Safety precautions to be observed while working on lathes

Objective: At the end of this lesson you shall be able to
- **state the precautions to be observed before starting work on a lathe, during work and after.**

Before starting the work

Ensure that the lubricating system is functioning.

The mating gears should be in proper mesh and the power feed levers are in neutral position.

The work area should be clean and tidy.

The safety guards should be in place.

During work

Never try to stop a rotating chuck with your hand. A rotating chuck is dangerous.

Switch off the machine before making any adjustment on the lathe.

It is dangerous to leave the chuck key in the chuck. Remove it immediately after use. (Fig 1)

Single point tools are sharp and dangerous. Be extra careful when using them.

Chips are sharp and dangerous. Never remove them with your bare hands. Use a chip rake or brush.

You must always know where the emergency stop switch is.

Fig 1

After work

Clean the lathe with a brush and wipe with cotton waste.

Oil the bed ways and lubricating points.

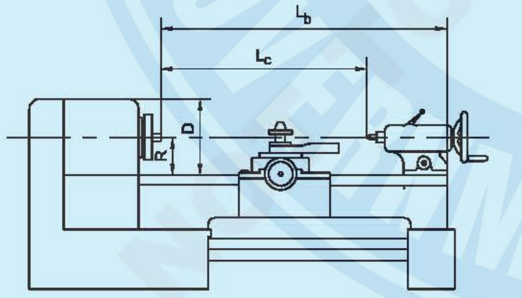
Clean the surroundings of the lathe, wipe the dirt and coolant and remove the swarf.

Specification of a centre lathe

Objective: At the end of this lesson you shall be able to
- **specify a centre lathe.**

Specification of a lathe (Fig 1)

Fig 1



R-RADIUS OF WORK (CENTRE HEIGHT OF LATHE)
D-DIAMETER OF WORK (SWING) L_C-LENGTH BETWEEN CENTRES L_b-LENGTH OF THE BED

A lathe is to be specified by the following.

The maximum diameter of a work that can be held.

The swing over bed. This is the perpendicular distance from the lathe axis to the top of the bed.

The length of the bed. The length of the bed-ways.

The maximum length of work that can be turned between centres.

The range of threads that can be cut. The capacity of the lathe. The swing over carriage.

The value of each division on the graduated collars of the cross-slide and compound slide.

Range of spindle speeds.

Range of feeds.

Size of the spindle bore.

Type of spindle nose.

The specifications help in communication between the seller and the buyer of the lathe.

It helps the operator of the lathe to decide whether the work in hand can be accommodated for performing the operations.

Constructional features of lathe

Objectives: At the end of this lesson you shall be able to
- **name the main parts of a lathe**
- **state the constructional features of lathe**
- **explain the principle of a lathe.**

Centre lathe is a machine which is used to bring the raw material to the required shape and size by metal removal. This is done by feeding a cutting tool against the direction of rotation of the work.

The machine tool on which turning is carried out is known as a lathe.

Lathe is a machine tool which holds the job in between the centre and rotates the job on its own axis. Due to this quality of holding the job from the centre and rotating the job, it is called centre lathe. Work can be held on a chuck and face plate. Chuck and face plate are mounted on the front of spindle. Cutting tool is fed against work after holding it in the tool post firmly. The work rotates on it own axis and tool is moved parallel to work. When tool moves parallel to axis it produces cylindrical surface and when it rotates at some angle, it produces taper surface.

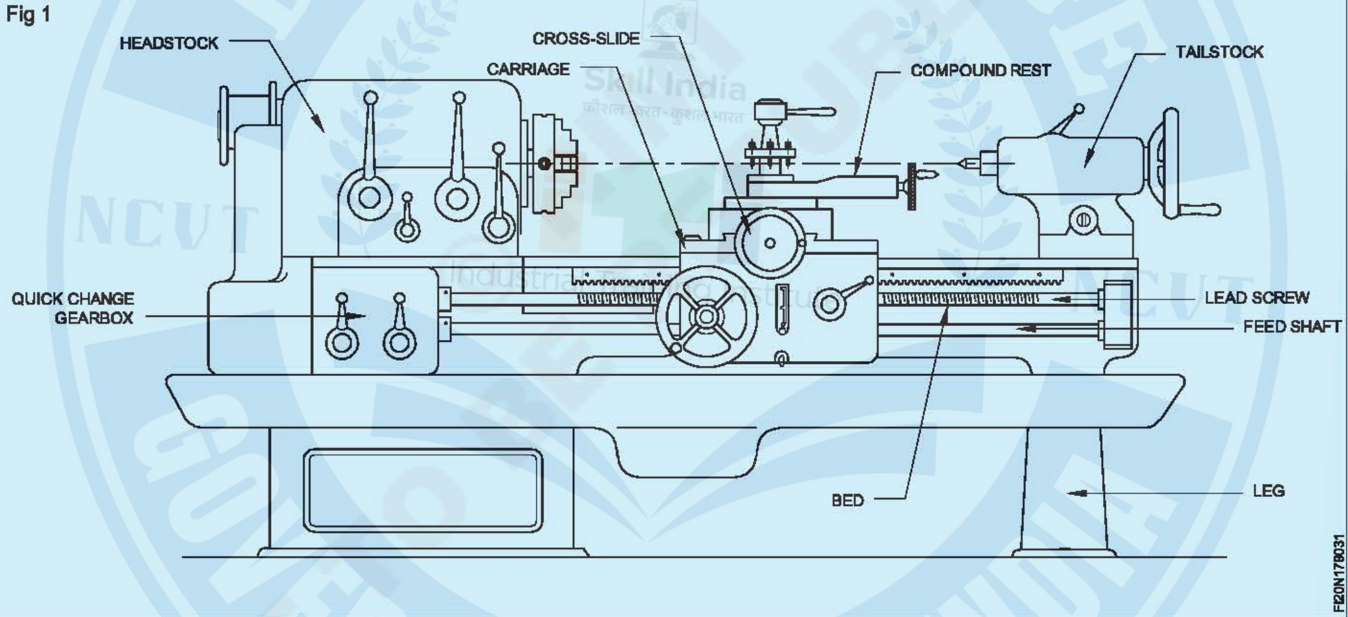
Constructional features of a lathe

A lathe should have provision :

- To hold the cutting tool, and feed it against the direction of rotation.
- To have parts, fixed and sliding, to get a relative movement of the cutting tool with respect to the rotation of the work.
- To have accessories and attachments for performing different operations.

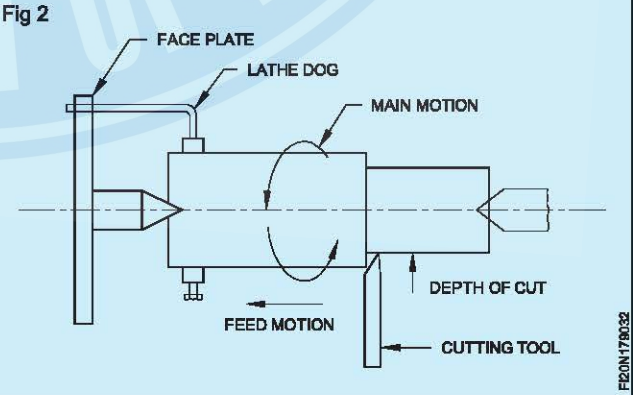
The following are the main parts of a lathe. (Fig 1)

Fig 1



- Headstock
- Tailstock
- Carriage
- Cross-slide
- Compound slide
- Bed
- Quick change gearbox
- Legs
- Feed shaft
- Lead screw

Fig 2



Working principle of Lathe (Fig 2)

Lathe main parts

Objectives: At the end of this lesson you shall be able to
- **name the parts**
- **state the functions of the parts**

Lathe bed

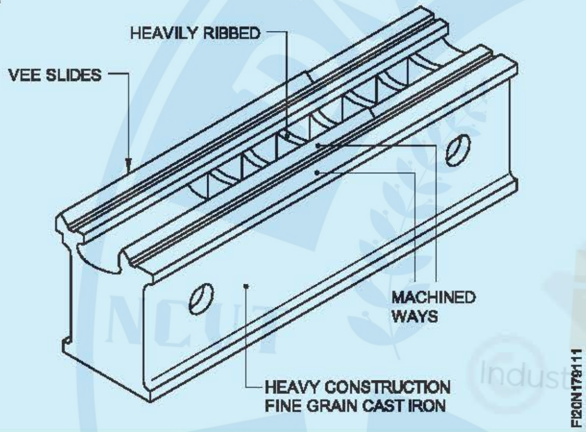
Functions of a lathe bed

The functions of a lathe bed are:

- To locate the fixed units in accurate relationship to each other.
- To provide slide-ways upon which the operating units can be moved.

Constructional features of a lathe bed (Fig 1)

Fig 1



The lathe bed generally consists of a single casting. In larger machines, the bed may be in two or more sections accurately assembled together. Web bracings are employed to increase the rigidity. For absorbing shock and vibration, the beds are made heavy.

A combined swarf and coolant tray is provided on lathes. This may be an integral part with the lathe bed.

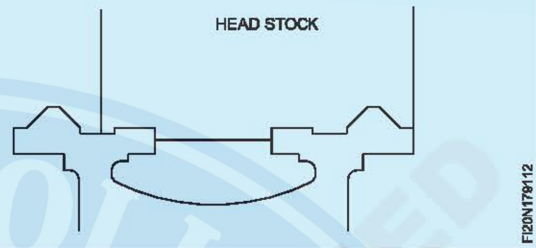
The bed is generally made by cast iron or welded sheet metal legs of box section. This provides the necessary working height for the lathe. Very often the electrical switch gear unit and the coolant pump assembly are housed in the box section of the legs at the headstock end.

Bed-ways (Fig 2)

The bed-ways or slide ways assist in accurate location and sliding of the accessories/parts mounted on this.

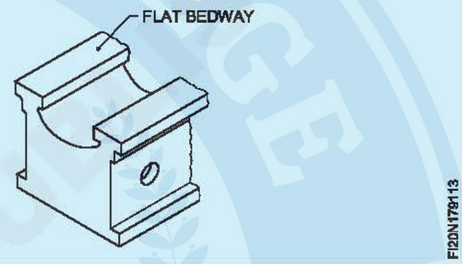
The bed-ways are of three types.

Fig 2



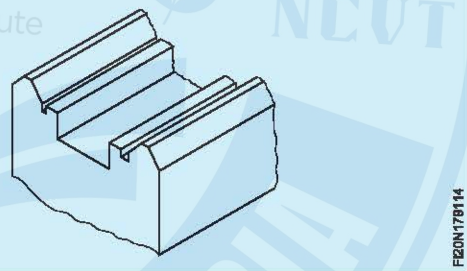
Flat bed-way (Fig 3)

Fig 3



'V' bed way (Fig 4)

Fig 4



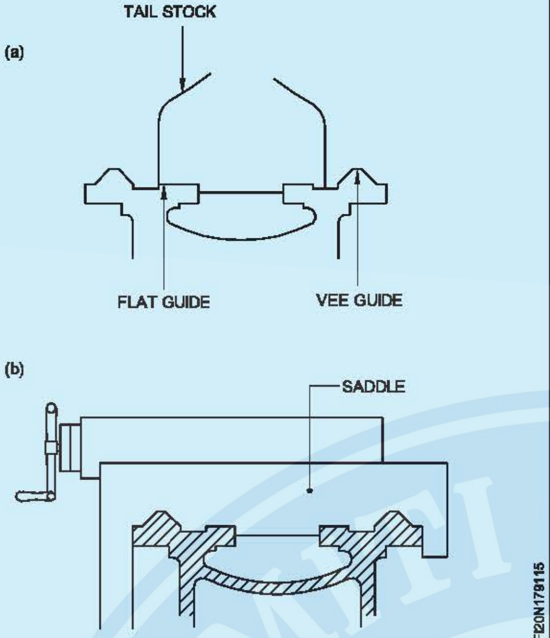
Combination bed way (Figs 5a & 5b)

Normally the bed-ways stop at a distance away from the headstock with a gap at this point. This enables to mount larger diameters of the work.

Some lathes have a detachable section of the bed, which can be fitted when desired, to enable the saddle to operate close to the headstock.

The bed-ways are highly finished by grinding. Some lathes have their bed-ways hand scraped. Some have their bed-ways hardened and ground. The wear-resisting qualities of bearing surfaces are improved by employing chilled iron castings.

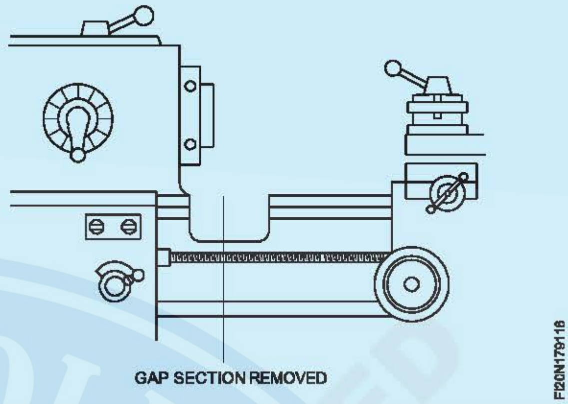
Fig 5



The beds are mostly made up of closely ground, grey cast iron.

Gap bedway (Fig 6)

Fig 6

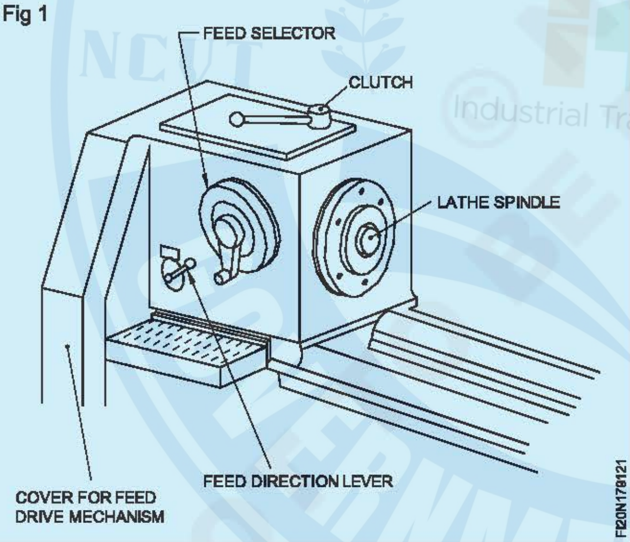


Headstock

Objectives: At the end of this lesson you shall be able to
- **state the function of the headstock**
- **differentiate between cone pulley headstock and all geared headstock.**

Functions (Fig 1)

Fig 1



To provide a means to assemble the work-holding devices. Transmit the drive from the main motor to the work.

To accommodate shafts, gears and levers for a wide range of varying work speeds.

To ensure arrangement for lubricating the gears, shafts and bearings.

Types of headstocks

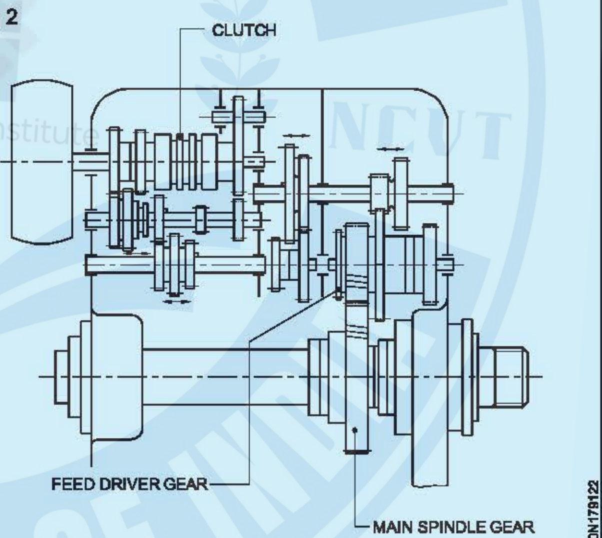
The following are the two types of headstocks.

1 All geared headstock.

2 Cone pulley headstock.

All geared headstock (Fig 2)

Fig 2



It is a box section casting having a removable top cover. It has internal webs for stiffening, and to take shaft bearings. It has an input shaft which is connected by means of 'V' belts to the main motor, and it runs at a constant speed. It is equipped with clutches and a brake.

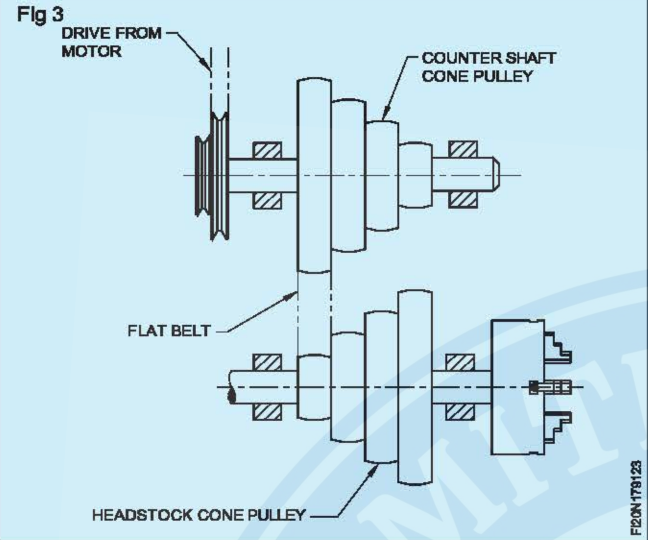
There may be two or more intermediate shafts on which sliding gears are mounted. The main spindle is the last driven shaft in the headstock assembly. The nose of the spindle is outside the headstock casting, and is designed to accommodate the work-holding devices.

The levers operating the forks for the sliding gears are situated outside in front of the headstock casting.

In the all-geared headstock, lubricating oil is filled for splash lubrication of the internal gears. A sight glass with an oil level mark is provided to see the oil level.

Cone pulley headstock (Fig 3)

Fig 3



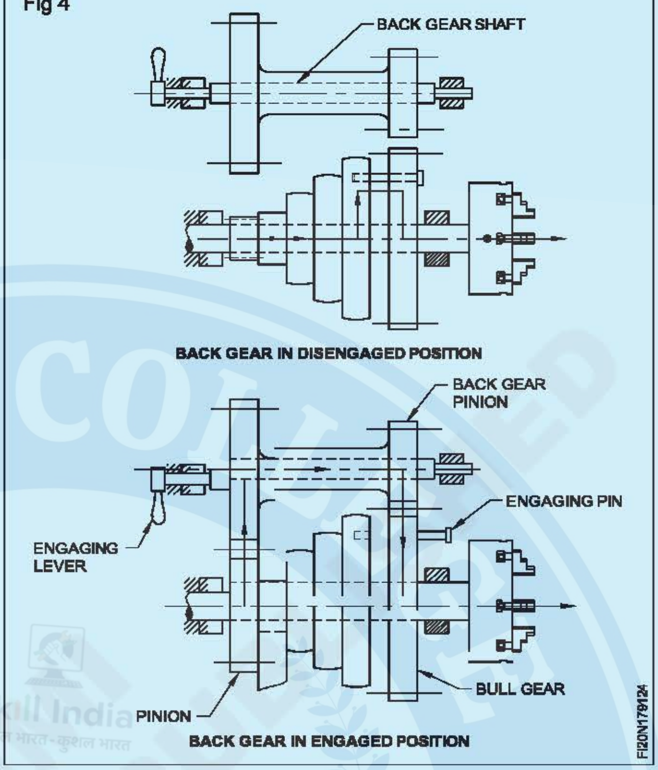
It has a stepped cone pulley mounted on the main spindle, and it is free to revolve. It is connected by means of a flat belt to a similar cone pulley, with steps arranged in the reverse order. This cone pulley gets the drive from the main motor.

The spindle is mounted on the bearing on the headstock casting and has a gear wheel called 'bull gear' keyed to it. A pinion is coupled to the cone pulley.

The back gear unit has a shaft which carries a gear and a pinion. The number of teeth of the gear and pinion on the back gear shaft corresponds to the number of teeth on the bull gear and pinion on the cone pulley. The axis of the back gear shaft is parallel to the axis of the main spindle. The back gear is engaged or disengaged with the cone pulley system by means of a lever. The back gear unit is engaged for reducing the spindle speeds. (Fig 4)

A three-stepped cone pulley headstock provides 3 direct ranges of speeds through a belt connection. With the back gear in engagement, 3 further ranges of reduced speeds can be obtained.

Fig 4



Advantages

- Can take up heavy load.
- Less noise during working.
- Easy to maintain.

Disadvantages

The number of spindle speeds is limited to the number of steps in the cone pulley.

It takes time to change the spindle speeds.

Carriage

Objectives: At the end of this lesson you shall be able to
- **state the functions of a carriage**
- **name the parts of a carriage.**

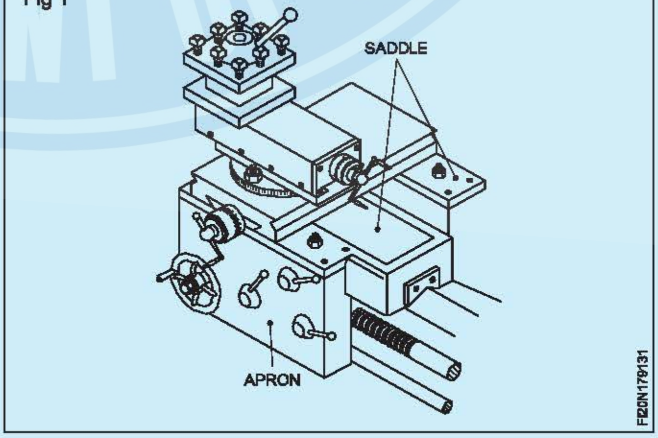
Carriage is the feature of a lathe that provides the method of holding and moving the cutting tool. (Fig 1) It can be locked at any desired position on the lathe bed. It consists of two major parts namely, apron and saddle.

Apron (Fig 2)

The apron is bolted to the front of the saddle. It contains mechanism for moving and controlling the carriage. The main parts of an apron are :

- Traversing hand wheel
- Feed lever
- Feed selector
- Lead screw engagement lever.

Fig 1



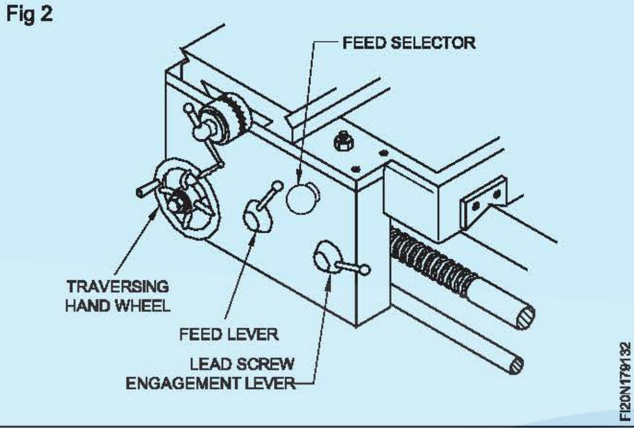


Fig 2

Saddle (Fig 3)

It is a 'H' shaped casting having 'V' guide grooves at the bottom face, corresponding to the lathe bed-ways for mounting on the lathe bed and for sliding.

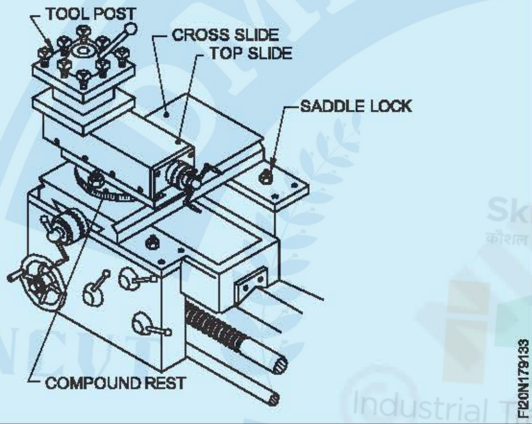


Fig 3

Parts of a saddle

Cross-slide

The cross-slide is mounted on the top of the saddle, and it provides cross movement for the tool. This is fitted at right angles to the bed and is moved by means of a screwed spindle, fitted with a handle. A graduated collar, mounted on the screw rod along with the hand wheel, helps to set the fine movements of the cross- slide.

Compound rest

The compound rest is fitted on the top and to the front of the cross-slide. The compound rest can be swiveled horizontally through 360°.

Top slide

The top slide is fitted on the top of the compound rest. It supports the tool post which holds the cutting tool. The top slide provides a limited horizontal movement for the cutting tool.

By swiveled the compound rest, the top slide can be set at an angle to the cross-slide (Fig 4). Usually the compound rest is set in such a way that the top slide is at right angles to the cross-slide.

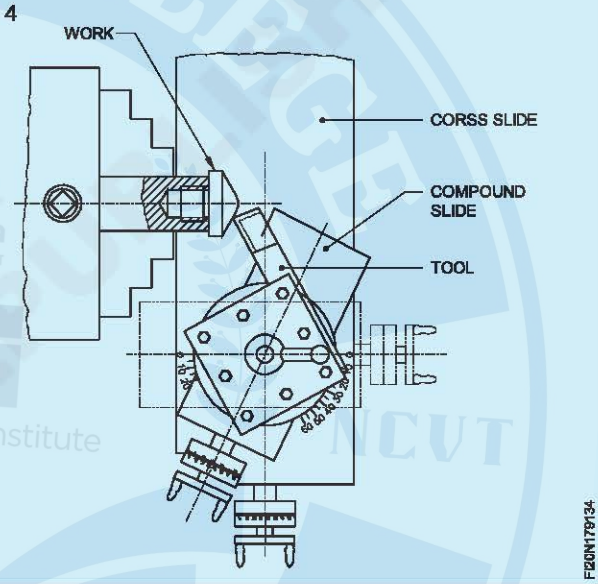


Fig 4

Tailstock

Objectives: At the end of this lesson you shall be able to
- **list the parts of a tailstock**
- **state the uses of a tailstock**
- **explain the function of a tailstock.**

Tailstock

It is a sliding unit on the bed-ways of the lathe bed. It is situated on the right hand side of the lathe. It is made in two parts namely the `base' and the `body'. The base bottom is machined accurately and has `V' grooves corresponding to the bed-ways. It can slide over the bed and can be clamped at any position on the bed by means of the clamping unit. The body of the tailstock is assembled to the base. Graduations are marked on the rear end of the base and a zero line is marked on the body.

When both zero lines coincide, the axis of the tailstock is in line with the axis of headstock.

The body and base are made out of cast iron. The parts of a tailstock are: (Fig 1)

a Base

b Body

c Spindle (barrel)

d Spindle locking lever

e Operating screw rod

f Operating nut

g Tailstock hand wheel

h Key

i Set screw/set over screw

j Clamping bolt

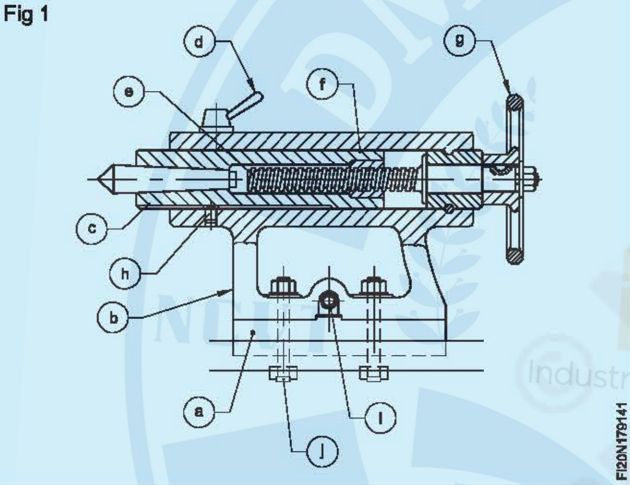
Functioning of a tailstock

By rotating the hand wheel, the barrel can be moved forward or backward. The barrel can be locked in any required position. The hollow end of the barrel at the front is provided with a Morse taper to accommodate the cutting tools with a taper shank. Graduations are sometimes marked on the barrel to indicate the movement of the barrel. With the help of the adjusting screws, the body can be moved over the base laterally, and the amount of movement may be read approximately referring to the graduations marked. This arrangement is to offset the centre of the tailstock as required for taper turning.

Purpose of the tailstock

To accommodate the dead center to support lengthy work to carry out lathe operations. (Fig 2)

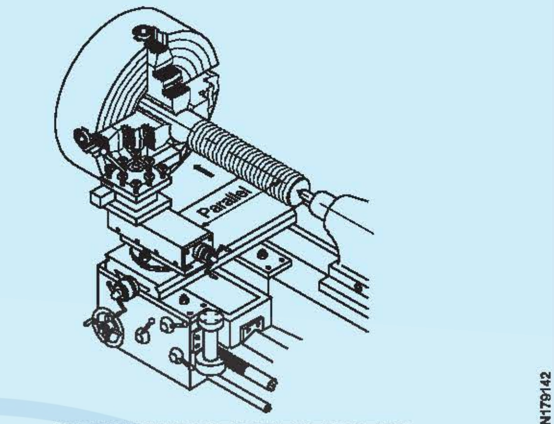
Fig 1



To hold cutting tools like drills, reamers, drill chucks provided with taper shank. (Fig 3)

To turn external taper by offsetting the body of the tailstock with respect to the base. (Fig 4)

Fig 2



LONG WORKPIECE HELD BETWEEN CENTRES

Fig 3

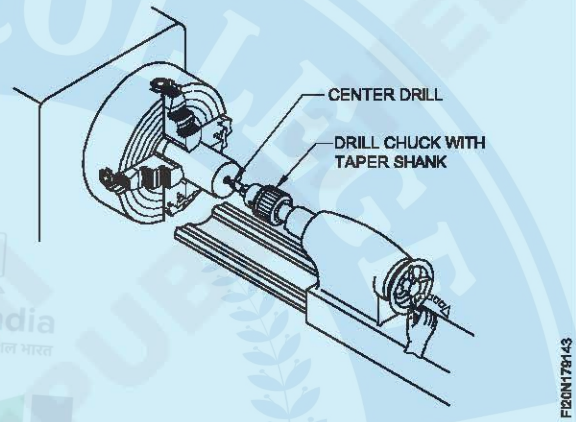
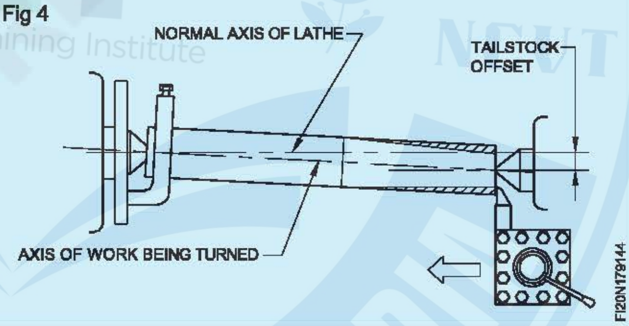


Fig 4

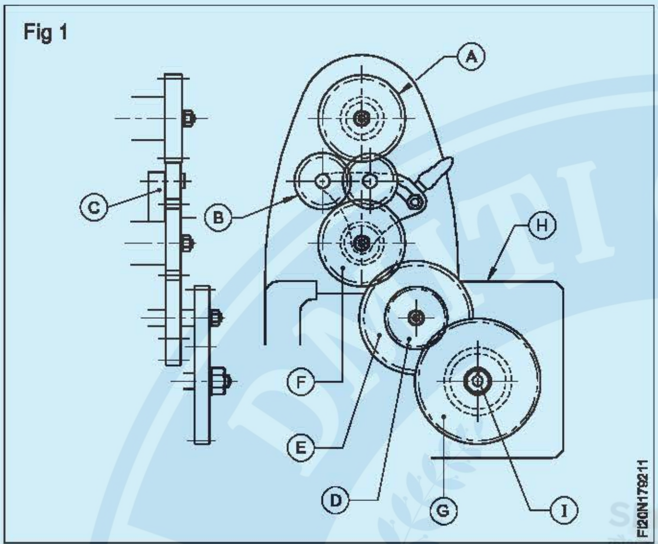


Feed & thread cutting mechanism

Objectives: At the end of this lesson you shall be able to
- **name the parts of the feeding mechanism**
- **state the functional features of the feeding mechanism.**

Feed mechanism (Fig 1)

Fig 1



The feed mechanism of a lathe enables automatic feeding for the tool longitudinally and transversely as needed. By automatic feeding the finish on the work will be better, the feeding of the tool will be at a uniform continuous rate and it takes less time to finish the operation while manual labour is avoided.

The feed mechanism comprises the following.

- Spindle gear (A)
- Tumbler gear unit (B)
- Fixed stud gear (C)
- Change gear unit (DEFG)
- Quick change gear box (H)
- Feed shaft / Lead screw (I)
- Apron mechanism (Fig 5)

The proportionate tool movement for each revolution of work is achieved through all the above units of the feed mechanism.

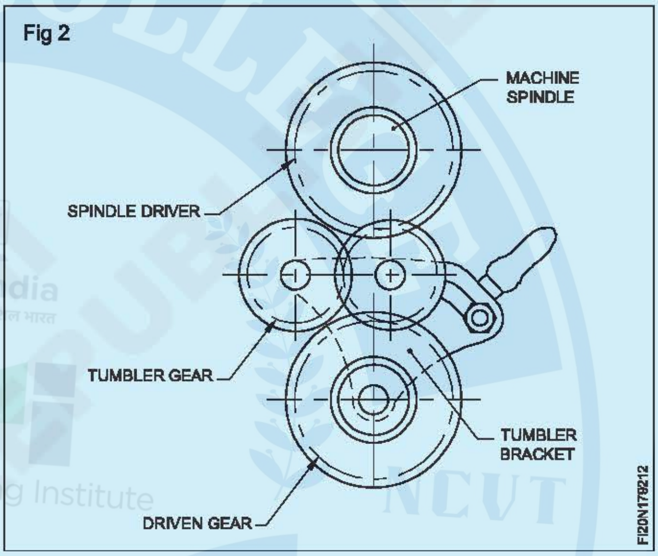
Spindle gear

The spindle gear is fitted to the main spindle, and it is outside the headstock casting. It revolves along with the main spindle.

Tumber gear unit

The tumbler gear unit set of three gears, having the same number of teeth and it connects the spindle gear to the fixed gear. It is also called the reversing gear unit as it is used to change the direction of feed of the tool for the same direction of rotation of the spindle. It can be engaged and disengaged with the fixed stud gear by the operation of the hand lever provided in the unit. (Fig 2)

Fig 2



The fixed stud gear

The fixed stud gear gets the drive from the main spindle gear through the tumbler gear unit and runs at the same number of revolutions per minute as the spindle gear on most lathes.

Change gear unit

The fixed stud gear transmits its drive through a change gear unit to the quick change gear box. The change gear unit has provision for changing the driver, the driven and the idler gears from the set of change gears available for the purpose of feed changing as an additional unit. (Fig 3)

Quick change gear box

The quick change gear box is provided with levers outside the box casting, and by shifting the levers, different gears are brought in mesh so that different feed rates can be given to the tool. A chart listing the different feed rates for the different positions of the levers is fixed to the casting, and by referring to the table, the levers may be engaged in position for the required feed rate. (Fig 4)

Fig 3

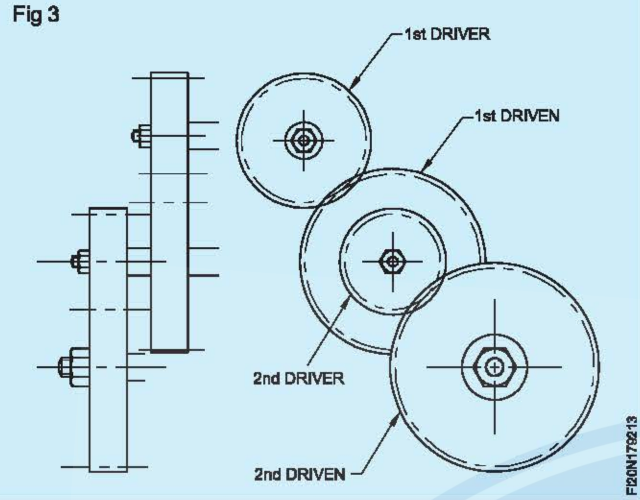
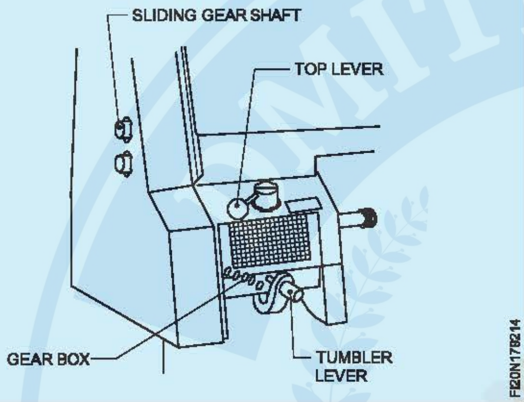


Fig 4



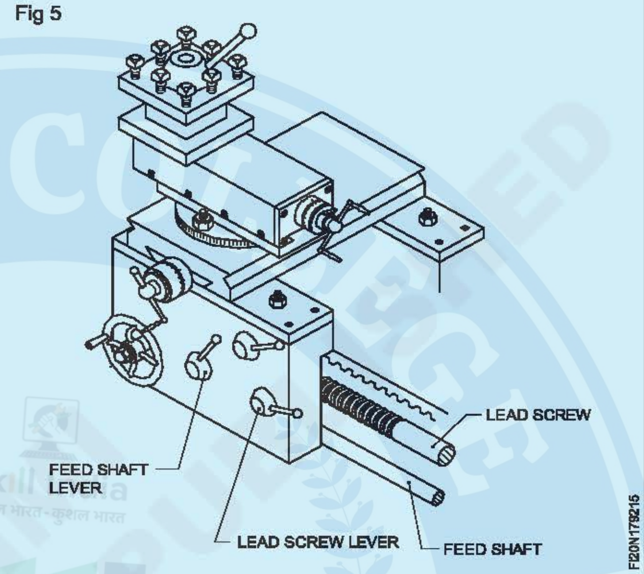
The feed shaft

The feed shaft gets its drive from the quick change gear box, and through the apron mechanism, the rotary movement of the feed shaft is converted into the linear movement of the tool.

The apron mechanism

The apron mechanism has the arrangement for transmitting the drive from the feed shaft to the saddle for longitudinal movement of the tool or to the cross-slide for the transverse movement of the tool. (Fig 5)

Fig 5



Thread cutting with simple and compound gear trains

Objective: At the end of this lesson you shall be able to
- **thread cutting with simple and compount gear trains.**

Change gear train

Change gear train is a train of gears serving the purpose of connecting the fixed stud gear to the quick change gearbox. The lathe is generally supplied with a set of gears which can be utilized to have a different ratio of motion between the spindle and the lead screw during thread cutting. The gears which are utilized for this purpose comprise the change gear train.

The change gear tran consists of driver and driven gears and idler gears.

Simple gear train

A simple gear train is a change gear train having only one driver and one driven wheel. Between the driver and the driven wheel, there may be an idler gear which does not affect the gear ratio. Its purpose is just to link the driver and the driven gears, as well as to get the desired direction to the driven wheel.

Fig 1 shows an arrangement of a simple gear train.

Fig 1

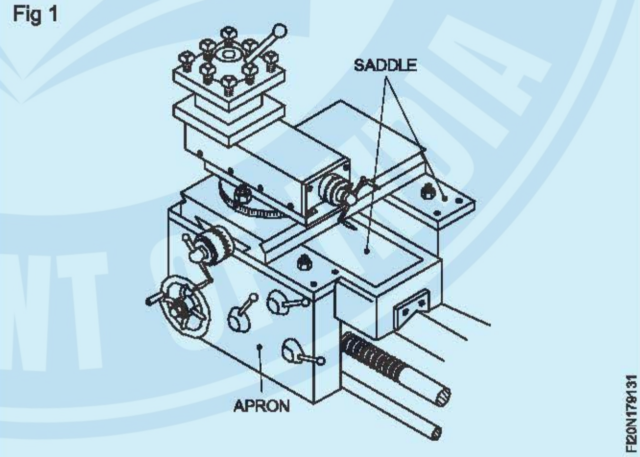
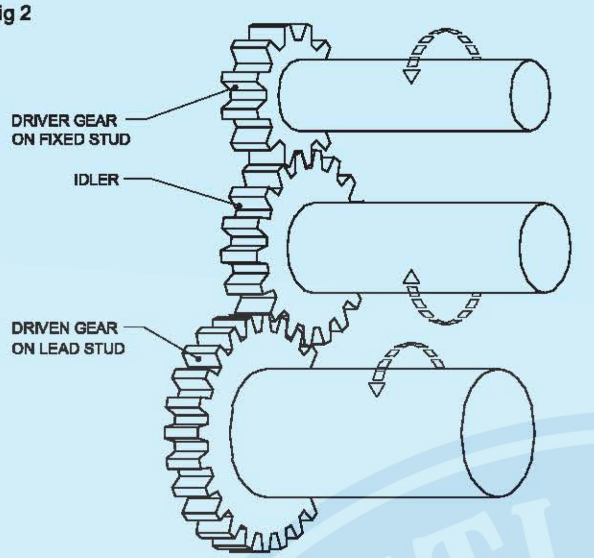


Fig 2 shows mountings of the driver and driven gears in a lathe.

Fig 2



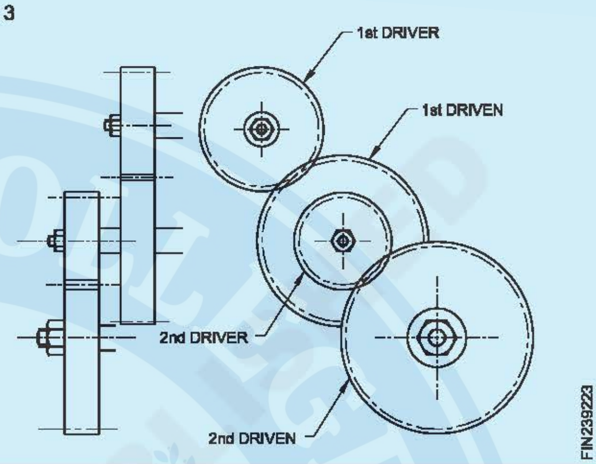
The driver gear and the driven gear are changed according to the pitch of the thread to be cut on the job.

Compound gear train

Sometimes, for the required ratio of motion between the spindle and the lead screw, it is not possible to obtain one driver and one driven wheel. The ratio is split up and then the change gears are obtained from the available set of gears which will result in having more than one driver and one driven wheel. Such a change gear tain is called a compound gear train.

Fig 3 shows the arrangement of a compound gear train.

Fig 3



Holding the job between centre and work with catch plate and dog

Objectives: At the end of this lesson you shall be able to
- **preparing work for turning between centre**
- **to set the catch plate**
- **working with catch plate and dog**

Turning work in-between centres avoids the need for truing the work. The work turned will be parallel through-out. But it requires great skill to perform operations especially like knurling, thread cutting, undercutting. It is limited to external operations only. The work needs the following preparations to be carried out before the actual operations are to be performed.

Face both sides of the work, and maintain the total length accurately within limits.

Choose the correct size and type of centre drill and do centre drilling at both ends.

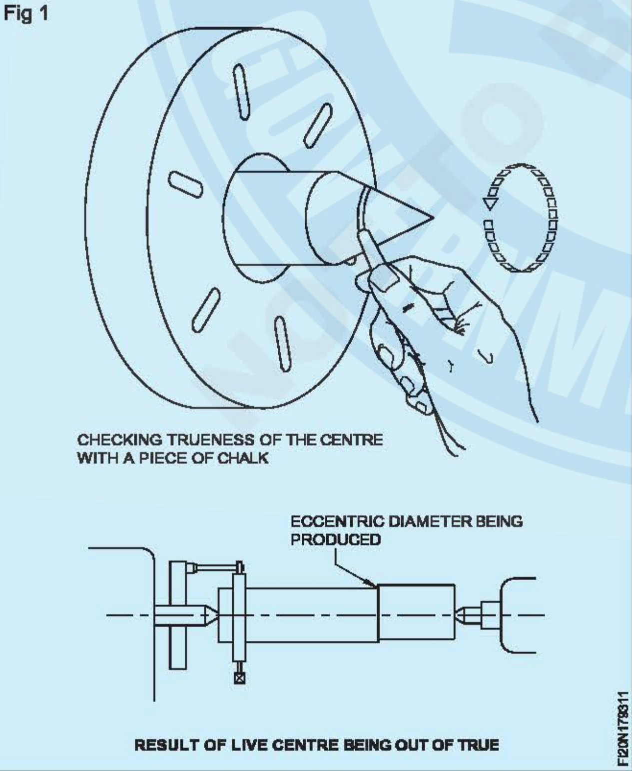
Diamantle the chuck from the spindle nose and assemble the driving plate or catch plate.

Assemble the spindle sleeve to the spindle nose and fix live centre to the sleeve.

Ensure that the spindle sleeve and live centre are free from damages, burrs and are thoroughly cleaned before assembly.

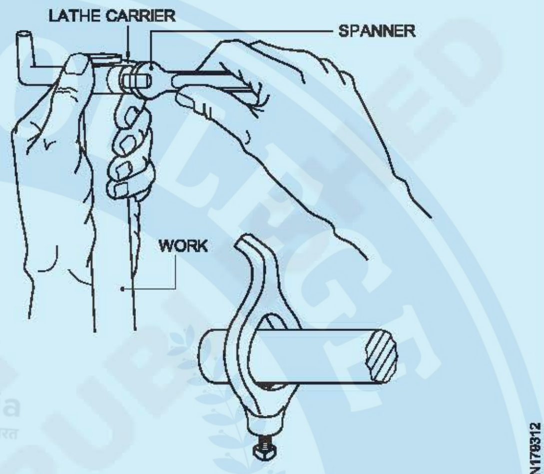
Check for the true running of the live centre. (Fig 1)

Fig 1



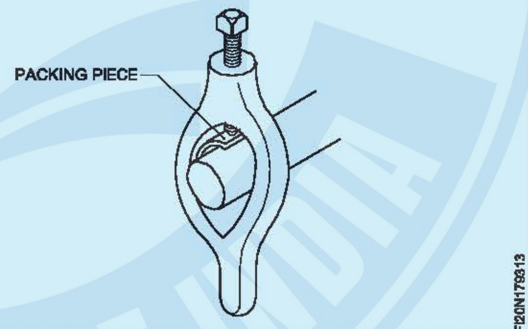
Select a suitable lathe carrier according to the diameter of the work and fasten it on one end of the work with the bent tail pointing outwards. (Fig 2)

Fig 2



Work that has a finished surface should be protected by inserting a small sheet of copper or brass between the end of the screw in the carrier and the work. (Fig 3)

Fig 3



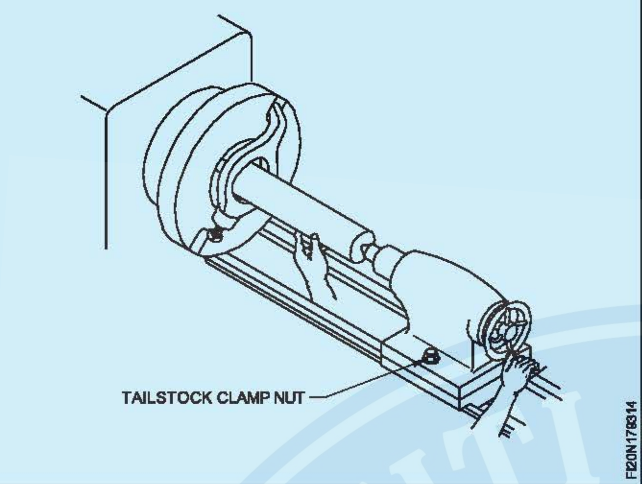
Apply a suitable lubricant (soft grease) to the centre hole of the workpiece to be engaged by the tailstock dead centre.

Move the tailstock to a position on the bed to suit the length of the workpiece. The tailstock spindle should extend approximately 60 to 100 mm beyond the tailstock.

Ensure there is sufficient space for the saddle to operate before clamping the tailstock to the bed.

Clamp the tailstock in position by tightening the tailstock clamp nut. (Fig 4)

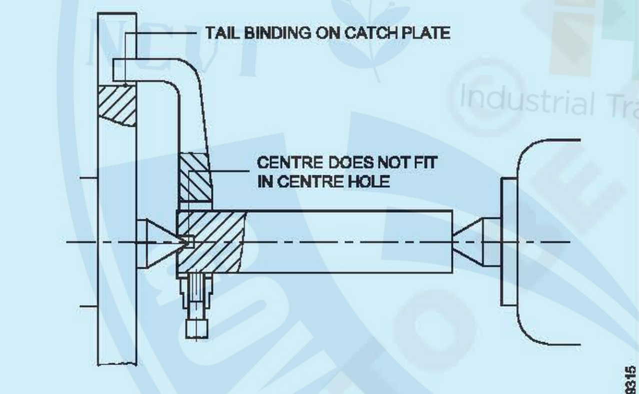
Fig 4



Engage the work-centre hole with the point of live centre and with the tail of the lathe carrier in the slot in the catch plate. Hold the work in this position with hand.

Ensure that the tail of the lathe carrier does not rest on the bottom of the slot in the driving plate. This will not permit the centre entering the centre hole of the work for proper seating. (Fig 5)

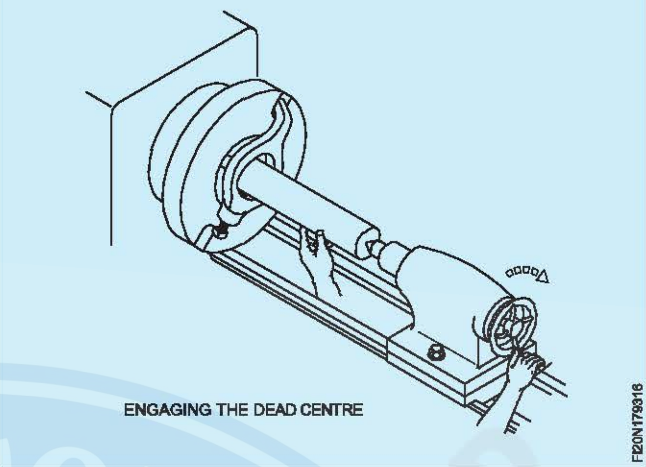
Fig 5



A LATHE CARRIER BINDING IN THE DRIVE PLATE CAUSES INACCURACIES SINCE THE WORK IS FORCED OFF THE LATHE CENTRE

Advance the tailstock spindle by the hand wheel rotation until the point of dead centre enters the centre hole of the work with proper seating eliminating all endwise movement. (Fig 6)

Fig 6



ENGAGING THE DEAD CENTRE

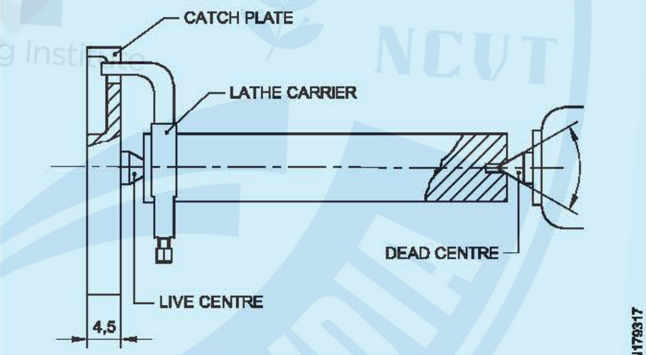
Move the tail of the carrier back and forth. At the same time adjust the hand wheel until only a slight resistance is felt.

Tighten the tailstock spindle clamp at this position and check that the resistance does not change. Set the machine for about 250 r.p.m. and allow the work to run for a few seconds.

Check once again for the resistance and adjust the tailstock spindle, if needed.

Work is now ready for operations. (Fig 7)

Fig 7



WORK HELD BETWEEN CENTRES

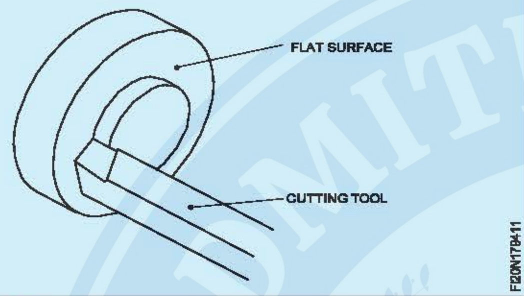
Before holding the work in between centres ensure that the centres are aligned.

Simple description of facing and roughing tool

Objectives: At the end of this lesson you shall be able to
- **state the purpose of facing**
- **setting the rough facing tool**
- **state the reasons for the defects**
- **state the remedies to overcome the defects in facing**

Facing: This is an operation of removing metal from the work-face by feeding the tool at right angles to the axis of the work. (Fig 1)

Fig 1



Purpose of facing

- To have a reference plane to mark and measure the step lengths of the work.
- To have a face at right angle to the axis of the work.
- To remove the rough surface on the faces of the work and have finished faces instead.
- To maintain the total length of the work.

Facing may be rough or finish facing. Rough facing is done to remove the excess metal on the face of the work by coarse feeding with more depth of cut, leaving sufficient metal for finishing. Rough facing is done by feeding the tool from the periphery of work towards the centre of the work. Finish facing is the operation to have a smooth face by removing the rough surface produced by the rough facing.

Finish facing is done by feeding the tool from the centre of the work towards the periphery. (Figs 2a and 2b)

Rough facing is done by choosing a spindle RPM according to the average diameter of the work, the recommended cutting speed, with a coarse feed and more depth of cut.

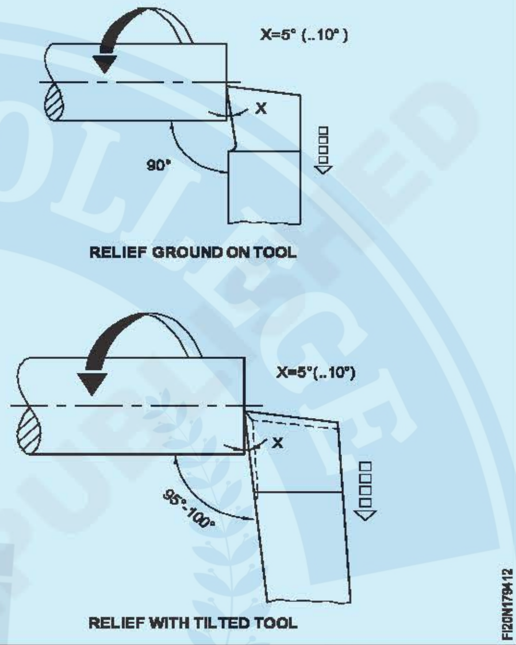
Finish facing is done by choosing a cutting speed about twice that of the cutting speed for roughing, with a fine feed rate of 0.05 mm approximately and with a depth of cut of not more than 0.1 mm.

The following are the defects found in facing work (Fig 3)

A concave face: This is caused by the tool digging into the work during the feeding as the tool is not clamped rigidly. By clamping the tool rigidly with minimum overhang, this defect can be avoided.

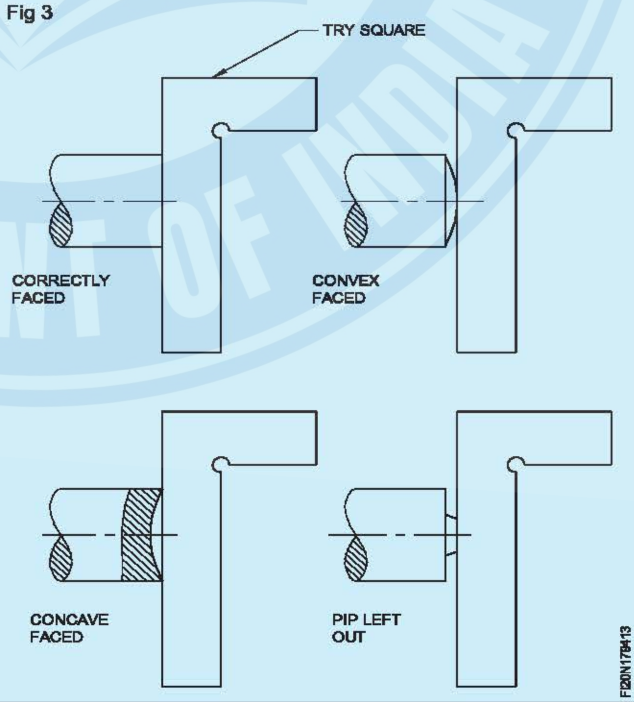
A convex face: This is caused by the blunt cutting edge of the tool and the carriage not being locked. To avoid this defect, re-sharpen the tool and use it; Also lock the carriage to the bed of the lathe.

Fig 2



A pip left in the centre : This is due to the tool not being set to the correct centre height. By placing the tool to the centre height, this defect can be avoided.

Fig 3



Nomenclature of single point cutting tools and multi point cutting tools

Objectives: At the end of this lesson you shall be able to
- **name the types of cutting tool**
- **state the nomenclature of single point cutting tools**
- **state the nomenclaure of multi point cutting tools**

Lathe cutting tools are divided into two groups. These are

1 Single point cutting tools

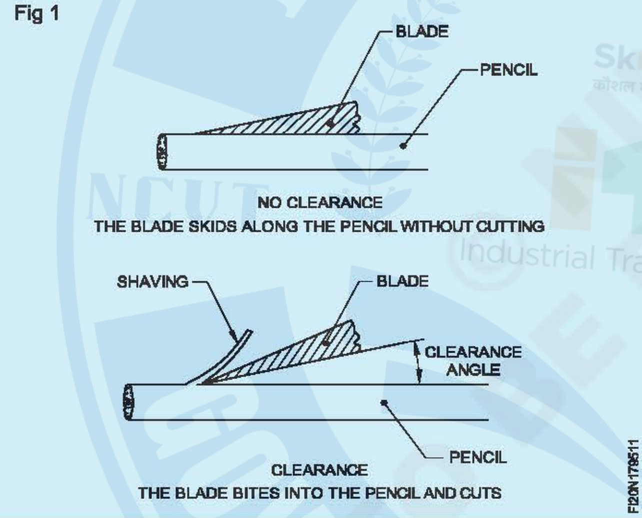
2 Multi point cutting tools

Single point cutting tool nomenclature

The tool acts like a wedge during turning. The wedge shaped cutting edge penetrates into the work and removes the metal. This necessitates the grinding of a tool cutting edge to a wedge shape.

When we sharpen a pencil with a pen knife by trial and error, we find that the knife must be presented to the wood at a definite angle, if success is to be achieved. (Fig 1)

Fig 1



If, in the place of a wooden pencil, a piece of soft metal such as brass is cut, it will be found that the cutting edge of the blade soon becomes blunt, and the cutting edge gets crumbled. For the blade to cut the brass successfully, the cutting edge must be ground to a less acute angle. (Fig 2)

The angle shown in Fig 1 is called as clearance angle and that shown in Fig 2 is a wedge angle.

Angles ground on a lathe cutting tool (Fig 3)

All the angles given below may not be located or found in every tool. As an example a roughing tool is chosen. The angles and clearances ground on this tool are:

1 Approach angle

2 Trail angle

3 Top rake angle

Fig 2

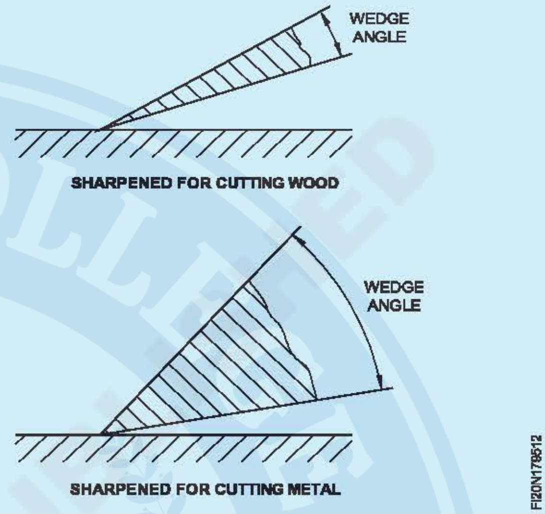
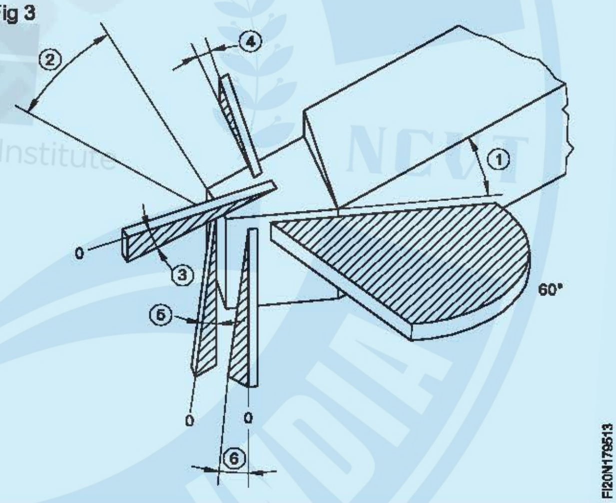


Fig 3



4 Side rake angle

5 Front clearance angle

6 Side clearance angle

Multi point cutting tools used in lathe are:

- Drill
- Reamed
- Tap
- Die

Tool selection based on different requirements

Objectives: At the end of this lesson you shall be able to
- **state the qualities of good cutting tool material**
- **state the factors to be remembered when selecting tool**
- **name the different types of tool**
- **name the shapes of the tool**

Cutting tool materials

Tool materials should be:

- Harder and stronger than the material being cut
- Tough to resist shock loads
- Resistant to abrasion thus contributing to long tool life.

Cutting tool material should possess the following qualities.

- Cold hardness
- Red hardness
- Toughness

Cold hardness

It is the amount of hardness possessed by a material at normal temperature. Hardness is the property by which it can cut/scratch other metals. When hardness increases, brittleness also increases, and a material, which has too much of cold hardness, is not suitable for the manufacture of cutting tools.

Red hardness

It is the ability of a tool material to retain most of its cold hardness property even at very high temperatures. While machining, the friction between the tool and the work, the tool and the chips, causes heat to be generated, and the tool loses its hardness, and its efficiency to cut diminishes. If a tool maintains its cutting efficiency even at increased temperatures during cutting, it can be said that it possesses the red hardness property.

Toughness

The property to resist breakage due to sudden load that results during metal cutting is termed as `toughness' This will reduce the breakage of the cutting edges of tools.

The following factors are to be considered, when selecting a tool material.

- Material to be machined.
- Condition of the machine tool.(rigidity and efficiency)
- The total quantity of production and the rate of production.
- The dimensional accuracy required and the quality of surface finish.
- The amount of coolant applied and method of application.
- Condition and form of material to be machined.

Grouping of tool material

The three groups under which tool materials fall are:

- ferrous tool materials
- non-ferrous tool materials
- non-metallic tool materials.

Ferrous tool materials

These materials have iron as their chief constituent. High carbon steel (tool steel) and high speed steel belong to this group.

Non-ferrous tool materials

These do not have iron, and they are formed by alloying elements like tungsten, vanadium and molybdenum. Stellate belongs to this group.

Carbides

These materials are also non-ferrous. They are manufactured by powder metallurgy technique. Carbon and tungsten are the chief alloying elements.

Non-metallic materials

These tool materials are made out of non metals. Ceramics and diamonds belong to this group.

High carbon steel is the first tool material introduced for manufacturing cutting tools. It has poor red hardness property, and it loses its cutting efficiency very quickly. Alloying elements like tungsten, chromium and vanadium, are used to produce high speed steel tool material. Its red hardness property is more than that of high carbon steel.

High speed steel is used for making solid tools, brazed tools and inserted bits. It is costlier than high carbon steel. Carbide cutting tools can retain their hardness at very high temperatures, and their cutting efficiency is higher than that of high speed steel. Due to its brittleness and cost, a carbide tool cannot be used as a solid tool. It is used as a brazed tool and throw away tool bit.

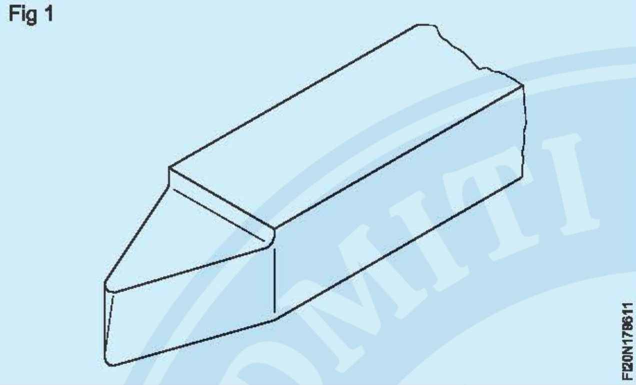
Lathe cutting tool types

The tools used on lathes are

- Solid type tools
- Brazed type tools
- Inserted bits with holders
- Throw-away type tools. (carbide)

Solid tools (Fig 1)

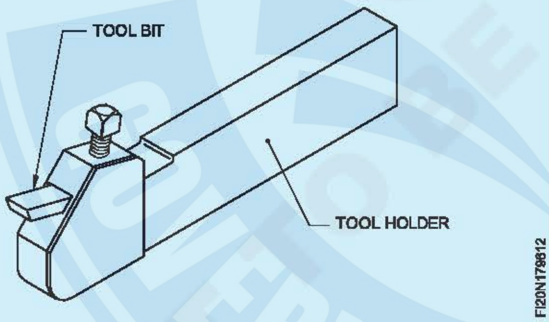
Fig 1



These are tools having their cutting edges ground on solid bits of square, rectangular and round cross-sections. Most of the lathe cutting tools are of the solid type, and high carbon steel and high speed steel tools are used. The length and cross-section of the tool depend upon the capacity of the machine, the type of tool post and the nature of the operation.

Inserted bits with holders (Fig 2)

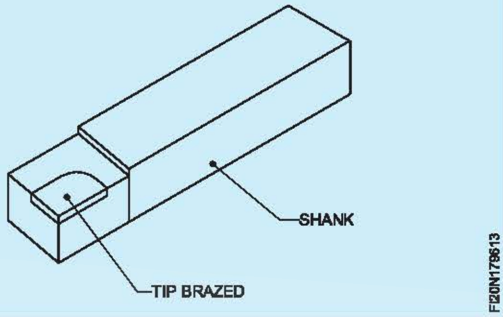
Fig 2



Solid high speed steel tools are costly; hence, they are sometimes used as inserted bits. These bits are small in sizes, and are inserted in the holes of the holder. These holders are held and clamped in the tool posts to carry out the operations.The disadvantage in this type of tools is that the rigidity of the tool is poor.

Brazed tools (Fig 3)

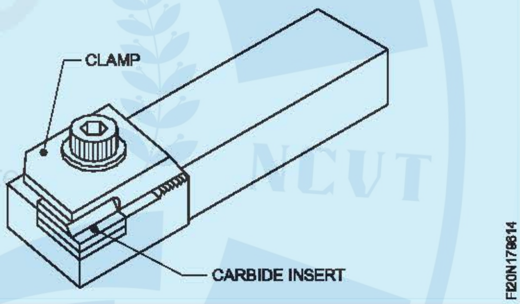
Fig 3



These tools are made up of two different metals. The cutting portions of these tools are of cutting tool materials, and the body of the tools do not possess any cutting ability, and are tough. Tungsten carbide tools are mostly of the brazed type. Tungsten carbide bits of square, rectangular and triangular shape are brazed to the tips of the shank. The tips of the shank metal pieces are machined on the top surface according to the shape of the fits so as to accomodate the carbide bits. These tools are economical, and give better rigidity for the tools than the inserted bits clamped in the tool-holders. This is applicable to high speed steel brazed tools also.

Throw-away type tools (Fig 4)

Fig 4



Carbide brazed tools when blunt or broken need grinding which is time consuming and expensive. Hence, they are used as throw-away inserts in mass production. Special tool- holders are needed and the carbide bits of rectangular, square or triangular shapes are clamped in the seating faces and machined on this type of special holders.

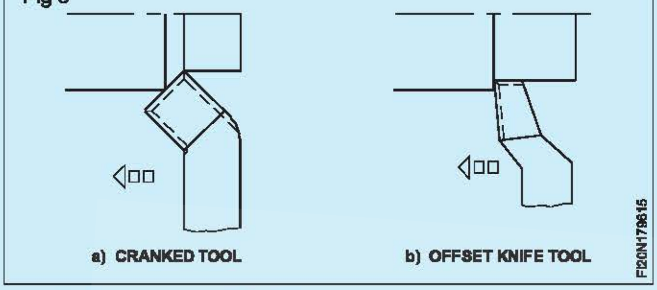
The seating faces are machined in such a way that the rake and clearances needed for the cutting bits are automatically achieved when the bits are clamped.

Lathe cutting tool shapes

Lathe cutting tools are available in a variety of shapes for performing different operations. Some of the lathe cutting tools generally used are:

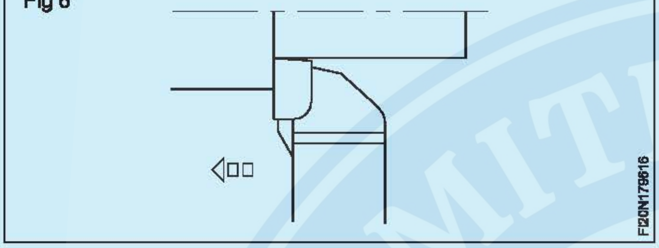
- Facing tool (Figs 5a and 5b)

Fig 5



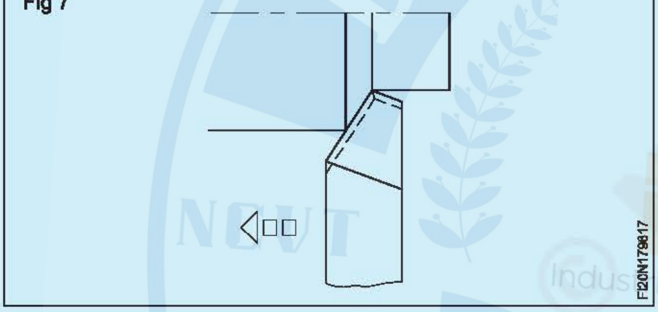
- Knife edge tool (Fig 6)

Fig 6



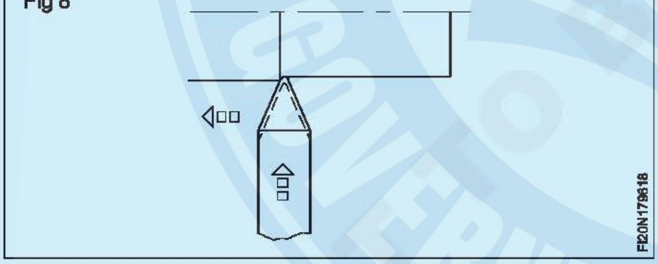
- Roughing tool (Fig 7)

Fig 7



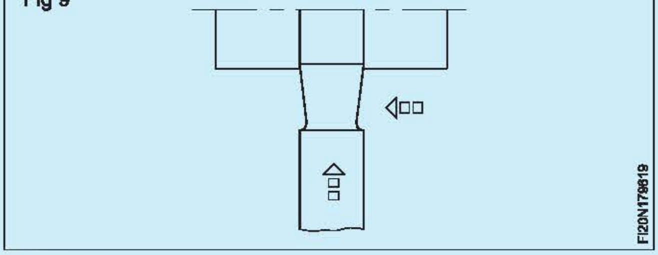
- Round nose finishing tool (Fig 8)

Fig 8



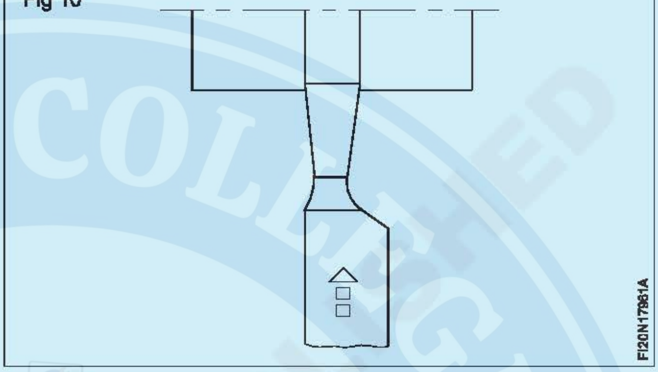
- Broad nose finishing tool (Fig 9)

Fig 9



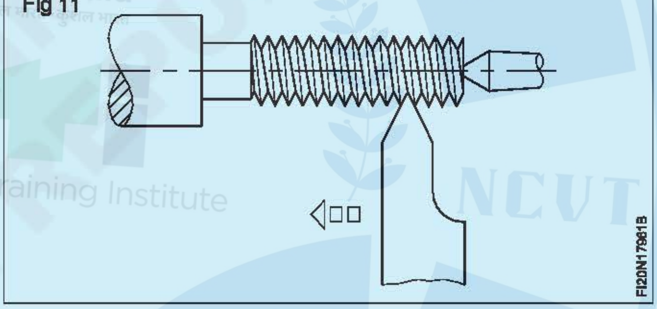
- Undercutting tool/parting off tool (Fig 10)

Fig 10



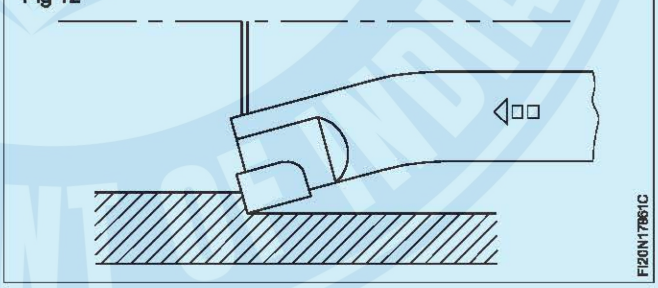
- External threading tool (Fig 11)

Fig 11



- Boring tool (Fig 12)

Fig 12

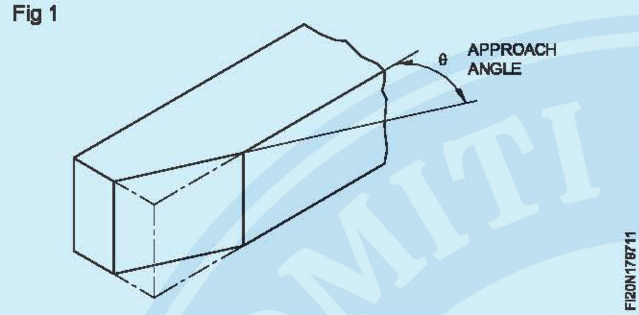


Necessity of tool angles

Objectives: At the end of this lesson you shall be able to
- **name the different angle of the tool**
- **state use of the each angle**
- **state the effect of the incorrect angle.**

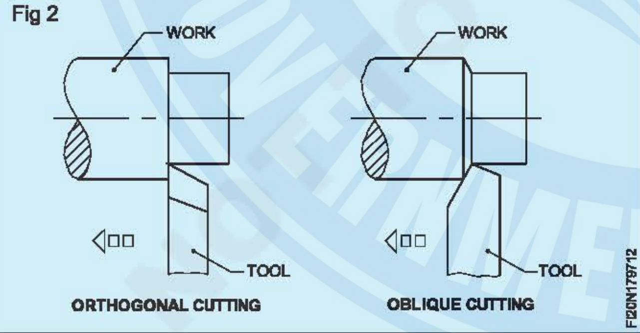
Approach angle (Fig 1)

Fig 1



This is also known as side cutting-edge angle. This is ground on the side of the cutting tool. The cutting will be oblique while cutting. The angle ground may range from 25° to 40° but as a standard a 30° angle is normally provided. The oblique cutting has the advantages over the orthogonal cutting, in which the cutting edge is straight. More depth of cut is given in the case of oblique cutting, since, when the tool is fed to the work, the contact surface of the tool increases gradually as the tool advances, whereas in the case of the orthogonal cutting, the length of the cutting edge for the given depth fully contacts the work from the beginning itself which gives a sudden maximum load on the tool face. The area over which heat is distributed is greater in oblique cutting. (Fig 2)

Fig 2

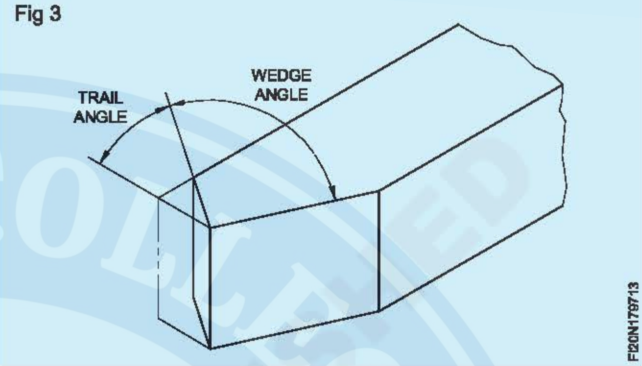


Trail angle (Fig 3)

It is also known as end-cutting edge angle, and is ground at 30° to a line perpendicular to the axis of the tool, as illustrated.

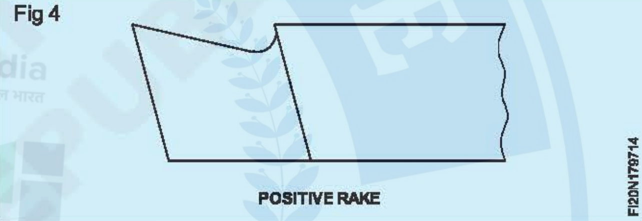
The approach angle and trail angle ground will form the wedge angle of 90° for the tool.

Fig 3



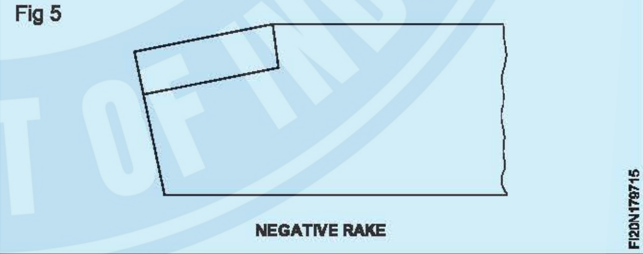
Top or back rake angle (Fig 4)

Fig 4



The rake angle ground on a tool controls the geometry of chip formation. Thereby, it controls the cutting action of the tool. The top or back rake angle of the tool is ground on the top of the tool, and it is a slope formed between the front of the cutting edge and the top of the face. If the slope is from the front towards the back of the tool, it is known as a positive top rake angle, and if the slope is from the back of the tool towards the front of the cutting edge, it is known as a negative back rake angle. (Fig 5)

Fig 5

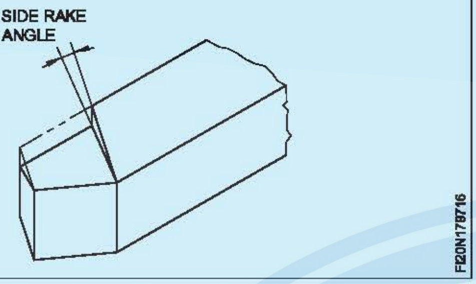


The top rake angle may be ground positive, negative or zero according to the material to be machined. When turning soft, ductile materials, which form curly chips, the positive top rake angle ground will be comparatively more than for turning hard brittle metals.

When turning hard metals with carbide tools, it is the usual practice to give a negative top rake. Negative top rake tools have more strength than tools with positive top rake angles.

Side rake angle (Fig 6)

Fig 6

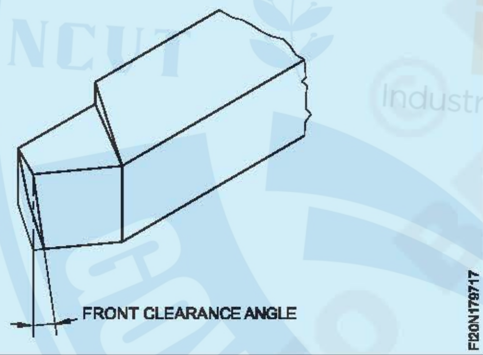


A side rake angle is the slope between the side of the cutting edge to the top face of the tool width wise. The slope is from the cutting edge to the rear side of the tool. It varies from 0° to 20°, according to the material to be machined.

The top and side rake, ground on a tool control the chip flow, and this results in a true rake angle which is the direction in which the chip that shears away from the work passes.

Front clearance angle (Fig 7)

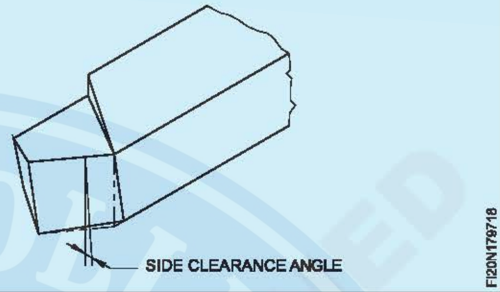
Fig 7



It is the slope between the front of the cutting edge to a line perpendicular to the axis of the tool drawn downwards which is known as the front clearance angle. The slope is from the top to the bottom of the tool, and permits only the cutting edge to contact the work, and avoids any rubbing action. If the clearance ground is more, it will weaken the cutting edge.

Side clearance angle (Fig 8)

Fig 18



The clearance angle is the slope formed between the side cutting edge of the tool with a line perpendicular to the tool axis drawn downwards at the side cutting edge of the tool. The slope is from the top of the side cutting edge to the bottom face. This is also ground to prevent the tool from rubbing with the work, and allows only the cutting edge to contact the work during turning. The side clearance angle needs to be increased when the feed rate is increased.

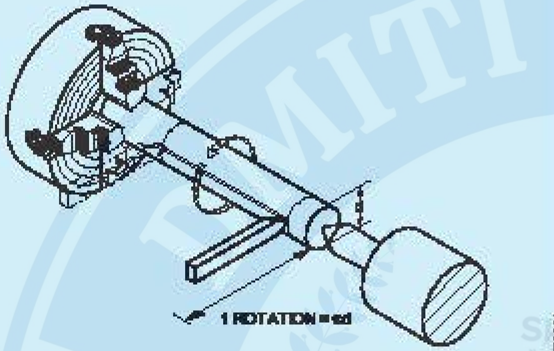
When grinding rake and clearance angles, it is better to refer to the standard chart provided with the recommended values and grind. However, actual operation will indicate the performance of the tool, and will indicate to us, if any modifications are needed for the angles ground on the tool.

Lathe cutting speed and feed, use of coolants, lubricants

Objectives: At the end of this lesson you shall be able to
- **distinguish between cutting speed and feed**
- **read and select the recommended cutting speed for different materials from the chart**
- **point out the factors governing the cutting speed**
- **state the factors governing feed.**

Cutting speed is the speed at which the cutting edge passes over the material, and it is expressed in metres per minute. (Fig 1)

Fig 1



CUTTING SPEED FOR TURNING OPERATION

When a work of a diameter 'd' is turned in one revolution the length of the portion of work in contact with the tool is $\pi \times d$. When the work is making 'n' rev/min, the length of the work in contact with the tool is $\pi \times D \times n$. This is converted into metres and expressed in a formula form as

$$V = \frac{\pi dn}{1000} \text{ metre/min}$$

where

V = cutting speed in m/min.

$\pi = 3.14$

d = diameter of the work in mm.

n = RPM.

When more material is to be removed in lesser time, a higher cutting speed is needed. This makes the spindle to run faster but the life of the tool will be reduced due to more heat being developed. The recommended cutting speeds are given in a chart. As far as possible the recommended cutting speeds are to be chosen from the chart and the spindle speed calculated before performing the operation. (Fig 2) Correct cutting speed will provide normal tool life under normal working condition.

Example

Find out the rpm of a spindle for a 50 mm bar to cut at 25 m/min.

$$V = \frac{\pi dn}{1000} \qquad n = \frac{1000V}{\pi xD}$$

$$\frac{1000x25}{3.14x50} = \frac{500}{3.14} = 159pm$$

Factors governing the cutting speed

- Finish required
- Depth of cut
- Tool geometry
- Properties and rigidity of the cutting tool and its mounting.
- Properties of the workpiece material
- Rigidity of the workpiece
- The type of cutting fluid used.

Feed (Fig 3)

The feed of the tool is the distance it moves along the work for each revolution of the work and it is expressed in mm/rev.

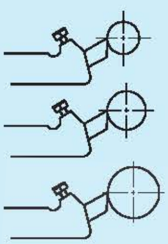
The factors governing the feed are:

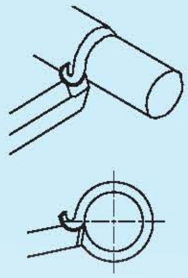
- Tool geometry
- Surface finish required on work
- Rigidity of the tool.

Rate of metal removal

The volume of metal removal is the volume of chip that is removed from the work in one minute, and it is found by multiplying the cutting speed, feed rate and the depth of cut.

Fig 2





Cutting speed 30 m / min	Length of metal passing over cutting tool in one revolution	Calculated RPM of spindle
Ø 25 mm	----------------78.56 mm	1528
Ø 50 mm	----------------157.12 mm	764
Ø 75 mm	----------------235.68 mm	509.3

Relationship of RPM to the cutting speed on different diameters.

Fig 3

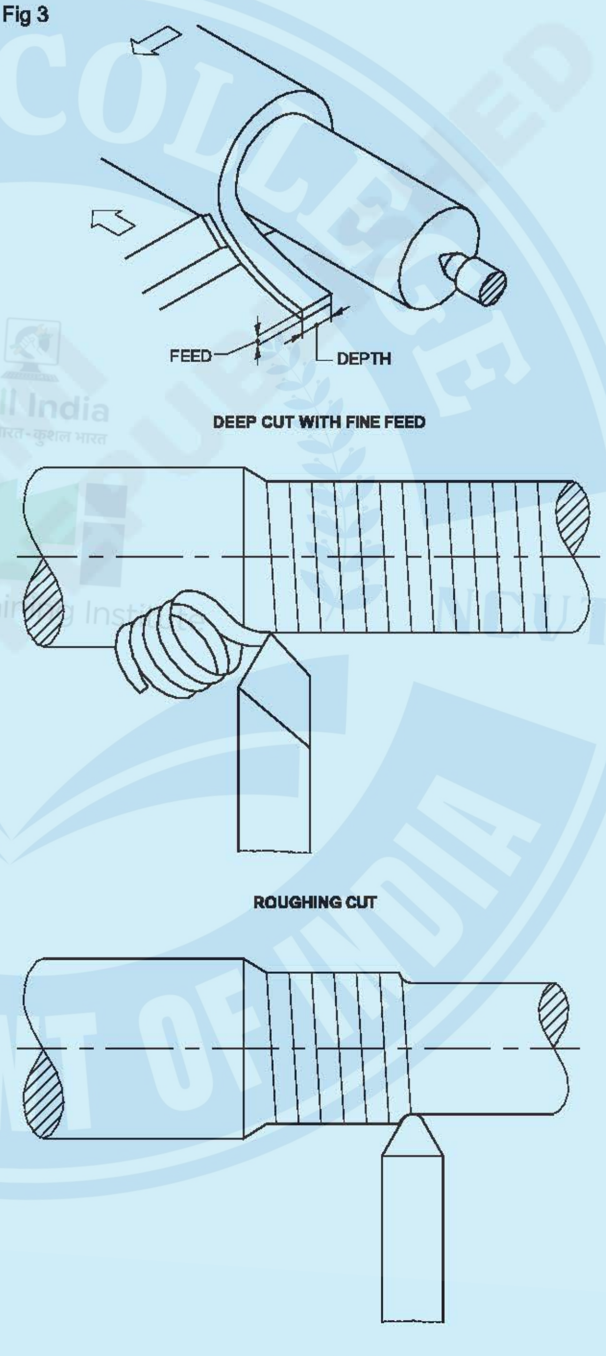


Table 1

Cutting speeds and feeds for H.S.S tool

Material being turned	Feed mm/rev	Cutting speed m/min
Aluminium	0.2-1.00	70-100
Brass (alpha)-ductile	0.2-1.00	50-80
Brass (free cutting)	0.2-1.5	70-100
Bronze (phosphor)	0.2-1.00	35-70
Cast iron (grey)	0.15-0.7	25-40
Copper	0.2-1.00	35-70
Steel (mild)	0.2-1.00	35-50
Steel (medium-carbon)	0.15-0.7	30-35
Steel (Alloy-high tensile)	0.08-0.3	5-10
Thermo-setting plastics	0.2-1.00	35-50

Note

For super HSS tools the feeds should remain the same, but cutting speeds could be increased by 15% to 20%.

A lower speed range is suitable for heavy, roughing cuts. A higher speed range is suitable for light, finishing cuts.

The feed is selected to suit the finish required and the rate of metal removal.

When carbide tools are used, 3 to 4 times higher cutting speed to that required for H.S.S. tools may be chosen.

Comparison of HSS and Carbide Tools

HSS Tool	Carbide Tool
• Ferrous tool material have iron as their chief constituent.	• Non-Ferrous tool material do not have iron.
• Alloying tungsten, chromium and vanadium to high carbon steel, high speed steel tool material is produced.	• Carbide cutting tools can retain their hardness at very high temperature that of high speed steel.
• Cutting speed is low.	• Cutting speed is high.
• Solid tool.	• It is a brazed tool bit and throw away tool bit die to brittleness.
• Cost low.	• Cost high.

Coolants & lubricants (Cutting fluids)

Objectives: At the end of this lesson you shall be able to
- **state the properties of cutting fluids**
- **state the purpose of using a cutting fluid**
- **name the different cutting fluids**
- **distinguish the characteristics of each type of cutting fluids**
- **select a proper cutting fluid to suit various materials and machining operations.**

Coolants (Cutting fluids)

Coolants (Cutting fluids) play an important role in reducing the wear of cutting tools.

Coolants (Cutting fluids) are essential in most metal cutting operations. During a machining process, considerable heat and friction are created by the plastic deformation of metal occurring in the shear zone when the chip slides along the chip tool interface. This heat and friction cause the metal to adhere to the cutting edge of the tool, and the tool may break down. The result is poor finish and inaccurate work.

The advantages of a cutting fluid is it :

- Cools the tool and the workpiece
- Lubricates the chip / tool interface and reduces the tool wear due to friction
- Prevents chip welding
- Improves the surface finish of the workpiece
- Flushes away the chips
- Prevents corrosion of the work and the machine.

A good cutting fluid should have the following properties.

- Good lubricating quality
- Rust resistance
- Stability both in storage and in use
- Resistant to separation from solution after it is mixed with water
- Transparency
- Relatively low viscosity
- Non-flammability

The following are the main purposes of cutting fluids.

- To cool the cutting tool and the workpiece as heat is generated during cutting operation because of friction between the tool and the workpiece.
- To cool the cutting edge of the tool and to prevent any wear on the tool.
- To prevent the formation of chip welding.
- To give a good cutting efficiency to the tool.
- To give a good surface finish on the job.
- To act as a lubricant for the tool and the machine.

The different types of cutting fluids are:

- Soluble mineral oils
- Straight mineral oils
- Straight fatty oils
- Compounded or blended oils
- Sulphurised oils.

Cutting fluids - Types and Characteristics

Soluble mineral oils

They are made from mineral oils with emulsifying material added to make for mixing with water. Soluble oil is diluted with water to form an emulsion. The water cools whilst the oil lubricates. The extent of dilution depends upon the type of operation.

Straight mineral oils

They are purely mineral oils. Lighter oils are used when cooling and lubrication are required. Heavier oils are used when lubrication is mainly essential. They are used on automats. They protect the machine parts and workpieces from rusting.

Lard oils

Lard oils are usually blended with mineral oils to prevent deterioration, reduce cost and destroy the objectionable odour. For machining under extreme conditions, they are an excellent lubricant.

Sulphurised oils

To suit extreme cutting conditions of modern tools sulphurised oils have been devised. The addition of sulphur improves performance on difficult operations. Its lubricating property prevents the welding of chip on to the tool.

Coolants (Cutting fluids) plan an important role in reducing the wear of cutting tools.

Recommended cutting fluids for various metals

Material	Drilling	Reaming	Threading	Turning	Milling
Aluminium	Soluble oil Kerosene Kerosene and Lard oil	Soluble oil Kerosene Mineral oil	Soluble oil Kerosene and Lard oil	Soluble oil	Dry Soluble oil Lard oil Mineral oil
Brass	Dry Soluble oil Mineral oil Lard oil	Dry Soluble oil	Soluble oil Lard oil	Soluble oil	Dry Soluble oil
Bronze	Dry Soluble oil Mineral oil Lard oil	Dry Soluble oil Mineral oil Lard oil	Soluble oil Lard oil	Soluble oil	Dry Soluble oil Mineral oil Lard oil
Cast iron	Dry Air jet Soluble oil Lard oil	Dry Soluble oil Mineral oil Lard oil	Dry Sulphurized oil Mineral oi Lard oil	Dry Soluble oil	Dry Soluble oil
Copper	Dry Soluble oil Mineral oil Lard oil Kerosene Oil	Soluble oil Lard oil	Soluble oil Lard oil	Soluble oil	Dry Soluble oil
Steel Alloys	Soluble oil Sulphurized oil Mineral oil Lard oil	Soluble oil Sulphurized oil Mineral oil Lard oil	Sulphurized oil Lard oil	Soluble oil	Soluble oil Mineral oil Lard oil
General purpose steel	Soluble oil Sulphurized oil Lard oil	Soluble oil Sulphurized oil Lard oil	Sulphurized oil Lard oil	Soluble oil	Soluble oil Lard oil

Lubricants

Objectives: At the end of this lesson you shall be able to
- **state the purpose of using lubricants**
- **state the properties of lubricants**
- **state the qualities of a good lubricant.**

With the movement of two mating parts of the machine, heat is generated. If it is not controlled, the temperature may rise resulting in total damage of the mating parts. Therefore a film of cooling medium with high viscosity is applied between the mating parts which is known as a 'lubricant'.

A 'lubricant' is a substance having an oily property available in the form of fluid, semi-fluid, or solid state. It is the lifeblood of the machine, keeping the vital parts in perfect condition and prolonging the life of the machine. It saves the machine and its parts from corrosion, wear and tear and it minimises friction.

Purpose of using lubricants

- Reduces friction
- Prevents wear
- Prevents adhesion
- Aids in distributing the load
- Cools the moving elements
- Prevents corrosion
- Improves machine efficiency

Properties of Lubricants

Viscosity

It is the fluidity of an oil by which it can withstand high pressure or load without squeezing out from the bearing surface.

Oiliness

Oiliness refers to a combination of wettability, surface tension and slipperiness. (The capacity of the oil to leave an oily skin on the metal.

Flash point

It is the temperature at which the vapour is given off from the oil (it decomposes under pressure soon).

Fire point

It is the temperature at which the oil catches fire and continues to be in flame.

Pour point

The temperature at which the lubricant is able to flow when poured.

Emulsification and de- emulsibility

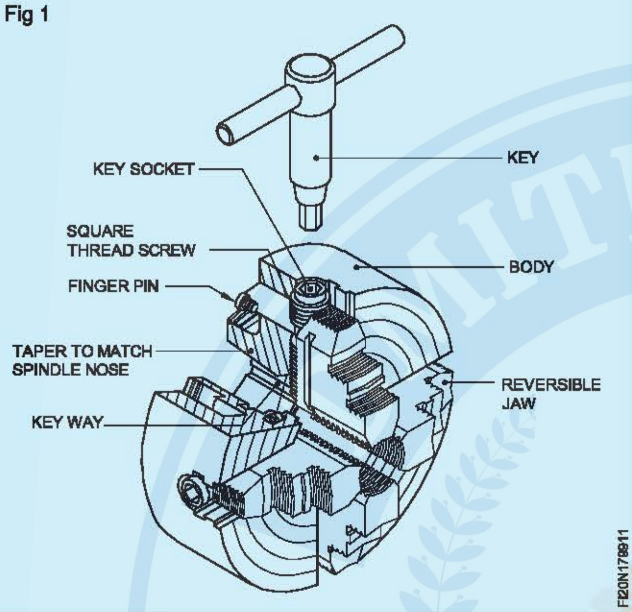
Emulsification indicates the tendency of an oil to mix immediately with water to form a more or less stable emulsion. De-emulsibility indicates the readiness with which subsequent separation will occur.

Chucks and chucking - the independent 4 jaw chuck

Objectives: At the end of this lesson you shall be able to
- **state the constructional features of a 4 jaw chuck**
- **name the parts of a 4 jaw chuck.**

4 Jaw chuck (Fig 1)

Fig 1



The four jaw chuck is also called as independent chuck, since each jaw can be adjusted independently; work can be trued to within 0.001" or 0.02mm accuracy, using this chuck.

This type of chuck is much more heavily constructed than the self-centering chuck, and has much greater holding power. Each jaw is moved independently by a square thread screw. The jaws are reversible for holding large diameter jobs. The independent 4 jaw chuck has four jaws, each working independently of the others in its own slot in the chuck body and actuated by its own separate square threaded screw. By suitable adjustment of the jaws, a workpiece can be set to run either true or eccentric as required.

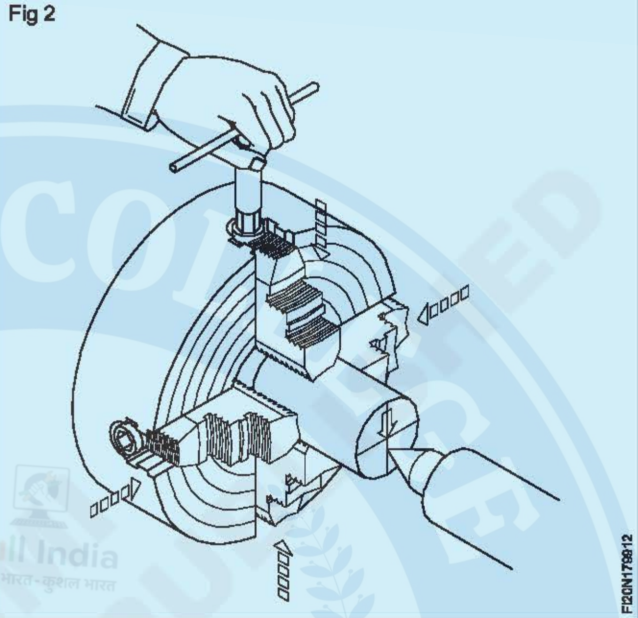
To set the job for the second time, it can be trued with the help of a dial test indicator.

The check on the workpiece should be carried out near the chuck and repeated as far from it as the workpiece permits, to ensure that the work is not held in the chuck at an angle to the axis of rotation.

The independent adjustment also provides the facility of deliberately setting the work off-centre to produce an eccentric workpiece. (Fig 2)

The parts of a 4 jaw chuck are:

Fig 2



- Back plate
- Body
- Jaws
- Square threaded screw shaft.

Back plate

The back plate is fastened to the back of the body by means of Allen screws. It is made out of cast iron/steel. Its bore is tapered to suit the taper of the spindle nose. It has a key way which fits into the key provided on the spindle nose. There is a step in front and on which the thread is cut. A threaded collar, which is mounted on the spindle, locks the chuck by means of the thread, and locates by means of the taper and key. Some chucks do not have back plates.

Body (Fig 1)

The body is made out of cast iron/cast steel and the face is flame-hardened. It has four openings at 90° apart to assemble the jaws and operate them. Four screw shafts are fixed on the periphery of the body by means of finger pins. The screw is rotated by means of a chuck key. The body, hollow in the cross-section, has equi-spaced circular rings provided on the face, which are marked by numerical numbers. Number 1 starts in the middle, and increases towards the periphery.

Jaws (Fig 1)

Jaws are made out of high carbon steel. hardened and tempered, which slide on the openings of the body. These jaws are reversible for holding hollow work.

The back side of the jaws are square-threaded which help in fixing the jaws with the operating screws.

Screw shaft (Fig 1)

The screw shaft is made out of high carbon steel, hardened, tempered and ground. The top portion of the screw shaft is provided with a square slot to accommodate the chuck key. On the body portion, a left hand square thread is cut. In the middle of the screw shaft, a narrow step is made and held by means of finger pins. The finger pins permit the screws to rotate but not to advance.

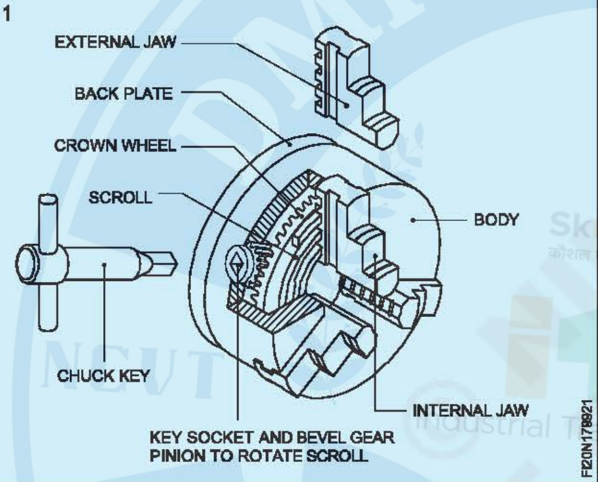
3 Jaw chuck

Objectives: At the end of this lesson you shall be able to

- **identify the parts of a 3 jaw chuck**
- **state the constructional features of a 3 jaw chuck**
- **distinguish between a 3 jaw chuck and a 4 jaw chuck**
- **state the merits and demerits of a 4 jaw chuck over a 3 jaw chuck**
- **specify a chuck.**

3 Jaw chuck (Fig 1)

Fig 1



The 3 jaw chuck is also known as a self-centering chuck. The majority of the chucks have two sets of jaws for holding internal and external diameters. Only perfect round work with equally spaced flats divisible by three should be held in a 3 jaw chuck.

From the construction of a 3 jaw chuck it is seen that the scroll not only clamps a component in place, it also locates the component. This is fundamentally a bad practice, since any wear in the scroll and/or the jaws impairs the accuracy of location. Further there is no means of adjustment possible to compensate for this wear.

The jaws of this type of chuck are not reversible, and separate internal and external jaws have to be used.

The parts of a 3 jaw chuck are:

- Back plate
- Body
- Jaws
- Crown wheel and
- Pinion.

Back plate (Fig 1)**:** The back plate is fastened at the back of the body by means of allen screws. It is made out of cast iron. Its bore is tapered to suit the taper of the spindle nose. It has a key- way which fits into the key provided on the spindle nose. There is a step in the front on which the thread is cut. The threaded collar, which is mounted on the spindle, locks the chuck by means of the thread and locates by mean of the taper and the key.

Body (Fig 1): The body is made out of cast steel, and the face is hardened. It has three openings 120° apart to assemble the jaws and operate them. Three pinions are fixed on the periphery of the body to operate the jaws by means of a chuck key. It is hollow in its cross-section. A crown wheel is housed inside the body.

Jaws (Fig 1): The jaws are made out of high carbon steel, hardened and tempered, which slide on the openings of the body. Generally there are two sets of jaws viz. external jaws and internal jaws. External jaws are used for holding solid works. Internal jaws are used for holding hollow works. Steps on the jaws increase the clamping range. The back side of the jaws is cut with scroll thread. Each jaw is numbered in a sequential manner, which helps in fixing the jaws in the corresponding numbered slots.

Crown wheel (Fig 1): The crown wheel is made out of alloy steel, hardened and tempered. On one side of the crown wheel, a scroll thread is cut to operate the jaws, and the other side is tapered on which bevel gear teeth are cut to mesh with the pinion. When the pinion is rotated by means of a chuck key, the crown wheel rotates, thus causing the jaws to move inward or outward, depending upon the rotation .

Pinion (Fig 1): Pinion is made out of high carbon steel, hardened and tempered. It is fitted on the periphery of the body. On the top of the pinion a square slot is provided to accommodate the chuck key. It has a tapered portion on which bevel gear teeth are cut, which match with the crown wheel.

Comparison between 3 jaw chuck and 4 jaw chuck

3 Jaw chuck	4 Jaw chuck
Only cylindrical, hexagonal work can be held.	A wide range of regular and irregular shapes can be held.
Internal and external jaws are available.	Jaws are reversible for external and internal holding.
Setting up of work is easy.	Setting up of work is difficult.
Less gripping power.	More gripping power.
Depth of cut is comparatively less.	More depth of cut can be given.
Heavier jobs cannot be turned.	Heavier jobs can be turned.
Workpieces cannot be set for eccentric turning.	Workpieces can be set for eccentric turning.
Concentric circles are not provided on the face.	Concentric circles are provided which help for approximate setting of jaws.
Accuracy decreases as the chuck gets worn out.	There is no loss of accuracy as the chuck gets worn out.

Merits of a 4 jaw chuck

- A wide range of regular and irregular shapes can be held.
- Work can be set to run concentrically or eccentrically at will.
- Has considerable gripping power, and hence heavy cuts can be given.
- The jaws are reversible for internal and external work.
- Work can be readily performed on the end face of the job.
- There is no loss of accuracy as the chuck gets wornout.

De-merits of a 4 jaw chuck

- Workpieces must be individually set.
- The gripping power is so great that fine work can be easily damaged during setting.

Merits of a 3 jaw chuck

- Work can be set with ease.
- A wide range of cylindrical and hexagonal work can be held.
- Internal and external jaws are available.

De-merits of a 3 jaw chuck

- Accuracy decreases as chuck becomes worn out.
- Run out cannot be corrected.
- Only round and hexagonal components can be held.
- When accurate setting or concentricity with an existing diameter is required, a self-centering chuck is not used.

Specification of chuck

To specify a chuck, it is essential to provide :

- Type of chuck.
- Capacity of chuck.
- Diameter of the body.
- Width of the body.

The method of mounting to the spindle nose.

Example

3 Jaw self-centering chuck.

Gripping capacity 450 mm.

Diameter of the body 500 mm.

Width of the body 125 mm.

Tapered or threaded method of mounting.

Method of cleaning the thread of the chuck mounting

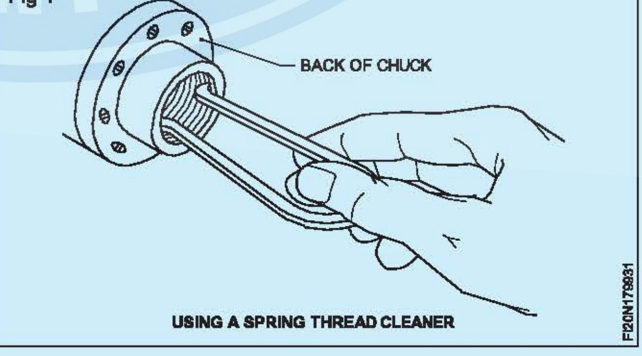
Objective: At the end of this lesson you shall be able to
- **state the uses of thread cleaner.**

Thread cleaners are used to clean all the mating parts of the chuck and spindle as, otherwise, the dirt on these surfaces could result in the following.

Cause the chuck to run out of true.

Damage the threads or taper on the spindle or chuck. (Fig 1)

Fig 1



Mounting and dismounting of chucks

Objective: At the end of this lesson you shall be able to
- **explain the method of mounting and dismounting chucks from spindle noses.**

To perform lathe operations on work materials, it may not be always possible to have only one type work-holding device fitted to the spindle. Hence it becomes an absolute necessity for dismounting the work-holding device already assembled to the spindle and mount that work-holding device which is needed for the work in hand.

For an easy understanding of different spindle noses and their applications, the mounting of different work-holding devices are illustrated.

When mounting a chuck on the headstock spindle, exercise care to prevent damage occurring to the chuck or spindle.

Damage may reduce the accuracy of the lathe. The points set out below are important and should be followed.

Before mounting

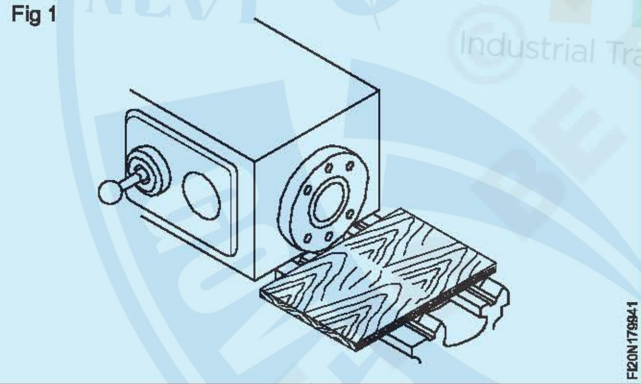
Before attempting to mount a chuck, ensure that it is the correct one for the lathe and for the job in hand.

Do not use power to mount a chuck on spindle noses.

To prevent such damage from occuring, take the following steps.

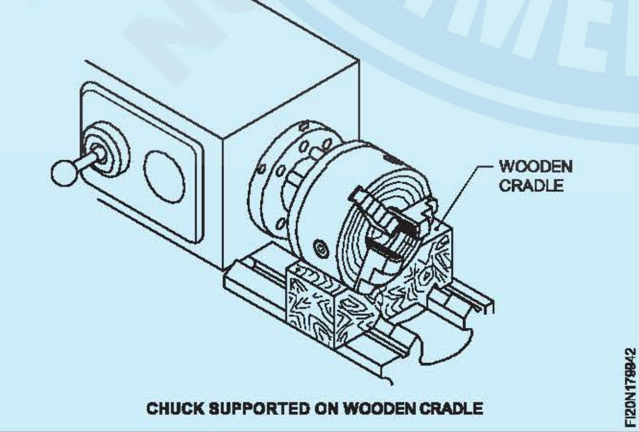
Place a wooden board on the lathe bed when mounting light chucks to prevent damage to the slideways. (Fig 1)

Fig 1



For large chucks place a wooden cradle between the chucks and the lathe bed. (Fig 2)

Fig 2



CHUCK SUPPORTED ON WOODEN CRADLE

In addition to protecting the bed slideways it makes fitting the chuck easier and safer.

Always seek assistance when mounting large and heavy chucks.

Lubricate the mating surfaces with a light film of oil.

After mounting

Set the speed-change lever to the slowest speed.

Turn on the power to the motor.

Switch on the motor.

Engage the clutch lever.

The chuck would now begin revolving.

Check that the diameter and face of the chuck are running true by observing the surfaces.

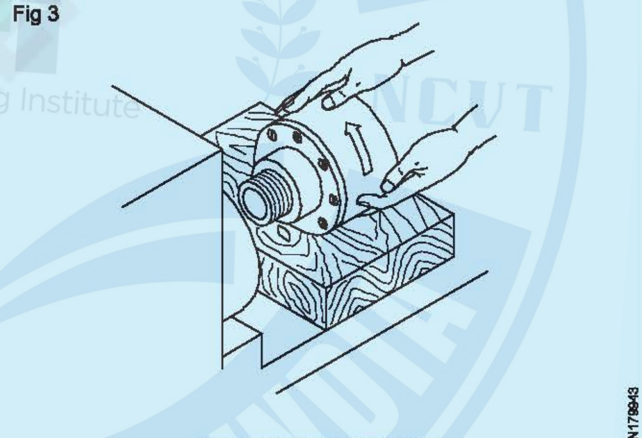
Mounting chuck on to the threaded spindle (Fig 3)

Switch off the motor.

Place the chuck on the wooden plank or cradle and slide it close to the spindle nose.

Turn the spindle anticlockwise by hand and engage the chuck on the spindle threads. (Fig 3)

Fig 3



MOUNTING THREADED CHUCK

Set the speed-change lever to the slowest speed. Screw the chuck in until it fits firmly on the spindle.

The chuck should easily screw into the spindle. If any resistance is felt, remove the chuck and check that the threads are clean and not damaged.

Mounting on tapered spindle (Fig 4)

Switch off the motor.

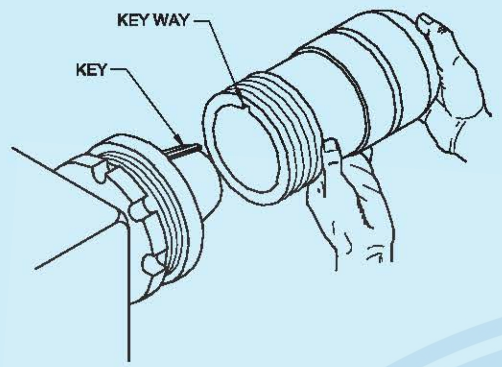
Hence the chuck on the wooden board or cradle and slide it close to the spindle nose.

Turn the spindle by hand until the key on the spindle nose lines up with the keyway in the chuck.

Set the speed-change lever to the slowest speed.

Push the chuck on to the spindle and turn the locking ring anticlockwise. (Fig 4)

Fig 4

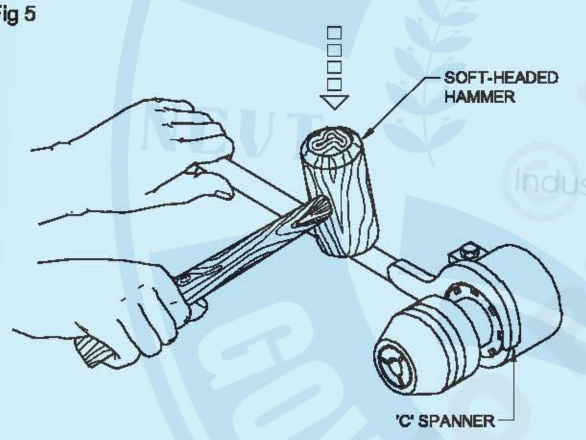


MOUNTING THREADED CHUCK

The figure given here illustrates a small chuck held with both hands and being mounted. Engage the special 'C' spanner on the locking ring.

The spanner should fit around the top of the locking ring with the handle pointing downwards. Grip the end of the handle with one hand and firmly strike the other end with the other hand in an anticlockwise direction. This would securely tighten the locking ring. (Fig 5)

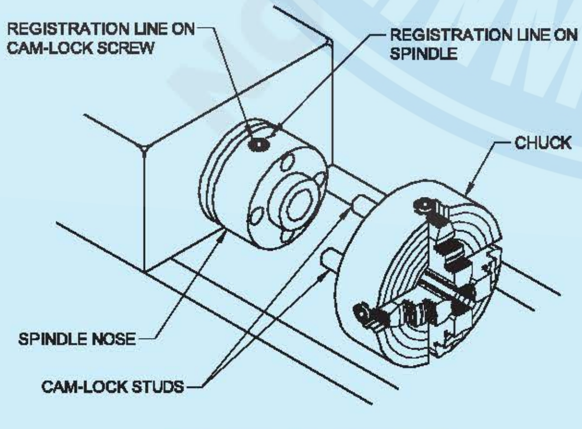
Fig 5



LOCK CHUCK ON SPINDLE

Mounting on a cam-lock spindle (Fig 6)

Fig 6



MOUNTING CHUCK ON CAM-LOCK SPINDLE

Switch off the motor.

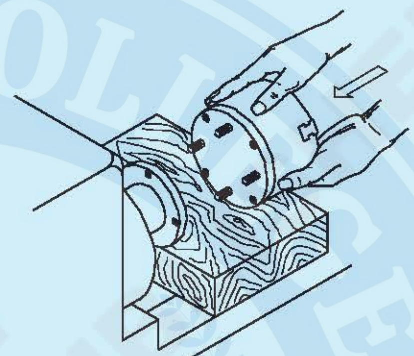
Place the chuck on a wooden board or cradle and slide it close to the spindle nose. Disengage the clutch to permit free rotation of the spindle. Insert the correct chuck key into a cam-locking screw on the spindle.

Turn each cam-locking screw so that the registration line is vertical or aligns with the corresponding line on the spindle. Turn the spindle by hand until the clearance holes on the spindle align with the cam-lock studs on the chuck.

Set the speed. Change lever to the slowest speed. Push the chuck on to the spindle. Tighten each cam-lock screw in a clockwise direction.

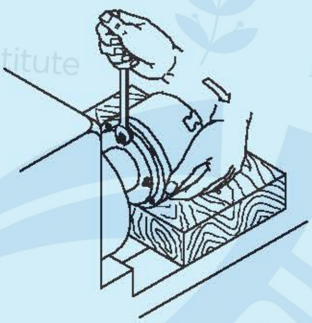
Mounting on to a bolted spindle (Figs 7 and 8)

Fig 7



MOUNTING CHUCK TO BOLTED SPINDLE

Fig 8



TIGHTEN NUTS FIRMLY

Switch off the motor.

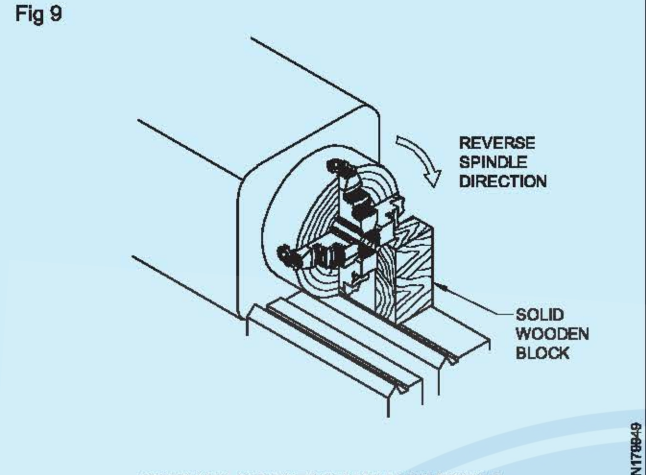
Place the chuck on a wooden board or cradle. Remove nuts and washers from the studs on the chuck. Disengage the clutch to permit free rotation of the spindle. Turn the spindle by hand until the key in the spindle lines up with the slot in the chuck. Set the speed- change lever to the slowest speed. Push the chuck on to the spindle. Fit washers and nuts to the studs.

Hold the chuck in position when fitting nuts.

Tighten the nuts in an anticlockwise direction using a spanner on the opposite nuts.

Dismounting chucks from a threaded spindle (Fig 9)

Fig 9



REMOVING SCREWED CHUCK FROM SPINDLE

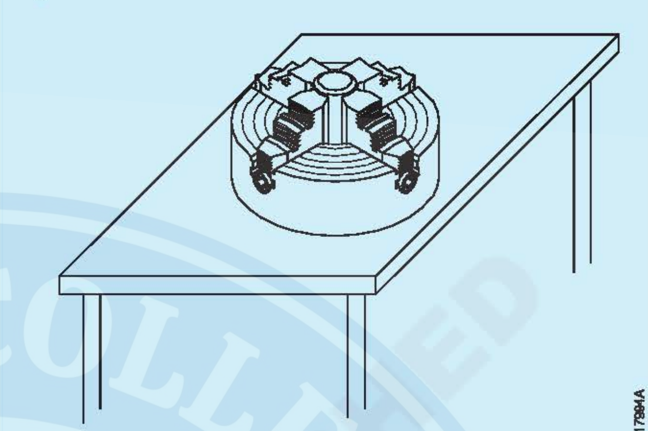
Switch off the motor. Set the speed change lever to the slowest speed. Place a solid wooden block between one of the chuck jaws and the rear of the lathe-bed.

The length of the wooden block should be slightly less than the centre height of the lathe.

Turn the lathe spindle clockwise by hand to loosen the chuck from the spindle nose.

Remove the wooden block. Place the wooden board or cradle on the lathe-bed. Unscrew the chuck from the spindle. Clean and store the chuck (Fig 10)

Fig 10



STORE ALL CHUCKS CAREFULLY AFTER USE

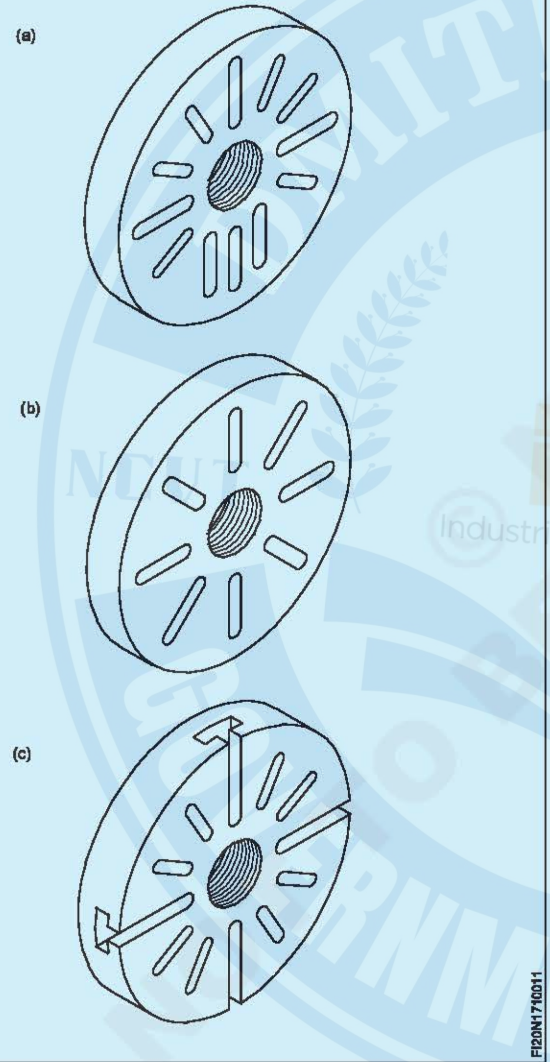
Face plate

Objectives: At the end of this lesson you shall be able to
- **state the types of face plate**
- **state the uses of face plates**

The different types of face plates are:

- Face plates with only elongated radial slots (Fig 1a)
- Face plates with elongated slots 'T' slots. (Fig 1b)
- Face plates with elongated radial slots and additional parallel slots. (Fig 1c)

Fig 1



Face plates are used along with the following accessories.

Clamps, 'T' bolts, Angle plate, Parallels, counterweight, Stepped block, 'V' Block etc.

Large, flat, irregular shaped workpieces, castings, jigs and fixtures may be firmly clamped to a face plate for various turning operations.

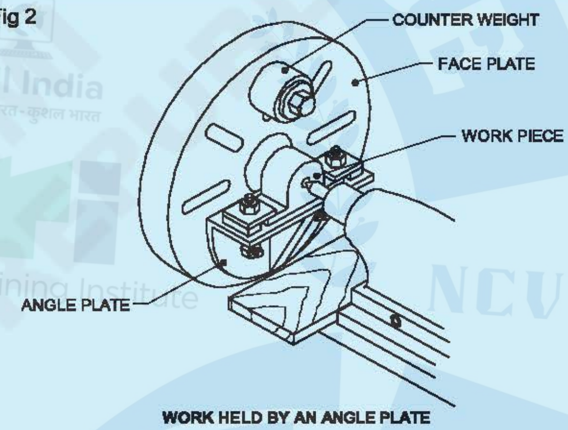
A work can be mounted on a face plate while the face plate is on the lathe spindle or on the workbench. If the workpiece is heavy or awkward to hold, the workpiece is mounted while the face plate is on the workbench. Before mounting the face plate set up to the spindle, it is advantageous to locate the workpiece on the face plate and centre the workpiece. Centre a punch mark or hole approximately on the face plate. This makes it easier to true the work after the face plate is mounted on to the spindle.

The poisition of the bolts and clamps is very important, if a workpiece is to be clamped effectively.

If a number of duplicate pieces are to be machined, the face plate itself can be set up as a fixture, using parallel strips and stop blocks.

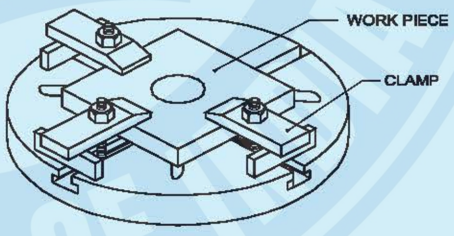
The application of the face plate with accessories in different set ups is shown in the sketches below. (Figs 2,3&4)

Fig 2



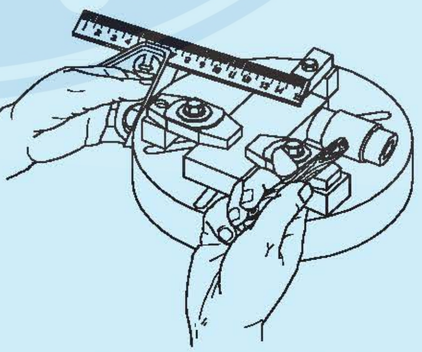
WORK HELD BY AN ANGLE PLATE

Fig 3



WORK CLAMPED DIRECTLY TO FACE PLATE

Fig 4



LOCATING WORK ON A FACE PLATE

Drilling

Objectives: At the end of this lesson you shall be able to
- **state the drilling process done in a lathe**
- **state the methods of holding the drill in the tail stock.**

Lathe can be used for drilling

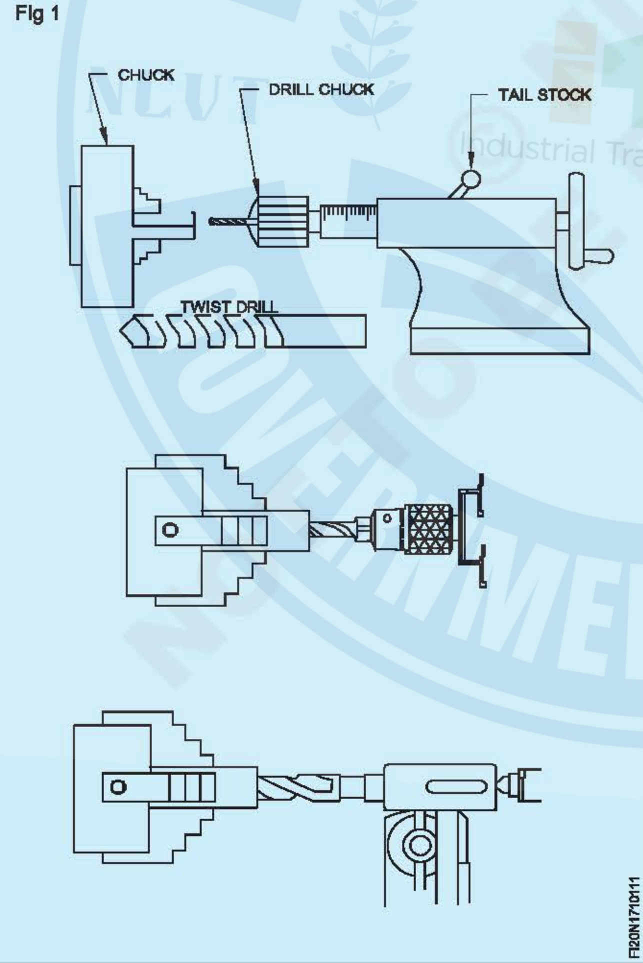
Before doing internal operation like boring, reaming and tapping. Although lathe is not a drilling machine time and effort are saved by using the lathe for drilling operations instead of changing the work to the other machines. Prior to drilling the end of the work piece on the lathe, the end face to be drilled must be spotted (center punched) and then centre drilled so that the drill will start properly

The head stock and tail stock spindle should be aligned for all drilling, reaming and tapping in order to produce a true hole.

Straight shank and taper shank drills can be held in the tailstock spindle as held in the drilling machine spindle using drill chuck sleeve and sockets. Since the tail stock spindle has the mores taper. (Fig 1)

Methods of holding drills in a tail stock (Fig 1)

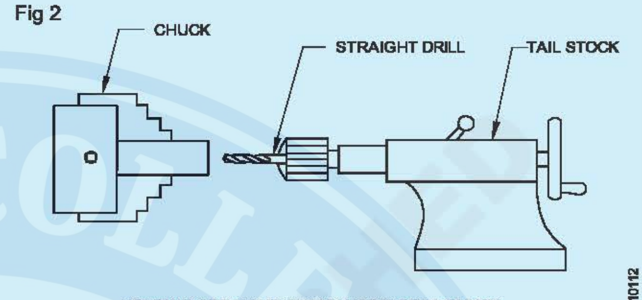
Fig 1



The different methods of holding drill in the tailshock are

- By using drill chuck (Fig 2)

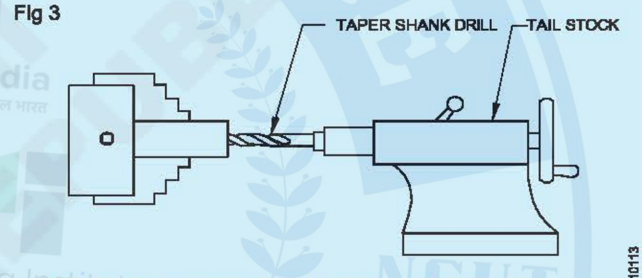
Fig 2



HOLDING STRAIGHT SHANK USING DRILL CHUCK

- By directly fitting in the tailstock spindle (Fig 3)

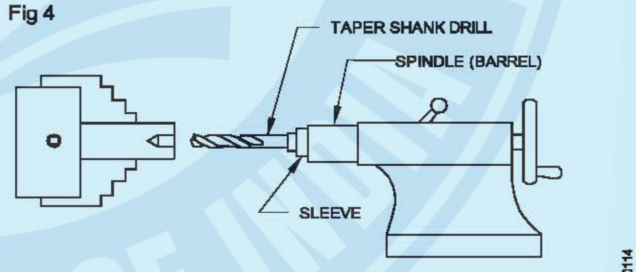
Fig 3



HOLDING TO TAPER SHANK DRILL DIRECTLY IN THE SPINDLE

- By using drill sleeve (Fig 4)

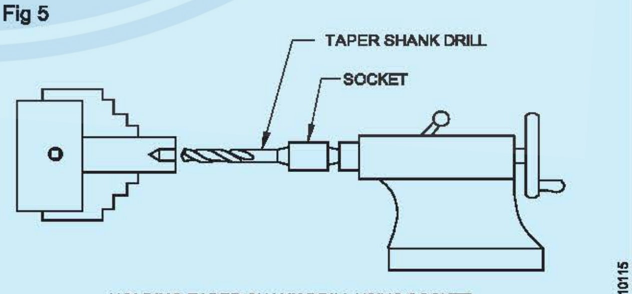
Fig 4



HOLDING TAPER SHANK DRILL USING SLEEVE

- By using drill socket (Fig 5)

Fig 5



HOLDING TAPER SHANK DRILL USING SOCKET

Boring & boring tools

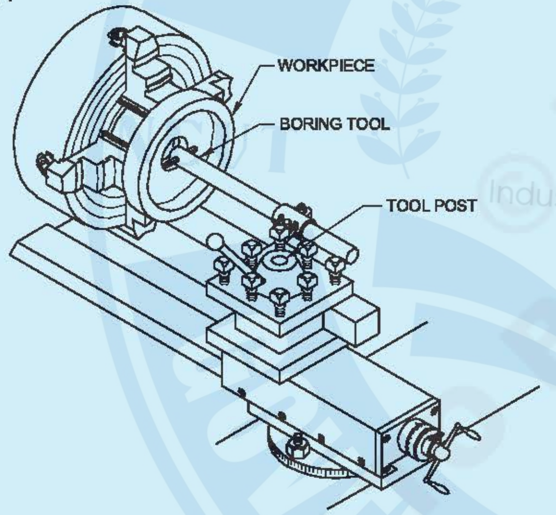
Objectives: At the end of this lesson you shall be able to
- **state the operation boring**
- **state the different types of boring tools.**

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.

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 adopted for boring small sized works. A solid forged tool is used for boring small holes, 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 cross-slide screw and the feed is effected by the longitudinal travel of the carriage. (Fig 1)

Fig 1



BORING OPERATION

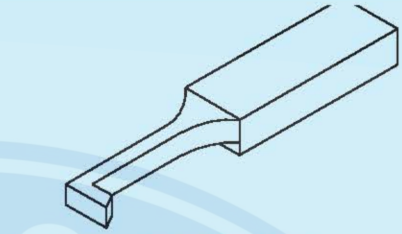
Types of boring tools

Solid forged tools

Solid forged boring tool is made from HSS with the end forged and ground. It resembles a left hand turning tool and the operation is performed from right to left. There are two types, solid boring tool (Fig 2) and solid forged bar in a tool-holder (Fig 3). They are used for light duty and on small diameter holes.

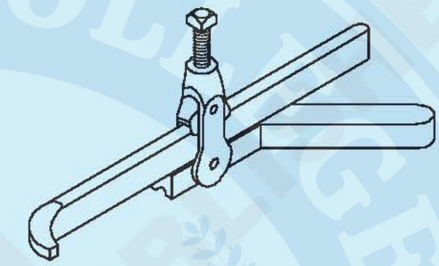
Advantages

- Regrinding is easy.
- Alignment is easy.
- Mounting and removal is easy.



SOLID BORING TOOL

Fig 3



A SOLID FORGED BAR IN A TOOL HOLDER

Boring bars with inserted bits

Square and round tool bits made from HSS are inserted and fixed in the boring bar. The inserts can be set at an angle of 30°, 45° or 90° in the bar. It is used for heavier cuts than those made by the solid boring tool.

For plain boring, the inserts are set square to the axis of the bar. For facing the shoulder, or threading up to the shoulder, the inserts are set at an angle.

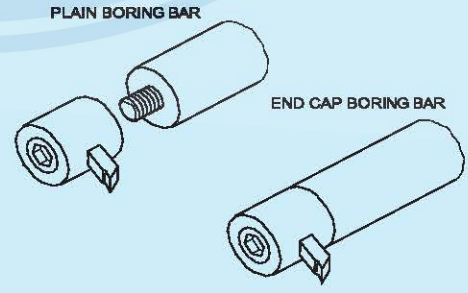
Boring bars used are of two types. (Fig 4)

- Plain boring bar
- End cap boring bar

Advantages

- Used for heavy duty boring operations.
- Tool changing is faster.
- Low cost
- Boring tools can either be set square or at an angle quickly.

Fig 4



BORING BARS

Tool setting

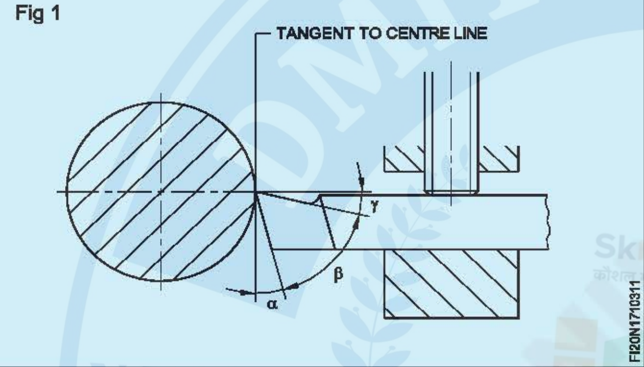
Objective: At the end of this lesson you shall be able to
- **set the tool in the tool post for performing the operation.**

For optimum cutting, the effective rake angle and clearance angle of the clamped tool must be equal to the ground angles of the tool. This requires clamping of the tool to have its axis perpendicular to the lathe axis, with the tool tip at the workpiece centre. (Fig 1)

It is difficult to determine the effective angles of the tool when it is not set to the centre height.

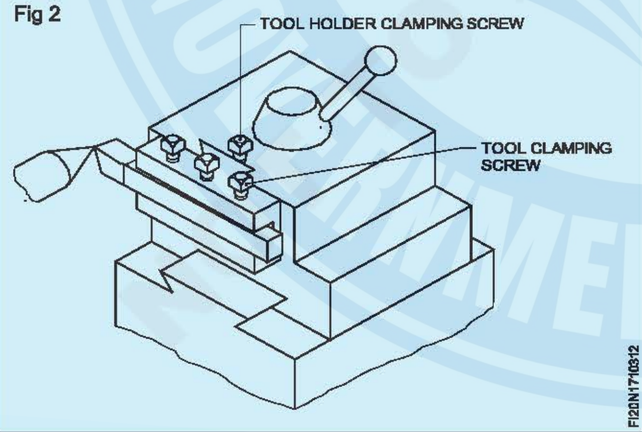
The tool nose can be set to the work centre by means of a tool-holder with adjustable height. (Fig 1)

Fig 1



The tool nose can be set to the exact centre height by placing the tool in the tool post on the shims or packing strips. These packing strips should be preferably a little less in width than the wideth of the tool but should never be more. The length of these strips should be according to the shank length and the tool seating face of the tool post. (Fig 2)

Fig 2



The procedure to follow is given below.

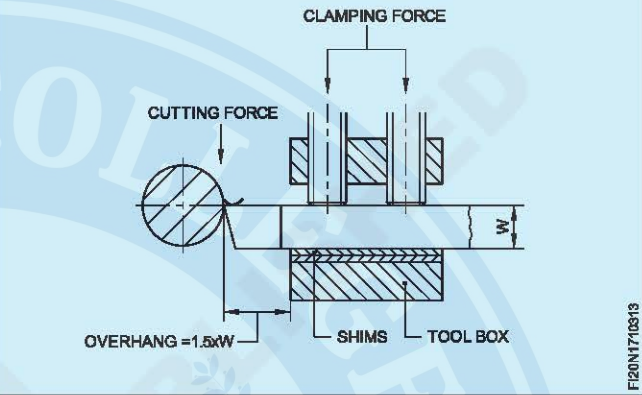
Clean the tool post seating face, and place the shims on the seating face.

Use a minimum number of shims for height adjustment.

Shims must be flushed with the edge of the seating face.

Place the tool in the tool post on the shims, with the rear butting against the wall of the seating face. (Fig 3)

Fig 3

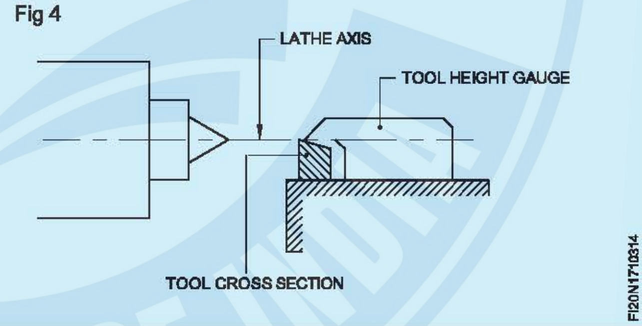


The unsupported length of the overhanging end of the turning tool should be kept to a minimum. As a rule, the overhanging length of tool is equal to the tool shank width x 1.5.

Tighten the tool with the centre screw of the tool post.

Check the centre height with a height setting gauge. (Fig 4)

Fig 4



Remove or add shims and check the height when the tool is tightened by the centre screw.

Tighten the other two tool-holding screw alternate applying the same amount of pressure.

When both the screws have a full gripping pressure, tighten the centre screw fully.

Check once again with a tool height setting gauge.

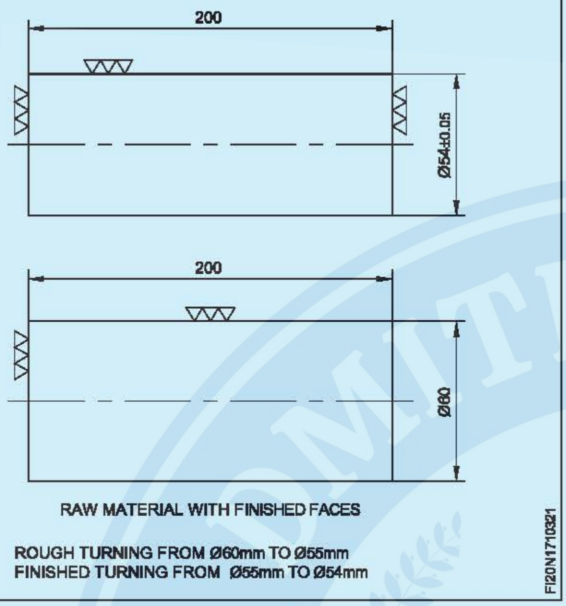
Note: The gauge should be made according to the size of the machine. If a gauge is not available, use a surface gauge and set the pointer tip to the dead centre height fixed in the tailstock. Use this as the height to which the tool is to be set.

Parallel or straight turning

Objectives: At the end of this lesson you shall be able to
- **define plain turning**
- **distinguish between the two stages of plain turning.**

Plain turning (Parallel turning) (Fig 1)

Fig 1

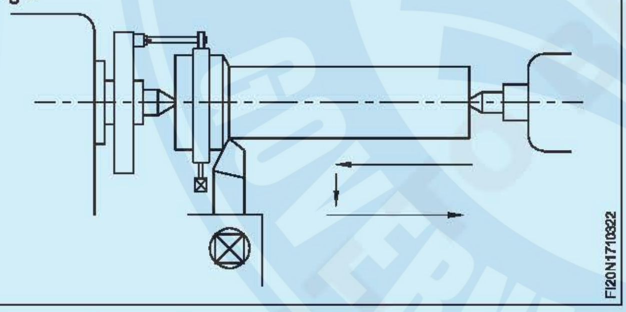


This operation involves removal of metal from the work and it has a cylinder for the full travel of the tool on the work, keeping the same diameter throughout the length.

Plain turning is done in two stages.

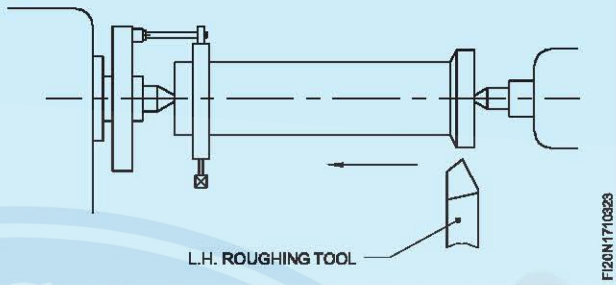
- Rough turning, using roughing tool or knife tool. (Fig 2)

Fig 2



The spindle speed is calculated according to the material being turned, the tool material and the recommended cutting speed.

Fig 3



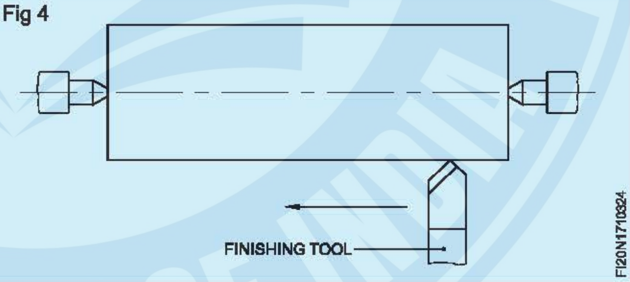
Rough turning: By rough turning the maximum amount of material is removed and the job is brought close to the required size, leaving sufficient metal for finishing. Surface finish and accuracy are not good. While rough turning, the spindle speed is less and the feed is more. A roughing tool or a knife tool is used.

While plain turning for roughing or finishing, long jobs are held between centres. It is necessary to change the ends to obtain a true parallel surface throughout the length. (Fig 3)

Finish turning: It is done, after the rough turning is completed to bring the size of the work to the required accuracy and good surface finish by removing the rough marks produced by the rough turning. For finish turning, the speed is higher (1 to 2 times more than for rough turning) and the feed is very less. A round nose finish turning tool or a knife with a larger nose radius than normal is used for finish turning.

- Finish turning using a finishing tool. (Fig 4)

Fig 4



Step turning

Objective: At the end of this lesson you shall be able to
- **define step turning**

Step turning

It is an operation of producing various steps of different diameters in the work piece as shown in Fig 1 & 2. This operation is carried out in the similar way as plain turning.

Fig 1

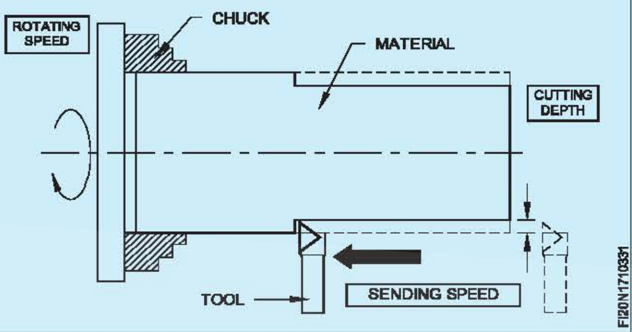
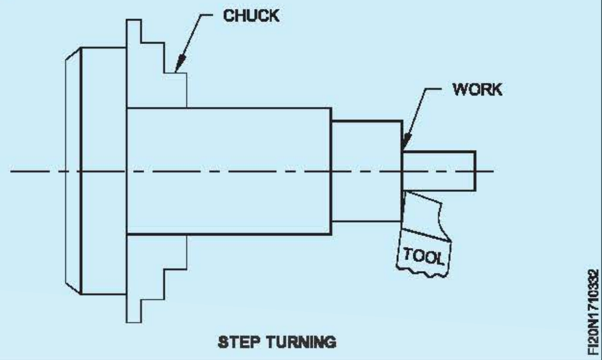


Fig 2



Grooving

Objectives: At the end of this lesson you shall be able to
- **state what is grooving**
- **name the types of grooves**
- **state the specific uses of each type of groove.**

Grooving

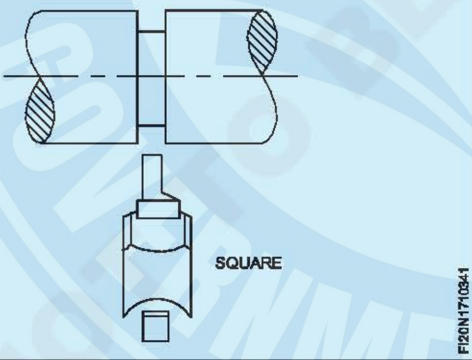
Grooving is the process of turning a grooved form or channel on a cylindrically turned workpiece. The shape of the cutting tool and the depth to which it is fed determine the shape of the groove.

Types of grooves

Square grooves

Square grooves are frequently cut at the end of a section to be threaded in order to provide a channel into which a threading tool may run. A square groove cut against a shoulder allows a matching part to fit squarely against the shoulder. (Fig 1)

Fig 1



When a diameter is to be finished to size by grinding, a groove is generally cut against the shoulder to provide clearance for the grinding wheel and to ensure a square corner.

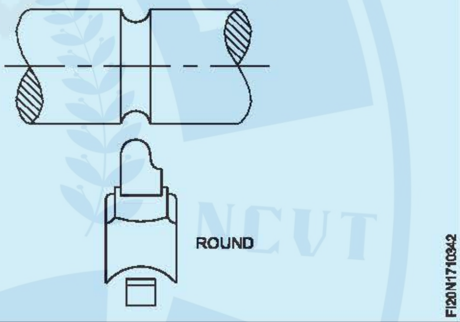
Square grooves are cut with a tool bit ground to the width of the square groove to be formed.

A square groove also serves the purpose of providing space for forks of shift levers in sliding gear assemblies.

Round groove

Round grooves serve the same purpose as square grooves. They are generally used on parts subjected to stress. The round groove eliminates the sharpness of the square corners and strengthens the part at the point where it tends to fracture. A tool bit with a round nose ground to the required radius is used to cut round grooves. (Fig 2)

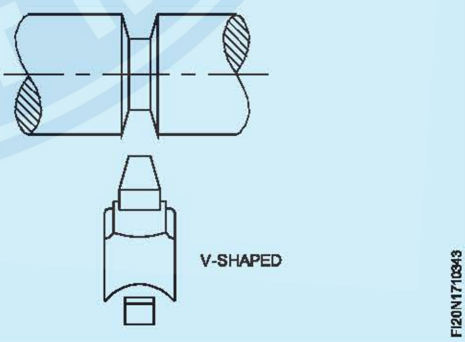
Fig 2



'V' shaped groove

'V' shaped grooves are most commonly found on pulleys driven by 'V belts. The 'V shaped groove eliminates much of the slip which occurs in the other forms of the belt drive. A 'V groove may also be cut at the end of a thread to provide a channel into which the threading tool may run. (Fig 3)

Fig 3

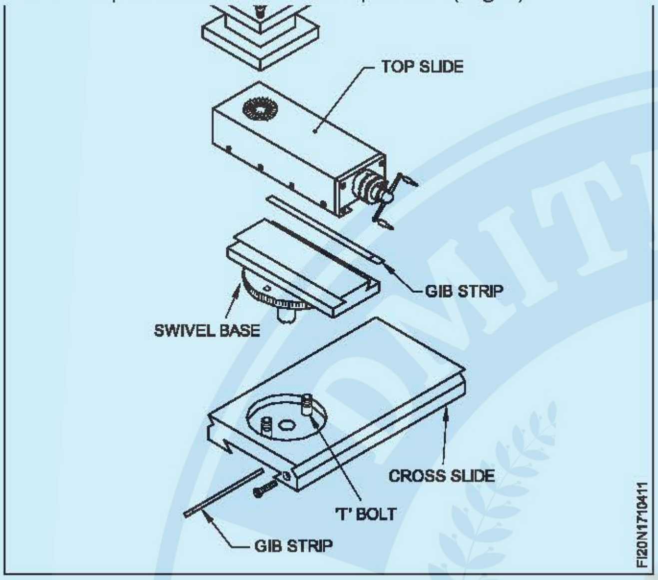


A tool bit ground to the desired angle is used to cut a shallow 'V' groove. Larger 'V' grooves such as those found on pulleys should be cut with the lathe compound rest to form each face of the groove individually.

Tool post

Objectives: At the end of this lesson you shall be able to
- **name the commonly used types of tool posts**
- **compare the features of different types of tool posts.**

The tool post holds and firmly supports the tool or tools. The tool post is fitted on the top slide. (Fig 1)



The commonly used types of tool posts are:

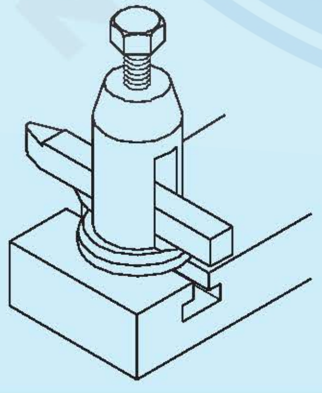
- American type tool post or single way tool post.
- Indexing type tool post or square tool post.
- Quick change tool post.

Single way tool post (Fig 2)

It consists of a circular tool post body and a pillar with a slot for accommodating the tool or tool-holder. A ring base, a rocker arm (boat piece) and a tool clamping screw complete the assembly of this type of tool post.

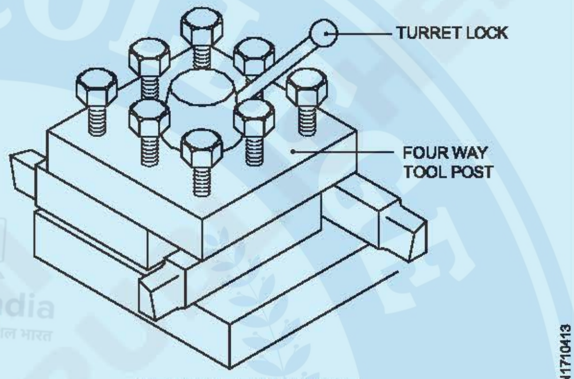
The tool is positioned on the boat piece and clamped. The centre height of the tool tip can be adjusted with the help of the rocker arm and the ring base. Only one tool can be fixed in this type of tool post. The rigidity of the tool is less as it is clamped with only one bolt.

Fig 2



Indexing type tool post (Fig 3): It is also called as square tool post or a four-way tool post. Four tools can be fixed in this type of tool posts, and any one can be brought into the operating position, and the square head is clamped with the help of the handle lever. By loosening the handle lever, the next tool can be indexed and brought in to the operating position. The indexing is manually.

Fig 3



INDEXING TYPE TOOL POST

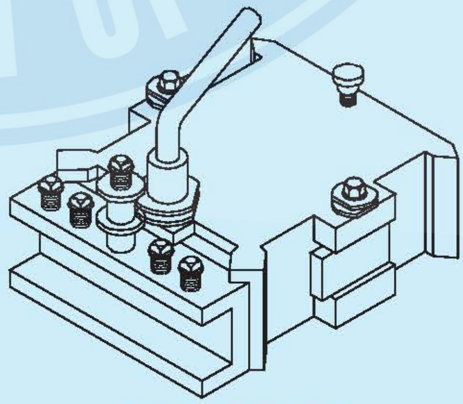
The advantages are as follows: Each tool is secured in the tool post by more than one bolt, and, therefore, the rigidity is more.

Frequent changing of the tool for different operations need not be done as all the four tools can be clamped at the same time.

The disadvantage is that skill is required to set the tools, and it takes more time to set to the centre height.

Quick change tool post (Fig 4): Modern lathes are provided with this type of tool posts. Instead of changing the tools, the tool holder is changed in which the tool is fixed. This is expensive and requires a number of tool-holders. But it can be set to the centre height easily, and has the best rigidity for the tool.

Fig 4



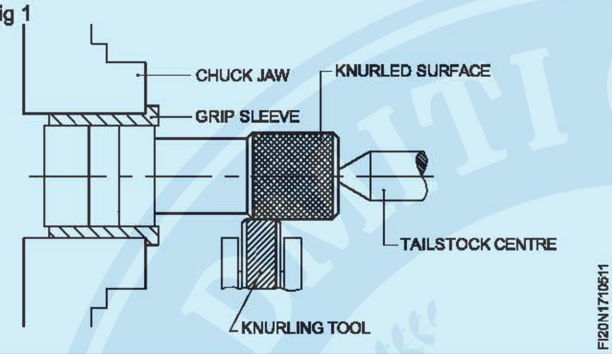
QUICK CHANGE TOOL POST

Lathe operation - Knurling

Objectives: At the end of this lesson you shall be able to
- **define knurling operation**
- **state the purpose of knurling**
- **list the different types of knurls and knurling patterns**
- **name the grades of knurls**
- **distinguish between the various types of knurling tool-holders.**

Knurling (Fig 1)

Fig 1



It is the operation of producing straight lined, diamond shaped pattern or cross lined pattern on a cylindrical external surface by pressing a tool called knurling tool. Knurling is not a cutting operation but it is a forming operation. Knurling is done at a slow spindle speed (1/3 the turning speed). However speed & feed given for knurling is to be divided according to the job material and the finish required.

Purpose of knurling

The purpose of knurling is to provide:

- A good grip and make for positive handling.
- Good appearance
- For raising the diameter to a small range for assembly to get a press fit.

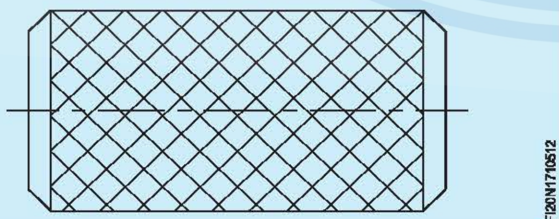
Types of knurls and knurling patterns

The following are the different types of knurling patterns.

Diamond knurling, Straight knurling, Cross knurling, Concave knurling and Convex knurling.

Diamond knurling (Fig 2)

Fig 2



It is a knurling of diamond shaped pattern. It is done by using a set of rolls. One roller has got right hand helical teeth and the other has left hand helical teeth.

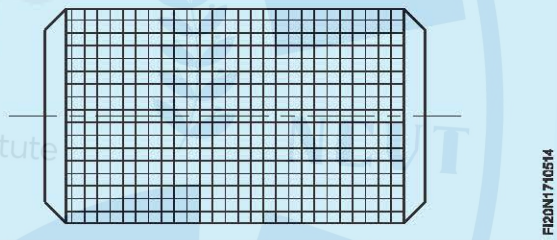
Straight knurling (Fig 3)

Fig 3

It is a knurling of straight lined pattern. This is done by using either a single roller or a double roller with straight teeth.

Cross knurling (Fig 4)

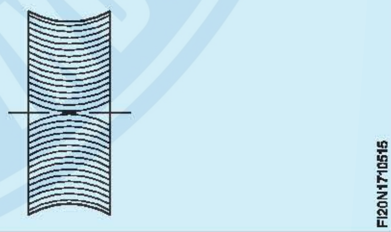
Fig 4



It is a knurling having a square shaped pattern. It is done by a set of rollers, one having straight teeth the other having teeth at right angles to the axis of knurl.

Concave knurling (Fig 5)

Fig 5



This is done by a convex knurl on a concave surface. This is done only by plunging the tool. The tool should not be moved longitudinally. The length of the knurling is limited to the width of the roller.

Convex knurling (Fig 6)

This is done by using a concave knurl on a convex surface. This is also done by plunging the tool.

Grades of knurling (Fig 7)

Fig 6

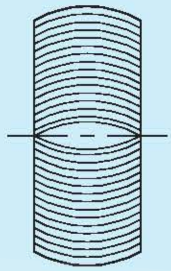
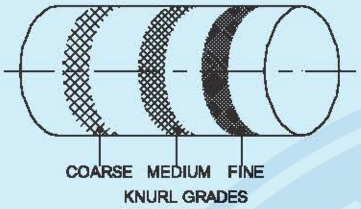


Fig 7



Knurling can be done in three grades.

Coarse knurling, Medium knurling and Fine knurling

Coarse knurling is done by using coarse pitched knurls of 1.75 mm pitch. (14 TPI)

Medium knurling is done by using medium pitched knurls of 1.25 mm pitch. (21 TPI)

Fine knurling is done by using fine pitched knurls of 0.75 mm pitch. (33 TPI)

Types of knurling tool-holders

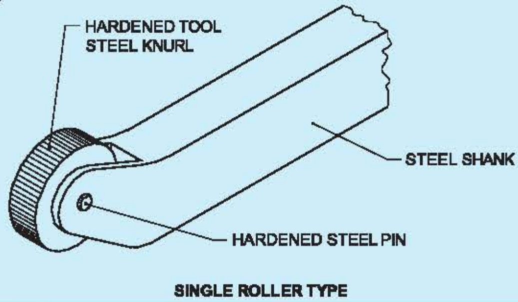
The different types of knurling tool-holders are:

- Single roller knurling tool-holders (parallel knurling tool-holders)
- Knuckle joint type knurling tool-holders
- Revolving type knurling tool-holders (universal knurling tool-holders).

A knurling tool-holder has a heat-treated steel shank and hardened tool steel knurls. The knurls rotate freely on hardened steel pins.

Single roller knurling tool-holder (Fig 8)

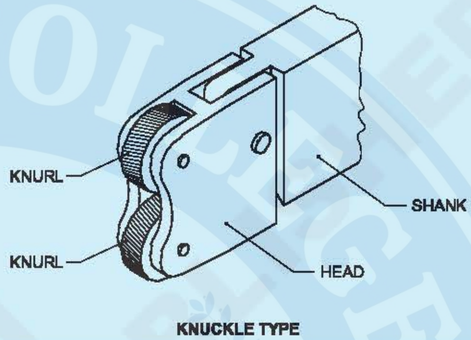
Fig 8



It has only one single roller which produces a straight lined pattern.

Knuckle joint type knurling tool-holders (Fig 9)

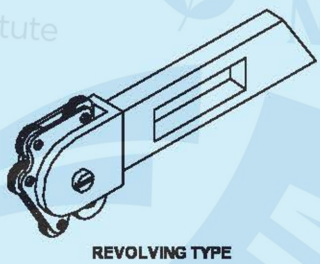
Fig 9



This tool holder has a set of two rollers of the same knurling pitch. The rollers may be of straight teeth or helical teeth. It is self-centering.

Revolving head knurling tool (Fig 10)

Fig 10



This tool-holder is also called a universal knurling tool-holder. It is fitted with 3 pairs of rollers having coarse, medium and fine pitches. These are mounted on a revolving head which pivots on a hardened steel pin. It is also self-centering.

Difference between different types of knurling tool-holders

Single roller	Knuckle joint	Revolving type
Only one roller is used	A pair of rollers are used	Three pairs of rollers are used
Only one pattern of knurling can be produced with this type of knurling tool-holder	Cross of diamond knurling pattern can be produced	Knurling patterns of different pitches can be produced
It is not self-centering	It is self-centering	It is self-centering

Knurling - Speed and Feed

The tables shown be used as a guide for determining the amount of end-feed or in-feed per revolution of the work. The rate of the feed for diamond pattern knurling is slower than that for straight or diagonal knurling.

Straight or Diagonal

End - FEED KNURLING

Approximate

FEED per REVOLUTION

T.P.I	Alum Brass	Mild Steel	Alloy Steel
12	.008"	.006"	.004"
16 - 20	.010"	.008"	.005"
25 - 35	.013"	.010"	.007"
40 - 80	.017"	.012"	.009"

Straight or Diagonal

IN - FEED KNURLING

Approximate

REVOLUTION

T.P.I	Alum Bras	Mild Steel	Alloy Steel
12	12	15	25
16-20	10	13	22
25-35	8	11	20
40-80	6	9	18

Standard tapers

Objectives: At the end of this lesson you shall be able to

- **define a taper**
- **state the uses of tapers**
- **state the method of expressing tapers**
- **state the methods to be adopted while specifying tapers**
- **distinguish between the features of self-holding and self-releasing tapers**
- **name the different types of self-holding tapers and state their features**
- **state the features of self-releasing tapers**
- **state the features of pin taper and keyway taper.**

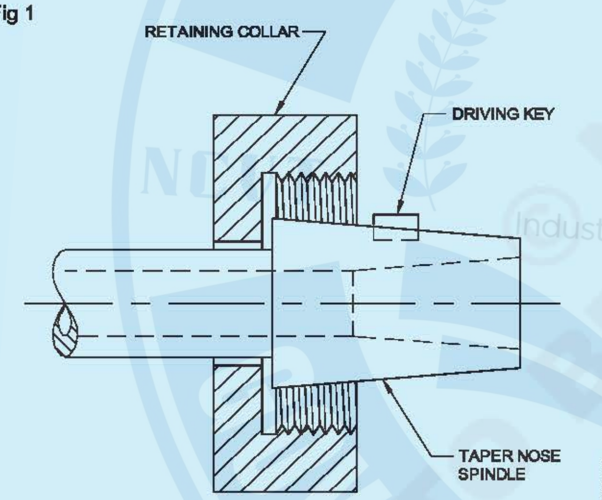
Definition of Taper: Taper is a gradual increase or decrease in the dimention along its length of the job.

Tapers are used for:

- Self-alignment/location of components in an assembly.
- Assembling and dismantling parts easily.
- Transmitting drive through assembly.

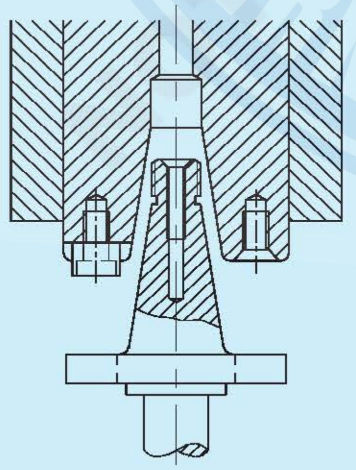
Tapers have a variety of applications in engineering assembly work.(Figs 1,2 & 3)

Fig 1



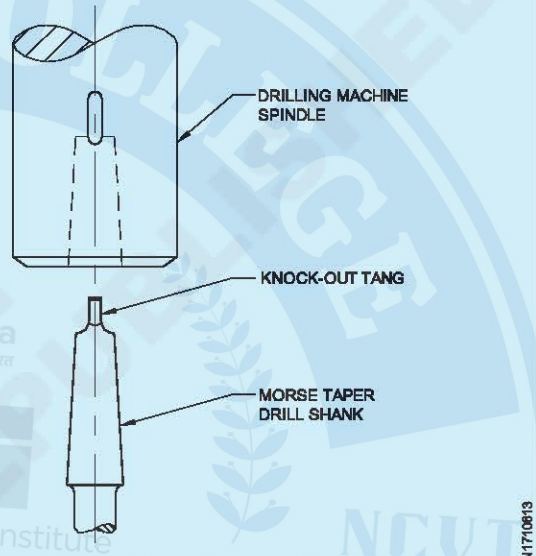
LATHE-TAPER NOSE SPINDLE

Fig 2



MILLING MACHINE SPINDLE

Fig 3

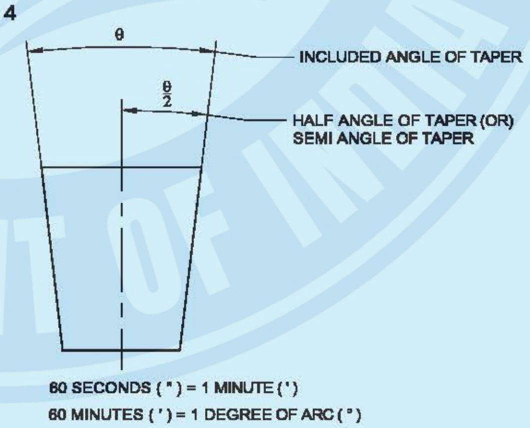


DRILLING MACHINE SPINDLE AND DRILL WITH TAPER SHANK

Tapers of components are expressed in two ways.

- Degree of arc (Fig 4)

Fig 4



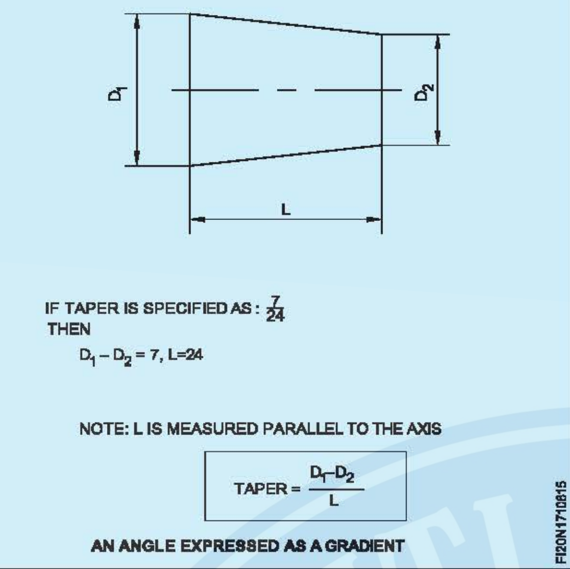
AN ANGLE EXPRESSED IN DEGREES OF ARC

- Gradient (Fig 5)

The method adopted for expressing tapers depends on:

- The steepness of the tapers
- The method adopted for measuring.

Fig 5



Specification of tapers

While specifying taper in drawings it should indicate the:

- Angle of the taper
- Size of the component. (Figs 6,7, 8 & 9)

Fig 6

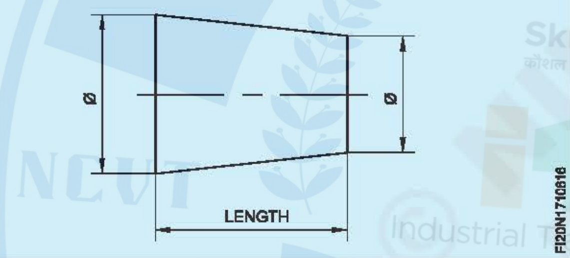


Fig 7

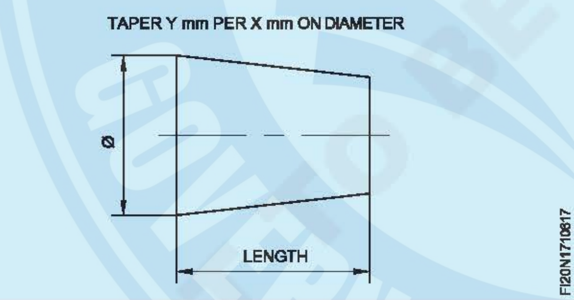


Fig 8

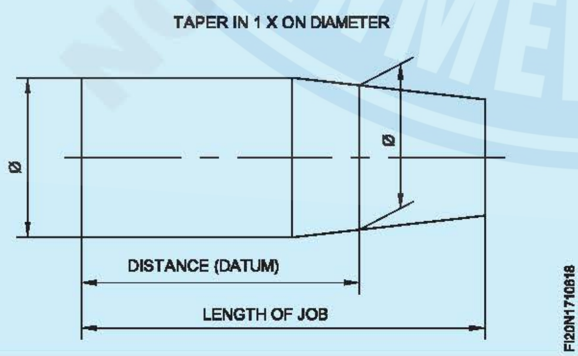
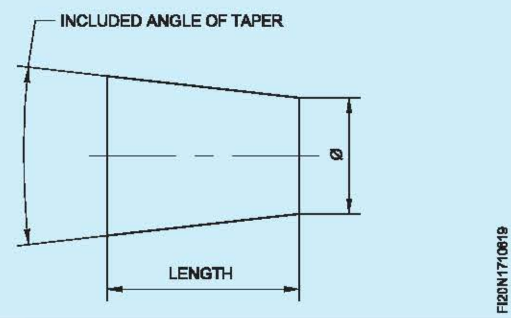


Fig 9



Standard tapers

Tapers for tool-holding

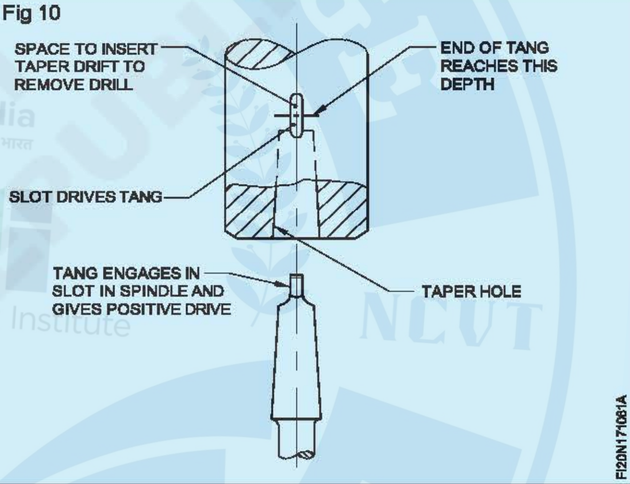
Two types of tapers are used for tool-holding on machines.

- Self-holding tapers
- Self-releasing tapers

Self-holding tapers

Self-holding tapers have less taper angle. These are used for holding and driving cutting tools like drills, reamers etc. without any locking device. (Fig 10)

Fig 10



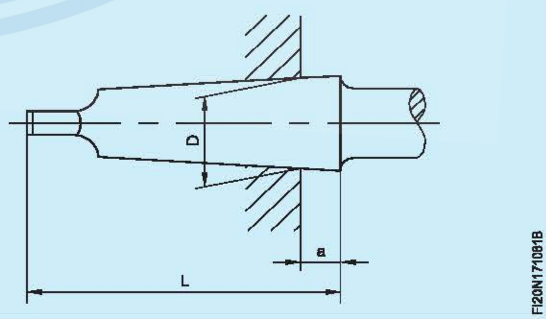
The standard tapers used for this are:

- The metric taper
- The morse taper.

Metric taper

The taper on diameter is 1:20. The commonly used shank sizes in metric tapers are metric 4, 6, 80, 100, 120, 160 and 200. The shank size indicating the metric taper is the diameter at D. (Fig 11)

Fig 11



Morse taper

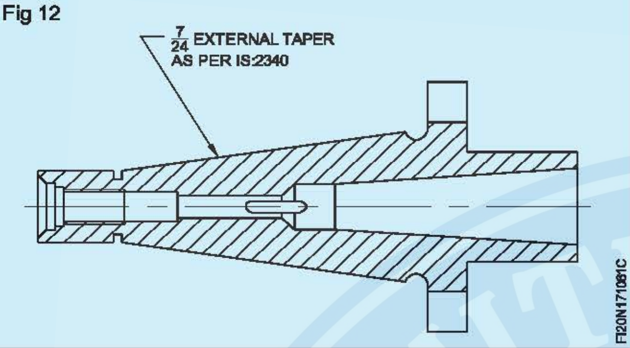
The commonly used taper shank sizes are:

0, 1, 2, 3, 4, 5 and 6.

The taper is varying according to the size of the Morse taper. It varies from 1:19.002 to 1:20.047.

Self-releasing 7/24 taper (Fig 12)

Fig 12



Spindle noses and arbors used on milling machines are usually provided with self-releasing tapers. The standard self-releasing taper is 7/24. This is a steep taper which helps in the correct location and release of the components in the assembly. This taper does not drive the mating component in the assembly. For the purpose of driving, additional features are provided.

The commonly used 7/24 taper sizes are: 30,40,45,50 and 60.

The taper of a 7/24 taper of No.30 will have a maximum diameter of (D) 31.75 mm and No.60, 107.950 mm. All other sizes fall within this range.

Tapers used in other assembly work

A variety of tapers are used in engineering assembly work. The most common ones are:

- pin taper
- key and keyway taper.

Pin taper

This is the taper used for taper pins used in assembly. (Fig 13)

The taper is 1:50.

The diameter of taper pins is specified by the small diameter.

Taper pins help in assembling and dismantling of components without disturbing the location.

Key and keyway tapers

This taper is 1:100. This taper is used on keys and keyways. (Figs 14 and 15)

Note: For further information about the tapers used for special application refer to: IS: 3458 - 1981.

Fig 13

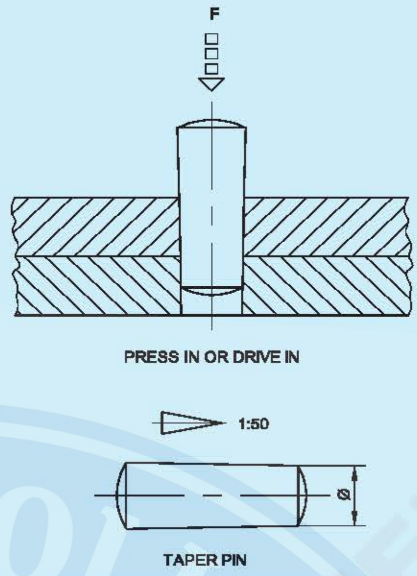


Fig 14

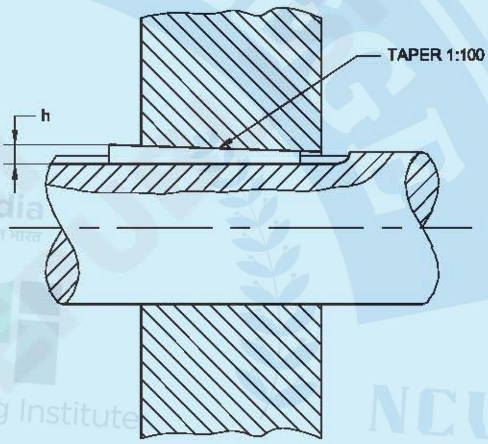
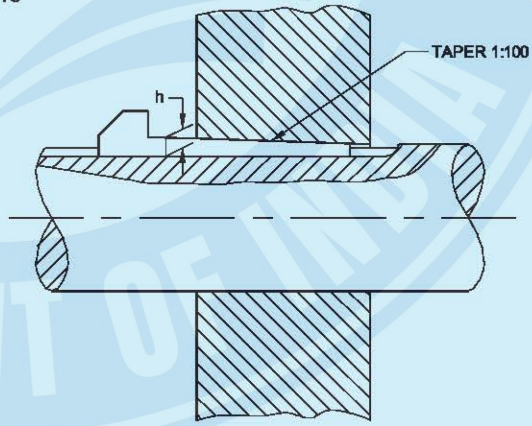


Fig 15



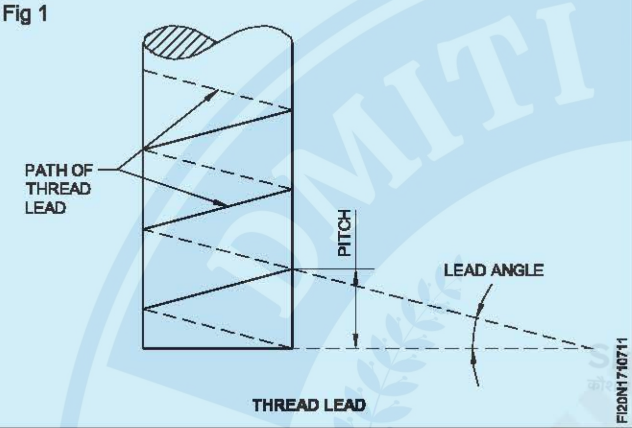
Screw thread

Objectives: At the end of this lesson you shall be able to
- **define screw thread**
- **state the use of screw thread.**

Definition

Thread is a ridge of uniform cross-section which follows the path of a helix around the cylinder or cone, either externally or internally. (Fig 1)

Fig 1



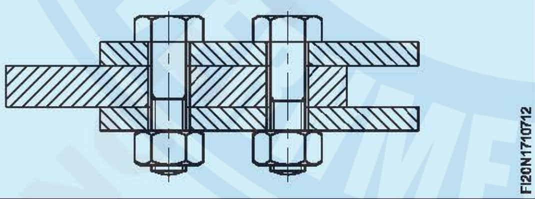
Helix is a type of curve generated by a point which is moving at a uniform speed around the cylinder or cone and at the same time, moves at a uniform speed parallel to the axis. (Fig 1)

Uses of Screw threads

Screw threads are used

- As fasteners to hold together and dismantle components when needed. (Fig 2)

Fig 2

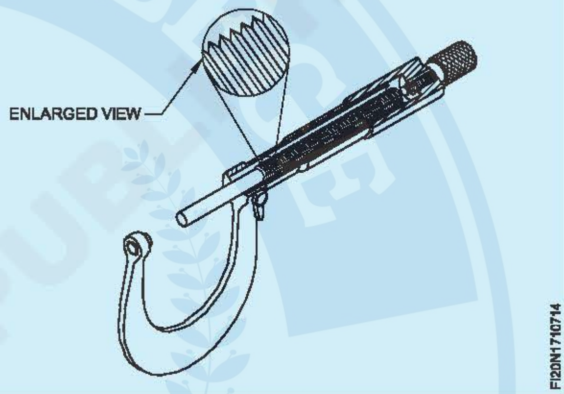


- To transmit motion on machines from one unit to another. (Fig 3)

Fig 3

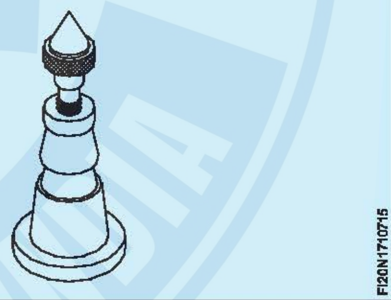
- To make accurate measurements. (Fig 4)

Fig 4



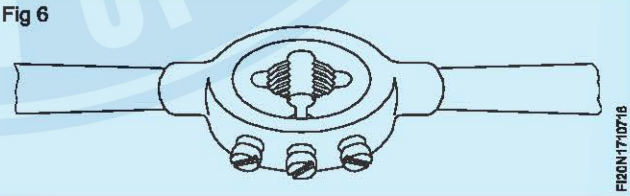
- To apply pressure. (Fig 5)

Fig 5



- To make adjustments. (Fig 6)

Fig 6



Square, worm, buttress and acme threads

Objectives: At the end of this lesson you shall be able to
- **identify square thread and specify its uses**
- **state the relationship between the pitch and the other elements of square threads**
- **identify the modified square thread and its applications**
- **identify the different forms of trapezoidal threads and their uses**
- **state the relationship between the pitch and the other elements of all the different forms of trapezoidal threads.**

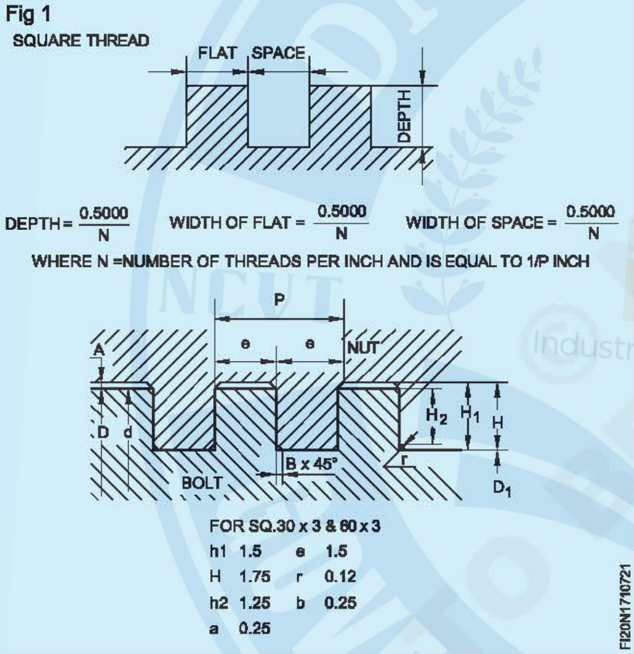
Square and trapezoidal threads

Square and trapezoidal threads have more cross-sectional area than 'V' threads. They are more suitable to transmit motion or power than 'V' threads. They are not used for fastening purposes.

Square thread

In this thread the flanks are perpendicular to the axis of the thread. The relationship between the pitch and the other elements is shown in Fig 1.

Fig 1



Square threads are used for transmitting motion or power. Eg. screw jack, vice handles, cross-slide and compound slide, activating screwed shafts.

Designation

A square thread of nominal dia. 60mm and pitch 9mm shall be designated as Sq. 60 x 9 IS: 4694-1968. The dimensions a, b, e, p, H_1, h_1, h_2 & d_1 are changed as per thread series (fine, normal & coarse).

Modified square thread

Modified square threads are similar to ordinary square threads except for the depth of the thread. The depth of thread is less than half pitch of the thread. The depth varies according to the application. The crest of the thread is chamfered at both ends to 45° to avoid the formation of burrs. These threads are used where quick motion is required.

Trapezoidal threads

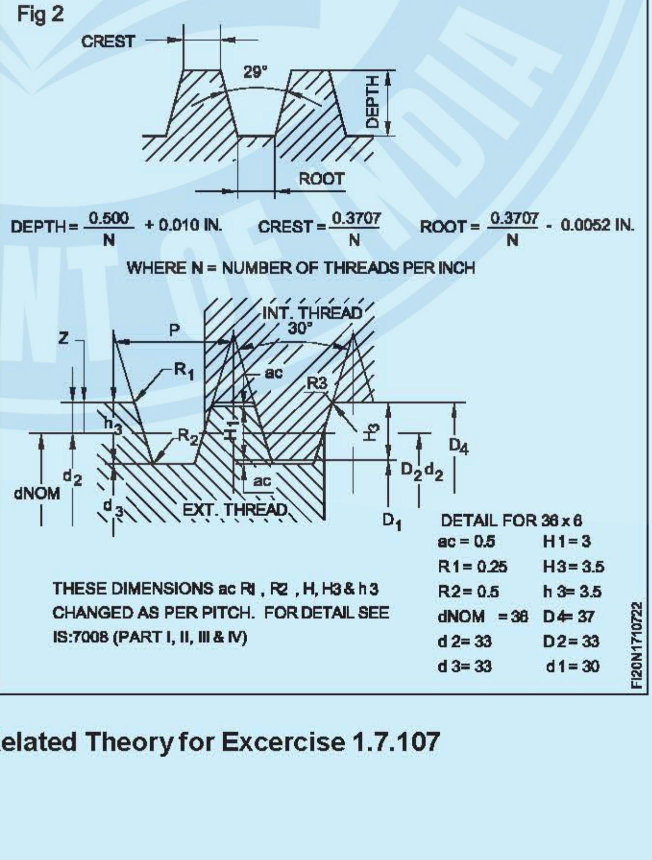
These threads have a profile which is neither square nor 'V' thread form and have a form of trapezoid. They are used to transmit motion or power. The different forms of trapezoidal threads are:

- Acme thread
- Buttress thread
- Saw-tooth thread
- Worm thread.

Acme thread (Fig 2)

This thread is a modification of the square thread. It has an included angle of 29°. It is preferred for many jobs because it is fairly easy to machine.

Fig 2



Acme threads are used in lathe lead screws. This form of thread enables the easy engagement of the half nut. The metric acme thread has an included angle of 30°. The relationship between the pitch and the various elements is shown in the figure.

Buttress thread (Fig 3)

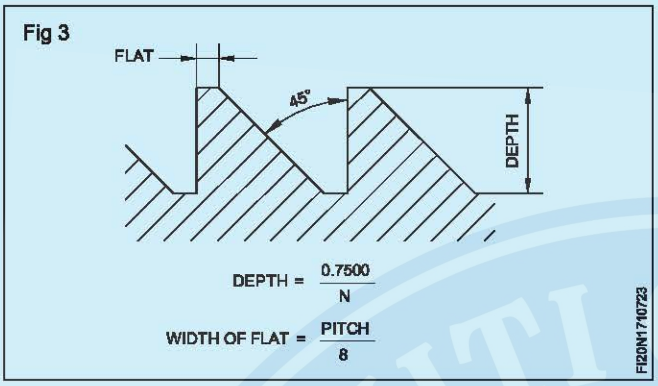


Fig 3

In buttress thread one flank is perpendicular to the axis of the thread and the other flank is at 45°. These threads are used on the parts where pressure acts at one flank of the thread during transmission. Figure 3 shows the various elements of a buttress thread. These threads are used in power press, carpentry vices, gun breeches, ratchets etc.

Buttress thread as per B.I.S. (Fig 4)

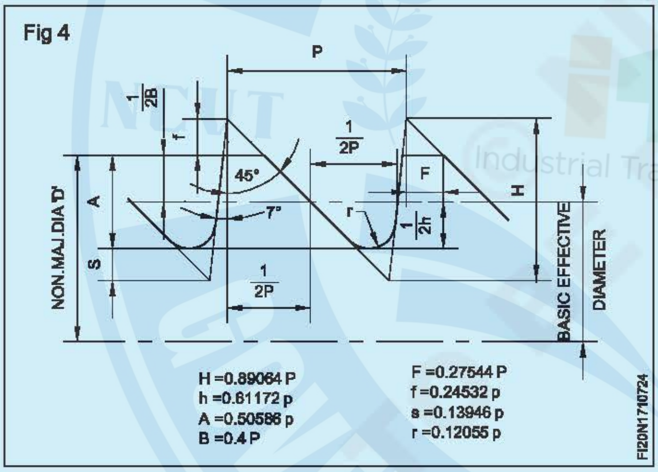


Fig 4

This is a modified form of the buttress thread. Figure 4 shows the various elements of the buttress thread. The bearing flank is inclined by 7° as per B.I.S. and the other flank has a 45° inclination.

Saw-tooth thread as per B.I.S. 4696

This is a modified form of buttress thread. In this thread, the flank taking the load is inclined at an angle of 3°, whereas the other flank is inclined at 30°. The basic profile of the thread illustrates this phenomenon. (Fig 5) The proportionate values of the dimensions with respect to the pitch are shown in Figs 6 and 7.

The equations associated with the dimensions indicated in the two figures (Figs 6 and 7) are given below.

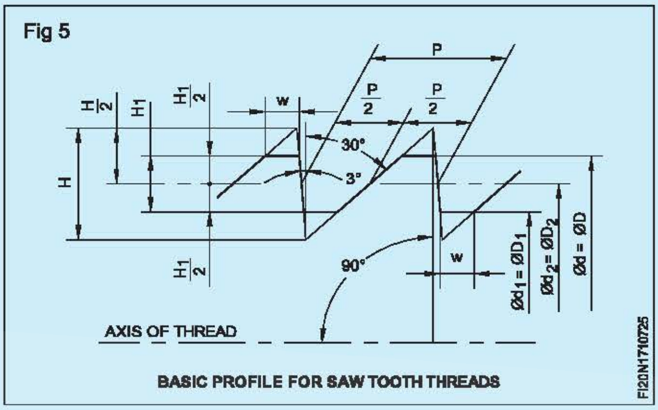


Fig 5

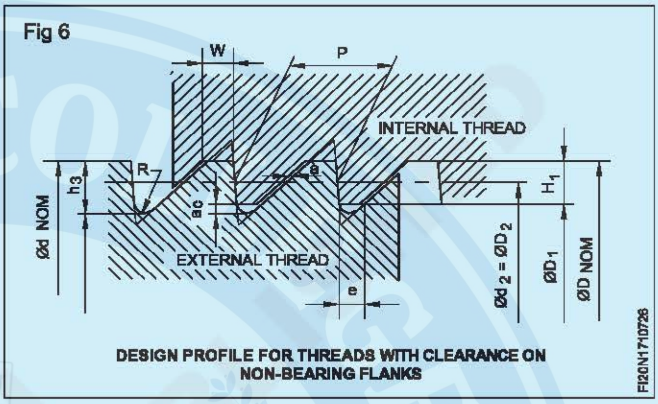


Fig 6

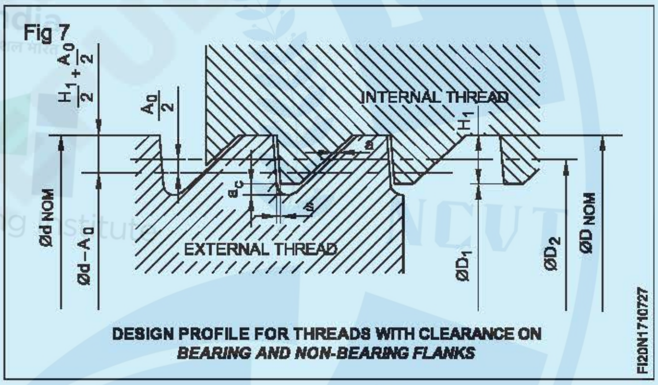


Fig 7

$H_1 = 0.75\ P$

$h_3 = H_1 + a_c = 0.867\ 77\ P$

$a = 0.1\ \sqrt{P}$ (axial play)

$a_c = 0.117\ 77\ P$

$W = 0.263\ 84\ P$

$e = 0.263\ 84\ P - 0.1\ \sqrt{P} = W - a$

$R = 0.124\ 27\ P$

$D_1 = d - 2\ H_1 = d - 1.5\ P$

$d_3 = d - 2\ h_3$

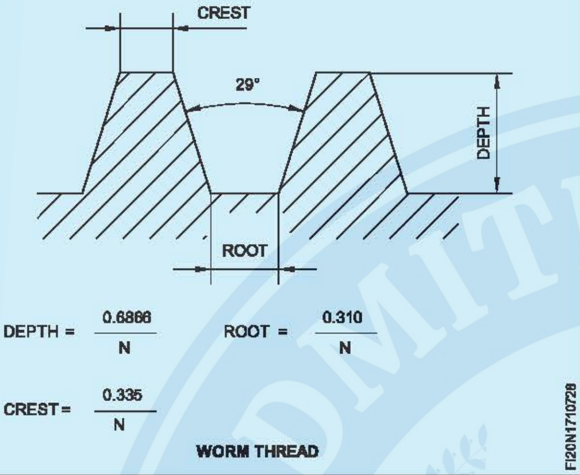
$d_2 = D_2 = d - 0.75\ P$

S = 0.314 99 A_o, where A_o = basic deviation (= upper deviation) for external thread in the pitch diameter.

Worm thread

This is similar to acme thread in shape but the depth of thread is more than that of acme thread. This thread is cut on the worm shaft which engages with the worm wheel. Figure 8 shows the elements of a worm thread.

Fig 8



WORM THREAD

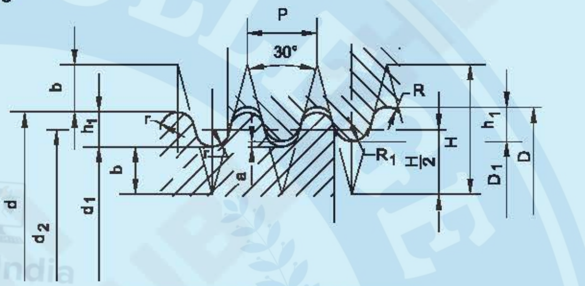
The worm wheel and worm shaft are used in places where motion is to be transmitted between shafts at right angles. It also gives a high rate of speed reduction. The worm wheel is generally cut by diametral pitch (D.P) or module pitch cutters. Diametral pitch (D.P) is the ratio between the number of teeth to the pitch diameter (P.D.) of the gear. Module is the ratio between the pitch diameter of the gear and the number of teeth of the gear.

The linear pitch of the worm thread must be equal to the circular pitch of the worm gear. When the worm gear is of D.P. then the linear pitch of the worm thread in mesh is equal to p/DP. When the worm gear is of module teeth, then the linear pitch of the worm thread is equal to module x p. In some of the lathes, a chart illustrates the position of levers of the quick change gearbox together with the change gear connections for cutting D.P. or module worm threads.

Knuckle threads

The shape of the knuckle thread is not trapezoidal but it has a rounded shape. It has limited application. The figure shows the form of knuckle thread. It is not sensitive against damage as it is rounded. It is used for valve spindles, railway carriage couplings, hose connections etc. (Fig 9)

Fig 9



BASIC PROFILE OF KNUCKLE THREAD

b = 0.68301.P
h_1 = 0.5P
a = 0.05P

DESIGNATION:- A KNUCKLE THREAD OF DIAMETER 60 mm & PITCH 4.233mm SHALL BE DESIGNATED AS k60x4.233 IS:4695

FI20N1710729

Principle of cutting screw thread in centre lathe

Objectives: At the end of this lesson you shall be able to
- **state the principle of thread cutting by a single point tool**
- **list the parts involved in the thread cutting mechanism and state their functions**
- **derive formula for change gear calculation.**

Principle of thread cutting

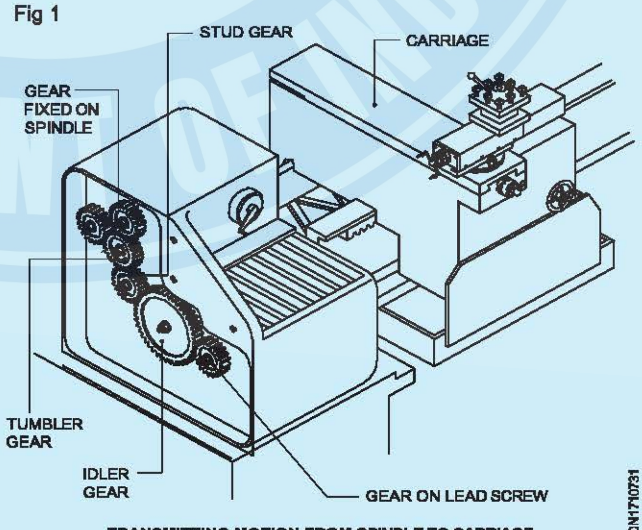
The principle of thread cutting involves producing a uniform helical groove on a cylindrical or conical surface by rotating the job at a constant speed, and moving the tool longitudinally at a rate equal to the pitch of the thread, per revolution of the job.

The cutting tool moves with the lathe carriage by the engagement of a half nut with the lead screw. The shape of the thread profile on the work is the same as that of the tool ground. The direction of rotation of the lead screw determines the hand of the thread being cut.

Parts involved in thread cutting

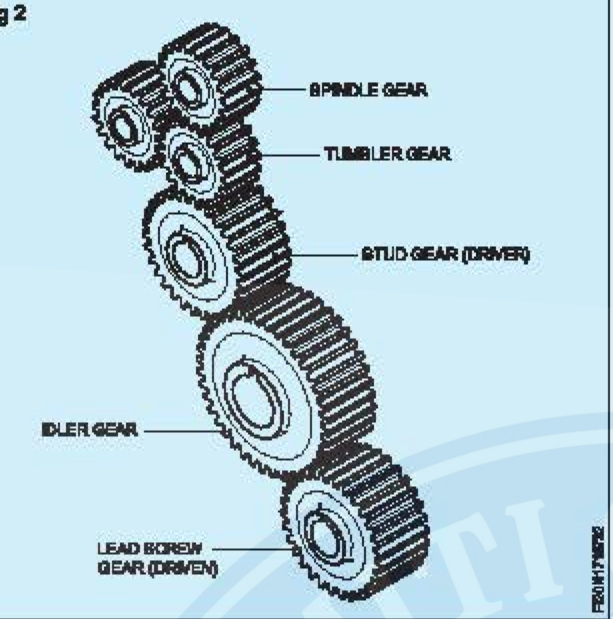
Figures 1 & 2 illustrate how the drive is transmitted from the spindle to the lead screw through a change gear arrangement. From the lead screw the motion is transmitted to the carriage by engaging the half nut with the lead screw.

Fig 1



TRANSMITTING MOTION FROM SPINDLE TO CARRIAGE

Fig 2



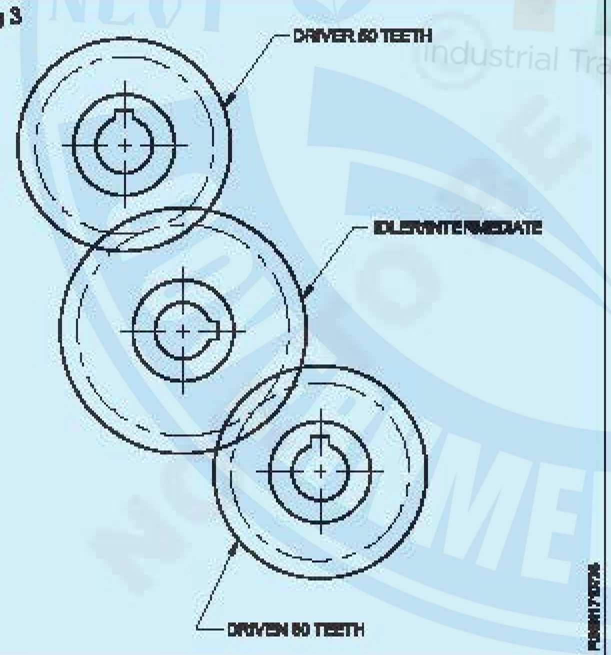
Derivation of the formula for change gears

Example

CASE 1 : To cut 4 mm pitch (lead) thread on the job in a lathe having a lead screw of 4 mm pitch.

When the job rotates once, the lead screw should make one revolution to move the tool by 4 mm. Hence, if the stud gear (Driver) has a 50 teeth wheel, the lead screw should be fixed with a gear of 50 teeth (Driven) to get the same number of revolutions as the spindle. (Fig 3)

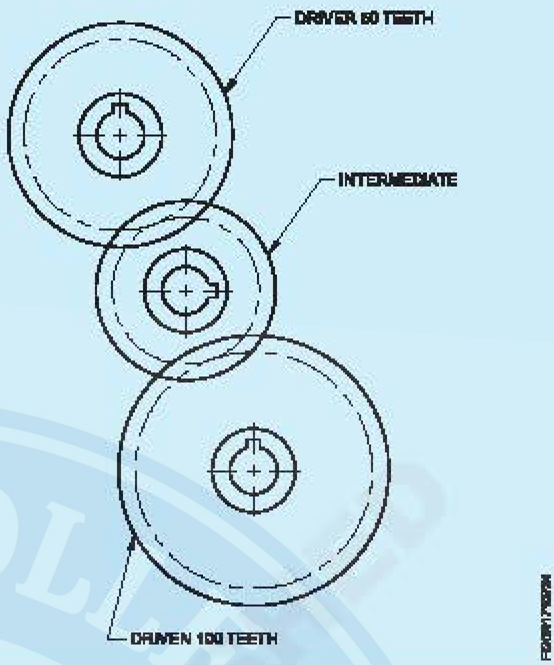
Fig 3



CASE 2 : To cut 2 mm pitch threads instead of 4 mm in the same lathe.

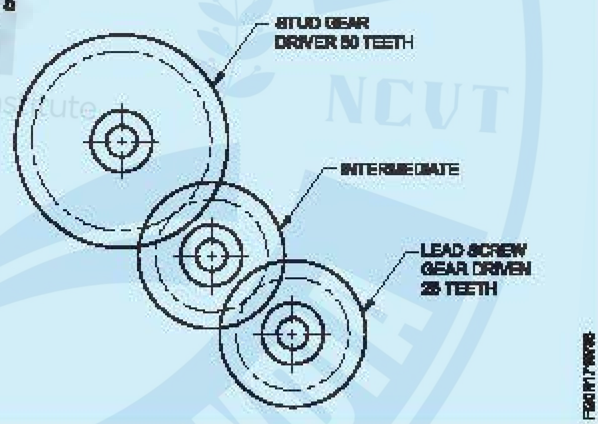
When the job makes one rotation, the lead screw should rotate 1/2 revolution so that the lead screw rotation is slower. Therefore, the driven wheel (lead screw gear) should be of 100 teeth if the driver (stud gear) is of 50 teeth. (Fig 4)

Fig 4



CASE 3 : If we have to cut a 8 mm pitch thread on a job, with a 4mm lead screw pitch, the tool should move 8 mm per revolution of the job. The lead screw should rotate 2 revolutions when the job makes one rotation, making the LS to run twice as fast as the spindle. So the driven wheel (lead screw gear) should be of 25 teeth if the driver wheel is of 50 teeth. (Fig 5)

Fig 5



Let us compare the above three examples.

Examples :	Case 1	Case 2	Case 3
Pitch(Lead) of job	4	2	8
Pitch(Lead) of L.S	4	4	4
Driver	50	50	50
Driven	50	100	25

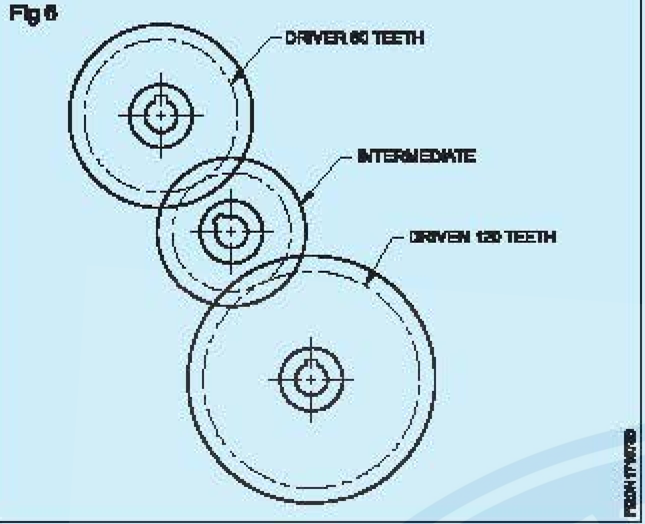
Stating the above in a formula,

$$\text{The gear ratio} = \frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of work}}{\text{Lead of lead screw}}$$

Solved examples

1 Find the change gears required to cut a 3 mm pitch on a job in a lathe, having a lead screw of 6 mm pitch. (Fig 6)

Fig 6



Ratio = Driver = Lead of work

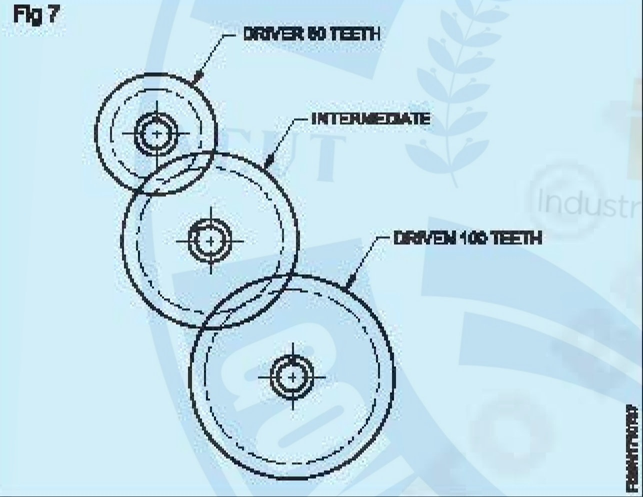
$$\text{The gear ratio} = \frac{3}{6} = \frac{3 \times 20}{6 \times 20} = \frac{60}{120}$$

Driver = 60 teeth

Driven = 120 teeth

2 Find the change gears required to cut a 2.5 mm pitch in a lathe, having a lead screw of 5 mm pitch. (Fig 7)

Fig 7



$$\text{Ratio} = \frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of work}}{\text{Lead of lead Screw}}$$

$$= \frac{2.5}{5} = \frac{2.5 \times 20}{5 \times 20}$$

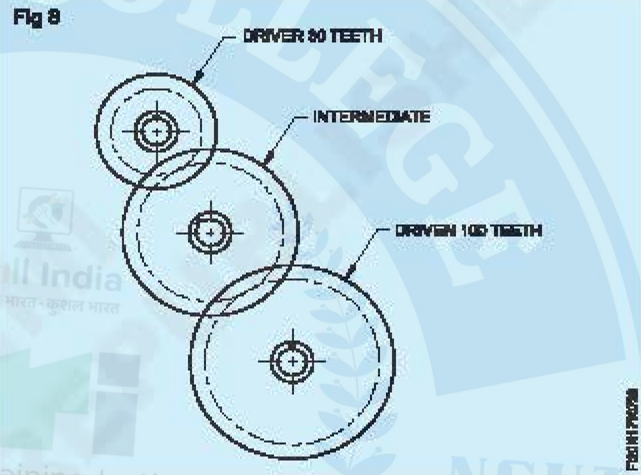
$$= \frac{50\ (\text{Driver})}{100\ (\text{Driven})}$$

3 Calculate the gears required to cut a 1.5 mm pitch in a lathe having a lead screw of 5 mm pitch. (Fig 8)

$$= \frac{1.5}{5} = \frac{3}{10} = \frac{3 \times 10}{10 \times 10}$$

$$= \frac{30\ (\text{Driver})}{100\ (\text{Driven})}$$

Fig 8



Principle of chasing screw thread

Objectives: At the end of this lesson you shall be able to
- **state the necessity of a thread chasing dial**
- **state the constructional details of a British thread chasing dial**
- **state the functional features of a British thread chasing dial.**

Thread chasing dial

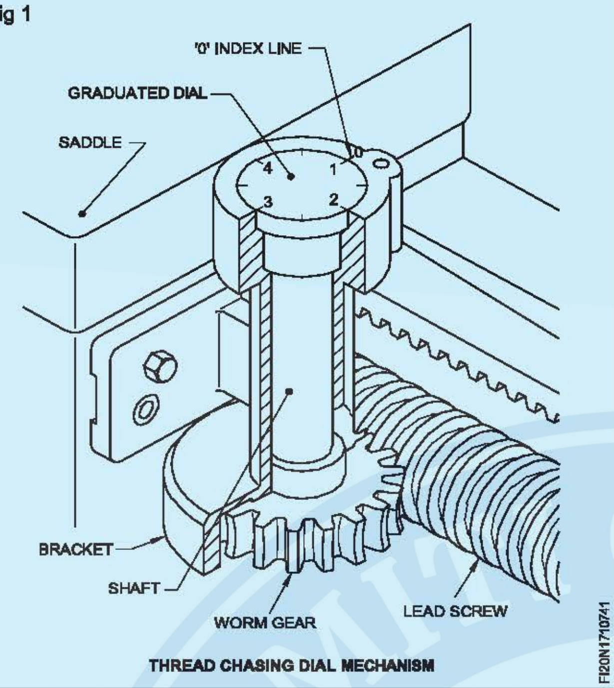
To catch the thread quickly and to save manual labour, use of a chasing dial is very common during thread cutting by a single point cutting tool. A thread chasing dial is an accessory.

Constructional details (Fig 1)

The figure shows constructional details of a British thread chasing dial. It consists of a vertical shaft with a worm wheel made out of brass or bronze, attached to the shaft at the bottom. On the top, it has a graduated dial. The shaft is carried on a bracket in bearing (bush) which is fixed to the carriage. The worm wheel can be brought into an engaged or disengaged position with the lead screw as needed. When the lead screw rotates it drives the worm wheel which causes the dial to rotate. The movement of the dial is with reference to the fixed mark ('O' index line).

The face of the dial is usually graduated into eight (8) divisions, having 4 numbered main divisions and 4 unnumbered subdivisions in between.

Fig 1



THREAD CHASING DIAL MECHANISM

The number of teeth on the worm gear is the product of the number of threads per inch on the lead screw and the number of numbered divisions on the dial.

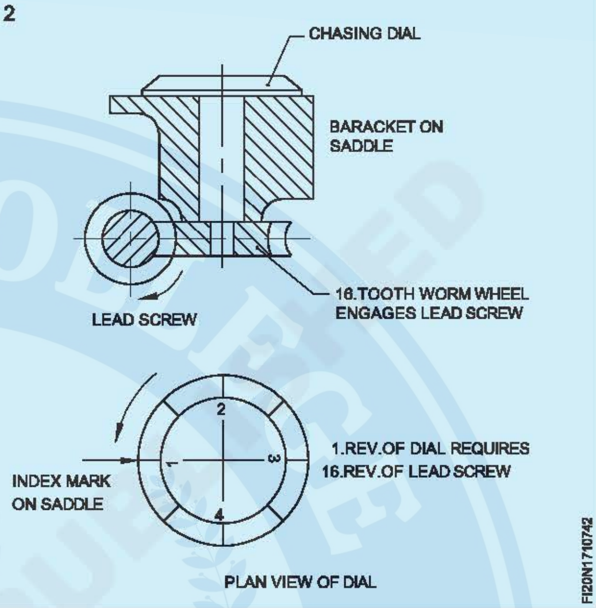
Each numbered division represents 1 inch travel of the carriage.

Let the worm wheel have 16 teeth, and the lead screw 4 TPI. The number of numbered graduations and unnumbered graduations are 4 each.

The half nut can be engaged 8 times for one revolution of the graduated dial. The movement of the carriage for one complete revolution of the dial is 4''. (Fig 2) Since the dial is having totally 8 graduations marked, each graduation represents 1/2'' travel of the carriage.

The chart given here shows the positions at which the half nut is to be engaged when cutting different threads per inch, when a British thread chasing dial with the above data is fitted to the lathe.

Fig 2



THREAD CHASING DIAL CHART

Threads per inch to be cut	Dial graduation at which the half nut can be engaged to catch the thread	Reading on the dial illustrated
Threads which are a multiple of the number of threads per inch of the lead screw.	Engage at any position the half nut meshes.	Use of dial unnecessary.

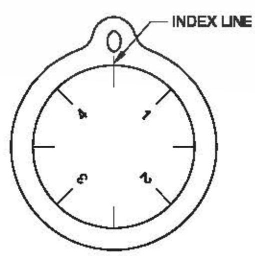
Example T.P.I. to be cut - 8

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I to be cut}} = \frac{4}{8} = \frac{1}{2}$$

Predetermined travel $= 1 \times \frac{1''}{4} = \frac{1''}{4}$

The predetermined travel of 1/4'' is represented by the dial position in the exact middle between any numbered division and adjacent un-numbered division. The half nut engagement can be done at any position at which it can be engaged (ie. 16 positions).

Referring to the dial is not necessary.

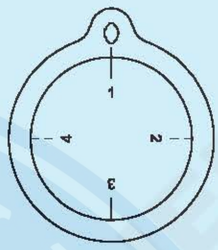
Even number of threads	Engage at any graduation on the dial. 8 positions	1 1 1/2 2 2 1/2 3 3 1/2 4 4 1/2	

Example T.P.I. to be cut - 6

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I to be cut}} = \frac{4}{6} = \frac{2}{3}$$

Predetermined travel $= 2 \times \frac{1''}{4} = \frac{1''}{2}$

The predetermined travel of 1/2'' is represented by dial movement from any numbered division to the next adjacent unnumbered division. The half nut can be engaged when any numbered or unnumbered graduation coincides with the zero line (8 positions).

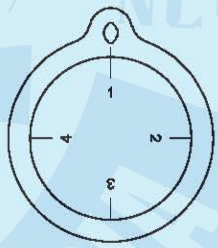
Odd number of threads	Engage at any main division. 4 positions	1 2 3 4	

Example T.P.I. to be cut - 5

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I. to be cut}} = \frac{4}{5} = \frac{4}{5}$$

Predetermined travel $= 4 \times \frac{1''}{4} = 1''$

The predetermined travel of 1'' is represented by the dial movement from any numbered division to the next numbered division or from any unnumbered division to the next unnumbered division. Therefore, if the first cut is taken when a numbered division of the dial coincides with zero, then the half nut engagement for successive cuts can be done when any numbered division coincides with the zero mark. If the first cut is taken when an unnumbered division coincides with the zero, then the half nut for successive cuts, is engaged when any unnumbered division coincides with the zero. (4 positions)

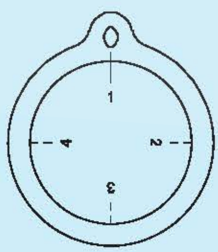
Half fractional number of threads	Engage at every other main division. 2 positions	1 & 3 or 2 & 4	

Example T.P.I. to be cut - 3 1/2

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I. to be cut}} = \frac{4}{3\ 1/2} = \frac{8}{7}$$

Predetermined travel $= 8 \times \frac{1''}{4} = 2''$

The half nut can be engaged only at opposite numbered or unnumbered graduations (2 positions).

Quarter fractional number of threads	Engage at the same main division. 1 position	1 or 2 or 3 or 4	

Example T.P.I. to be cut - 2 3/4

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I. to be cut}} = \frac{4}{2\,3/4} = \frac{16}{11}$$

Predetermined travel = 16 x 1"/4 = 4"

The half nut can be engaged to catch the thread only when the same numbered or unnumbered graduated line, at which the first cut is taken, coincides with the zero line (1 position only).

Example T.P.I. to be cut - 1 3/8

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I. to be cut}} = \frac{4}{1\,3/8} = \frac{32}{11}$$

Predetermined travel = 16 x 1"/4 = 4"

The half nut engaged for the first cut should remain at the engaged position till thread cutting is completed and the machine is reversed as it takes a long time to cover the predetermined travel arrived at by calculation.

Example T.P.I. to be cut - 1 3/8

$$\frac{DR}{DN} = \frac{\text{T.P.I. on lead screw}}{\text{T.P.I. to be cut}} = \frac{4}{1\,3/8} = \frac{32}{11}$$

Predetermined travel = 32 x 1"/4 = 8"

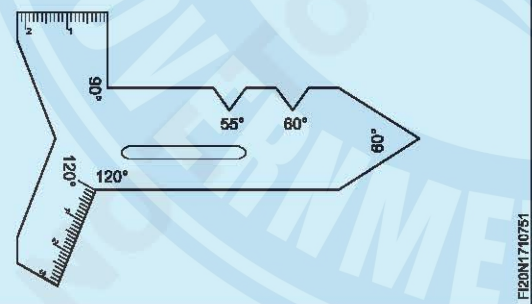
The half nut engaged for the first cut should remain at the engaged position till thread cutting is completed and the machine is reversed as it takes a long time to cover the predetermined travel arrived at by calculation.

Centre gauge

Objectives: At the end of this lesson you shall be able to
- **define centre gauge**
- **write the uses of centre gauge.**

Centre gauge: (Fig 1)

Fig 1



Centre gauges and fish tail gauges are gauges used in lathe work for checking the angles when grinding the profiles of single point screw cutting tool bits and centers. In the image, the gauge on the left is called a fishtail gauge or centre gauge, and the one on the right is another style of center gauge.

These gauges are most commonly used when hand grinding threading tool bits on a bench grinder, although they may be used with tool and cutter grinders.

When the tool bit has been ground to the correct angle, they may then be used to set the tool perpendicular to the workpiece.

They can incorporate a range of sizes and types on the one gauge, the two most common being metric or UNS at 60^0, and BSW at 55^0. Gauges also exist for the acme thread form.

Tool setting - external thread

Objective: At the end of this lesson you shall be able to
- **tool setting to cut external thread by half angle method.**

Check the diameter of the workpiece to be threaded by referring to the drawing.

> **To provide thread clearance, it is good practice to turn the diameter of the workpiece undersize depending upon the requrired.**

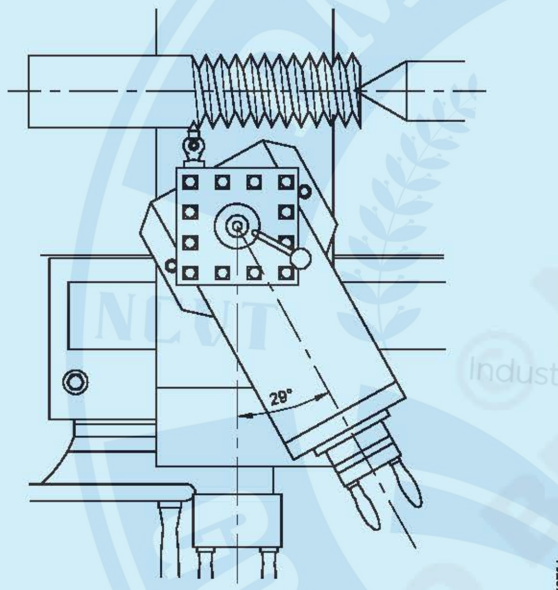
Set the lathe spindle speed to about one fourth of the turning speed.

Set the gerarbox according to the pitch of thread to be cut.

Swivel the compound slide to 90° from the horizontal position to bring it in line with the cross-slide.

Swivel to the right 1° less than the half included angle of the thread it is a right hand thread. (Fig 1)

Fig 1



COMPOUND REST SET AT 29° TO THE RIGHT TO CUT A 60° INCLUDED ANGLE RIGHT HAND THREAD

> **The angle to which the compound rest is set affects the cutting action of the cutting tool by producing a shearing action on the trailing edge of the tool. This produces a smooth cut.**

Set the tool in the tool post with a minimum overhand perpendicular to the axis and also set with a centre gauge. (Fig 2)

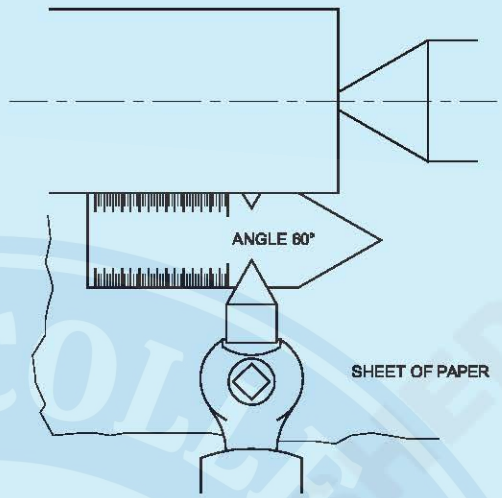
Mark out the length of the workpiece to be threaded.

Chamfer the end of the workpiece surface with the leading edge of the cutting tool to a depth, just greater than the minor diameter of the thread to be cut.

Advance the cutting tool to the work surface by operating the cross-slide hand wheel.

When the tip of the tool just touches the work surface, stop further advancement and set the cross-slide and compound slide graduated collars to zero.

Fig 2



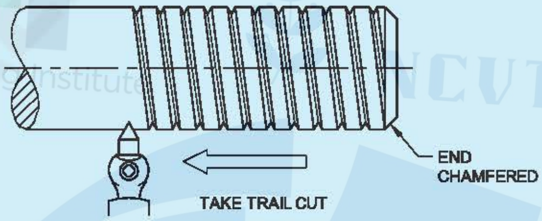
Move the carriage to the right until the end of the tool clears the work.

Feed the tool in about 0.1 mm using the top slide hand wheel.

Engage the half nut referring to be chasing dial.

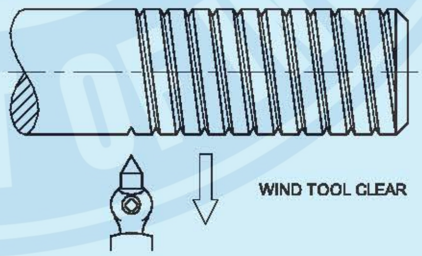
Take a trial cut along the workpiece to be threaded. (Fig 3)

Fig 3



At the end of the trial cut, withdraw the tool immediately, winding it clear off the workpiece by operating the cross slide hand wheel and simultaneously reversing the machine. (Fig 4)

Fig 4



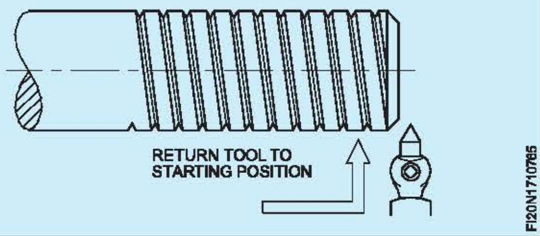
Allow the carriage to move to the right till it is cleared from the end of the work, and stop the machine. (Fig 5)

Check the thread formation with a pitch gauge.

Advance the tool by the cross-slide hand wheel toll zero position.

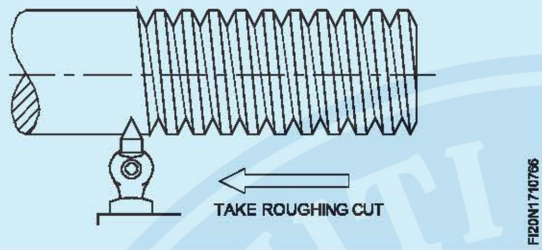
Give depth of cut with the top slide handle.

Fig 5



Start the machine and allow the tool to cut the thread. (Fig 6)

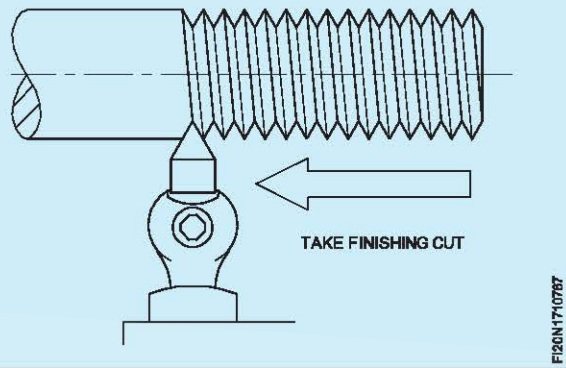
Fig 6



Use plenty of coolant during threading.

Repeat the steps till the required depth is reached. (Fig 7)

Fig 7



Note: At the end of each cut, the tool is withdrawn from the work by the cross-slide hand wheel and the carriage is brought to the starting point. The cross-slide hand wheel is brought to zero position and a depth of cut is given by the top slide.

Cutting an internal thread

Objectives: At the end of this lesson you shall be able to
- **tool setting to cut an internal thread.**

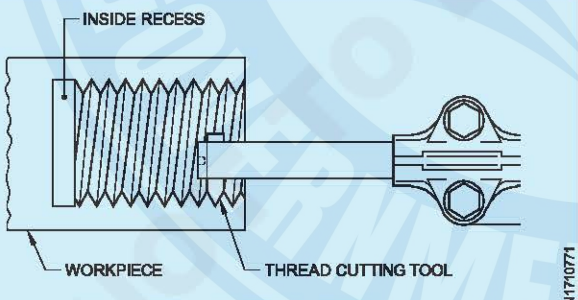
Mount the job on four jaw chick/three jaw chuck/ collect.

Drill and bore the job to the core diameter of the thread to required length/through hole.

For a blind hole, cut a recess at the end of the bore enough to permit the cutting tool to clear thread.

The recess must be larger than the major diameter of the thread. (Fig 1)

Fig 1



CUTTING AN INTERNAL THREAD

Chamfer the front end to 2x45°.

Set the compound rest at 29° to cut 60° included angle as shown in Fig 2.

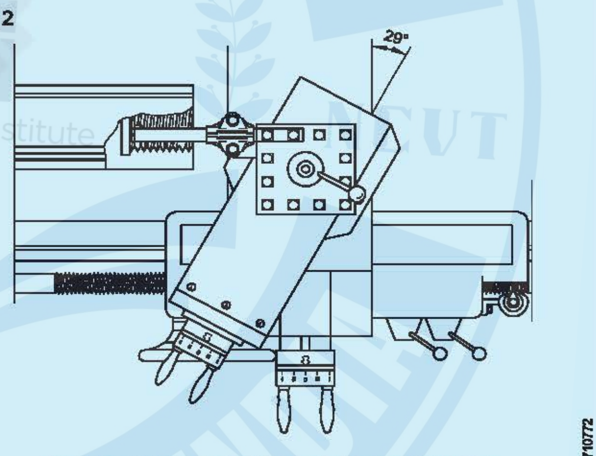
Set the gear box levers to the required pitch.

Fix the correctly ground threading tool in a boring bar.

Fix the boring bar parallel to the lathe centre line and set the point of the cutting tool to lie on the centre.

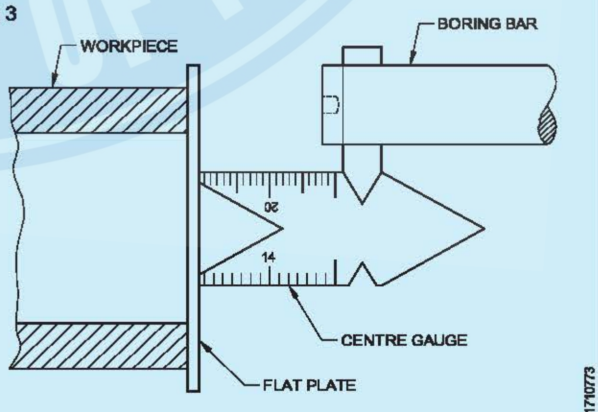
Align the cutting tool with a help of centre gauge as shown in Fig 3.

Fig 2



COMPOUND REST SET AT 29° TO THE LEFT TO CUT A 60° INCLUDED ANGLE INTERNAL THREAD

Fig 3



ALIGNING THE CUTTING TOOL WITH A CENTRE GAUGE

Mark the boring bar to indicate the required depth to entry into the bore.

Ensure that the boring bar does not foul anywhere on the job.

Reverse the cross slide until the tool point just touches the bore.

Set the cross-slide and compound slide graduated collars to zero.

Withdraw the cutting tool from the bore.

Set the spindle speed to 1/3 of the calculated r.p.m.

Start the machine.

Adjust the depth of cut to 0.1 mm.

Engage the half nut.

At the end of the cut, simultaneously reverse the chunk and clear the tool just away from the thread.

Ensure that the tool should not touch the thread in both side of the bore.

When cutting tool comes out of the bore stop the machine.

Give the depth of cut and run the machine in forward direction. Similarly finish the thread until final depth is achieved.

Check the finished thread with a thread plug gauge or a threaded bolt.

Screw pitch gauge

Objectives: At the end of this lesson you shall be able to
- **state the purpose of a screw pitch gauge**
- **state the features of a screw pitch gauge.**

For obtaining accurate results while using the screw pitch gauge, the full length of the blade should be placed on the threads. (Fig 1)

Fig 1