contact with the work material.

Front clearance angle : The front clearance angle prevents the front flank of the tool from rubbing against the work. A minimum clearance angle is given to provide maximum support to the tool cutting edges by increasing the lip angle. The front clearance angle should be increased for large diameter work.

Side clearance angle : The side clearance angle prevents the side of the tool from rubbing against the work when longitudinal feed is applied. The side clearance angle depends upon the amount of feed given. Larger feed will require greater side clearance angle.

Nose radius : The nose of a tool is slightly rounded in all turning tools. The functions of nose radius are as follows :

1. Greater nose radius clears up the feed marks caused by the previous shearing action and provides better surface finish. All finish turning tools have greater nose radius than rough turning tools.
2. It increases the strength of the cutting edge, tends to minimize wear taking place in sharp pointed tool with consequent increase in tool life.
3. Accumulation of heat is less than that in a pointed tool which permits higher cutting speeds.
4. Slight reduction in cutting force may be obtained.

Very large nose radius may cause chatter. For rough turning, nose radius is usually 0.4 mm and for finish turning it varies from 0.8 to 1.6 mm.

Side cutting edge angle : The side cutting edge angle of turning tools vary from 0° to 90°. The following are the advantages of increasing side cutting edge angle.

1. It increases tool life as, for the same depth of cut, the cutting force is distributed on a wider surface.
2. It diminishes chip thickness for the same amount of feed and permits greater cutting speed.
3. It dissipates hear quickly for having wider cutting edge.
4. It improves surface finish.

Very large side cutting edge angle produces chatter or bending on a slender workpiece. The usual value of the side cutting edge angle is 15°

although angle from 30° to 40° are sometimes used.

Two extreme geometrical values of side cutting edge angle are (
and 90°. A knife edge turning tool has 0° side cutting edge angle, and it
cutting edge is perpendicular to the work surface. This type of tool is use
for turning slender work as no bending stress is developed when the tool i
fed into the work. The end thrust is taken up by the live centre.

A square nose tool having side cutting angle equal to 90° has it:
cutting edges parallel to the work surface. This type of tool is used fo
finish turning where a very fine depth of cut and coarse feed may be given

End cutting edge angle : The main function of the end cutting edge angl
is to prevent the trailing front cutting edge of the tool from rubbing agains
the work. The end cutting edge angle ranges from 8° to 15°. A large end cutting edge angle unnecessarily weakens the tool.

Lip angle : The amount of cutting angle or lip angle determines the strength of the cutting edge. As the lip angle depends upon the amount of rake and clearance angle provided on the tool, the lip angle is maximum when the rake and the clearance angle are minimum. In the case of a negative rake tool, lip angle increases with the rake angle. Large lip angle permits machining harder metals, applying heavier depth of cut, and rotating the work at higher cutting speed. It also increases tool life, and improves dissipation of heat.

Recommended tool angles : It has been observed from the foregoing discussion that the tool angles vary under different machining conditions. Tables 3.3, 3.4 and 3.5 show average recommended tool angles for different work and tool materials.

TABLE 3.3 THE RECOMMENDED ANGLES FOR HIGH CARBON AND HIGH SPEED TURNING TOOLS

Material	*Front rake deg.*	*Front clearance deg.*	*Side rake deg.*	*Side clearance deg.*
Mild steel	10–12	6–8	10–12	6–8
Stainless steel	5–7	6–8	8–10	7–9
Aluminium	30–35	8–10	14–16	12–14
Brass	0–6	8–10	1–5	10–12
Cast iron	3–5	6–8	10–12	6–9
Copper	14–16	12–14	18–20	12–14

TABLE 3.4 RECOMMENDED ANGLES FOR STELLITE TURNING TOOLS

Material	*Front rake deg.*	*Front clearance deg.*	*Side rake deg.*	*Side clearance deg.*	*Side cutting edge deg.*	*End cutting edge deg.*
Mild steel	8–15	7	8–15	7	10	15
Stainless steel	8–15	7	8–15	7	10	15
Aluminium	10–20	7–8	12–15	7–8	10	10
Brass	4	5	4	5	10	10
Cast iron	0	5–6	5	5–6	0–15	10
Copper	20	8	10	10	10	10

3.44 INFLUENCE OF TOOL HEIGHT ON TOOL ANGLES

Clearance and rake angles are dependent on the position of the cutting tool in relation to the axis of rotation of the work. If the tool point is on the centre line and the base of the tool is horizontal, the rake and clearance angles are operative under their values as ground on the tool.

If the cutting edge of a turning tool is held below the centre of the part to be turned, the rake angle is reduced and the clearance angle is increased. If the tool angle is set sufficeently below the centre line, the effective rake angle may even be negative. This results in an unfavorable working position. The flow of chips is obstructed, the cutting edge of the tool may seize and break and the cutting pressure is increased. As a consequence, the surface quality of the workpiece will be impaired.

If the turning tool is held in such a way that its cutting edge is above the centre of the workpiece the rake angle is increased, whereas the clearance angle is reduced. The flow of chips is improved, whereas the friction between the flank of the cutting tool and the workpiece is increased.

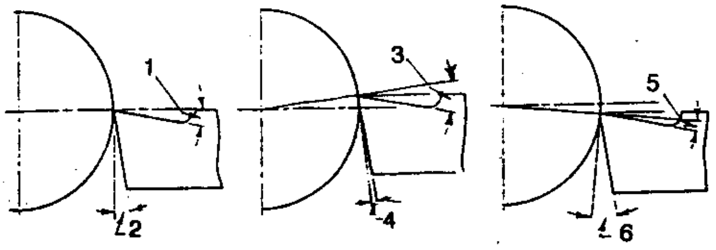


Figure 3.77 Influence of tool height on tool angles

Tool nose at the centre height: 1. True rake, 2. True clearance angle
Tool nose above the centre height: 3. Effective rake, 4. Effective clearance angle.
Tool nose below the centre height: 5. Effective rake, 6. Effective clearance angle.

Therefore, as a rule, the correct position is that the tool is set in horizontal position with its nose touching exactly to the centre of the part to be turned. In many cases of rough and finish turning 1 per cent variation above or below the centre line is permitted. Fig.3.77 the centre line of the work.

3.45 TYPES OF TOOL

In a lathe work different operation require different types of tools which are described below :

TABLE 3.5 RECOMMENDED ANGLES CEMENTED CARBIDE TURNING TOOLS

Material	*Front rake deg.*	*Front clearance deg*	*Side rake deg.*	*Side clearance deg.*	*Side cutting edge deg*	*End cutting edge deg.*	*Nose radius mm.*
Mild steel	0–(–7)	5–10	+ 6 (–7)	5–8			0–1
Stainless steel	0–(–7)	5–10	+ 6 (–7)	5–10			0–1
Aluminium	0–10	6–10	10–20	6–10	0–45	5–15	0.5–1.5
Brass	0–(–7)	6–8	+ 8 (–5)	6–8			0.3–0.5
Cast iron	0–(–7)	5–8	+ 6 (–7)	5–8			0.5–1.1
Copper	0–4	6–8	15–20	6–8			0.5–1.5

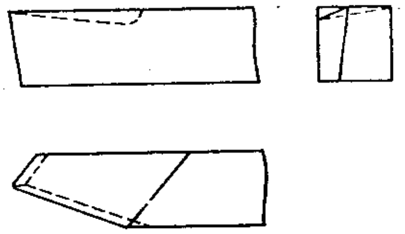


Figure 3.78 Rough turning tool

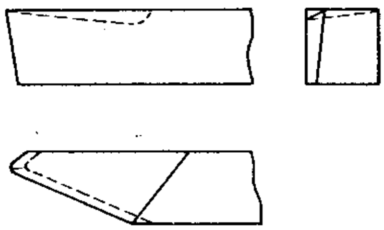


Figure 3.79 Finish turning tool

Turning tool : There are mainly two classes of turning tool: (1) rough turning tool, and (2) finish turning tool.

Rough turning tool : The main function of a rough turning tool is to remove maximum amount of metal in a minimum time that the tool, work, and the machine will permit. Fig. 3.78 illustrates a typical rough turning tool. The cutting angle is so ground that it can withstand maximum cutting pressure.

Finish turning tool : A finish turning tool is used to remove a very small amount of metal. The tool angle is so ground that it can produce a very smooth and accurate surface. Fig.3.79 illustrates a typical finish turning tool.

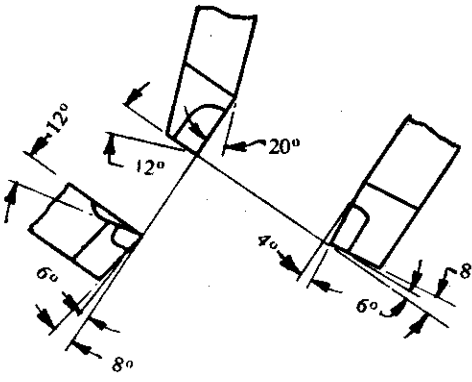


Figure 3.80 Cemented carbide tipped turnir tool

Fig.3.90 illustrates a h.s.s. boring tool. The rake and clearance angles are similar and can be given for turning mild steel according to Table 3.3. For turning any other metal the rake and clearance angles are changed keeping front cutting edge angle and side cutting edge angle approximately constant. Fig.3.80 illustrates a cemented carbide tipped turning tool conforming Indian standard specification. The end view given shows true rake and clearance angles of the cutting edge. Fig.3.81(a)(b)(c) illustrates a stellite tipped and diamond pointed turning tool.

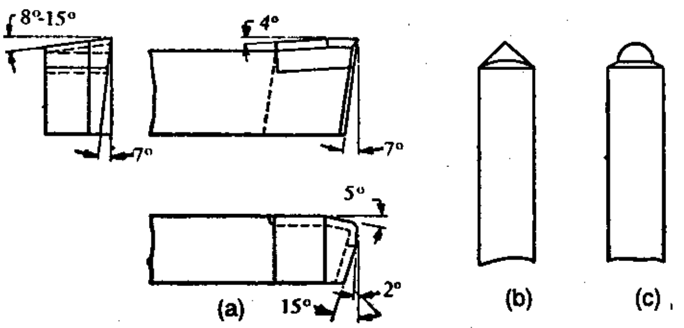


Figure 3.81 Stellite tipped and diamond pointed turning tool
(a). Stellite tipped turning tool, (b). Diamond pointed rough turning tool, (c). Diamond pointed finish tool.

Chamfering tool : A straight turning tool may be used as a chamfering tool when the cutting edges are set at the angle of chamfer. Where a large number of chamfer works are to be performed a special chamfering tool with its side cutting edge angle ground to the angle of chamfer used. A chamfering tool is shown in Fig.3.52.

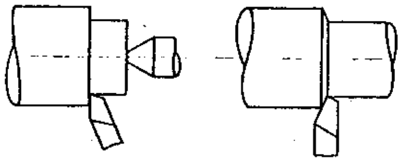


Figure 3.82 Shoulder turning tool

Shoulder turning tool : A square shoulder is turned by a knife edge turning tool or facing tool. A beveled shoulder may be turned by a straight turning tool having a side cutting edge angle and zero nose radius as illustrated in Fig. 3.82. A filleted shoulder may be turned by a straight turning tool with a nose radius corresponding to the fillet radius of the work.

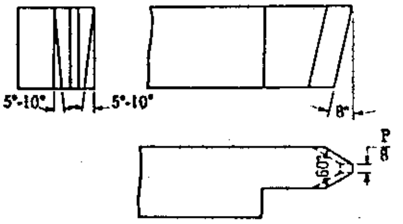


Figure 3.83 h.s.s. thread cutting tool

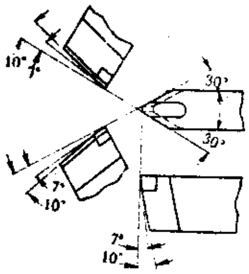


Figure 3.84 Cemented carbide tipped thread cutting tool

External thread cutting tool : Metric, B.S.W., or American "V" threads are formed by a single point thread cutting tool with its cutting edges ground to the shape and size of the thread to be cut. The shape of the tool is determined by the included angle at the nose of the tool which should correspond to the angle of the thread. It may be 60° for metric threads or 55° for B.S.W. threads. The size or cross-section of the cutting edges of the tool depends upon the pitch of the thread. Fig.3.83 illustrates a h.s.s. thread cutting tool.

So for machining different screw threads having different pitches separate tools are used to generate accurate threads. The nose of the tool may be pointed, flat or rounded according to the shape of the root of the thread. A thread tool gauge may be used to check the shape and size of the tool after it has been ground. No top rake is given to the tool if it removes metal on both of inside cutting edges and the successive cuts are given by feeding the tool perpendicular to the work surface. Some amount of side rake is given to the tool when it removes metal from one of its side cutting edge only and successive cuts are given by the compound slide which is swiveled to the half angle of the thread. The side clearance on a thread straight turning tool, because the effective side clearance is reduced by the helix angle of the thread. Fig.3.84 and 3.85 illustrates cemented carbide and diamond pointed thread cutting tools.

Tool for cutting square threads : The side clearance of the tool for cutting square thread is of prime importance in order to prevent the tool from interfering or rubbing against the vertical flank of the thread. As a rule, the forward side clearance angle is determined by adding 5° to the helix angle of the thread and trailing side clearance is obtained by subtracting 5° from the helix angle. If ϕ be the forward side clearance angle and θ be the trailing side clearance angle, then from the formula :

$$\angle\phi = 5^\circ + \left(\tan^{-1} . \frac{lead}{\pi \times core\, dia.} \right) \qquad 3.12$$

$$\angle\phi = \left(\tan^{-1} . \frac{lead}{\pi \times outside\, dia.} \right) - 5^\circ \qquad 3.13$$

The width of the cutting edge should be equal to half the pitch of the thread.

Small clearance angle of 1° to 2° are provide at the side of the tool in order to prevent the surface from rubbing with the work. A square thread cutting too is shown in Fig. 3.86.

Figure 3.85 Diamond pointed thread cutting tool

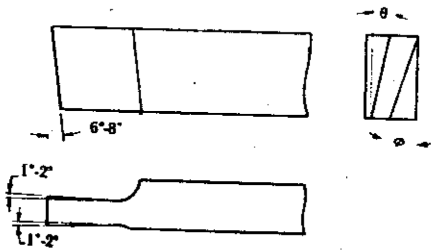


Figure 3.86 Square thread cutting tool
ϕ. Forward side clearance,
θ. Trailing side clearance.

Internal thread cutting tool : The cutting edge of the tool is exactly similar to an external thread cutting tool but the front clearance angle is sufficiently increased as in a boring tool. The tool may be forged type or bit type or bit type and held on a boring bar. The tool may be tool must be set square with the work.

Facing tool : The facing tool removes metal by its side cutting edges. So no top rake is necessary in a facing tool. Fig.3.87 illustrates a h.s.s. facing tool intended for finishing operation. The tool having 0° side cutting edge angle and 34° end cutting edge angle can be accommodated in the space between the end of the work and 60° dead centre leaving a clearance 2° on both sides.

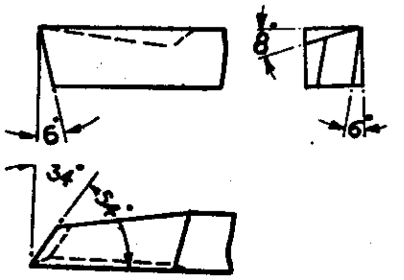


Figure 3.87 h.s.s. facing tool

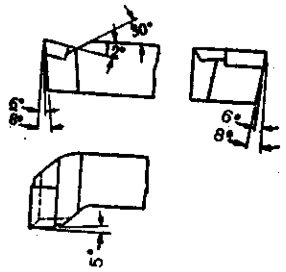


Figure 3.88 Cemented carbide cranked facing tool

Fig.3.88 illustrates a cemented carbide cranked facing tool conforming the Indian standard specification. The standard shank sections are 20x20, 25x25, 30x30, 32x32, 40x40, and 50x55 all expressed in mm. The length of the tool may be 125, 140, 170, 200 and 240 mm and the nose radius varies from 0.5 to 1.6 mm.

Grooving tool : Grooving tool is similar to a parting-off tool illustrated in Fig. 3.95. The cutting edges may be made square, rounded or "V" shaped according to the shape of the groove to be cut.

Forming tool : Turning curved profiles may be effected by using (1) ordinary lathe tools, (2) flat forming tools, and (3) circular forming copying attachment is used to reproduce the form of a template. Flat forming tools are made of two types: (1) simple forming tools, and (2) flat dovetail forming tools.

Simple forming tools have their cutting edges ground to the shape of the groove, undercut, or thread to be cut. Flat dovetail forming tools have wider cutting edges corresponding to the shape desired. Dovetail end of the tool is fitted in special tool holder. No front ranges from 10° to 15°. Regrinding is always done on the top face of the tool which does not alter the form of the tool. Circular form tools are preferred in production work as a very long cutting surface can be used resulting in a long tool life. The centre of the tool is set slightly above the centre line of the work to provide an effective front clearance angle on the tool. The tool will rub against the work if the centres are on the same height. The tool centre is usually higher than the centre line of the lathe by 1/20 to 1/10 of the tool diameter. This height is termed 'offset'. Regrinding is done by grinding the flat face only. Fig.3.64 illustrates a form tool.

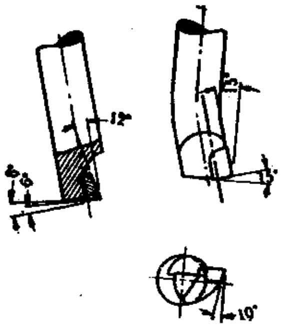


Figure 3.89 Cemented carbide tipped boring tool

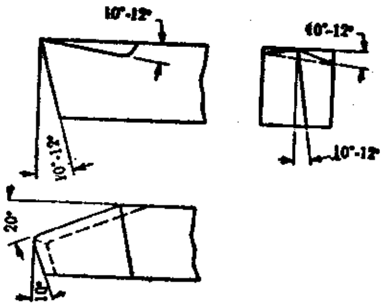


Figure 3.90 h.s.s. tool bit for boring bar

Boring tool : A boring tool is similar to a left hand external turning tool so far its cutting edge is concerned. A cemented carbide tipped boring tool is illustrated in Fig. 3.89. The tool may be a bit type inserted in boring bar or holder, or forged type having a tool shank. Fig. 3.90 shows a h.s.s. tool bit inserted in boring bar. A boring bar is made of mild steel with slots or holes cut into it to accommodate the tool bit which may be locked by an Allen screw. the amount of projection of the cutting edge of the tool from the centre of the bar determined the finished hole diameter of the work. The bit is generally inserted at right angles to the centre line of the bar for boring a continuous hole passing from one end to the other end. The bit is set at an angle to the axis projecting beyond the end of the bar for boring a blind hole. A tool bit having two cutting edges at its two ends is used for quick machining. A wide double bladed cutter may be inserted in the boring bar for finish boring operation. Fig. 3.91 shows different designs of boring cutter mounted on a boring bar. Two or more bits may be inserted in a boring bar for boring different diameters in one setting.

Boring bar are held in the tailstock for boring small holes ranging from 12 to 100 mm. For boring larger hole diameters, boring bars are gripped by two clamp blocks and held in the tool post. For precision boring or boring in odd size work that is supported on crossslide, the bar may be supported on centres and is made to revolve.

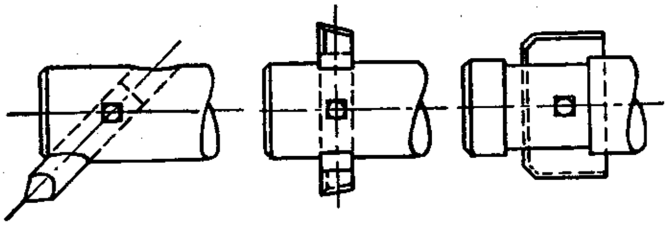


Figure 3.91 Boring cutters on boring bar

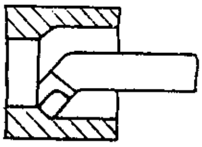


Figure 3.92 Tipped boring tool in operation

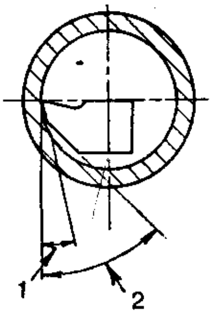


Figure 3.93 Double clearance of a boring tool

1. Primary clearance,
2. Secondary clearance.

Although a forged tool is commonly used it is not a very rigid unit as a very small cross-section of the tool is used to enable the tool to pass into the work. For this reason holes less than 100 mm cannot be conveniently bored by a forged tool. Fig.3.92 shows a tipped boring tool in operation.

In a boring tool, the tool cutting edge must have sufficient front clearance in order to clear the work. In order to strengthen the tool point double clearance, primary and secondary, may be provided. Fig.3.93 shows a boring tool with double clearance. The smaller the hole diameter the larger should be the front clearance. Larger clearance angle necessitates reduction in rake angle in boring tool. The nose of the tool may be straight or round according to the type of finish desired.

Counterboring tool : Counterboring operation can be performed by an ordinary boring tool. The tool cutting edge is so ground that it can leave a shoulder after turning. A counterbore having multiple cutting edges is commonly used.

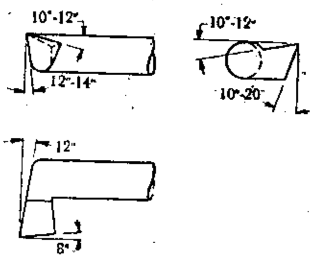


Figure 3.94 Undercutting tool

Undercutting tool : Undercutting or grooving tool has a point and form of the cutting edge exactly similar to the form of the required groove. Clearance angle is given at all the wider than the width of the tool cutting edge, longitudinal feed is employed. Fig.3.94 illustrates an undercutting tool. The front clearance angle depends upon the bore of the work.

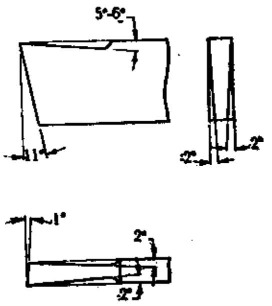


Figure 3.95 h.s.s. Parting off tool

Parting off tool : A parting off tool is usually forged and may be used as bits for cemented carbide tipped tools. Parting tool is made as narrow as possible to remove minimum of metal. The length of the cutting tool which penetrates into the work should be slightly longer than the radius of the barstock being machined. As the tool penetrates deep into the work, clearance is provided all around the tool cutting edge to prevent it from rubbing against the work surface. As the tool is purely end cutting it has no side rake; slight back rake may be provided on the tool to promote easy flow of the chips. Fig.3.95 and 3.96 illustrates a high speed and cemented carbide parting off tool.

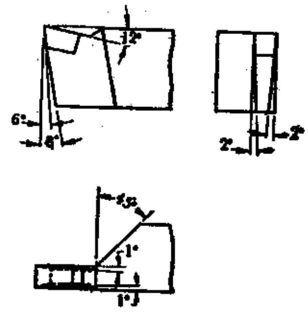


Figure 3.96 Cemented carbide tipped parting off tool

Right hand tool : A right hand tool illustrated in Fig.3.97 is that which is fed from right to the left hand end of the lathe bed, i.e. from the tailstock to the headstock end when operations like turning, thread cutting etc. are performed. A right hand tool may be known by its cutting edge which is formed on its left hand end when viewed from the top with its nose pointing away from the operator.

Left hand tool : A left hand tool illustrated in Fig.3.98 is that which is fed from Left to the right hand end of the lathe bed, i.e. from the headstock to the tailstock end. A left hand tool may be used for left hand thread cutting operation or turning operation which leave a shoulder on the right hand end of the workpiece. A left hand tool has its cutting edge formed on its right hand end when viewed from the top with its nose left hand tool may also be used for facing operation.

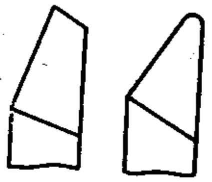


Figure 3.97 Right hand tool

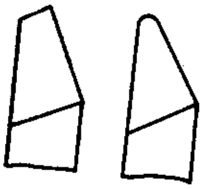


Figure 3.98 Left hand tool

Figure 3.99 Round nose tool

Round nose tool : A round nose turning tool shown in Fig.3.99 may be fed from left to the right or from right to the left hand end of the lathe bed ways. For this reason they have no back rake and side rake. In some cases a small back rake may be provided on the tool. A round nose turning tool is generally used for finish turning operation.

3.46 CUTTING SPEED

The cutting speed (v) of a tool is the speed at which the metal is removed by the tool from the workpiece. In a lathe it is the peripheral speed of the work past the cutting tool expressed in meters per minute.

$$\text{Cutting speed} = \frac{\pi d n}{1{,}000} \quad m/\text{min}. \qquad 3.14$$

where, d is the diameter of the work in mm,
and n is the r.p.m. of the work.

In the British system, cutting speed is expressed in feet per minute and diameter of the work in inches.

$$\text{Cutting speed} = \frac{\pi dn}{12} \text{ feet/min.} \qquad 3.15$$

where *d* is the diameter of the work in inches, and *n* is the r.p.m. of the work.

The cutting speed, direction of feed and depth of cut to be given to a workpiece are illustrated in Fig.3.100.

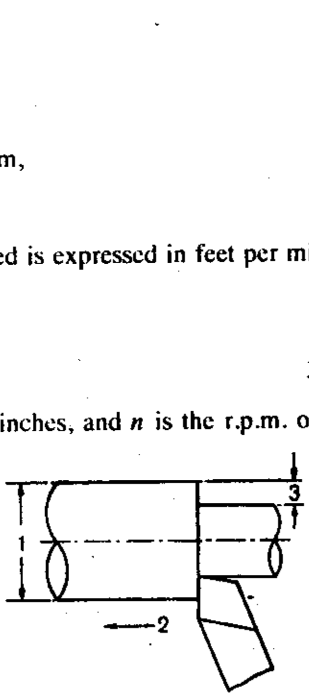


Figure 3.100 Cutting speed, feed and depth of cut
1. Diameter on which cutting speed is calculated. 2. Feed. 3. Depth of cut.

Example 3.21 : A steel shaft of 25 mm diameter is turned at a cutting speed of 50m per min. Find the r.p.m. of the shaft.

$$\text{Cutting speed} = \frac{\pi dn}{1{,}000} \text{ m/min.}$$

$$\text{or,} \quad 50 = \frac{\pi \times 25 \times n}{1{,}000}$$

$$\text{or,} \quad n = \frac{50 \times 1000}{\pi \times 25} = 637 \text{r.p.m.}$$

In practice when the calculated speed is not available in the machine the next lower value is selected.

3.47 FEED

The feeds of a cutting tool in a lathe work is the distance the tool advances for each revolution of the work. Feed is expressed in millimeters per revolution.

In the British system it is expressed in inches per revolution.

Increased feed reduces cutting time. But increased feed greatly reduces the tool life. The feed depends on factors such as size, shape, strength and method of holding the component, the tool shape and its setting as regards overhang, the rigidity of the machine, depth of cut, power available, etc. Coarser feeds are used for roughing and finer feeds for finishing cuts.

3.48 DEPTH OF CUT

The depth of cut *(t)* is the perpendicular distance measured from the machined surface to the uncut surface of the workpiece. In a lathe the depth of cut is expressed as follows :

$$\text{Depth of cut} = \frac{d_1 - d_2}{2} \qquad 3.16$$

where, d_1 = diameter of the work surface before machining,
and d_2 = diameter of the machined surface.

Other factors remaining constant, the depth of cut varies inversely as the cutting speed. For general purposes, the ration of the depth of cut to the feed varies from 10 : 1.

3.49 MACHINING TIME

The machining time in lathe work can be calculated for a particular operation if the speed of the job, feed length of the job is known.

If s is the feed of the job per revolution expressed in mm per revolution and l the length of the job in mm, then number of revolutions of the job required for a complete cut will be :

If the r.p.m. of the work is n, time taken to revolve the job through l/s number revolutions for a complete cut will be:

$$\frac{l}{s \times n} \text{ min.}$$

$$\text{Therefore, the time taken for a complete cut} = \frac{l}{s \times n} \text{ min.} \qquad 3.17$$

Example 3.22 : Find the time required for one complete cut on a piece of work 350 mm long and 50 mm in diameter. The cutting speed is 35 meters per minute and the feed is 0.5 mm per revolution.

$$\text{Cutting speed} = \frac{\pi dn}{1{,}000} = \frac{\pi \times 50 \times n}{1{,}000}\ 35$$

or, $$n = \frac{1000 \times 35}{\pi \times 50} = 222.5$$

Number of revolutions required for complete cut

$$= \frac{350}{0.5} = 700$$

Time required for complete cut = $\frac{700}{222.5} = 3.14$ min.

TABLE 3.6 AVERAGE CUTTING SPEED EXPRESSED IN m/min FOR DIFFERENT OPERATIONS IN A LATHE USING A H.S.S TOOL

Material	*Turning.*	*Thread cutting*	*Drilling*	*Reaming*
Cast iron	15–19	7–8	22–31	6–8
Mild steel	25–31	9–10	28–35	10–15
Aluminium	60–90	20–25	60–90	25–30
Brass	120	25–30	60–90	20–30

TABLE 3.7 AVERAGE CUTTING FEED AND DEPTH OF CUT FOR DIFFERENT TOOL MATERIAL

Material	*h.s.s v.m.p.m.*	*Stellite v.m.p.m*	*Cemented carbide v.m.p.m*	*Feed mm/rev*	*Depth of cut mm*
Cast iron	15–19	30	63	0.2 to 0.8	0.5 to 1 for finishing operation 2 to 5 for roughing operations
Mild steel	25–31	55	80		
Aluminium	60–90	120	180		
Brass	120	300	360		

3.50 CUTTING TOOL SIGNATURE

The *signature* is a sequence of numbers listing the various angles, in degrees, and the size of the nose radius. This numerical method of identification has been standardized by the American Standard Association.

The seven elements that comprise the signature of a single point cutting tool are always stated in the following order : back rake angle, and nose radius. Thus a tool with a shape specified as

8-14-6-6-6-15-4

has 8° back rake, 14° side rake, 6° end relief, 6° end or side relief, 6° end cutting edge and 15° side cutting edges angles, and 4 mm nose radius.

Plan approach angle : In cutting tool terminology used in the U.S.S.R., there is another angle called *plan approach angle.* This is the complementary angle to the slide-cutting-edge angle used in cutting tool terminology in India, Great Britain and the U.S.A.

This is the angle between the projection of the side cutting edge on the basic plane and the direction of feed. It is sometimes called the *entering angle.*

REVIEW QUESTIONS

1. What is the main function of a lathe ? List various types of lathes.
2. State the operations which may performed on a lathe.
3. Describe in brief an engine lathe.
4. How a lathe is specified ? Discuss.
5. What are the basic parts of an engine lathe ? Discuss the function of head stock.
6. What is the function of a back gear ?
7. (a) Differentiate between the motion of cross slide and a compound rest. (b) What are the different types of lathe bedways ? Why there is a gap in the bed ?
8. Why chucks are used ? List various types of chucks used in lathes. Describe any one in brief.
9. What is a mandrel ? Why they are used in lathes ? List different types of mandrels.
10. What are the different machining operations performed on a lathe by holding workpieces between centres or chucks ?
11. Distinguish rough and finish turning.
12. Define taper. How is the amount of taper expressed ? Name different methods of taper turning done on a centre lathe drawing simple sketches ?
13. Explain with schematic diagram the principle of thread cutting on a lathe. Find out the relation between ratio of change gears to the work pitch and lead screw pitch. The pitch of a lead screw is 6 mm and the pitch of the thread to be cut in 1.5 mm. Find the change gear. (30:120)

14. What is a thread chaser ? Briefly describe it.
15. Why turning is produced on a surface ? How it is produced on a lathe.
16. What is spinning ? Sketch and describe the spinning process on lathe.
17. How turning tools are classified ? list various turning tools.
18. Why brazed tipped tools are produced ? What are the techniques of brazing the tips with the tool shank ? Describe one.
19. What are the influences of cutting tool angles on machining ? Briefly state the effect of rake angle during cutting.
20. Describe various rakes of turning tools.
21. Sketch a square thread cutting tool and label tool angles.
22. Define cutting tool signature of a turning tool.