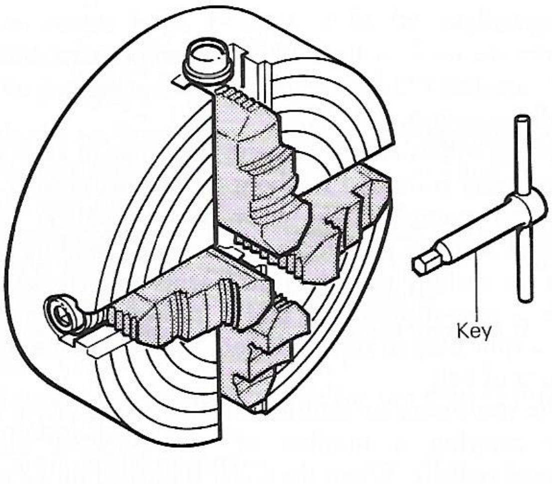

Figure 13.21 Four-jaw chuck.

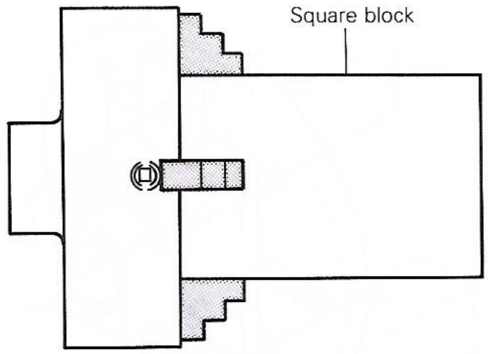

Figure 13.22 Work in four-jaw chuck.

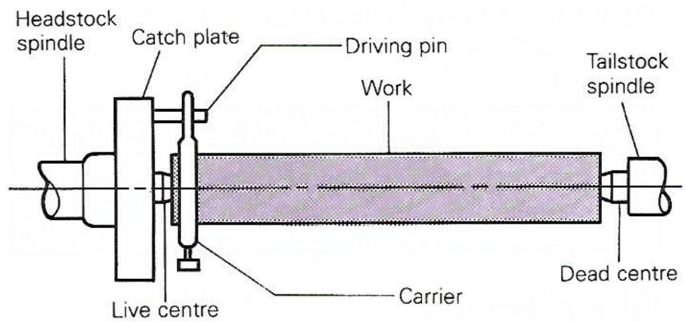

Figure 13.23 Work between centres.

> Note that this jaw is only suitable for holding round and hexagonal bars; it cannot grip irregular shapes, or geometrical shapes such as square and octagonal bars.

Using the four-jaw chuck

The four-jaw chuck (Figure 13.21) is also a geared scroll type. It has four jaws that move independently of each other. It can therefore be used to hold work of any shape (Figure 13.22). A high degree of concentricity can be obtained with work held in a four-jaw chuck compared with the three-jaw chuck, although it needs skill and takes longer to set up work.

The procedure for holding work in the four-jaw chuck is as follows.

1. Open the jaws independently to accommodate the work.
2. Insert the work and adjust the jaws to just touch it.
3. Check the alignment or concentricity with a scribing block or a dial indicator.
4. Make necessary adjustments and then grip the work firmly. Remove the key.
5. Turn the machine on and carry on the operation.

Holding work between centres

Workpieces that project a long way from the chuck tend to wobble, making it difficult to carry out operations. To avoid this problem, the work is held between two centres, one fitted in the headstock spindle and the other in the tailstock (Figure 13.23).

A high degree of concentricity can be obtained with this method of holding work, provided the centres are correctly aligned. You can carry out an alignment test by using a parallel test mandrel and a dial test indicator, or by sending the tailstock centre forward to touch the headstock centre, and using the offset adjustment screw on the tailstock to make the necessary adjustment. The test must be done before the work is supported on the centres.

The procedure for holding work between centres is as follows.

1. Centre both ends of the workpiece.
2. Remove the chuck and fit the catch plate in its place.
3. Fit the two centres, one in the morse taper of the headstock (live centre) and the other in that of the tailstock (dead centre).
4. Test for correct alignment.
5. Support the workpiece between the two centres after the dog has been cramped on the end fitting the live centre to serve as a driver. Apply grease at the ends of the centres.
6. Run the work and ensure correct support of the work on the centres. The adjustment can be made by turning the tailstock handle clockwise or anticlockwise to tighten or loosen respectively.
7. Clamp the tailstock sleeve in position. The work is now ready for operation.

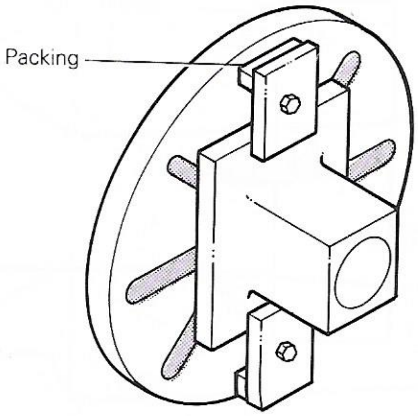

Figure 13.24 Work on face plate.

Note that when the work is very long, it should be supported by the travelling steady to prevent deflection.

Using the face plate

A workpiece whose shape or size does not allow the use of a chuck is often bolted or clamped to a face plate. The face serves as a datum surface, against which the workpiece is securely held (Figure 13.24).

The procedure for using the face plate is as follows.

1. Remove the chuck and screw on the face plate.
2. Mount the work on the face plate and clamp or bolt it down securely.
3. Check for correct alignment and balancing.
4. If the work is out of balance, attach a weight in a suitable position to overcome the out-of-balance forces.
5. Run the work and carry out the required operations.

Using collets

Collets are used to hold finished components and bar stock securely. There are two types: spring collets and multibore collets.

Spring collets are in sets, and the range of each individual collet is limited to about 0.05 of its normal size. A typical example is the draw-in collet chuck, which provides a quick and accurate means of holding small parts for making models, instruments and clocks. It is made of heat-treated steel, and is in the form of a sleeve with a split bore to receive round, square and hexagonal sections of bar.

The **multibore** or **multisize collet** consists of a steel body carrying a number of spring-loaded blades, arranged radially. When the collet is pushed into a conical housing, the blades move with a parallel grip with a range of movement of more than 3 mm. When the pressure is released, the springs retract the blades, allowing them to move forward, thereby releasing the work. Some collets of this type are in the form of key-operated chucks; others are lever- or power-operated chucks, which can be operated or closed while the machine is running.

When using the draw-in type of collet (Figure 13.25), you should use the following procedure to hold the work:

1. Remove the chuck from the headstock.
2. Insert a tubular drawbar in the spindle housing from the back of the headstock and screw it on the collet.
3. Fix the work in the collet.
4. Turn the drawbar to draw the collet into the morse taper housing or nose of the spindle to grip the work.
5. Start the machine and carry on the operation.

Using a mandrel

A mandrel is a precision round bar of carbon steel, hardened and ground with a very slight taper so that it can be entered in a finished bore and pressed home tightly.

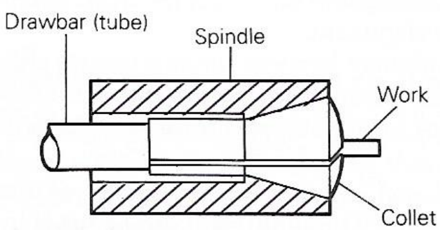

Figure 13.25 Work held in collet.

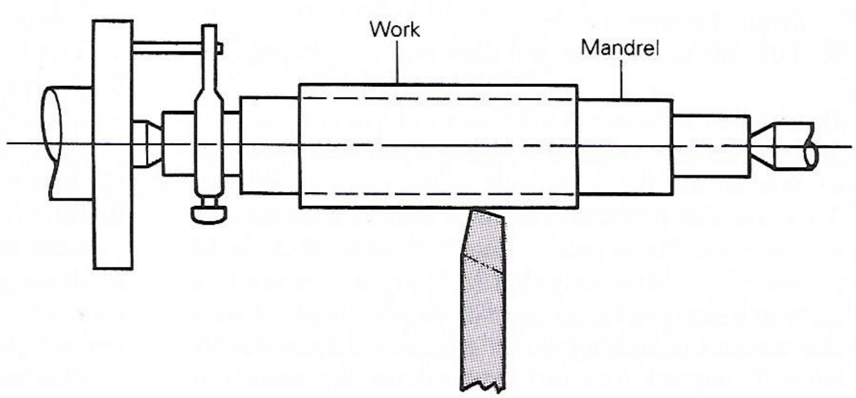

Figure 13.26 Using a mandrel.

It has centre holes at both ends for supporting it between centres (Figure 13.26).

The procedure for using a mandrel is as follows.

1. Fix the work on the mandrel and press home tightly.
2. Remove the chuck.
3. Fix the catch plate.
4. Insert centres in the headstock spindle nose and the sleeve nose of the tailstock.
5. Clamp a dog at one end of the mandrel.
6. Support the mandrel with the work between the centres.
7. Run the machine and check for the correct pressure on the centres.
8. Carry out the operation.

Lathe turning tools

The cutting tools used on all types of turning machines, boring machine, shapers and planners are known as **single-point tools.** They are so called because they have a single cutting edge at one end only. They are classified into three groups: solid tools, tool tips and tool bits (Figure 13.27).

Turning tools having the cutting edge integral with the shank are known as **solid tools.** The cutting end is forged to the appropriate shape and then ground to its final form. Solid tools are often made from high-speed steel.

When either high-speed steel, stellite or cemented carbides are brazed or welded to a low-carbon steel shank to form a cutting edge, the tool is referred to as a **tipped tool.** Tipped tools can be used at a high temperature without losing their hardness.

Turning tools that are inserted in a tool holder or a boring bar and held by adjusting screws or clamps are known as **tool bits.** Most tool bits are made from high-speed steel.

Tool angles

For a turning tool or cutter to cut efficiently, it must have a single cutting edge or point. This means that the front, the side and the top of the tool must be cleared away from the work, leaving only the cutting point to be in contact. To accomplish this the tool is ground in such a way that clearance and rake angles are formed (Figure 13.28).

The **clearance angle** is the angle formed between the flank face of the tool and a line tangential to the work surface at the cutting edge. There are two clearances: the **front clearance,** which is measured from the front of the tool; the **side clearance,** which is measured from the side of the tool.

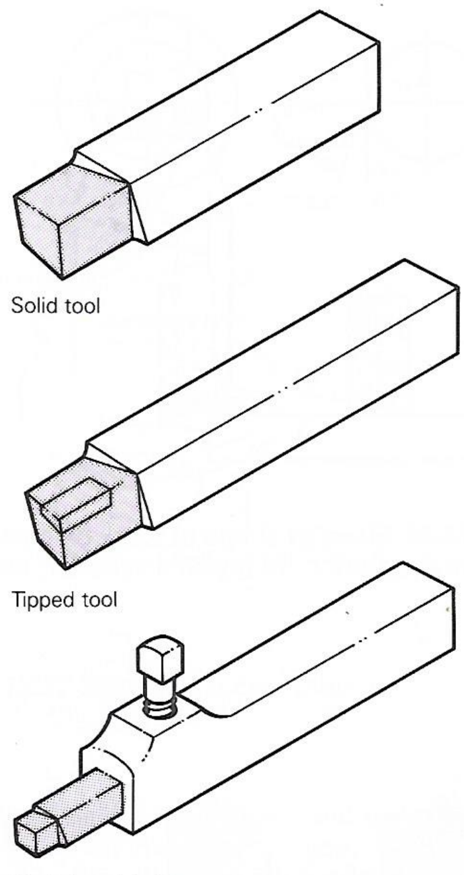

Figure 13.27 Lathe turning tools.

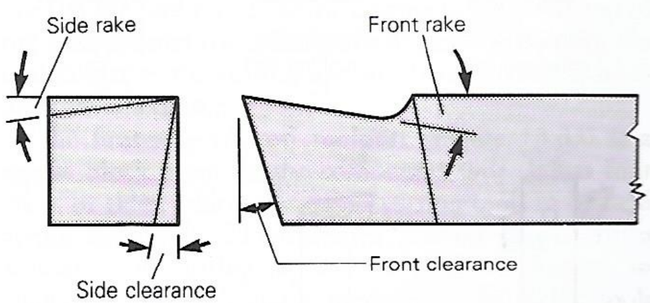

Figure 13.28 Tool angles.

The clearance angle is normally affected by the shape of the work (Figure 13.29). The three basic shapes that affect the clearance are:

1. **external cylinders,** i.e. turning, for which a clearance angle between 5° and 10° is required;
2. **internal cylinders,** i.e. boring, which requires above 8° with secondary clearance depending upon the diameter of the work;

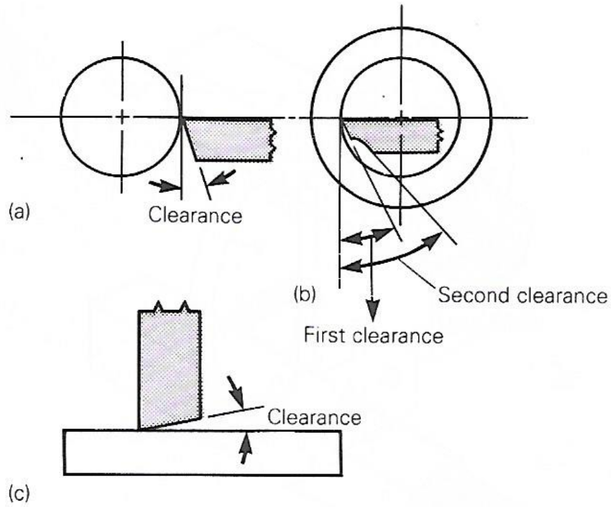

Figure 13.29 Effect of shape of work on clearance: (a) external cylinder; (b) internal cylinder; (c) flat surface.

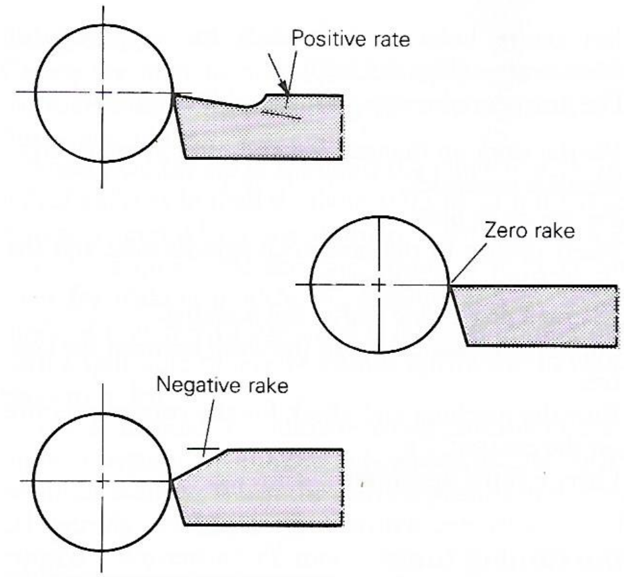

Figure 13.30 Types of rake angle.

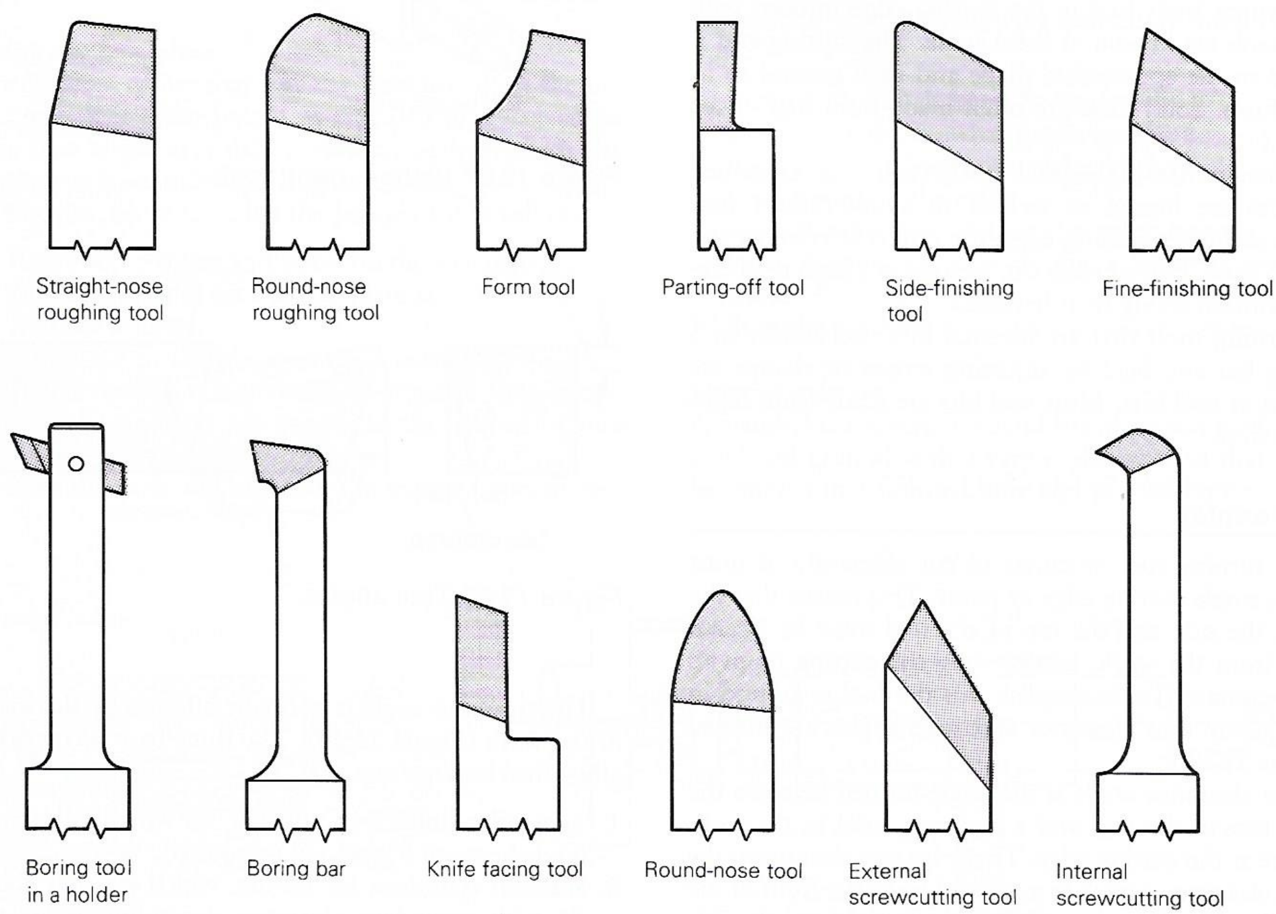

Figure 13.31 Common lathe tool shapes.

3. **flat surfaces**, i.e. shaping, which needs between 6° and 8° clearance.

Rake is the angle between the top face of the tool and a line normal to the surface or the end face of the work at the cutting edge. **Top or front rake** is the slope at the face of the tool. Its actual value may vary according to the material being cut. Generally, for free-cutting materials such as mild steel and aluminium, a large top rake is required. Crystalline materials such as cast iron and cast brass require a small top rake. The **side rake** is the slope of the cutting face relative to the end of the work, measured from the horizontal plane. **True rake** is the actual slope of the cutting face as represented to the work. Normally, it is a combination of the top and side rake angles.

If the front cutting angle, i.e. the front clearance plus the wedge angle, is less than 90°, the rake angle is said to be **positive**. The rake angle is said to be **negative** if the front cutting angle is greater than 90°. If there is no top rake, the tool is said to have **zero rake** (Figure 13.30).

Rake angles influence chip formation, tool wear, cutting force, surface finish and permissible cutting speed. Table 13.1 lists suggested tool angles for high-speed steel tools. Figure 13.31 shows a selection of common tool shapes.

Holding and setting lathe cutting tools

There are many factors that influence the efficiency of work produced on the lathe. They include the state of the machine itself, the accuracy of the measuring instrument used, and the way the cutting tool is held. Incorrect packing and incorrect setting of the cutting tool, for example, can adversely affect the quality of the work produced.

Types of toolpost

There are three main types of toolpost used for holding lathe cutting tools: the ring and rocker toolpost, the four-way turret toolpost and the quick-change toolpost.

The **ring and rocker toolpost** (Figure 13.32) is commonly fitted on American lathes, and is therefore often referred to as an American toolpost. With this type, tools are quickly adjusted at centre height by moving the rocker, which beds on the loose ring around the main post. The main post has its base in a tee form, which fits into the tee slot on the top of the compound slide. It also has a slot in the middle portion to accommodate the tool holder or the cutting tool. At the top of the post is fitted the locking screw used for bolding the tool holder or the cutting tool firmly on rocker. It takes one tool at a time.

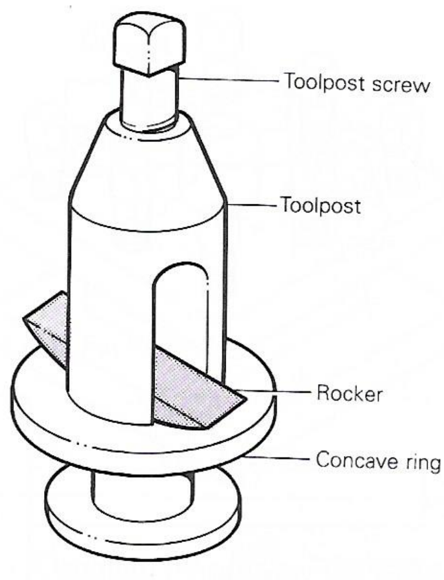

Figure 13.32 Ring and rocker toolpost.

The **four-way (turret) toolpost** (Figure 13.33) is a square block fitted on the compound slide. It has four ways, or sides, which allow four cutting tools to be held at the same time. This makes it possible to perform a sequence of cutting operations without having to change the tool. All that is needed is to fix all the tools required for the operations at the start. As a tool completes its cutting operation the post is turned through

Table 13.1 Suggested tool angles for high-speed steel

Material to be cut	Front clearance (degrees)	Side clearance (degrees)	Top rake (degrees)	Side rake (degrees)
Mild steel	8–10	6–10	18–20	18–20
High-carbon steel	6–8	6–8	6–10	6–10
Cast iron	8–10	6–10	5–10	7–10
Brass	6–10	6–10	0–5	0–5
Aluminium	8–10	8–10	25–40	15–20

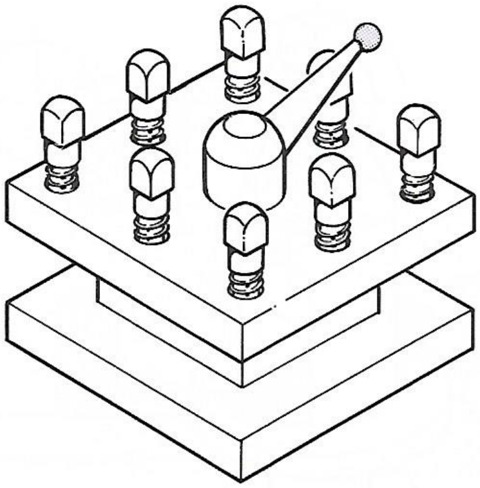

Figure 13.33 Four-way (turret) toolpost.

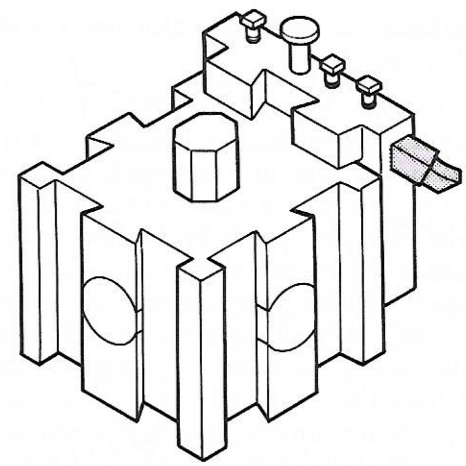

Figure 13.34 Quick-change toolpost.

90° to allow the next tool to come into use. With this type, the tool needs to be packed to adjust to the centre height.

The **quick-change toolpost** (Figure 13.34) is a modern slotted-block toolpost that dispenses with the need for packing to adjust the tool to centre height. With this type, the adjustment is done by means of a screw. There are separate tool holders that fit the four faces of the main block and are easily removed and returned to exactly the same setting. When carrying out multiple operations, the tools required for the operation may be held in the tool holders and fitted at the start. When one operation is completed the next tool may just be turned to come into use. If this is not required, the tool holder may be removed and another one carrying the next required tool slotted easily in its place.

Holding the cutting tool

To ensure efficient cutting, the cutting tool must be rigid enough to withstand the cutting force. The factors that are likely to affect the rigidity of the cutting are the tool overhang and the position of the packing piece or pieces.

Excessive **tool overhang** (Figure 13.35) will make the tool deflect when cutting, thereby creating chattering and producing a rough surface. Always ensure that the tool overhang is as small as possible. An incorrectly placed **tool packing** may also cause excessive tool overhang (Figure 13.36), particularly if the front clamping screw is not supported by the packing. This may cause the tool to be broken or bent, especially when tightening the front clamping screw. It is advisable to keep the number of packing pieces down to a minimum to ensure greater rigidity. One piece of packing is better than a number of pieces that add up to the same size.

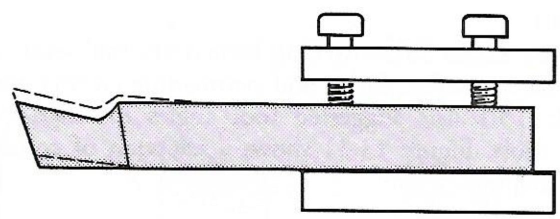

Figure 13.35 Tool overhang.

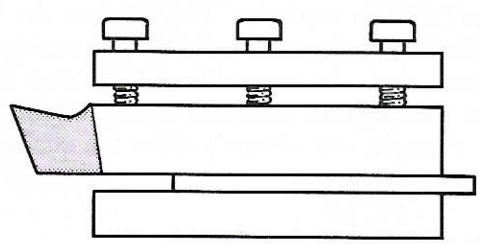

Figure 13.36 Incorrect tool packing.

Tool setting

Another factor that can affect the efficiency of the work is the setting of the cutting tool at the centre height of the lathe. Incorrect setting also affects the rake angle and the clearance angle of the cutting tool.

Cutting tools are normally ground to specific rake and clearance angle. To maintain these angles, the tool must be carefully set with the cutting edge on the horizontal centre line of the work. In other words, the cutting tool must always be set at the centre height of the lathe for all operations. At this condition the tool cuts correctly (Figure 13.37(a)).

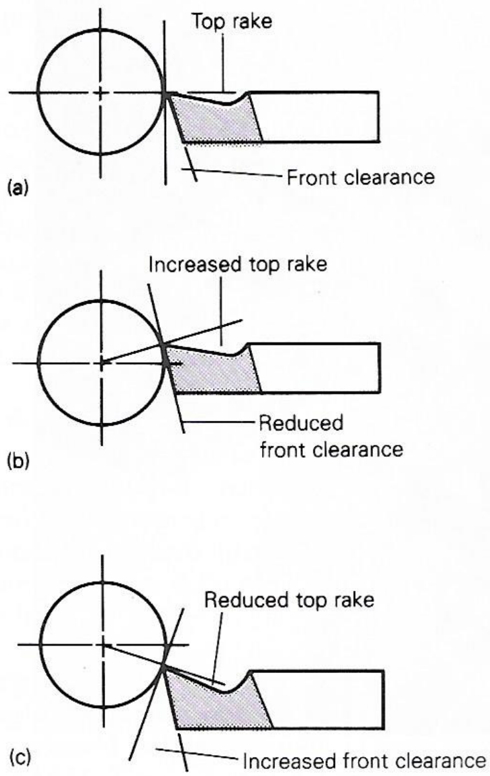

Figure 13.37 Tool set: (a) on centre line; (b) above centre line; (c) below centre line.

If the cutting tool is set **above** the centre line, the rake angle is increased above the value of the ground angle and the front clearance angle is decreased below the value of the ground angle. This results in the tool rubbing. When the cutting tool is set **below** the centre line, the value of the rake angle is decreased while the value of the clearance is increased. Consequently, the tool digs into the work.

Lathe operations

Lathe operations, in this context, refer to the types of work that can be carried out on the lathe. Because the lathe is so versatile, there are a great number of jobs that can be undertaken on it. This section deals with just a few of the common ones that can be performed in the school workshop.

Facing

Facing (Figure 13.38) is the process of squaring the end of work on the lathe. In this operation the cutting tool is traversed perpendicular to the axis of rotation of the work using the cross-slide handwheel. Sometimes it is necessary to lock the lathe carriage during cutting to prevent the carriage from being forced away from the work by the pressure of the cut.

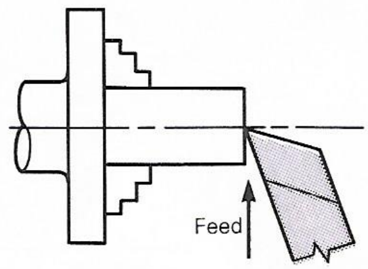

Figure 13.38 Facing.

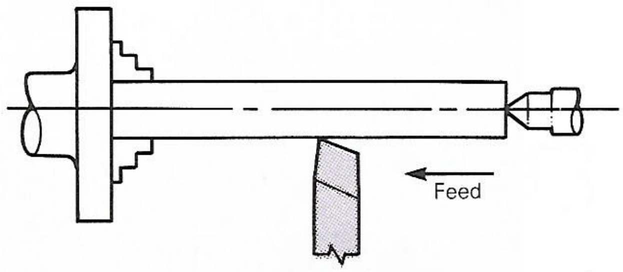

Figure 13.39 Parallel turning.

The procedure for facing is as follows.

1. Hold the work in the chuck firmly, allowing just enough projection.
2. Set the tool at the centre height of the lathe. Check for adequate clearance.
3. Start the machine. Ensure that the chuck key is removed first.
4. Take a reasonable depth of cut and traverse the tool across the bed using the cross-slide handwheel.
5. Repeat step 4 until the desired length is faced.
6. Clean the swarf and all metal chippings from the lathe bed.

Parallel turning

Turning is the process of reducing the diameter of a shaft on the lathe using a turning tool. When the traversing of the tool is parallel to the axis of the lathe, the operation is referred to as parallel turning (Figure 13.39). In this case the cutting tool, which is fixed in the toolpost, travels by the carriage along the bed.

The procedure for parallel turning is as follows.

1. Hold the work firmly in the chuck.
2. Face the projecting end of the work.
3. Centre drill the faced end. This is necessary only when the length to be turned is long and needs to be supported.
4. Fix a centre in the tailstock sleeve, apply grease on the tip of the centre and support the work on it. Ensure that the tailstock has not been offset from the true axis.

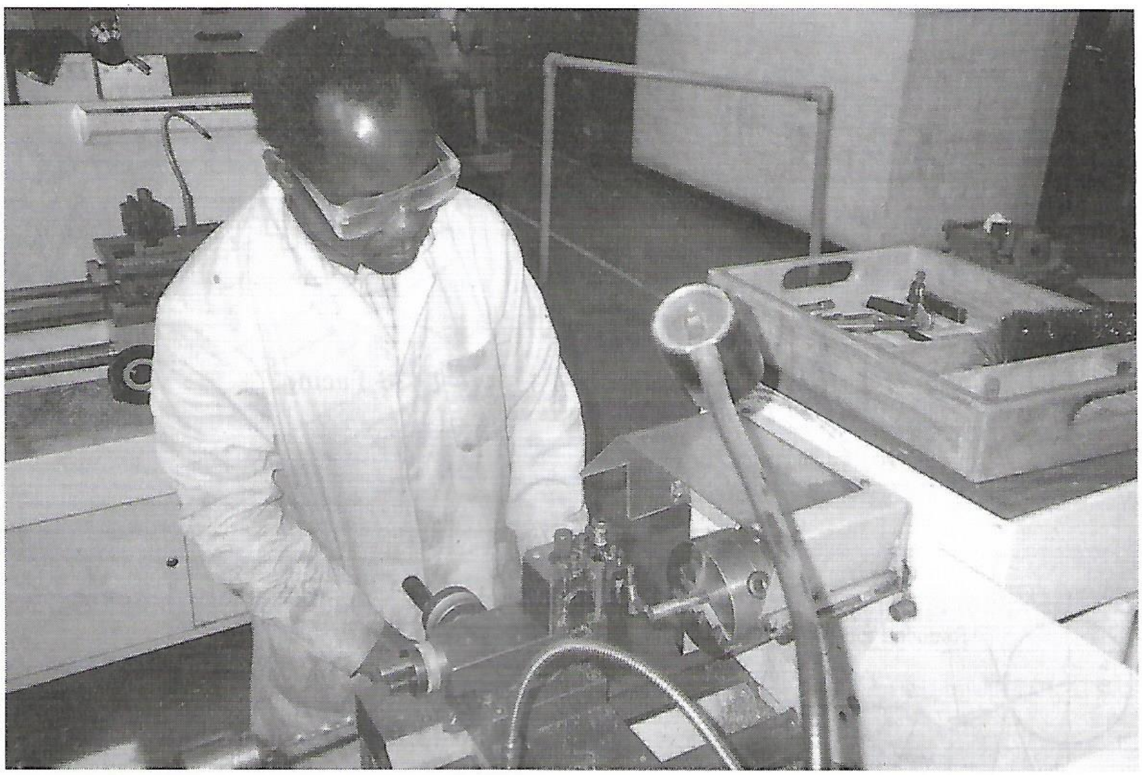

Figure 13.40 Parallel turning operation in progress.

5. Run the work; use the cross-slide handwheel to bring the cutting tool to the end of the work and take a reasonable depth of cut.
6. Traverse the tool longitudinally to the required length using the carriage handwheel or the carriage automatic feed.
7. Return the tool to the start and take another depth of cut.
8. Engage the automatic feed to traverse the tool to the length.
9. Repeat steps 7 and 8 until you have achieved the required diameter.
10. Make sure that the lathe bed is cleaned of swarf and metal chippings.

Figure 13.40 shows parallel turning in progress.

Parting off

Parting off is the making of a recess on a piece of work using a parting-off tool (Figure 13.41). A piece of work can also be cut off using this method. To ensure rigidity and avoid excessive vibration, part off close to the chuck. Ensure adequate tool clearance.

> Remember: too much clearance will weaken the tool. It is advisable to lock the saddle when cutting.

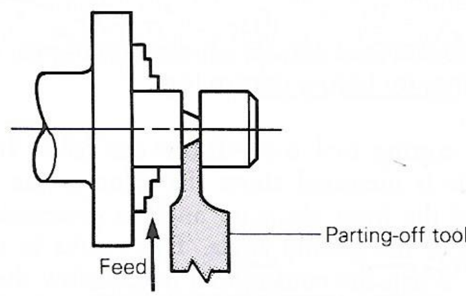

Figure 13.41 Parting off.

The procedure for parting off is as follows.

1. Set the parting-off tool at the centre height of the lathe.
2. Bring the tool to the required position and clamp the saddle.
3. Run the work at a low speed and feed perpendicular to the work using the cross-slide handwheel. Maintain a constant feed during cutting.
4. Remove the remaining piece from the chuck and clean up the lathe bed.

Boring

Boring is the act of enlarging a drilled hole using a boring tool (Figure 13.42). The operation is similar to turning except that the operation in this case is internal. The diameter is enlarged, instead of reducing it as in turning.

The procedure for boring is as follows.

1. Hold the work firmly in the chuck.
2. Face the projecting end.
3. Centre drill and pilot drill as close to the diameter of the hole to be bored as possible.
4. Hold the boring tool in the toolpost and set to centre height. It must be as rigid as possible to avoid deflection. Ensure adequate clearance.
5. Run the work at a reasonable speed.
6. Take a reasonable depth of cut by using the cross-slide handwheel feeding outwards: that is, towards you.
7. Use the carriage handwheel or the carriage automatic feed to traverse the tool to the required length.
8. Return the tool to the starting point and repeat steps 6 and 7. Repeat this operation until you have obtained the required diameter.
9. Remove the work from the chuck and clean up the lathe bed.

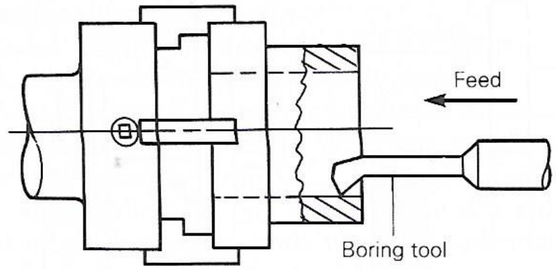

Figure 13.42 Boring.

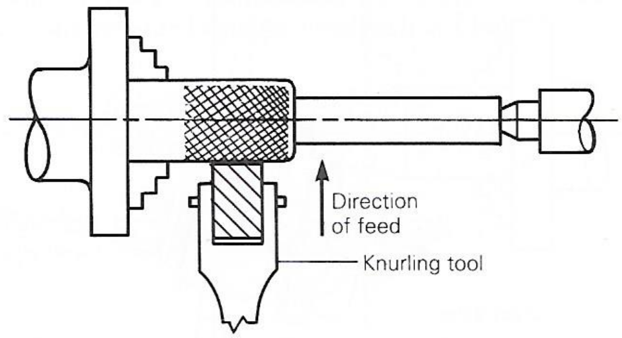

Figure 13.43 Knurling.

Knurling

Knurling is the process of making the impression of a pattern on round handles and tools to form a handgrip (Figure 13.43). The tool used is known as a knurling tool. This consists of a holder carrying two hardened wheels with their faces serrated at an angle. When they are pressed against a round surface, the surface is impressed with cross-marks. This is known as diamond knurling. Some wheels have straight serrated faces, and the impressions made by a set of these wheels are straight: hence it is known as **straight knurling**.

The procedure for knurling is as follows.

1. Set the knurling tool in the toolpost and adjust it to square up with the work.
2. Run the work at a slow speed and press (feed) the wheels hard onto the work at the right-hand side until you can see the pattern start to form.
3. Engage the carriage automatic feed and allow the tool to travel slowly to the required length.
4. Return the wheels (tool) to the starting point without disengaging them. Use plenty of oil if the job is steel. You can also use the carriage handwheel to traverse the tool to and fro.
5. Repeat steps 3 and 4 until the marks are deep enough.
6. Take the work off and clean up the lathe bed.

Taper turning

Quite often, you will need to turn or bore surfaces of conical shape. When the centre lathe is used to produce such surfaces, the process is known as taper turning. Figure 13.44 shows some of the most commonly used methods for turning tapers.

When a flat tool is accurately ground to the required angle, carefully set relative to the work, and then fed in by hand while the work revolves at slow speed, a very short taper is produced. This method of producing taper is knows as the **form tool method** (Figure 13.44(a)). The disadvantage of this method is that it cannot be used to produce long tapers.

With the **compound-slide method**, the compound slide is set to one half of the included angle of taper required, and the tool is traversed by the compound-slide handwheel (Figure 13.44(b)). This method is suitable for both external and internal tapers. As the length of tool travel is confined to the length of the slide screw, this method cannot be used for tapers that are longer than the tool travel. The operation can only be carried out by hand, and the finish obtained is often inferior.

With the **offsetting tailstock method**, the work is held between the two centres, with the tailstock centre set out of alignment with the headstock centre (Figure 13.44(c)). The amount of taper for a given length between centres will depend on the amount of offset. Steep tapers cannot be produced by this method because of the limited amount of the tailstock movement. However, it is suitable for long shallow external tapers. The main disadvantage of this method is that

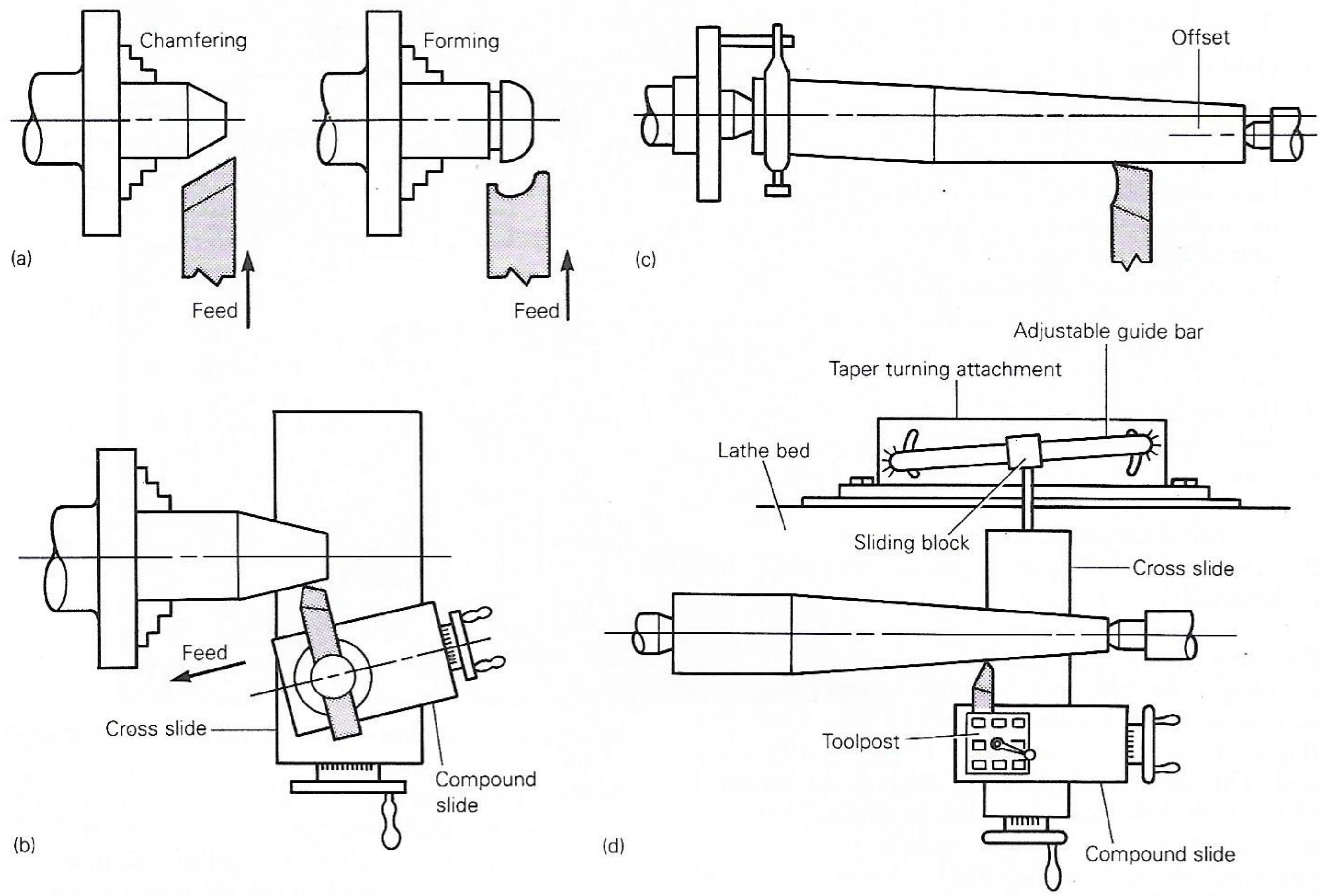

Figure 13.44 Taper turning: (a) use of form tools; (b) compound slide; (c) offset; (d) taper turning attachment.

excessive wear takes place in the work centres, mostly at the tailstock end, and hence causes inaccuracies. When producing duplicate parts, make sure that the length of each piece and the depth of the centre holes are identical, otherwise the tailstock must be adjusted for each individual workpiece.

The **taper-turning attachment**, which is attached to the lathe bed, provides a convenient, rapid and accurate method for producing various tapers, including external tapers, taper bores, and taper screws (Figure 13.44(d)). The basic function of this attachment is to provide a lateral movement to the cross-slide and the cutting tool at the same time as the longitudinal movement of the carriage. Briefly, its advantages are as follows.

1. The alignment of the lathe centre is not disturbed.
2. It can be used for repetitive work once it has been set, no matter the length of the work.
3. It can be used for taper boring.
4. It can be set very accurately and quickly.

Screw cutting on the lathe

Many people, including students and even some teachers, are fascinated by using a lathe to cut a screw. This leads most of them to want to carry out this operation long before they have mastered other simple operations.

Screw cutting on the lathe is not as complicated as it might seem. To cut an accurate screw, it is essential to maintain the correct relationship between the movement of the carriage and the revolution of the work. This can be achieved by means of the leadscrew, which is driven by a train of gears from the spindle (stud).

You therefore need to understand gear trains, and know how to calculate the appropriate gear ratios.

Gear trains

The speed ratio between the leadscrew and the stud is controlled by the gears driven between them. This relation depends only on the number of teeth on the gears. The arrangement of the gears is known as the gear train. There are two types of gear connection: the simple train and the compound train (Figure 13.45).

For a **simple train** gear arrangement (Figure

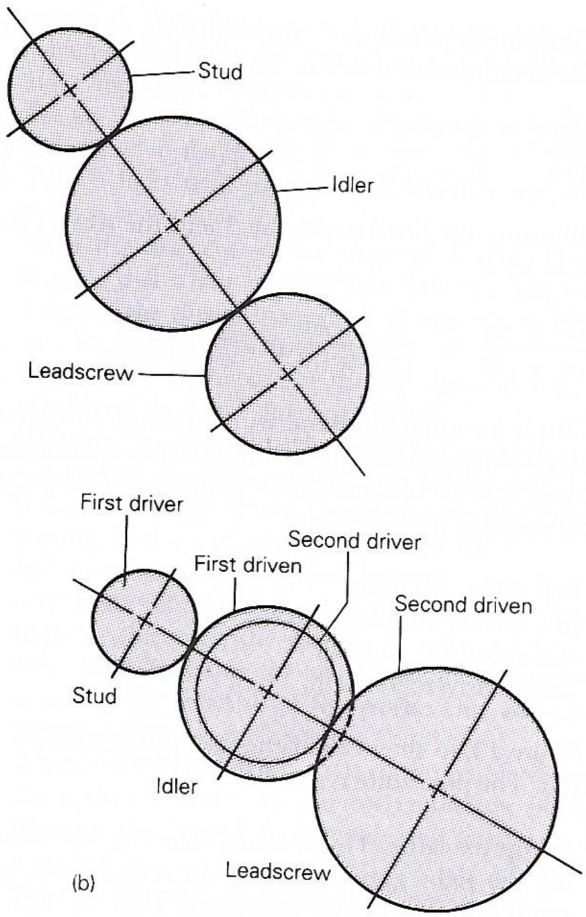

Figure 13.45 Gear trains: (a) simple train; (b) compound train.

13.45(a)), the gear on the stud and the one on the leadscrew are connected by an idler (intermediate) gear. The size of the intermediate gear does not matter. It can be determined by the size of the gap between the driver (stud) gear and the driven (leadscrew) gear.

A **compound train** is a connection in which the speed change requires two pairs of change gears. The two pairs must occupy the same space between the driver gear and the driven gear. The first pair of gears connect the stud to the intermediate pin, and the first gear of the second pair, also mounted on the intermediate pin and keyed to the other gear, drives the leadscrew gear. Note that the gears on the intermediate pin are not idlers; one is a driver gear and the other is a driven gear.

For each different pitch required, the gear train must be dismantled, reassembled, and brought back into mesh by adjusting the swing plate. Modern lathes are fitted with a quick-change gear box driven from a simple train, which, with one wheel change, provides a wide range of drive speeds.

Determination of gear ratio

We need to derive a formula for calculating gear ratio. First, consider the following propositions.

What happens if we use a gear ratio of 1:1 between the stud and the leadscrew? When the spindle turns once, the leadscrew turns once; the tool moves along one pitch of the leadscrew, and therefore cuts a thread of identical pitch on the work. However, with a ratio of 2 to the stud and 1 to the leadscrew, the carriage will move one pitch of the leadscrew when the work turns twice, and a thread of a pitch of one-half that of the leadscrew will be produced. Such a screw will have twice the number of threads per 25 mm or per inch (TPI). (Threads per 25 mm measures the number of threads in 25 mm measured along a screw such as a bolt.)

$$\text{But} \quad \frac{\text{Spindle turns}}{\text{Leadscrew turns}} = \frac{2}{1}$$

$$\text{meaning} \quad \frac{\text{Number of teeth of driver}}{\text{Number of teeth of driven}} = \frac{1}{2}$$

This is because a small gear rotates faster than a large one with which it meshes. A general formula can then be derived from the above explanation:

$$\frac{\text{Driver teeth}}{\text{Leadscrew teeth}} = \frac{\text{Leadscrew turn}}{\text{Spindle turns}}$$

$$= \frac{\text{Threads per 25 mm on leadscrew}}{\text{Threads per 25 mm on work}}$$

$$\text{But pitch} = \frac{1}{\text{Threads per 25 mm}}$$

and so

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Threads per 25 mm on leadscrew}}{\text{Threads per 25 mm to cut}}$$

$$= \frac{\text{Pitch to cut}}{\text{Pitch of leadscrew}}$$

Lathes are often supplied with a set of gears, ranging from 20 teeth to 120 teeth in steps of 5 teeth. Imperial lathes are supplied with one gear having 127 teeth to enable metric thread to be cut on the imperial (English) lathe.

When you calculate the gear ratio, representing driver/driven, remember to convert the answers to figures equal to the teeth of gears available for use on the lathe for the operation.

EXAMPLE 1

Calculate the required gears to cut a screw with 8 threads per 25 mm on a lathe with a leadscrew of 5 threads per 25 mm.

Solution:

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Threads per 25 mm of leadscrew}}{\text{Threads per 25 mm to cut}}$$

$$= \frac{5}{8} \times \frac{25}{25}$$

Cancel out the 25s and multiply both numerator and denominator by 5:

$$\frac{5}{8} \times \frac{5}{5} = \frac{25}{40}$$

This provides a simple train, with a gear of 25 teeth on the stud driven through an intermediate gear to a gear of 40 teeth on the leadscrew. That is:

- stud: 25 teeth
- leadscrew: 40 teeth.

EXAMPLE 2

Calculate the gears for cutting a screw of 27 threads per 25 mm on a lathe with a leadscrew of 4 threads per 25 mm.

Solution:

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Threads per 25 mm of leadscrew}}{\text{Threads per 25 mm to be cut}}$$

$$= \frac{4}{27} \times \frac{25}{25}$$

Cancel the 25s and multiply through by 5:

$$\frac{4 \times 5}{27 \times 5} = \frac{20}{135}$$

This cannot be a simple train, as a gear of 135 teeth is beyond the supplied gears. It must therefore be converted to a compound train as follows.

1. Convert the numerator into a multiple of 4 × 5 and the denominator into a multiple of 9 × 15:

$$\frac{20}{135} = \frac{4 \times 5}{9 \times 15} = \frac{4}{9} \times \frac{5}{15}$$

2. Multiply the first fraction by 5 and the second fraction by 6:

$$\frac{(4 \times 5)}{(9 \times 5)} \times \frac{(5 \times 6)}{(15 \times 6)} = \frac{20 \times 30}{45 \times 90}$$

This is a compound train, with a gear of 20 teeth on the stud driving a gear of 45 teeth on the intermediate, and a gear of 30 teeth on the intermediate driving a gear of 90 teeth on the leadscrew:

- first driver: 20 teeth
- first driven: 45 teeth
- second driver: 30 teeth
- second driven: 90 teeth.

Figure 13.46 shows the lathe set up for cutting a vee thread. The procedure is as follows.

1. Set up the lathe. This includes selecting and mounting the right gears, selecting the appropriate speed and feed, setting the compound slide to half the

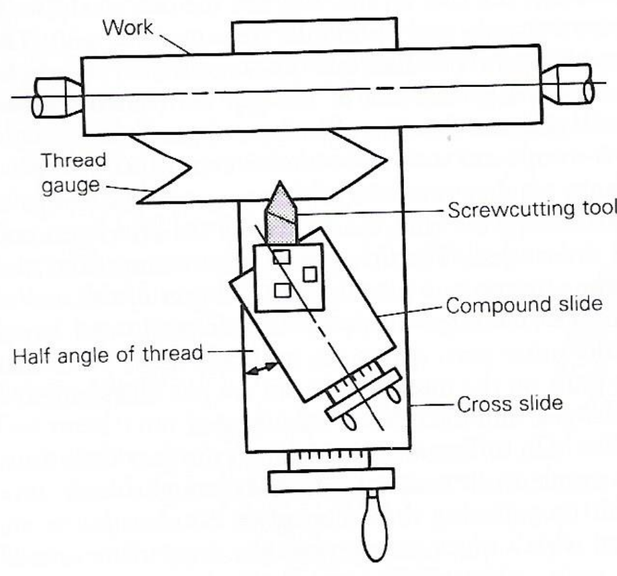

Figure 13.46 Setting up for screw cutting.

required thread angle and mounting the cutter, which must be on the centre line of the lathe and at right angles to the work.
2. Set up the work. It must be chamfered to produce a clean start to the thread.
3. Take a light trial cut to check whether the desired pitch is being cut.
4. Take the first cut to the required length. Use the chasing dial when engaging the half nut. Engage on 1 or 3 if the thread has a pitch of odd number and 2 or 4 for a pitch of even number.
5. At the end of the cut, withdraw the tool from the work with the cross-slide handwheel.
6. Traverse the carriage back so that the tool is just past the end of the work in readiness for the cut. If the lathe is not fitted with a thread-chasing dial or when cutting metric thread on an English lathe, once the half nuts have been engaged for taking the first cut, they must remain engaged during the entire operation. In this case, the carriage is returned for each new cut by reversing the whole machine. This means reversing the rotation of the work. This will ensure correct pick-up at each pass.
7. When the thread has been cut to the required depth, use a thread chaser of the corresponding pitch to trim up the thread form while the lathe speed is increased.
8. Take off the work and clean up the machine.

The milling machine

The milling machine is one of the classes of machine tools that remove metals by feeding the work against a revolving cutting tool.

Types of milling machine

There are two types of milling machine: horizontal and vertical.

The **horizontal milling machine** is the most popular machine, commonly found in schools and colleges (Figure 13.47). It has an overhang, called a **knee**, which slides up and down the front of the machine and to which the cross-slide and the adjustable worktable are attached. It has a horizontal arbor onto which the cutter is fixed.

The horizontal machine is manufactured in two types. The worktable of the **universal machine** may be swung round in the same way as a machine vice. This enables the machine to be used for cutting helices, such as the flutes of a twist drill. It is normally supplied with a dividing head and a vertical head as well, which enables vertical milling operations to be carried out. The **plain pattern** is often made on more robust lines, and is used mostly in industry for production work, where heavy cuts are required, rather than a wide range of movements.

Figure 13.48 shows the horizontal milling machine in use.

The **vertical milling machine** has a vertical head, and the cutter is fixed into a spindle that is normally in a vertical position (Figure 13.49). The worktable traverses perpendicular to the vertical axis of the machine.

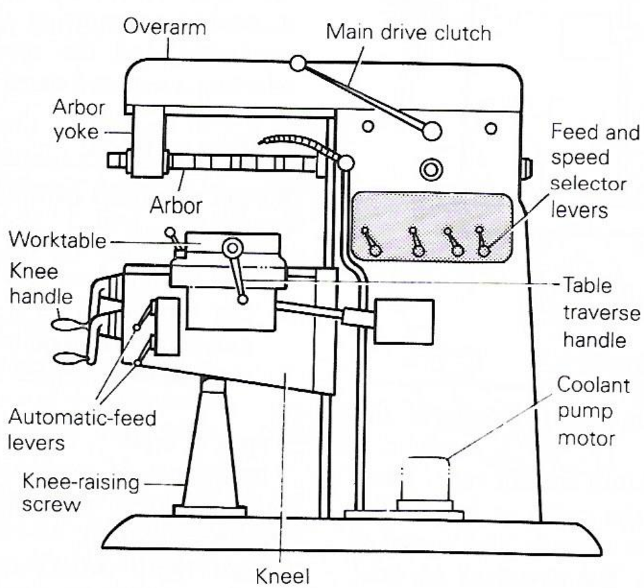

Figure 13.47 Horizontal milling machine.