Know Your Lathe

A Screwcutting Lathe Manual

1988 Edition

Wheatley, Halifax, West Yorkshire, England

Additional safety information is available in the form of wall chart no. SEP/S233-5 for school workshops from The Royal Society for the Prevention of Accidents (RoSPA, Cannon House, The Priory, Queensway, Birmingham B4 6BS).

Other related documents are:

BS. 4163 1975 – Recommendations for Health and Safety in Workshops of Schools and Colleges from The British Standards Institution.

BS. 5304 1975 – Code of Practice Safeguarding of Machinery from The British Standards Institution (101 Pentonville Road, London N1 9ND).

Machine Tool Trades Association – 1978 Code of Practice Safeguarding of Turning Machines (62 Bayswater Road, London W2 3P4).

*For further guidance on safe working practice see the Engineering Industry Training Board's instruction manuals, and, for lathe work in particular, module H2 Turning 1, (Safety practice for engineering trainees) and H23 Turning 11, (For engineering craftsmen). These are available from the Publications Dept. of the Engineering Industry Training Board, P.O. Box 176, 54, Clarendon Rd., Watford WD1 1LB.

Contents

SEE PREFACE FOR GENERAL SAFETY GUIDANCE

IMPORTANT

Boxford Lathes are fitted with limit switches for operator safety.
BEFORE MACHINE WILL FUNCTION CHECK:

1. Headstock lever is correctly located and screw engaged.
2. Door to drive compartment on cabinet is correctly closed.
3. Change gear guard is closed.
4. Saddle limit switch is not depressed (if fitted).
5. Selector switch is not in the isolate position.
6. Chuck guard (if fitted) is closed.

Preface

SAFETY GUIDANCE NOTES: BOXFORD LATHES

Whilst every effort is made in the design and manufacture of Boxford Lathes to provide a fundamentally safe machine, attention should be given to the following notes in the interests of safe lathe usage.

1. Personal safety precautions to be observed when operating a lathe.

   Always wear suitable clothing – overalls or a strong apron tied at the back and safety shoes are best.

   Never wear loose floppy garments.

   Never wear rings, watches, ties, gloves, bandages or anything which could become caught in the moving parts of the machine.

   Always wear your clothing buttoned up and roll up long sleeves or button the cuffs.

   Never allow long hair to hang loosely.

   ALWAYS WEAR SAFETY GLASSES OR PROTECTIVE VISOR.

   Always use a barrier cream on the hands.

   Never wash hands in machine coolant or wipe your hands with rags used to clean the machine.

   Never keep tools or other sharp objects in your pockets.

   ALWAYS REPORT ANY ACCIDENT HOWEVER SMALL IMMEDIATELY IT HAPPENS.

   ALWAYS ASK A QUALIFIED SUPERVISOR IF IN DOUBT.

   ALWAYS REMEMBER THAT THE MOST EFFECTIVE SAFETY PRECAUTION IS TO ADOPT SAFE WORKING PRACTICES*.

Operating a Boxford Lathe Safely

Know your lathe – Read and understand the manual provided before attempting to use the machine.

Keep lathe work areas clean and keep area surrounding the lathe tidy, and the floor free from grease or oil.

ENSURE YOU KNOW HOW TO STOP THE LATHE BEFORE STARTING IT.

Before you operate the lathe – make sure it is switched off at the mains and that the spindle control switch is in the off position.

Then ensure that tool/tools are firmly clamped in the toolpost.

Select spindle speed required via belt drive and/or back gear change lever. (On vari-speed lathe speed dial must not be moved when the spindle is stationary).

Select feed rate required via change wheels and/or quick change gear box. (Not applicable to TUD m/c).

Ensure feed control knob and half nut lever are disengaged and select required position for feed reverse gear lever and feed change lever. (Not applicable to TUD m/c).

Check that carriage lock is released.

Check that all compartment doors are fully closed and lower chuck guard into place (when fitted).

Check that work piece is securely held in work holding device, e.g. chuck, faceplate, between centres.

Then and only then switch on the mains and start spindle in required direction.

STOP THE MACHINE IMMEDIATELY ANYTHING UNEXPECTED HAPPENS AND OBTAIN QUALIFIED ASSISTANCE.

Never attempt to move back gear lever, reverse gear lever or feed change lever whilst machine is running.

ALWAYS STOP MACHINE AND SWITCH OFF AT THE MAINS BEFORE OPENING BELT DRIVE COMPARTMENT OR CHANGEWHEEL GUARD DOORS.

Always stop machine and switch off at the mains when leaving machine unattended.

ALWAYS REMOVE CHUCK KEY FROM CHUCK IMMEDIATELY AFTER USE OR USE SPRING LOADED SAFETY CHUCK KEY.

NEVER TOUCH REVOLVING PARTS OF THE MACHINE OR WORK PIECE.

Never remove swarf from the machine with the bare hands and never whilst the machine is running.
Always switch off at the mains before cleaning the lathe.
Always use correct size spanners.
Never use cracked or chipped turning tools.
Always ensure work piece and/or chip guards are in position (where these are provided) before starting the machine.
Always retreat the tool to a safe position before removing the work piece from the machine.
Always obtain qualified assistance when handling heavy or awkwardly shaped work piece, chuck and/or faceplate, and check that large work pieces clear bed and saddle etc. by rotating them manually before starting the spindle.
Be careful of, and remove if possible all work piece burrs and sharp edges.
Never lean on the lathe and NEVER INTERFERE WITH THE LATHE OR ITS ELECTRICAL EQUIPMENT.

Machine Capacity and User Responsibility

The dimensions of a work piece which can be accommodated on a Boxford Lathe are limited only by the physical restrictions of the machine itself and responsibility for the following points with respect to machining a work piece must inevitably lie with the user.

1. The operator must possess the required degree of skill and experience to undertake the work planned or adequate qualified supervision must be provided.
2. **Suitable work holding and/or supporting equipment must be provided for each work piece to be machined , i.e. chucks, steadies revolving centres, etc.
3. **Suitable lathe tools, drills, drill chucks etc. must be provided and correctly mounted in the lathe.
4. *Suitable cutting speeds and feeds should be selected and safe working practices adopted.
5. **Suitable work piece or chip guards should be provided and consistently used.

**See Boxford Lathe accessories and equipment lists.

Lathes General — Chapter 1

The lathe, most important of machine tools, is the one that has made most of the others possible. It is the basic fact behind Britain's great engineering industry. Much progress has followed since Henry Maudslay incorporated the principles of the screw cutting lathe in the small lathe he built in 1797, yet all the later developments have been built upon his firm foundation.
Essentially a lathe consists of an accurately machined bed upon which are mounted the headstock providing a means of rotation, a tailstock providing a means of location for cutting tools or centre to support a workpiece, and the saddle unit which can be moved along the bed and is accurately located upon it, and upon which the cutting tools are mounted.

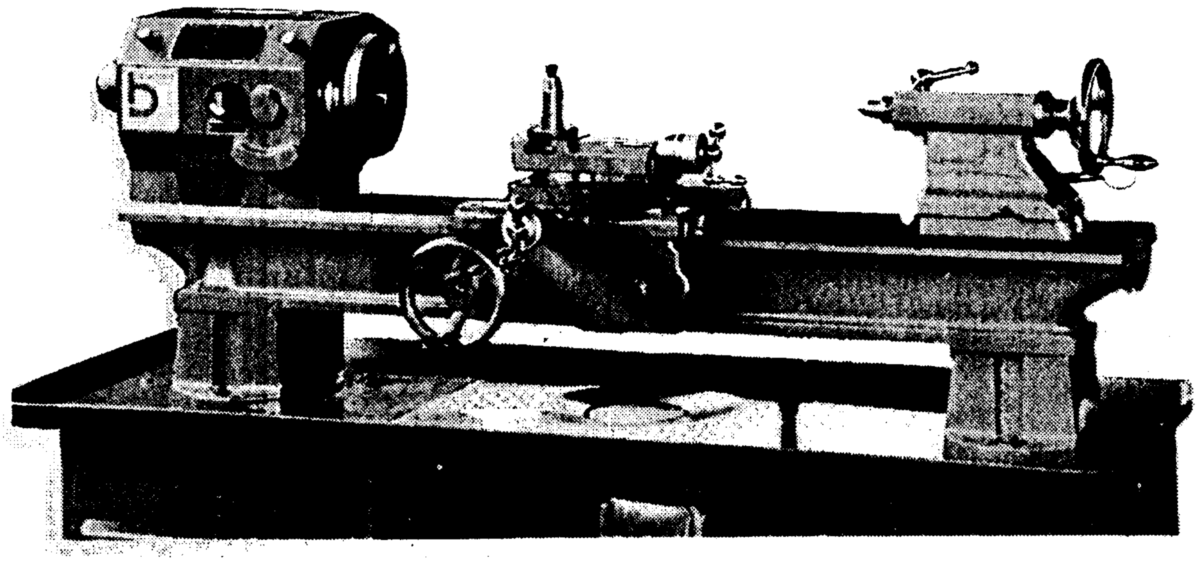


Fig. 1. Simple Training Lathe (Model TUD)

The saddle is provided with a cross slide which can move accurately at right angles to the bed and an apron which contains mechanisms for controlling the movement of the saddle along the bed or the cross slide across the bed.
Simple lathes, particularly those used for initial training, have these basic elements. All movements of the saddle and cross slide are manually controlled, the cross slide by a handle and the saddle by a handwheel. The headstock spindle rotation is obtained from an electric motor controlled by a switch or starter and changes of spindle speed are obtained by using a belt driven countershaft between the motor and headstock spindle.
To enable the lathe to be used for cutting screw threads, the rotation of the headstock spindle must control the movement of the saddle along the bed of the machine. The relationship between the headstock spindle through a series of change gears, the main leadscrew (situated at the front of the lathe and parallel to the bed) and the apron attached to the saddle, is obtained by engaging the split nuts of the apron with the main leadscrew.
To obtain different relationships between the headstock spindle and the movement of the saddle along the bed it is necessary to provide different sets of gearing, and this is done by either changing loose gears in the train between the headstock spindle and the main leadscrew or by introducing a gear box

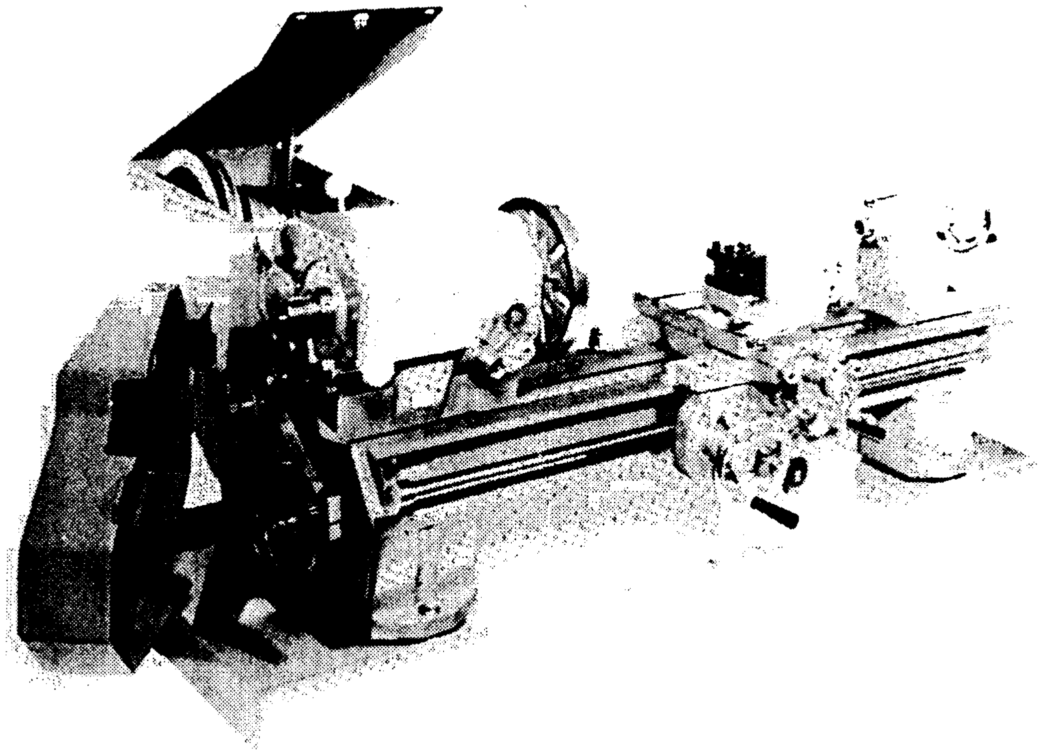


Fig. 2. Standard Change Gear Bench Lathe (Model ME 10)

into the gear train. When a gear box is fitted, changing the gear train is accomplished quickly and easily without the use of loose gears.

Other refinements to these basic elements are introduced on all modern lathes making them more easily and quickly used. A means of reversing the rotation of the main spindle is provided by an electric reversing switch. A reverse gear in the gear train between the headstock spindle and main lead-screw enables the main leadscrew to be reversed in direction relative to the main spindle thus allowing L.H. or R.H. screw threads to be cut, it also has a neutral position, in which only manual control of the cutting tools is obtained.

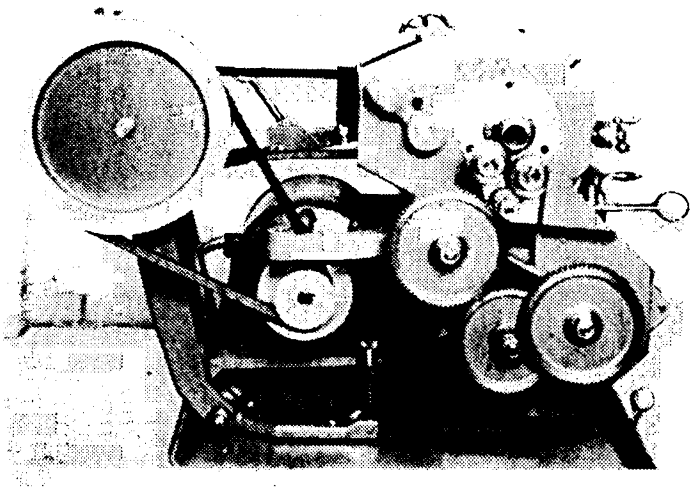


Fig. 3. End View of Standard Change Gear Lathe

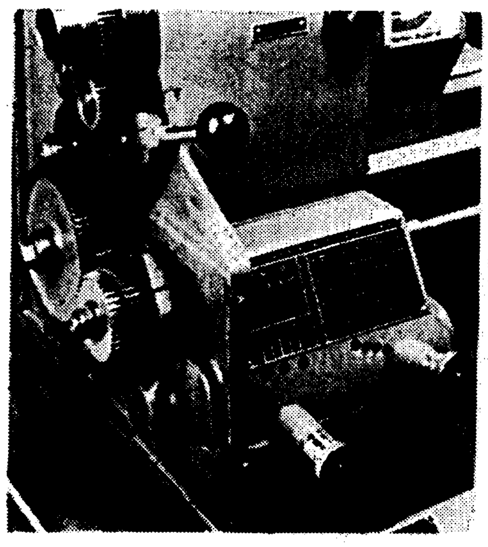


Fig. 4. Quick Change Gear Box

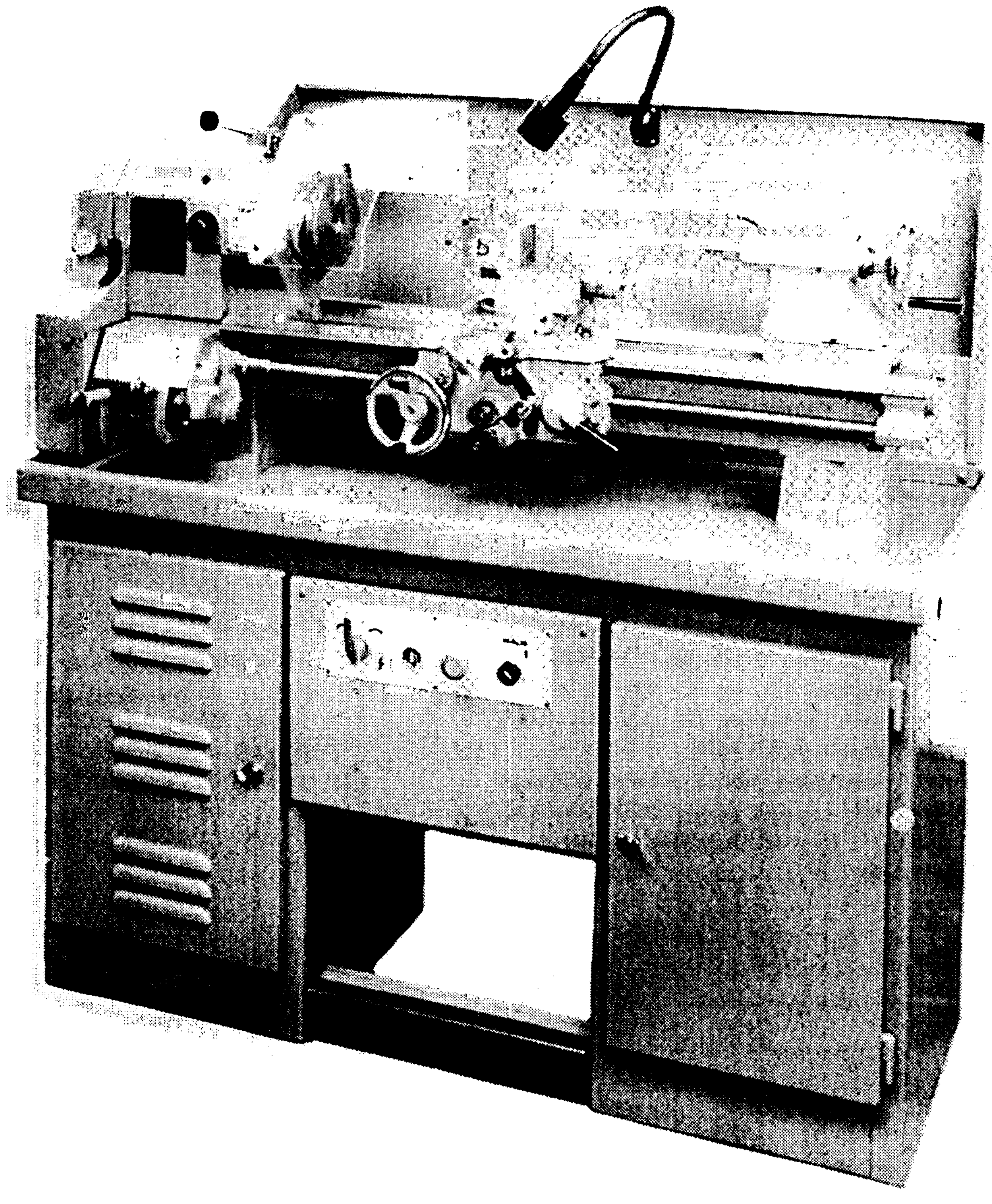


Fig. 5. Underneath Drive Cabinet Mounted Lathe (Model AUD)

A further refinement on screwcutting machines is the provision of a friction clutch, which enables a horizontal keyway in the main leadscrew to be used for providing power feeds to both the saddle and cross slide independent of the split nuts, thus the leadscrew and split nuts need only be used when actually cutting screws thus preserving the accuracy of the machine and simplifying the construction as one shaft acts as main leadscrew and feed shaft.

To enable a wider range of headstock spindle speeds to be obtained such as are used when screwcutting, a headstock back gear can be engaged which reduces the spindle speeds to allow large diameter work pieces or screw threads to be cut and gives increased power at low speeds.

Machines are designed for either bench or cabinet mounting, the bench machines having the countershaft and motor drive unit positioned to the rear of the headstock as in Fig. 2, thus allowing the whole unit to be mounted on a bench or work table.

Those machines which are cabinet mounted are usually supplied with the drive unit and countershaft situated beneath the headstock in the cabinet base and in both cases changes of speed can be obtained by moving the vee belt from one position to another on the multi-step pulleys.

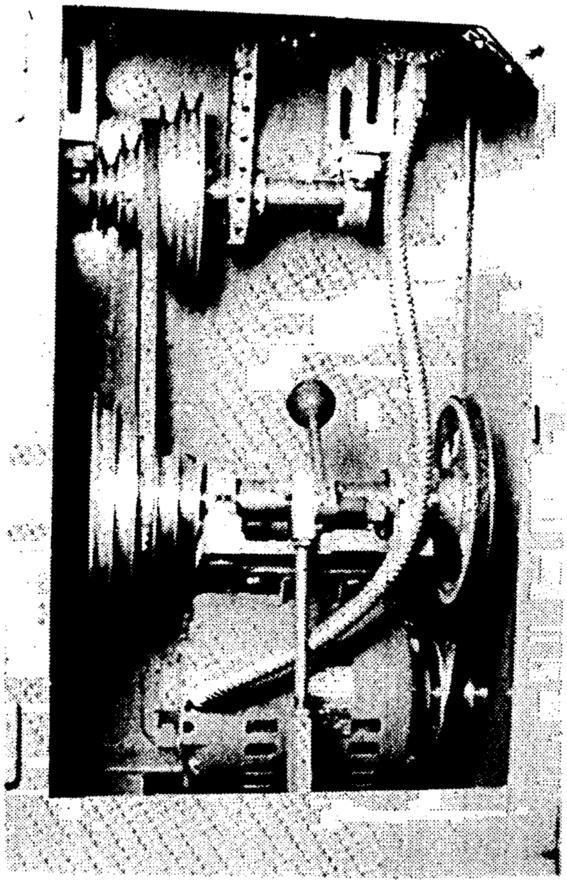


Fig. 6. Standard Underneath Drive Unit

The latest machine in the Boxford range is the model 500 VSL vari-speed, in which an infinitely variable motor drive unit controlled by a handwheel at the front of the cabinet gives stepless variable speeds and makes the changing of belts unnecessary. This model can also be equipped with an Electric brake unit fitted to the motor shaft which brings the machine quickly to a stop.

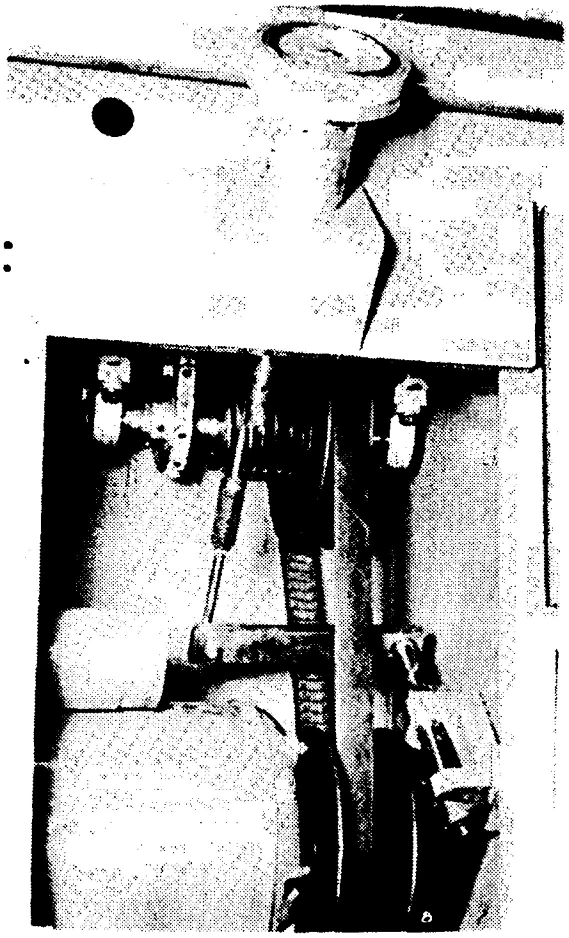


Fig. 7. Vari-Speed Drive Unit

A Lathe's Size and Capacity

Machine tool manufacturers both here and on the Continent describe a lathe by the height of spindle and tailstock centres above the lathe bed. In the U.S.A. and Canada, however, it is usual to refer to the swing or maximum diameter that can be swung in the lathe headstock. This usually being double the height of centres. Other important dimensions are the maximum swing over saddle wings and swing over cross slide which can limit the size of work which can be turned in the lathe.

The bed length of a lathe refers to the overall length of the bed, but the useful length of the bed or distance between centres is the maximum length of workpiece that can be machined between centres in the lathe.

Lathe Bed

The lathe bed is the foundation of the machine and it is essential that this is robust and rigid in construction, accurately machined and levelled and correctly installed. On Boxford machines the headstock and tailstock are fitted to the inner V and flat of the bed, and the saddle or carriage is guided on the two outer V's, all these being machined at the same time to ensure accuracy of alignment.

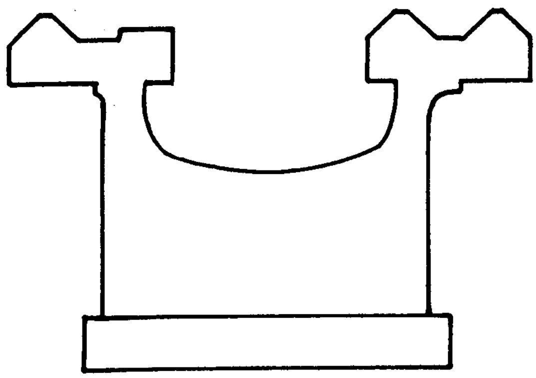


Fig. 8. Lathe Bed Formation

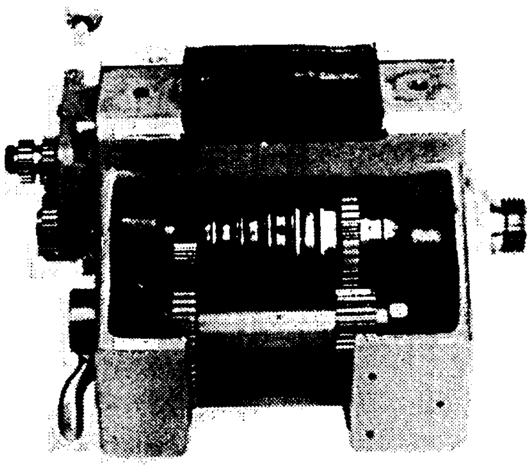


Fig. 9.
Bench Model Lathe Headstock
with Back Gear

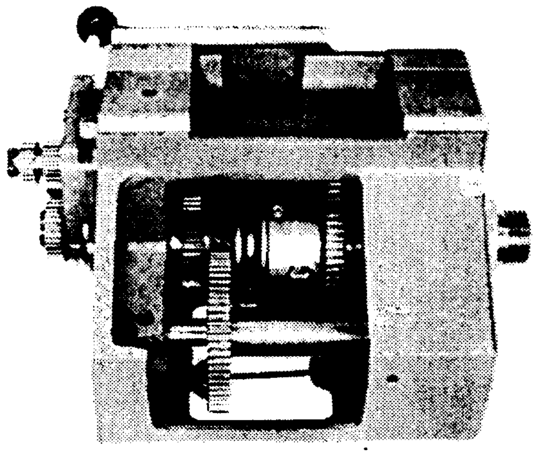


Fig. 10.
Underdrive Lathe Headstock
with Back Gear

The Headstock

The lathe headstock illustrated in Figs. 9 and 10 is used to rotate the workpiece or tools. It is the most important unit of the lathe and provides the power.

The headstock spindle must be accurately located in precision bearings and be in line with the ways of the bed.

Training lathes such as the BOXFORD TUD, see Fig. 1, have only a direct belt drive to the spindle, no back gear being fitted. Changes of speed are

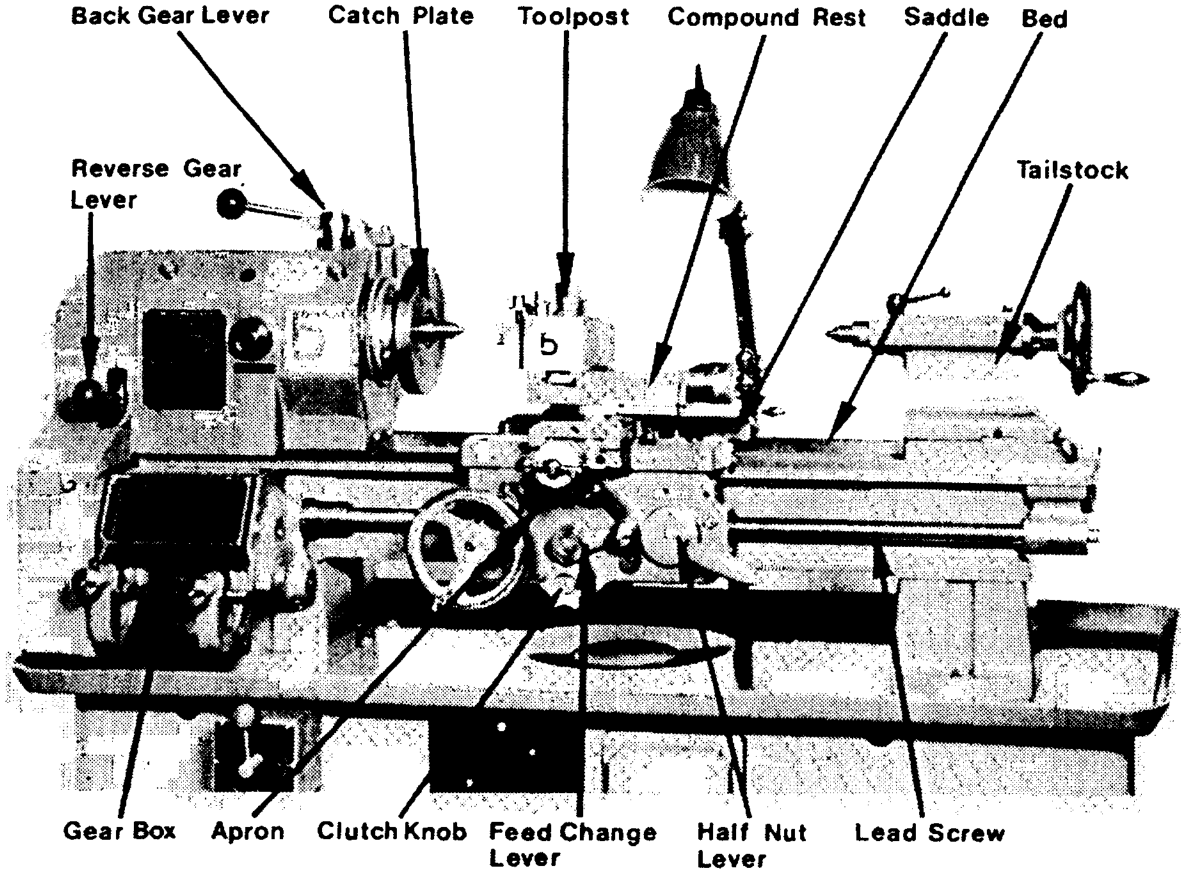


Fig. 11. The Main Parts of a Lathe

obtained by belt changes on the underdrive countershaft unit giving 5 speeds in all.

Bench lathes Models ME 10, A, B and C have a back gear fitted and this is engaged by moving the back gear and sliding gear levers on the headstock. DO NOT ATTEMPT TO ENGAGE THE BACK GEARS WHILST THE SPINDLE IS REVOLVING. FAILURE TO OBSERVE THIS WILL DAMAGE THE GEARS. The back gear lever at the left-hand side of the headstock engages the gearing and the sliding gear lever at the front of the headstock disengages the sliding gear from the pulley drive pins. The pulley on bench machines has four vee belt positions (see Fig. 9) and changing the belt from the countershaft and from the motor to the countershaft and using the back gear provides 16 spindle speeds. To re-engage the sliding gear with the pulley drive pins, it may be necessary to rotate the headstock spindle slowly when moving the sliding gear lever to the right.

WHEN BACK GEARING IS ENGAGED THE HEADSTOCK PULLEY ROTATES ON THE MAIN SPINDLE AND LUBRICATION OF THE PULLEY AND BACK GEAR IS OF PARAMOUNT IMPORTANCE.

See detail under maintenance. This applies to all machines fitted with back gear.

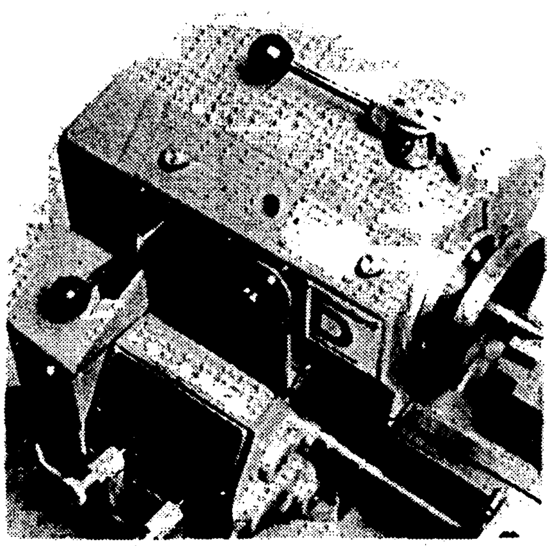


Fig. 12. Headstock (Underdrive)

Underdrive lathes models AUD, BUD, CUD, Mark II also have back gears fitted and the same principles apply, particular care being taken with correct lubrication, details under maintenance section.
The back gear on underdrive machines is operated by a single lever mounted on top of the headstock, two movements of the lever are necessary. The lever is normally locked into position by one or other of the socket head cap screws fixed in the lever. These screws engage the contacts of a micro-switch situated in the headstock casting and prevent the lever being depressed. These contacts cut off all electric power to the drive unit when the locking screw is unscrewed. The delay in unscrewing the screw ensures that the machine is stationary before the back gear lever can be depressed and moved thus safeguarding the gear train.
A spindle lock is provided on the front of the headstock of the Boxford TUD and all underdrive machines (see Fig. 12 above). This can be engaged when the machine is in direct drive by applying pressure to the knob with the thumb and rotating the lathe spindle by hand until the lock engages. This is used when removing or fitting chucks, faceplates etc. Changing the belt from the drive unit to the countershaft and using the back gear provides 10 spindle speeds on all underdrive machines.
The re-engagement of the sliding gear with pulley driving pins on underdrive machines presents no problem as engagement is automatically made when the machine is re-started.
The Vari-speed Boxford Lathe headstock is similar to that fitted to the UD machines but the back gear lever is not locked in position by socket head cap screws. The contacts, controlling the power supply to the motor, cut off the supply as soon as the lever knob is depressed and this lever can thus be used as an emergency stop. When changing to or from back gear on this machine the operator depresses the lever and waits until the machine is stationary before moving the lever, otherwise damage to the gear train will result.
These machines have an infinitely variable speed range between 50 and 2,000 r.p.m. In direct drive the speed range is from 2,000 to 250 r.p.m. and with back gear engaged 400 to 50 r.p.m. (On machines operating on 60 cycles supply speeds are 55 to 2400 r.p.m.)

Carriage (See Fig. 11)
The lathe carriage consists of five basic components.

1. The SADDLE which moves along the outer vee ways of the bed and provides a longitudinal movement and has on top a dovetail location for the CROSS SLIDE at right angles to the longitudinal movement. The rear of the saddle is held on to the bed ways by a spring-loaded gib under the bed shear, and the front of the saddle by the rack gearing and the carriage lock under the bed shear. The lock being provided to lock the saddle to the bed ways in any position, or to reduce free movement. The

cross slide leadscrew is mounted on the front of the saddle and is fitted with a three ball handle and dull chrome friction dial engraved in either decimals of an inch or on metric machines in millimetres. The leadscrew is located between two ball thrust races to eliminate wear, and engages with the bronze nut attached to the cross slide and provides micrometer movement of the cross slide at right angles across the bed of the machine.

2. The CROSS SLIDE as mentioned above can be translated across the dovetail ways of the saddle at right angles to the bed of the machine and is fitted with an adjustable gib strip which engages one side of the dovetail ways. Adjustment to the gib is made by tightening or slackening the self-locking grub screws along the side of the cross slide.
   The cross slide provides a central location hole with an engraved scale around it. The compound rest is located on this hole and locked into any position by the two socket head screws at the right and left of the cross slide.

3. The COMPOUND REST has a grooved spigot on the underside which locates accurately in the hole in the cross slide and an engraved mark on the base of the compound rest moves over the scale engraved on the cross slide.
   The compound rest can be locked firmly to the cross slide in any position on the scale, by means of the two clamping screws.
   The top face of the compound rest provides dovetail location for the top or tool slide. The compound rest leadscrew is mounted on the front of the tool slide and is fitted with a three ball handle and dull chrome friction dial engraved in either decimals of an inch or on metric machines in millimetres. The leadscrew mounted on the tool slide is located between two ball thrust races to eliminate wear and engages with the bronze nut attached to the compound rest, and provides micrometer movement of the tool slide. The swivel movement of the compound rest has many uses more fully described later in the text.

4. The TOOL OR TOP SLIDE can be moved along the ways of the compound rest by means of the leadscrew and is fitted with an adjustable gib strip which engages one side of the dovetail ways. Adjustment to the gib is made by tightening or slackening the self-locking grub screws along the side of the tool slide. The top of the tool slide has a flat location and tee slot upon which is mounted one or other of the toolholders.

5. The APRON is mounted on the front and underneath the saddle.
   (a) The simplest apron as fitted to the BOXFORD TUD Lathe merely provides a handwheel and gearing which locates in the rack on the underside of the bed shears, and gives manual control of the movement of the saddle along the bed ways, and is used on training lathes where no screwcutting can be carried out.
   (b) The apron fitted to the Model C and CUD Lathes is a simple apron which can be used either under manual control by using the handwheel or for screwcutting when the half nut lever engages the nuts with the main leadscrew. With this apron power controlled move-

ment of the saddle can be obtained either for screwcutting or fine cutting feeds.

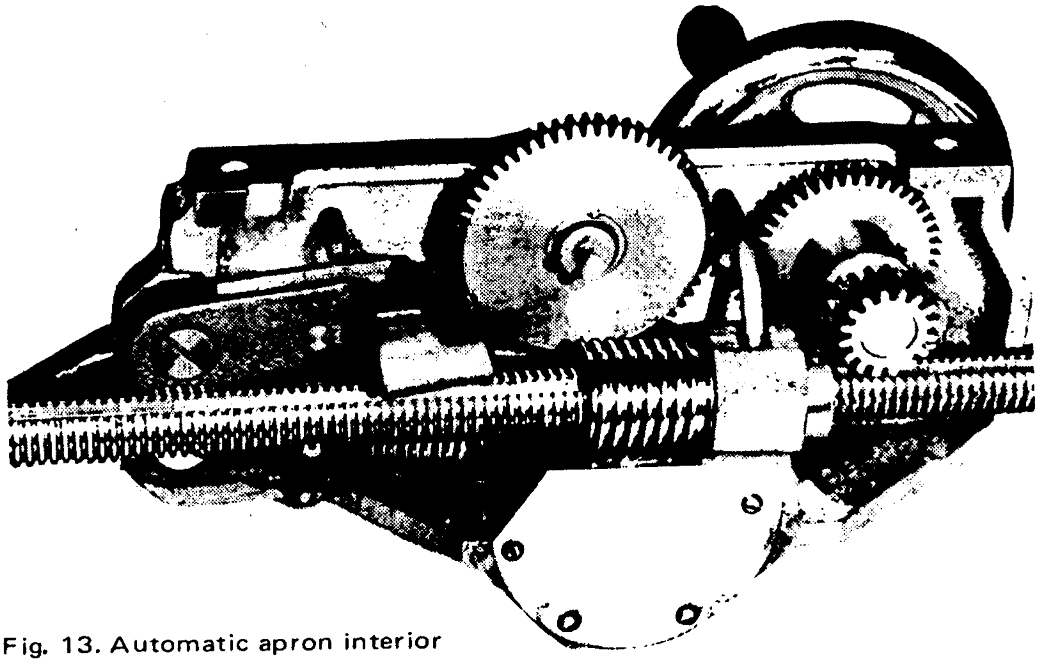


Fig. 13. Automatic apron interior

(c) The Automatic apron fitted to models ME 10, A, B, AUD, BUD and Vari-speed lathes, can be used either under manual control or for screwcutting or to provide a power longitudinal or cross feed using a friction clutch.

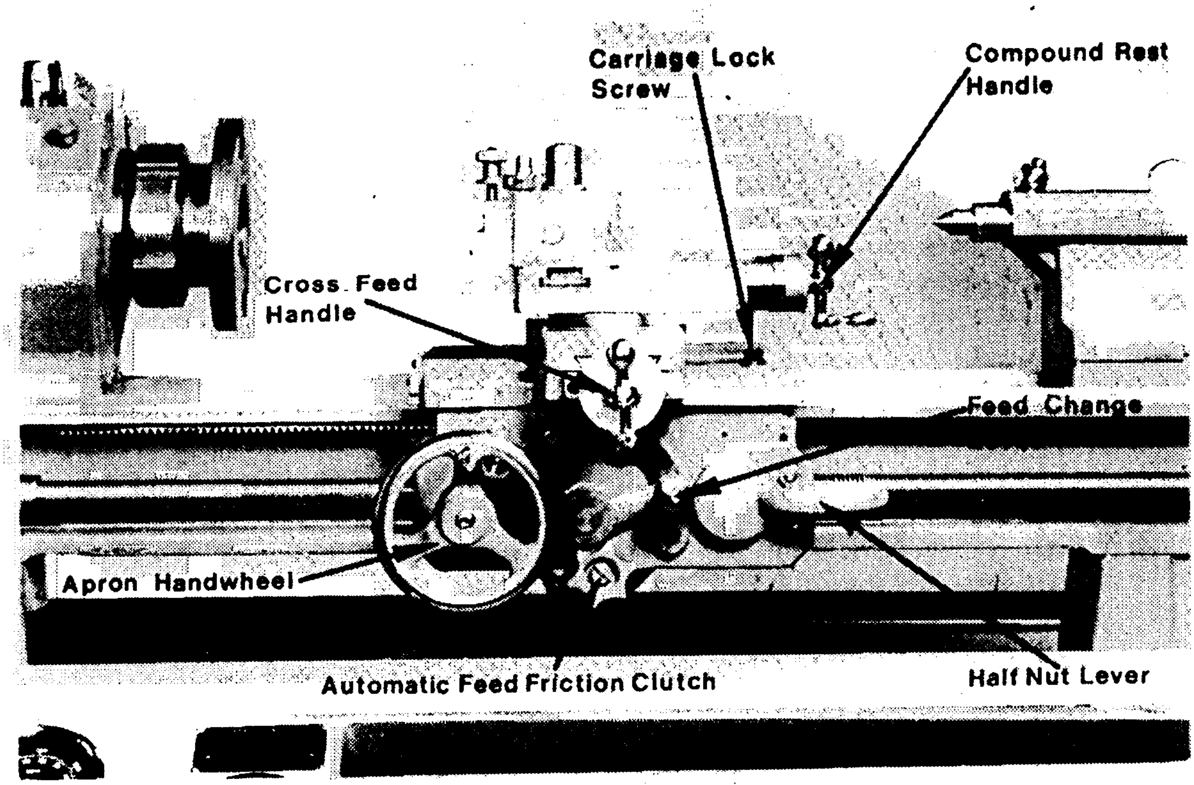


Fig. 14. Operating Parts of Lathe Carriage and Apron

When screwcutting, the half nut lever is lifted upwards and as in the simple screwcutting apron this makes the half or split nuts engage the threads on the main leadscrew. With the automatic apron the half nuts can only be engaged when the feed change lever is in the central position.
When power longitudinal or cross feeds are required with the automatic apron, the drive for the saddle or cross slide is obtained from a keyway in the main leadscrew which drives a worm and worm wheel in the apron, the power drive being obtained through a friction clutch inside the worm wheel. Thus the one shaft provides a leadscrew for accurate screwcutting, and a feed shaft, the screw thread and nuts being used only when actually screwcutting. To engage the power longitudinal feed the feed change lever should be moved to the top position and the knob controlling the friction clutch turned clockwise until the clutch engages.
To obtain power cross feed of the cross slide, the feed change lever should be moved to the bottom position and the knob controlling the friction clutch turned clockwise.
When the feed change lever is in either the top or bottom position it is impossible to operate the half nut lever as this is interlocked. Similarly when the feed change lever is in the central position to enable the half nuts to be engaged, it is impossible to engage a feed with the friction clutch knob.

Tailstock

The lathe tailstock or loose head consists of a base casting which locates on the inner V way and flat of the bed and has a tenon at right angles. Upon this tenon is mounted the tailstock body and a screw at each side of the body locates on a raised boss on the base.
By tightening and slackening one or other of these screws it is possible to translate the tailstock body across the bed of the lathe and this can be used for taper turning. See details under Taper Turning.

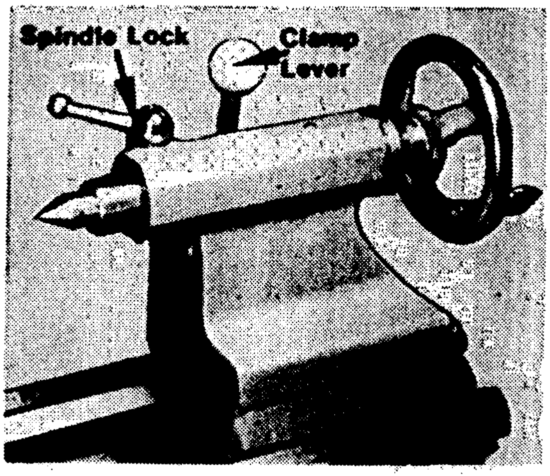


Fig. 15. Tailstock

The central position of the body is shown by the engraved lines at the rear of the tailstock. See Fig. 111.
The tailstock body contains the chute or spindle, which is bored out to a Morse Taper to take drill chucks, centre drills, etc. The spindle can be advanced or withdrawn from the body by operation of the handwheel and is suitably engraved in both metric and inch graduations. The spindle can be clamped in any position by means of the spindle lock handle.
The eccentric clamp lever when moved to the rear, releases the clamp and allows the tailstock to be freely moved along the bed to any desired position. Raising the eccentric clamp lever will lock the tailstock firmly to the bed.
With standard morse tapers, if the spindle is fully withdrawn into the body of the tailstock by using the handwheel the spindle will automatically eject the morse taper from the spindle.

Reverse Gear
All lathes other than training lathe TUD are fitted with a reverse gear which is situated at the left hand side of the headstock and is the first gearing in the gear train from headstock spindle to main leadscrew, the reverse gears driving the stud gear shaft and stud gear. See Fig. 149.

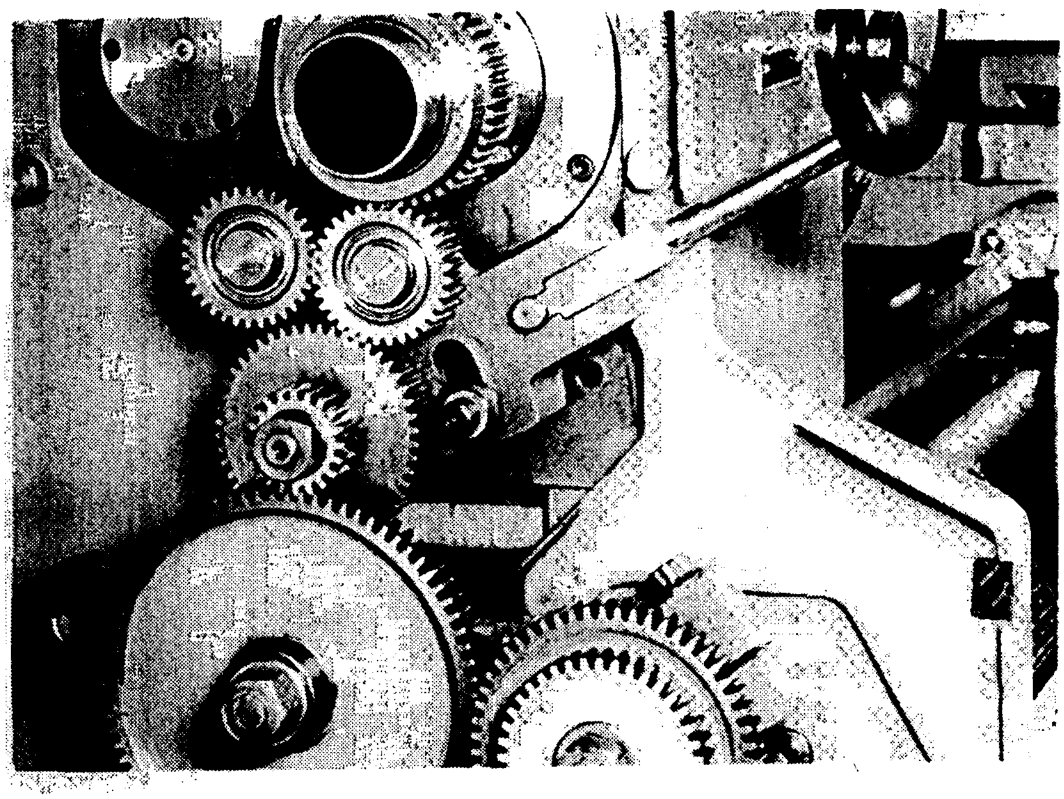


Fig. 16. Reverse gear

Before moving the reverse gear lever the machine MUST BE STATIONARY OR DAMAGE WILL RESULT. Pull the reverse gear lever to the right to disengage the locking plunger and move the lever up or down as required.
In the top position of the lever with conventional change gear set up (as in Fig. 16) the main leadscrew will revolve in the same direction as the main spindle; in the bottom position the main leadscrew will revolve in the opposite direction to the main spindle, and in the central position the change gear train is disengaged from the main spindle.
The reverse gear is used to select direction of feed relative to headstock spindle rotation and can be used for cutting either right-hand or left-hand threads.
The reverse gears are normally made of steel but alternatives in tufnol reinforced plastic material are available where no coarse feeds are required and high speeds are in general use. These tufnol gears will not stand up to the same degree of shock loading as steel gears but tend to run quieter at high speeds. (Fitted as standard on the ME 10.)
Lubrication of the reverse gears and particularly the stud gear shaft is of the greatest importance, see detail under lubrication.

Change Gears
The change gears are the loose gears with keyway which fit either on the stud gear shaft or the end of the leadscrew (see Fig. 149). They are connected

together by the idler gearing mounted on the change gear quadrant. On machines not fitted with a gear box, sufficient change gears are supplied to enable various screw threads and feeds to be obtained. The idler gears have plain bores and are mounted on hardened bushes located in any position along the slotted arms of the quadrant. The quadrant pivots about the axis of the leadscrew and can be clamped in any suitable position. The idler gears should be arranged on the quadrant to mesh with the leadscrew gear and then the whole quadrant raised until the first idler gear meshes with the stud gear thus completing the drive from headstock to leadscrew.

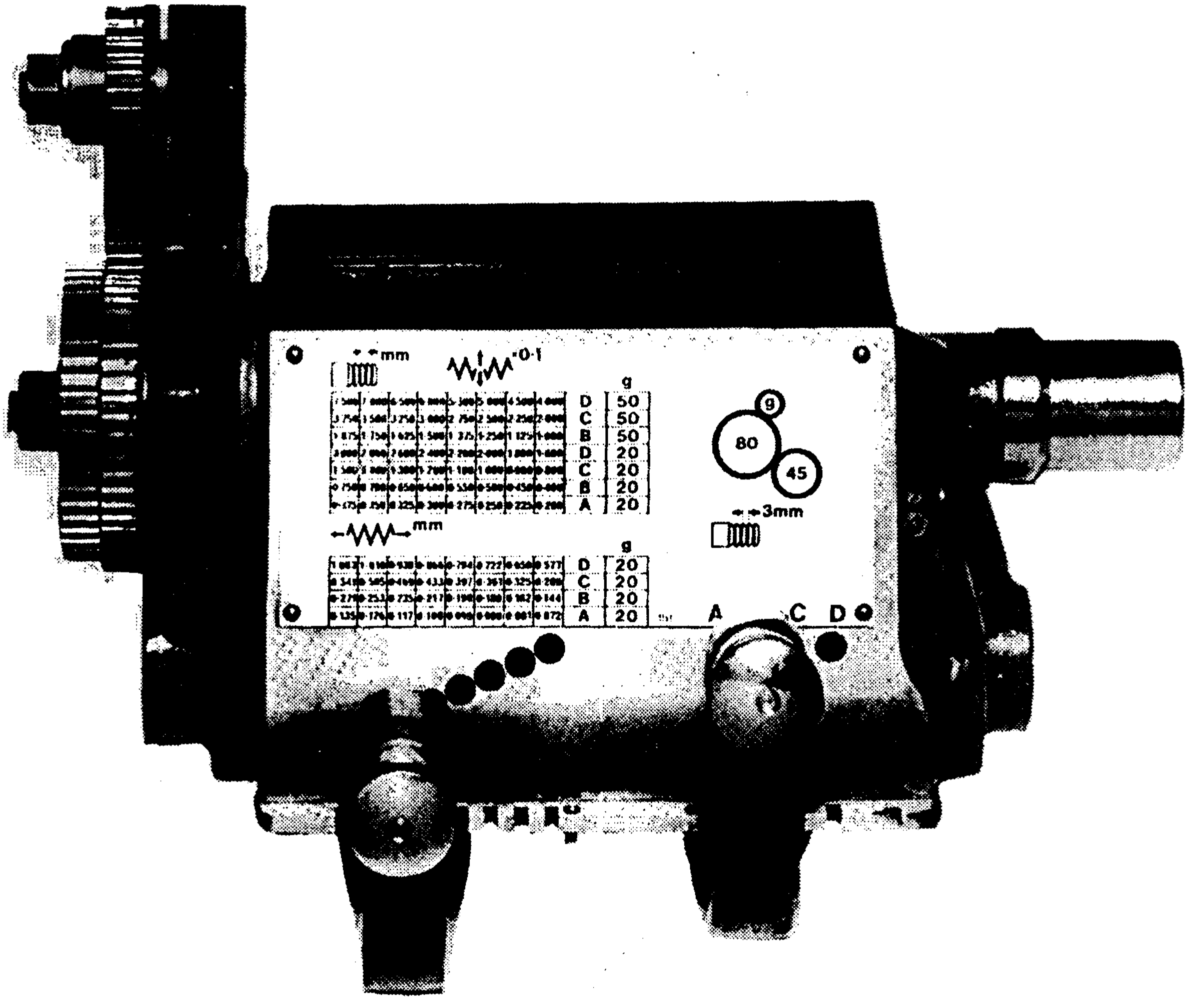


Fig. 17. Quick Change Gear Box

Quick Change Gear Box

The quick change gear box provides a wide range of feeds and screw thread pitches by moving the gear box levers. The gear box is fixed to the left hand end of the lathe bed in line with the leadscrew and a quadrant carries the idler gear which connects the stud gear spindle of the headstock with the input shaft of the gear box. The various pitches which can be obtained are shown on the chart fixed to the front of the gear box and change from one position to another is effected by pulling the knob on the lever outwards to withdraw the plunger and then lowering the handle until it is free to slide sideways to the appropriate position. A notched guide plate is fixed at the bottom of the gear box wall to guide the levers into the various positions. THE GEAR BOX LEVERS SHOULD NOT BE MOVED WHEN THE MACHINE IS IN MOTION but it may be necessary to rotate the headstock spindle by hand to enable the gears to be engaged.

LATHE DRIVE UNITS

Horizontal Motor Drive for Bench Models

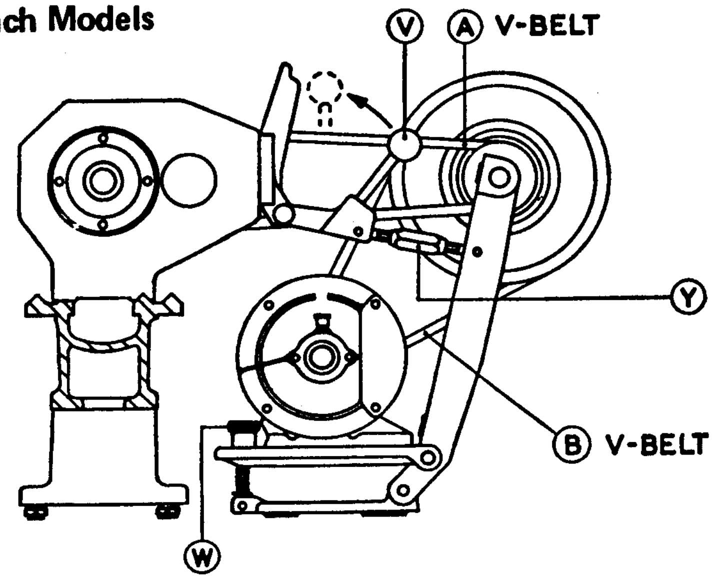


Fig. 18. Diagram of Bench Model Countershaft

The bench model drive unit as shown in Fig. 18, consists of a countershaft which pivots on a base fastened to the bench or cabinet top behind the headstock, and a motor platform which pivots on the countershaft. An adjustable belt tension lever is fixed between the countershaft and the headstock and when this lever is pulled towards the front of the machine, tension in both drive belts is released, enabling either belt to be changed from one position to another. A separate adjustment is provided to the motor platform so that once the correct tension in the headstock drive belt has been obtained, tension in the motor drive belt can be adjusted by means of the knurled knob. Drive belts should not be overtight as this causes unnecessary wear in the countershaft bearings and also requires more power to drive the machine. The horizontal drive unit is fully guarded by a hinged sheet steel guard fastened to the rear of the headstock.

Underneath Drive Unit

The standard underneath drive unit is situated in the left-hand compartment of the cabinet base, directly beneath the headstock (see Fig. 6). A $\frac{3}{4}$ H.P. electric motor is mounted on a hinged platform carrying the countershaft which the motor drives by means of a vee belt. Adjustment for tension of the belt being made by adjusting the motor position on the platform.

The position of this hinged platform is controlled by the belt tensioning device fixed to the motor platform. When the lever is pulled forward and downwards, the hinged platform is raised releasing all the tension between the 5 step pulleys on the countershaft and intermediate shaft enabling the drive belt to be readily changed from one step to another.

The final drive to the headstock is by link belting through the cabinet base tray and the lathe foot, the link belting at the rear of the headstock being guarded by a simple fixed sheet steel guard. The link belting is tensioned by removing links as necessary, easy access to it being obtained by removing the sheet steel guard fitted to the rear of the headstock.

Vari-Speed Drive Unit (See Fig. 7)

The Vari-Speed drive unit which is also housed in the left-hand compartment of the cabinet base directly beneath the headstock consists of a manually operated variable speed pulley mounted on the $1\frac{1}{2}$ H.P. motor of the series 500 model VSL/LOO machines which drives a spring-loaded automatic variable speed pulley on the intermediate shaft by means of a special vee belt.

The intermediate shaft is mounted on ball bearings suspended from the cabinet tray and the final drive to the headstock is by link belting through the cabinet tray and lathe foot. Speed adjustment is by means of a conveniently placed handwheel at the front of the cabinet base which operates a lever linkage to give movement to the variable speed pulleys. The handwheel is fitted with a mechanical dial indicator which registers the spindle speed of the machine. (Earlier models are fitted with an electrical impulse tachometer and separate handwheel.) A magnetic brake is an optional feature on the model 500 VSL drive unit.

All rotating shafts on both standard underneath drive and vari-speed drive units are mounted on sealed grease packed ball bearings which require no further lubrication.

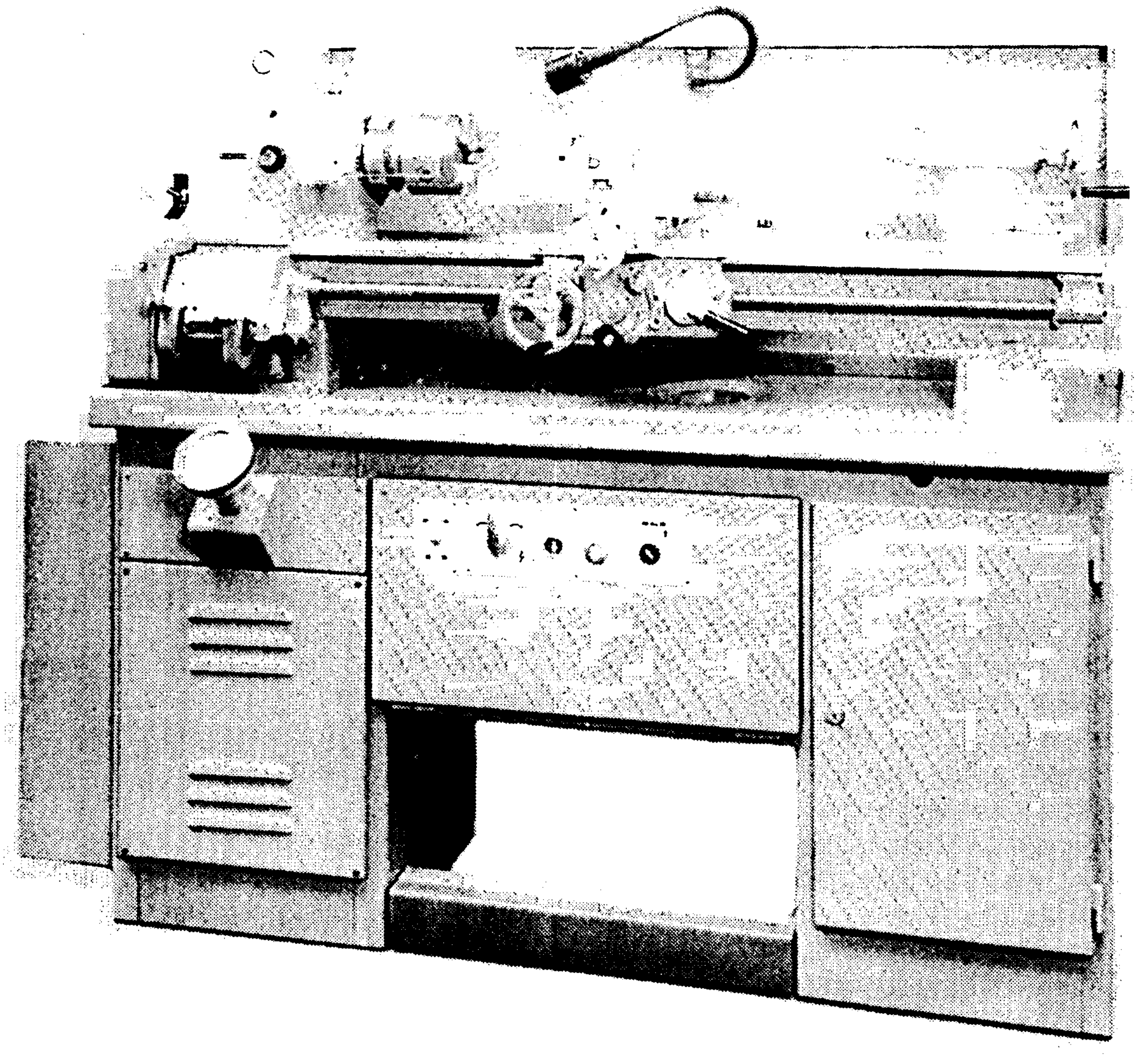


Fig. 19. Boxford 500 VSL Vari-Speed Lathe

Installation & Maintenance — Chapter 2

A lathe is a machine of the utmost precision. It has been built to a very fine degree of accuracy by the manufacturer so that it will produce first class work. Never, therefore, be careless about the way it is unpacked nor slip-shod in the manner of installing it in the workshop.
For example, when removing it from the case be very cautious when using a hammer or crowbar. A blow from either of them may cause serious damage to the lathe. There may be instruction books or small parts in the packing material, so examine it carefully. Before any attempt is made to set up the lathe be sure that all the reference books and instruction charts have been read and properly understood so that no mistake will occur.
Use paraffin on a stiff brush to go over the new lathe and clean it, afterwards wiping it with a clean cloth. Then the unpainted surfaces must be covered with a film of good-quality machine oil to protect the metal from rust. From time to time the old oil should be removed and replaced by a new coat. Dust, chips and any sort of dirt must never be allowed to accumulate on the lathe. To keep it covered with a dust cover when not in use is a great help towards cleanliness. These simple precautions are easy to observe and, with the finished surfaces kept clean and well oiled, the lathe will look and work like new for a long time.
A plastic dust cover can be supplied for Boxford Lathes as extra equipment.

The Need for a Solid Floor
Next to a good lathe, a solid, level foundation is required, if first-rate work is to be secured. A concrete foundation is the most satisfactory one. If a wooden floor is used, it must either be unusually substantial or strongly braced to prevent the sagging and vibration generally associated with it.
The cabinet may be fastened securely to the floor if desired. Unless the lathe is level its weight will cause the lathe bed to twist and thus throw the headstock out of alignment with the bed V-ways, resulting in the lathe turning and boring taper.
UNLESS A LATHE IS LEVEL AND UNSTRESSED ON A SOLID FLOOR IT CANNOT TURN OUT ACCURATE WORK.

LEVELLING

Bench Lathe
The underside of the feet of bench lathes are provided with screwed levelling jacks (or foot adjusting screws) which screw into the base of the foot.
When the lathe is in position on the bench or cabinet, a spirit level can be placed across the bed, and along the flat to check the level. Adjust the levelling jacks until the bubble in the level is central ensuring that all jacks are taking the weight of the machine and that the machine cannot be rocked on the jacks.
Bolt the machine rigidly to its base by bolts passing through the centre of the jacks, making sure that the fixing bolt washer is resting on the foot and not on top of the adjusting screw.
A suitable wooden bench for lathe mounting can be constructed from timber at least 50 mm (2") thick and approx. 720 mm ($28\frac{1}{2}$") from the ground. This bench should be rigidly fixed to the floor.

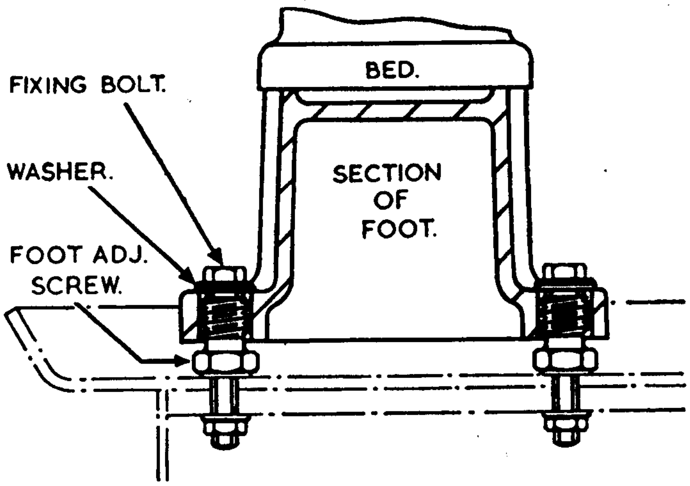


Fig. 20. Lathe Foot with Levelling Screws

Cabinet-mounted Lathes

The cabinet should rest on a solid floor and by using taper wedges it should be levelled using a spirit level placed first along and then across the bed, this is particularly important if suds equipment is to be used. Care must be taken to ensure that the cabinet is evenly supported and cannot be rocked on the wedges.

All Underdrive Lathes have the bed and feet mounted on the cabinet base when the cabinet and the bed are unstressed, the cabinet being on a level floor, thus, when the machine is levelled as directed above the lathe itself will be level and unstressed.

Once level it can, if required, be securely bolted to the floor. If the floor is concrete, which is ideal, a suitable type of expansion bolt is the Rawlbolt E.19—a loose-bolt type requiring a 18.25 mm ($\frac{23}{32}$") hole to a depth of 60 mm ($2\frac{3}{8}$')—the machine may also be grouted into position before bolting to ensure it maintains its stability.

Anti-vibration mountings can be used if required but the effect of these is nullified if the machine is bolted to the floor.

Checking the Lathe Level

When you are satisfied that the lathe is level, put a steel bar of at least 25 mm (1") diameter in a chuck and machine two collars of equal diameter 80 to 100 mm (3" or 4") apart, as shown in Fig. 21. Take a very light finishing cut across both these collars without altering the adjustment of the tool bit. Measure the diameter of each collar carefully with a micrometer.

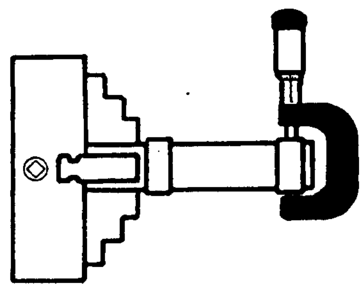


Fig. 21. Method of Checking Levelling of Lathe

If both collars are not equal in diameter the levelling is not accurate enough, or the alignment of the headstock has been disturbed. The levelling should be rechecked using a spirit level on vee blocks across the bed and along the flat, and the headstock location should be checked by first releasing the two bolts beneath the headstock, sliding the headstock to and fro along the bed and then reclamping securely making sure that the reverse gear is correctly aligned with the gears in the gear train (see p.64). The final levelling can be corrected by adjusting wedges under the cabinet base or the screwed jacks under the feet at the tailstock end of the lathe until the collars are turned to exactly the same diameter.

Lubricating the Lathe

To ensure smooth working of a new lathe every one of its bearings must be oiled before the lathe is started. A lathe should be oiled twice a day for the first week of its working life. After that oiling once a day will be sufficient. The lathe must never be oiled whilst it is running. Machine oil, not engine oil, is the correct lubricant to use.

Get into the habit of oiling the lathe in the same way every time. It avoids mistakes and ensures that no oil holes are overlooked.

A lathe does not need flooding in oil. A few drops applied to each oil hole is quite enough. Any oil in excess of that will run out of the bearings and get on to the lathe and collect dirt.

Chippings, dust and dirt of any kind should never be allowed to accumulate. When the process of oiling the lathe and drive has been completed, all excess oil must be wiped away from the bearings with a clean cloth or piece of waste. Treat surplus oil, rust, dirt and chippings as the lathe's worst enemy. At all times and at all costs, always keep the lathe clean.

Lubricating the Lathe

*1. HEADSTOCK SPINDLE BEARINGS. These are grease packed and require a small amount (1 cc. approx.) of new grease every three to six months, introduced through screw cap greasers at the front of the headstock.

2. SPINDLE PULLEY (Bench Model). Remove the small screw in the bottom of the second groove in the cone pulley and fill the oil reservoir twice a day for the first week and daily thereafter when back gear is in use. See Fig. 9.

2. SPINDLE PULLEY (U/Drive and VSL/LOO). Set back gear lever (top of headstock) to direct drive position, depress spindle locking plunger at front of headstock and rotate spindle by hand until lock is engaged. Depress ball spring oiler (visible through hole in front of headstock) with spout of oil can and oil at least daily when back gear is in use.

3. BACK GEAR SLEEVE. Fill oil reservoir daily (on Underdrive machines through oiler in centre of shaft at end of headstock) when back gear is in use. See Figs. 9 and 10.

4. HEADSTOCK GEARING (Bench Model). Apply oil to groove in sliding gear daily. (Fig. 9).

4. HEADSTOCK GEARING (U/Drive and VSL/LOO). Gears and shifter mechanism are treated with grease and should be inspected periodically by removing cover from top of headstock. Any addition should be of a similar type grease. (Fig. 10).

5. REVERSE GEAR BRACKET. Oil the bracket and gears at least daily when in use. (Fig. 16).

6. GEAR BOX and LEADSCREW BEARINGS. Oil daily.

7. TAILSTOCK. Oil at least weekly.

8. APRON BEARINGS. Oil daily.

9. LEAD SCREW AND HALF NUTS. Oil every hour, when in use. (Fig. 15).

10. CROSS SLIDE and TOOL SLIDE SCREWS and BUSHES—Oil at least weekly.

11. CARRIAGE V-WAYS and DOVETAILS. Keep clean and well oiled. (Inspect saddle wipers occasionally).

12. COUNTERSHAFT BEARINGS (Bench machines). Oil daily. See Fig. 18.

13. MOTOR BEARINGS. These are usually grease packed and sealed for life, but if not, they will only require attention after 12 months. A small amount of grease, one turn of the greaser cup (if fitted), or one or two drops of oil every three to six months.

14. BED V-WAYS and FLAT—Keep clean and well oiled.

15. UNDERDRIVE BELT TENSION LINKAGES etc. Oil weekly.

*16. { INTERMEDIATE SHAFT PULLEY—Oil daily through oiler at end of shaft external to cabinet.
COUNTERSHAFT BEARINGS—Oil daily.

17. MOTOR BEARINGS. See (13) above.

18. VARI-SPEED DRIVE UNIT.
   - (i) Oil linkage and leadscrew occasionally.
   - (ii) Give grease cup on manually operated pulley one turn every three to six months.
   - (iii) Inspect sleeves of variable speed pulleys and smear with moly-slip or similar lubricant if necessary every six months.

*N.B.—All rotating shafts on recent underneath drive units are mounted on bearings greased and sealed for life, no attention being required.

Headstock spindle bearings on recent machines are greased for life, no attention being required.

Recommended Lubricants

MAKE	GREASE	OIL
Castrol Industrial	Spheerol AP2	Magna E.D.
Duckham	Duckham's LB10	Duckham's SE15
Esso	T.S.D. 807	Esstic 55
Mobil	Mobilux No. 2	Mobil Vactra Heavy
B.P.	B.P. Energrease LS2	B.P. Energol HP30
Texaco	Regal AFB2	Regal PE. (R & O)
Shell	Alvania 2 (or RA)	Carnea 41
Sternol	Sternoline L.H.T.	Merlin 71
Caltex	Regal Starfak Premium 2	Regal PE. (R & O)

ALTERNATIVELY Grease—Any good quality bearing grease as used in garages.
Oil—Any light machine oil S.A.E. 20 grade or similar.

Lathe Tools and their Application

Chapter 3

The correct type of lathe tool must always be used if the lathe is to machine efficiently and accurately. The tool must have a keen and well-supported cutting edge which has been ground for the particular metal which is being machined. It is important that it be set at the correct height.

Figs. 27, 28 and 29 show high-speed steel tool bits mounted in forged steel holders. These are the most popular types of lathe tool.

Many classes of work cannot readily be executed with the normal finishing tools. For such work it is necessary to use the boring tool, parting-off tool, threading tool and knurling tool.

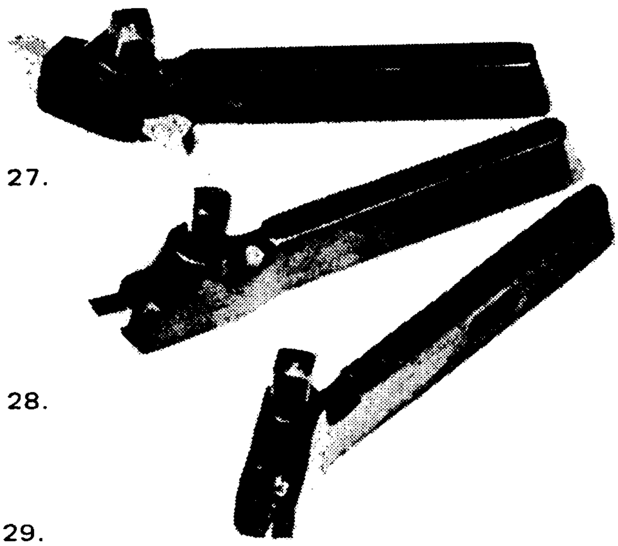


Fig. 27.

Fig. 28.

Fig. 29.

Left-Hand, Straight and Right-Hand Turning Tools

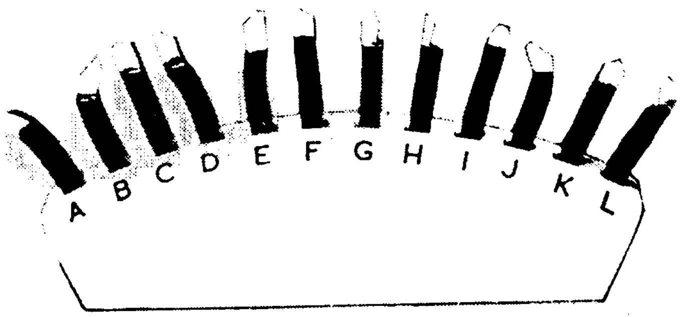


Fig. 30. Set of H.S.S. Tools in Wooden Stand

Fig. 31. Thread-form Cutting Tool

Fig. 32. Straight Parting-off Tool

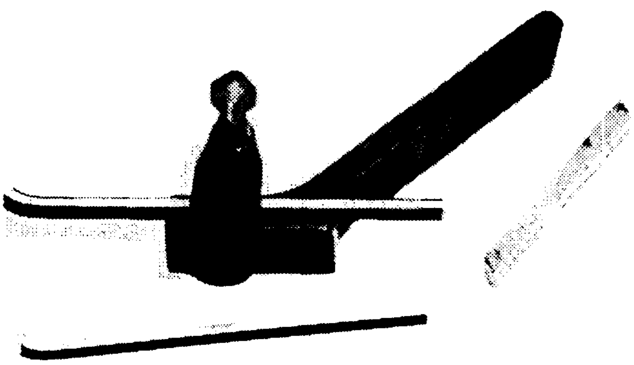


Fig. 33A. American type Boring Bar Holder

Fig. 34. Knurling Tool

Fig. 33B. 'Nulok' type Boring Bar

Correct Height of Tool Cutting Edge

For ordinary straight cutting the correct height of the tool cutting edge is about 5 degrees above centre, or 1 mm per 20 mm diameter of the work ($\frac{3}{64}$" per inch of diameter), as shown in Fig. 35. Take the position of the tool bit into consideration when grinding the various angles, because the height of the tool bit determines the amount of front clearance which is needed to permit free cutting.

Always place the cutting edge of the tool bit exactly on centre, as indicated in Fig. 36, for every type of taper turning and boring, as well as for cutting screw threads and turning brass, copper and similar tenacious metals.

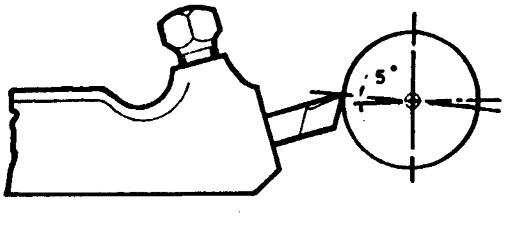


Fig. 35. Tool Cutting Edge above Centre

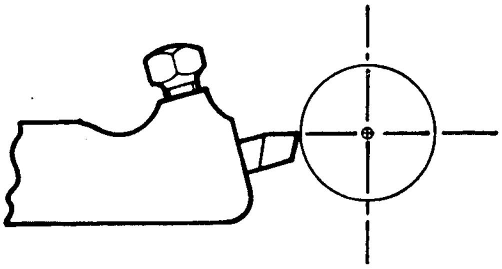


Fig. 36. Tool Cutting Edge on Centre

Tool Angle Variations

The tool angle or angle of keenness is the name given to the included angle of the tool bit's cutting edge. It varies with the texture of the work to be machined. When soft steel is being turned, for example, a rather acute angle is required. Hard steel or cast iron, on the other hand, demands a well-supported cutting edge and consequently the angle is not so acute.

Experience proves that the most efficient tool angle for machining soft steel is an included angle of 61 degrees, as illustrated in Fig. 37. The cutting edge's included angle must be increased to approximately 71 degrees when ordinary cast iron is being machined. Chilled iron or the very hard grades of cast iron will require an included angle of as much as 85 degrees.

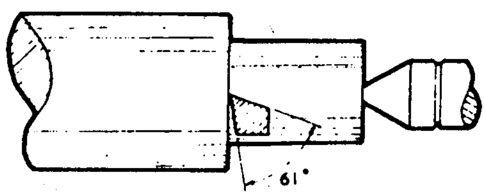


Fig. 37. Angle of Tool for Steel

Grinding Lathe Tool Bits

When grinding tool bits the angle made by the tool bit with the bottom of the tool holder must always be taken into account.

The side clearance shown in Fig. 38 is necessary so that the cutting edge can advance freely without the heel of the tool rubbing against the work.

Fig. 39 shows the front clearance which permits the cutting edge to advance freely as the tool is fed to the work. If this clearance is too great it will tend to weaken the cutting edge and probably fracture the tool. Insufficient clearance, on the other hand, will prevent the tool from cutting.

Free cutting is also helped by the side rake and back rake shown in Figs. 38 and 39. Very little side or back rake is needed for cast iron, hard bronze or hard steel.

The angle of keenness shown in Fig. 37 varies from 60 degrees for soft steel to almost 90 degrees for cast iron, hard steel, bronze and similar metals.

The various steps in grinding a tool bit for general machine work are illustrated in Figs. 40 to 44. The quality of the finish and the length of the tool's life are increased through honing the cutting edge as shown in Fig. 45.

Fig. 38. Side Clearance and Side Rake for Tool Bit

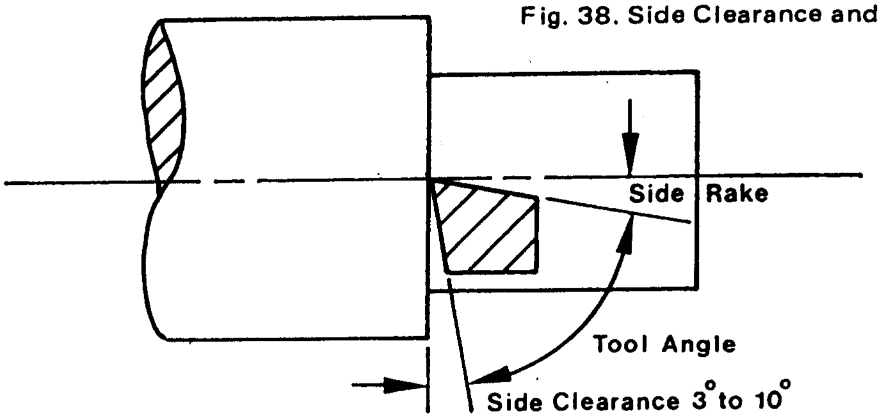
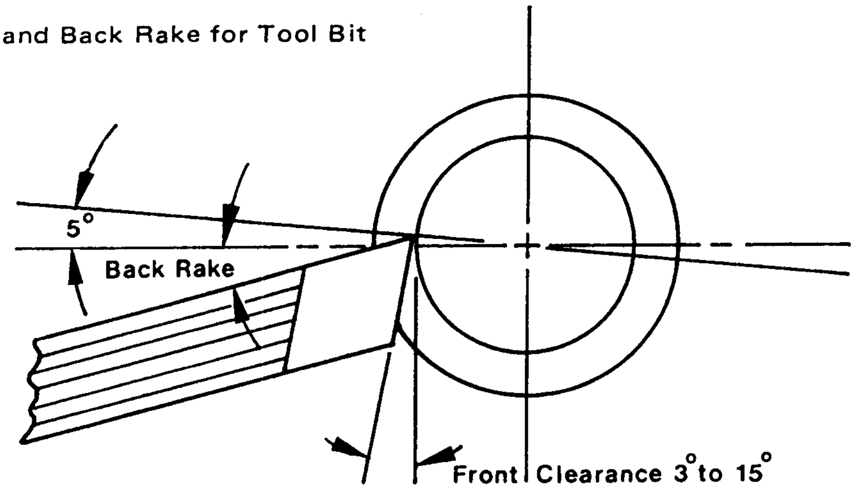


Fig. 39. Front Clearance and Back Rake for Tool Bit



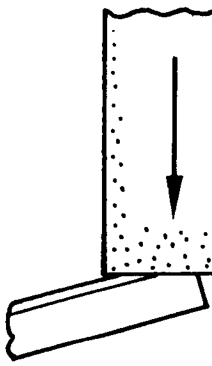


Fig. 40. Grinding Left Side of Tool Bit

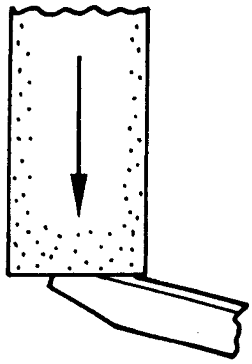


Fig. 41. Grinding Right Side of Tool Bit

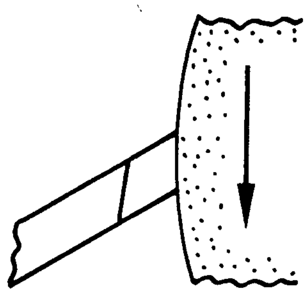


Fig. 42. Grinding Front of Tool Bit

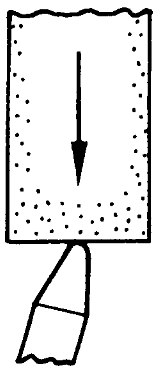


Fig. 43. Rounding Tool Bit

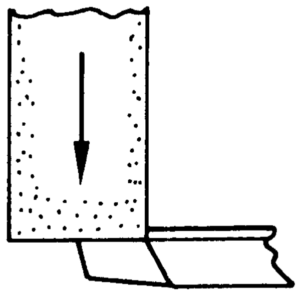


Fig. 44. Grinding Side and Back Rake of Tool Bit

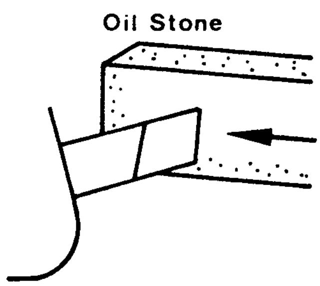


Fig. 45. Honing Cutting Edge of Tool Bit

Rough Turning

Heavy roughing cuts, to reduce the diameter of a steel shaft to the approximate size desired, can be made with the excellent tool pictured in Figs. 46 and 47. It cuts freely without producing a very smooth finish. When using this tool enough stock should be left for a finishing cut with the round-nosed tool shown an Fig. 48.

Fig. 47 shows the shape to which the tool should be ground. Figs. 38 and 39 give instructions on grinding the correct front clearance, etc.

The tool's cutting edge should be straight, with a slightly rounded point. A very small radius of approximately 0.4 mm ($\frac{1}{64}$") at the point will prevent the tool point from breaking down without hampering the free-cutting qualities of the tool.

The cutting edge of this tool should have an included angle or tool angle of about 61 degrees for ordinary machine steel. The angle should be increased for a harder grade of alloy or tool steel, but when free-cutting steel is being machined the angle ought to be rather less than 61 degrees.

If the cutting edge is honed with a small oil stone the tool will not only cut better but will have a longer life.

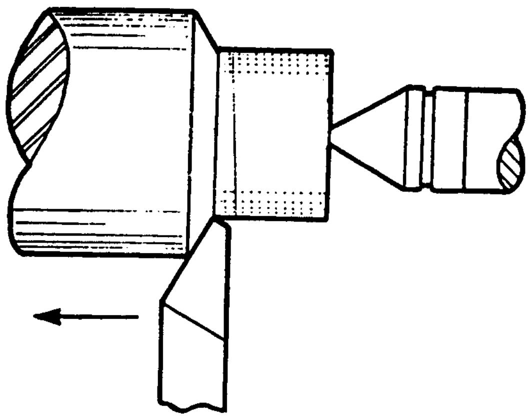


Fig. 46. A Roughing Tool

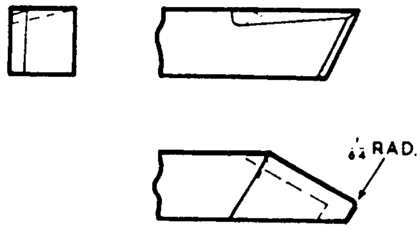


Fig. 47. Detail of Roughing Tool Bit

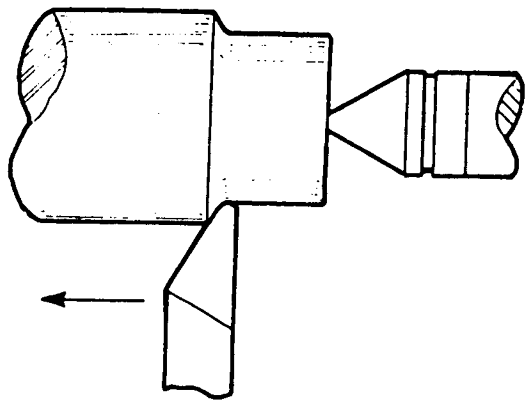


Fig. 48. A Finishing Tool

Finish Turning

A round-nosed turning tool for taking finishing cuts is shown in Figs. 48 and 49. This tool is very similar in shape to the more pointed one shown above for rough turning, except that the tool's point is rounded approximately 1 mm to 1.5 mm radius ($\frac{1}{32}$" to $\frac{1}{16}$").

If, when it has been ground, the cutting edge of this tool is well honed with an oil stone and a fine power traverse is employed, a very smooth finish can be obtained.

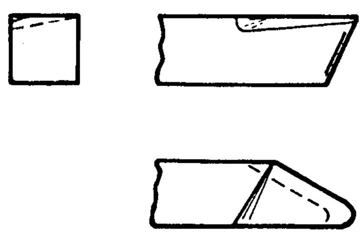


Fig. 49. Detail of Finishing Tool Bit

Round-Nosed Tool

A very convenient tool for reducing the diameter of a shaft in the centre is the round-nosed turning tool ground flat on top to allow for feeding in either direction, as indicated by the arrows in Fig. 50. The shape of the tool bit is shown in Fig. 51 and, by referring to Figs. 38 and 39, the correct angle for front and side clearance can be obtained.

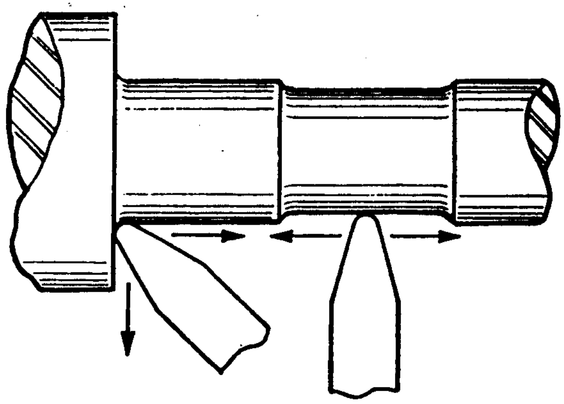


Fig. 50. Round Nosed Tool

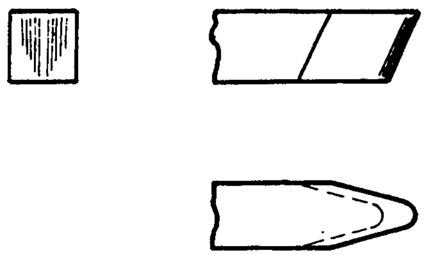


Fig. 51. Detail of Round Nosed Tool Bit

Right-Hand Tool

For general all-round machine work the most common type of tool is the right-hand turning tool shown in Fig. 52. This tool can be used for machining work from right to left, as indicated by the arrow. The shape of the tool is shown in Fig. 53 and the correct angles of clearance are given in Figs. 38 and 39.

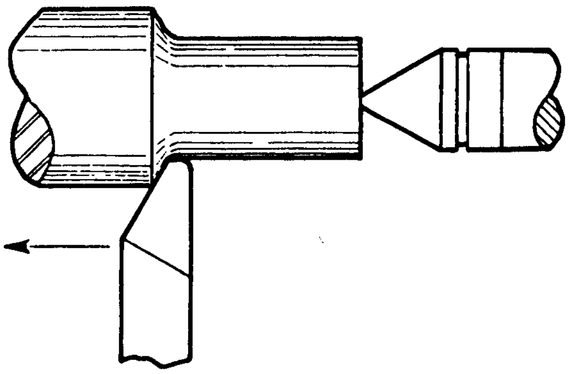


Fig. 52. Right-Hand Turning Tool

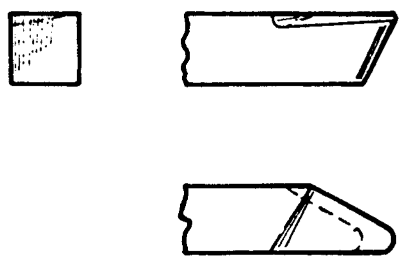


Fig. 53. Detail of Right-Hand Turning Tool Bit

Left-Hand Tool

The exact opposite of the right-hand turning tool in Figs. 52 and 53 is the left-hand turning tool illustrated in Figs. 54 and 55. It is designed for machining work from left to right.

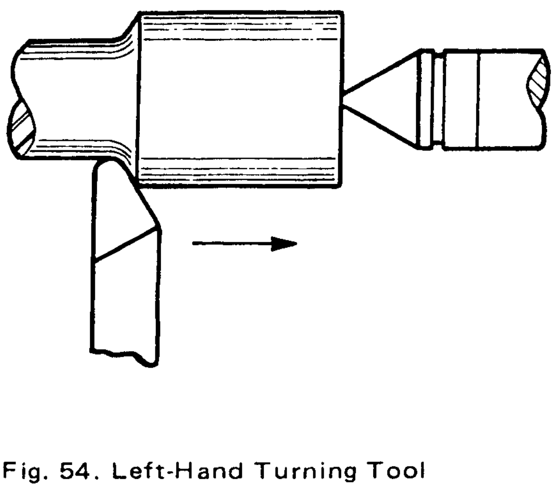


Fig. 54. Left-Hand Turning Tool

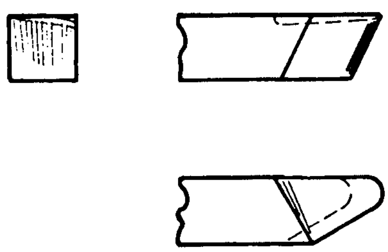


Fig. 55. Detail of Left-Hand Turning Tool Bit

Right-hand Facing Tool

Facing the ends of shafts and machining work on the right side of a shoulder is done by means of the right-hand facing tool, which should be fed from the centre outwards, as indicated by the arrow in Fig. 56. The tool's sharp point is ground to an angle of 58 degrees to prevent any interference with the tailstock centre. Great care must be taken, when this tool bit is in use, to prevent the end of the tool from bumping against the lathe centre. If this should happen it will tend to damage both the centre and the tool's cutting edge. Details of the correct angle for front and side clearance are given in Figs. 38 and 39.

Left-hand Facing Tool

Figs. 58 and 59 illustrate the left-hand facing tool, the reverse of the right-hand facing tool shown in Figs. 56 and 57. This tool is used for facing the left-hand side of the work and the method of application is shown in Fig. 58.

Thread Cutting Tool

The standard type of tool bit for cutting screw threads is shown in Figs. 60 and 61. The tool bit is usually ground flat on top, as shown in Fig. 60 and the tool point must be ground to an angle of 55° for Whit. or 60° for Metric form. If this tool bit is carefully ground and set it will produce screw threads of perfect formation. When this type of tool is used to cut screw threads in steel, the work must always be flooded with a good-quality oil to produce a good, clean thread. When good-quality oil is not available a soluble cutting oil will suffice.

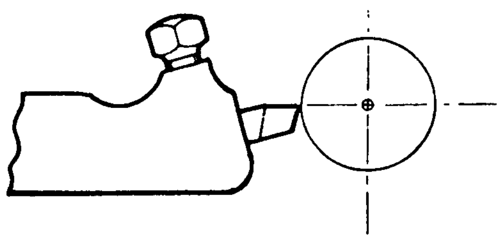


Fig. 60. Thread Cutting Tool

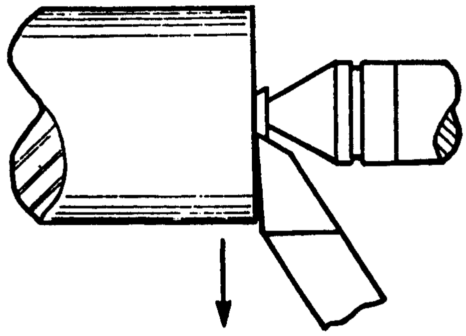


Fig. 56. Right-Hand Facing Tool

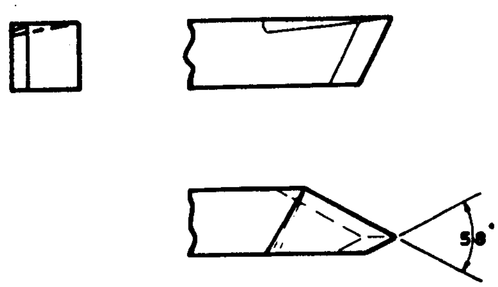


Fig. 57. Detail of Right-Hand Facing Tool Bit

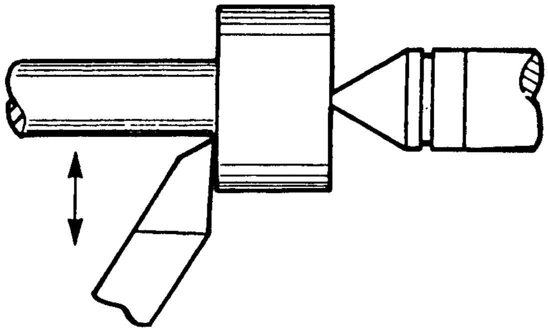


Fig. 58. Left-Hand Facing Tool Bit

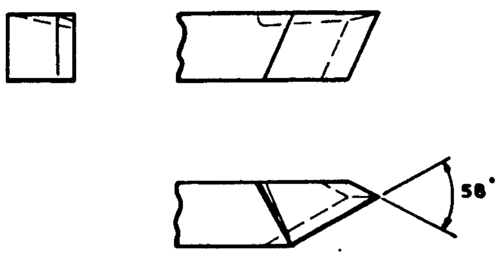


Fig. 59. Detail of Left-Hand Facing Tool Bit

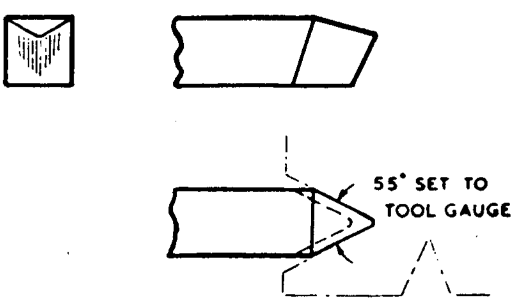


Fig. 61. Detail of Thread Cutting Tool Bit

Brass Turning Tool
The difference between this tool and the round-nosed turning tool shown in Fig. 50 is that the top is ground flat to avoid any side or back rake. By this means the tool is prevented from digging into the work and chattering.

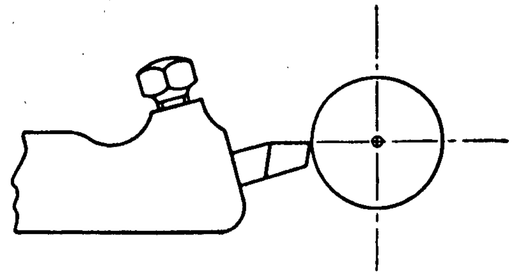


Fig. 62. Brass Turning Tool

Parting-Off Tool
Fig. 64 shows how this tool must always be set exactly on centre. It is sharpened by grinding the end of the tool blade to an angle of 5 degrees as illustrated in Fig. 65.
If there is sufficient taper on the blade sides to provide clearance they do not need grinding. It is common practice to grind the front face of the blade to an angle of up to 10 degrees which helps to remove the pip from the component parted off. When parting-off steel a top rake of approx. 8 degrees should be used together with a good supply of coolant if available. With cast iron a coolant is not required.

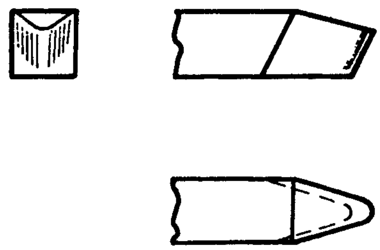


Fig. 63. Detail of Brass Turning Tool Bit

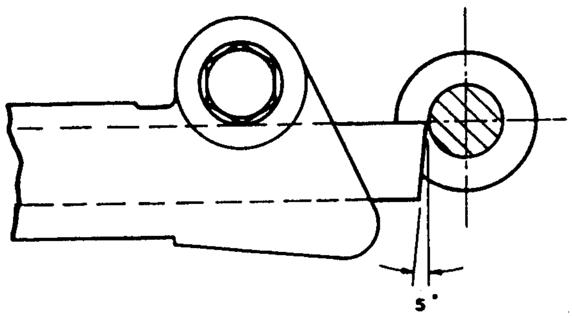


Fig. 64. Parting-off Tool

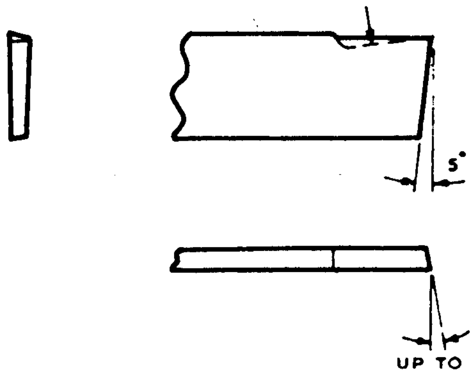


Fig. 65. Detail of Parting-off Blade

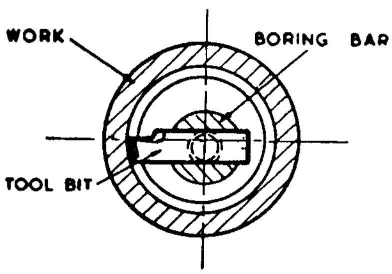


Fig. 66. Boring Tool

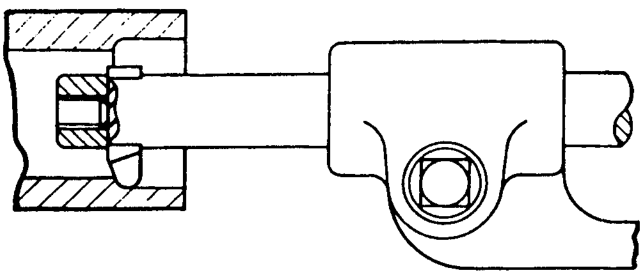


Fig. 67. Boring Tool Bit and Boring Bar

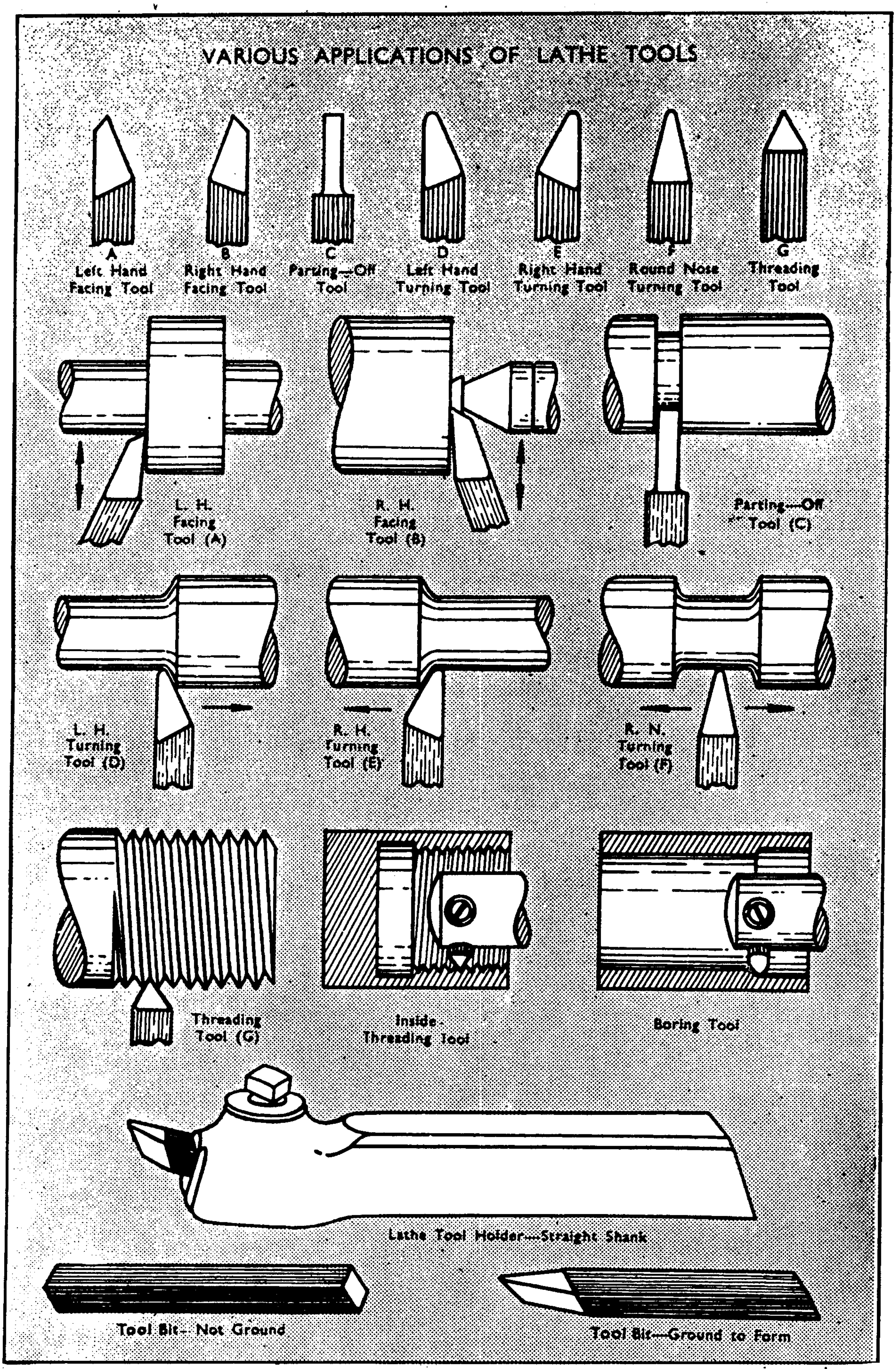


Fig. 68. Tool Bit Shapes and Applications

Boring Tool and Inside Threading Tool
The boring tool is ground just in the same way as the left-hand turning tool shown in Figs. 54 and 55. The only difference is that the front clearance of the boring tool must be ground at a slightly greater angle to prevent the heel of the tool rubbing in the bore of the work. The grinding of the inside threading tool is carried out in the same manner as for the screw thread cutting tool shown in Figs. 60 and 61. The front clearance must again be increased for the same reason as explained for the boring tool.

Tungsten Carbide Tools
When manufacturing operations demand the maximum cutting speeds, tungsten carbide tipped cutting tools are used. They are very efficient for the machining of cast iron, copper, brass, bronze, aluminium and the non-metallic abrasive materials such as fibre, hard rubber and plastics. The cutting speed, which depends on the depth of the cut or feed, may vary between 30 and 300 metres per minute (100 to 1,000 surface feet per minute.)
Tungsten carbide tipped tool bits require a special grade of grinding wheel because the ordinary type will not grind them satisfactorily. Good cutting-edge support is needed to avoid chipping. There should be just sufficient clearance to allow the tool to cut freely.

Tools for Non-Ferrous Materials
The response to machining of such non-ferrous materials as bronze, copper, aluminium and plastics varies considerably. As a consequence they frequently require specially ground cutting tools. The cutting speeds also vary greatly, with a serious effect upon the life of the tool and the surface finish. The following table (Table 1) gives suggested cutting speeds and tool angles for machining the commonest non-ferrous materials together with steels and cast iron.

Machining Soft Metals
Keen-edged tools with greater clearance, and possessing more front and side rake, are needed for aluminium, magnesium alloys and similar relatively soft metals than for the harder ones. The tool's cutting edge is sometimes placed high above the centre in order to achieve more back rake. This obviously cannot be done when turning tapers and facing. It is therefore necessary to make frequent adjustments when the diameter of the work varies.
Tenacious metals like pure copper must be machined with a cutting tool that has been honed to a very keen cutting edge to prevent tearing of the work and the production of a rough finish. The best results are obtained by light cuts at medium feeds with a round-nosed tool whose nose radius is 1 mm to 1.5 mm ($\frac{1}{32}$" to $\frac{1}{16}$").

Machining Plastics
To-day the range of plastic materials is so wide and varied that considerable study and resourcefulness is required in the grinding of cutting tools. There are, for example, hot-set moulded plastics which frequently contain abrasive fillers. These will dull any cutting edge very quickly unless it is well supported by minimum clearance angles. Some other types of plastic, on the other hand, must be given very large clearance angles if the tool is to be prevented from dragging. Laminated plastics and vulcanised rubber have to be machined at high speeds because, at low speeds, the tool's cutting edge is quickly dulled.

Table I. Cutting Speeds and Tool Angles for various materials.

MATERIAL	CUTTING SPEED		TOOL ANGLES IN DEGREES			
	Metres per Minute	Feet per Minute	Front Clearance	Side Clearance	Back or Top Rake	Side Rake
Aluminium — soft alloy ...	120 — 240	400 — 800	9	9	30	15
Aluminium — hard alloy ...	90 — 180	300 — 600	9	9	30	15
Brass	90 — 180	300 — 600	7	6	0	5
Bronze — free cutting ...	90 — 180	300 — 600	5	5	0	2
Bronze — tough	30 — 60	100 — 200	10	12	8	10
Cast Iron	20 — 30	60 — 100	5	4	10	9
Copper	20 — 45	60 — 150	5	5	20	25
Die-Castings (zinc)	60 — 90	200 — 300	8	8	8	10
Magnesium Alloy	180 — 300	600 — 1,000	10	10	8	6
Plastics — cast resin ...	60 — 180	200 — 600	10	12	30	25
Plastics — laminated ...	60 — 180	200 — 600	7	7	25	25
Steel — mild	30 — 60	100 — 200	8	6	20	15
Steel — high carbon ...	10 — 25	35 — 80	6	5	10	5
Steel — stainless	20 — 45	60 — 150	8	6	8	5
Wood	150 — 300	500 — 1,000	15	15	25	25

Accurate Measurements — Chapter 4

All good machine work depends upon accuracy of measurements. Without this it is impossible to produce reliable work. The ability to take such accurate measurements is not a gift men are born with, but is acquired only as a result of practice and experience. Measurements should always be taken with a steel rule or micrometer that has been accurately graduated. Wood rulers or inferior steel rules should never be used. They are invariably inexact and reliance on them will cause inferior work.
Experience will teach a mechanic to obtain measurements to a remarkable degree of accuracy by means of a steel rule and calipers. This result is achieved through the development of a sensitive touch and the careful setting of the calipers so that they "divide the line" graduated on the scale.

Outside Calipers
Fig. 69 illustrates an excellent way of setting an outside caliper to a steel rule. The caliper is held in the right hand, with the rule in the left. One end of the caliper is placed against the end of the rule and supported there by the index finger of the left hand, whilst the necessary adjustment is made by the thumb and first finger of the right hand.

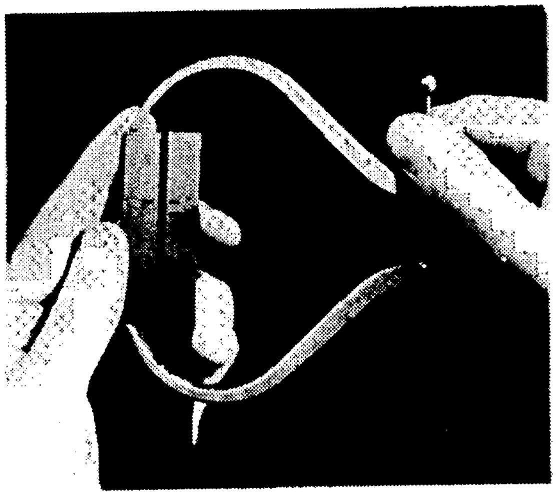


Fig. 69. Setting an Outside Caliper

Measuring with Calipers
Fig. 70 shows the proper way to apply an outside caliper in order to measure the diameter of a shaft or cylinder. The caliper, held exactly at right angles to the centre line of the work, is pushed gently to and fro across the diameter of the cylinder to be measured. If properly adjusted, the caliper should slip easily over the shaft on its own weight. The caliper must never be forced, since this will cause it to spring and result in an inaccurate measurement.

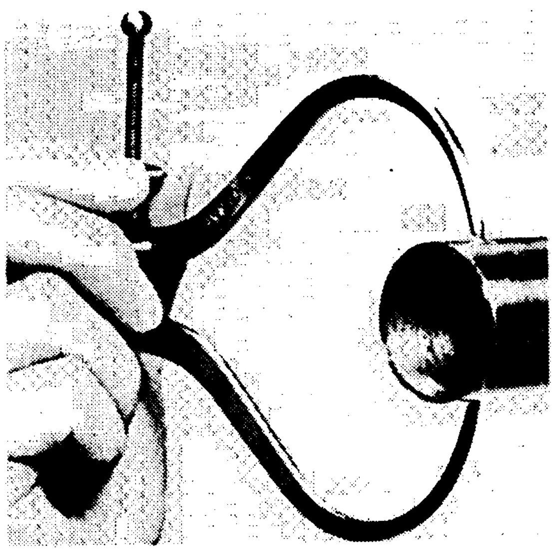


Fig. 70. Measuring with an Outside Caliper

Inside Calipers

When it is desired to set an inside caliper for a definite dimension, the end of the rule is placed against a flat surface. The end of the caliper is placed at the end and edge of this rule. The latter, of course, is held square with the flat surface. Then the other end of the caliper is adjusted to the dimension required. See Fig. 71.

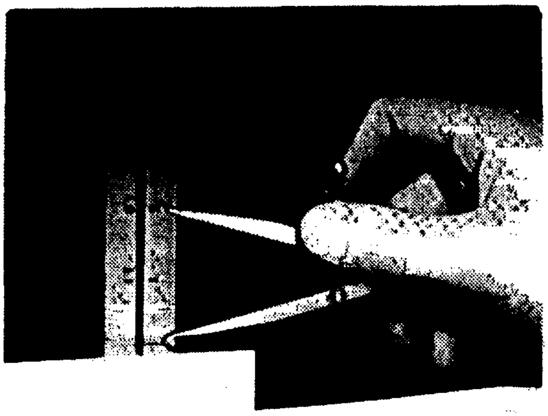


Fig. 71. Setting an Inside Caliper

Measuring Inside Diameters

When an inside diameter has to be measured, the caliper is placed in the hole as indicated in Fig. 72.

The hand is raised slowly and the caliper is then adjusted until it will slip into the hole with a very slight drag. It is very important that the caliper is held square across the diameter of the hole.

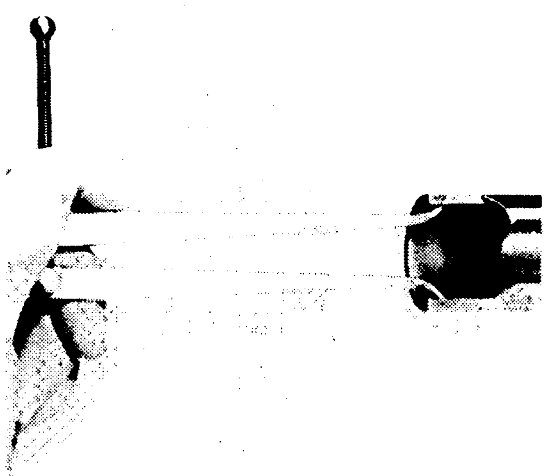


Fig. 72. Measuring with an Inside Caliper

Transferring Measurements

When measurements have to be transferred from an outside caliper to an inside one, the point of one leg of the inside caliper must rest on a similar point of the outside one, as shown in Fig. 73. With this contact point as a pivot, the inside caliper is moved between the other point of the outside caliper and is adjusted by the thumb screw until the touch indicates that the measurement is just right.

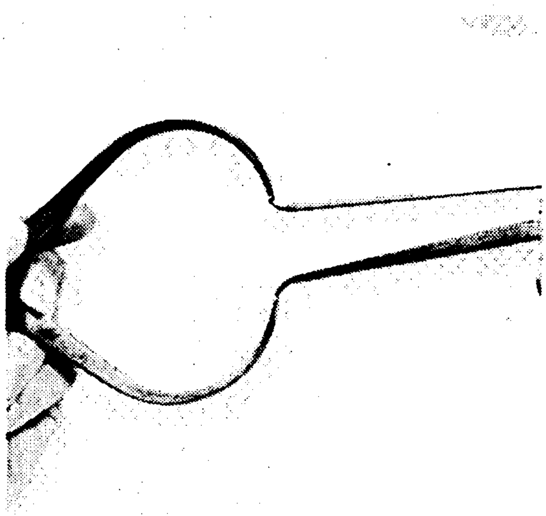


Fig. 73. Transferring Measurement from Outside to Inside Caliper

Hermaphrodite Caliper

Fig. 74 shows a hermaphrodite caliper which is set from the end of the scale in exactly the same way as the outside caliper.

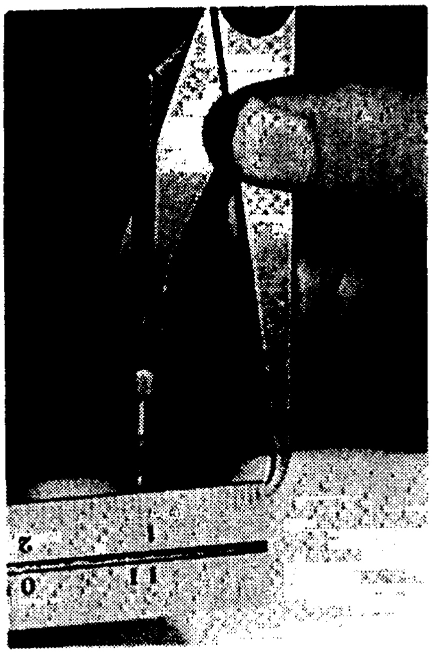


Fig. 74. Setting Hermaphrodite Caliper

Caliper Feel

All contact measurements are bound to rely upon the sense of touch or feel possessed by the operator. It is thus important to develop this sense as highly as possible and observe the following simple rule; never grip the caliper tightly. Always hold it gently and lightly in the finger tips. A tight grip considerably weakens the sense of touch which has to be relied on to secure accuracy.

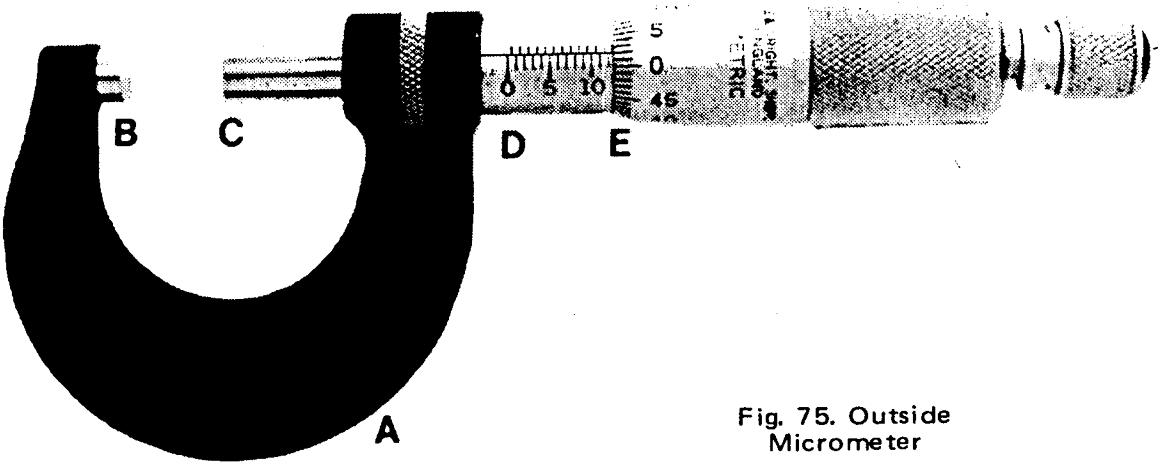


Fig. 75. Outside Micrometer

The Micrometer

Each graduation on the micrometer barrel "D" represents one turn of the spindle "C" or 0.5 mm. Each tenth graduation is numbered and represents 5.0 mm, since 10 X 0.5 mm is 5.0 mm.

On the thimble "E" are fifty graduations each of which represents 0.01 mm (equivalent to ·0004").

The total of the readings on the barrel and the thimble give the micrometer reading. Twenty five graduations are visible on the barrel of the illustration above. Each one represent 0.5 mm so that 25 X 0.5 equals 12.5 mm, and as the thimble is at the second graduation the micrometer reading is 12.5 plus 2 X 0.01 or 12.52 mm.

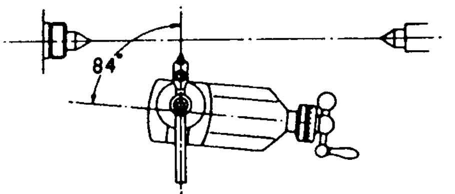
N.B. Each second graduation on the barrel "D" is marked below the horizontal line and this gives a clear reading of every second turn of the spindle "C" and thus represents each 1.0 mm.

Fig. 76. Outside Micrometer in Use

Fig. 77. Inside Micrometer in Use

Using Micrometers

Accurate measurements can be obtained consistently with micrometers, but the sense of feel referred to earlier must be cultivated so that the operator knows when he has the correct tension of the micrometer against the work. Proper care must always be exercised in taking measurements and readings. Internal measurements are taken with an inside micrometer, which is read in the same way as the outside micrometer described on page 31. Master slip gauges should be used frequently to check the micrometer readings and insure against any inaccuracy developing in the instruments themselves.



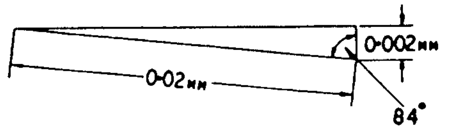


Fig. 78. Compound Rest Set for Fine Cross-Feed

Compound Rest Angle for Fine Cuts

If the compound rest is set to 84 degrees 16 minutes ($84\frac{1}{4}°$), each graduation on the micrometer collar represents an angular movement of 0.02 mm (·0008"), and a cross-feed movement of 0.002 mm (·00008"), or a reduction in diameter of 0.004 mm (·00016"). This angular setting gives a $\frac{1}{10}$ ratio of cross-feed to angular movement, so on machines in Imperial measurement ·001" compound rest movement gives ·0001" cross-feed or a reduction in diameter of ·0002".

This is a great help in adjusting the cutting tool when fine precision finishing cuts involving fractional thousandths in depth are being taken.

The same method can also be used most advantageously for final grinding operations with the tool post grinding attachment on the lathe.

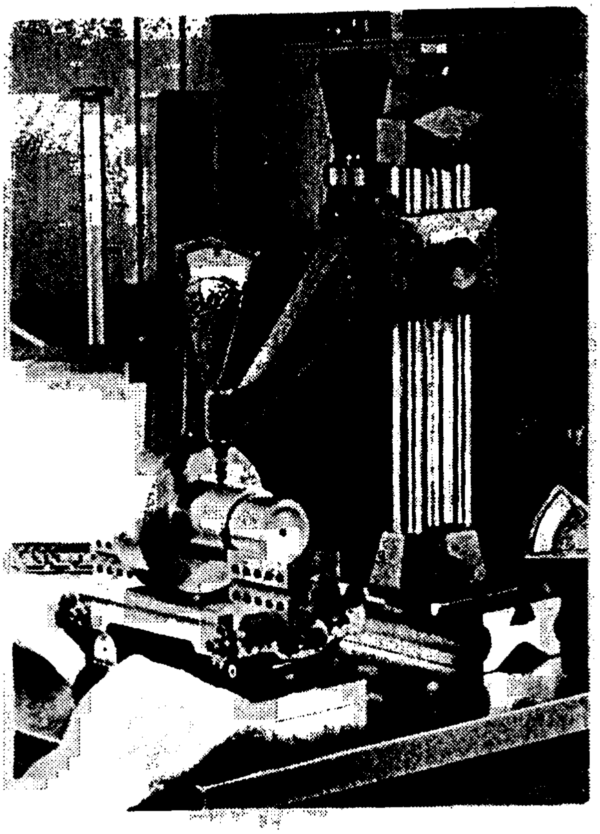


Fig. 79. Checking a Taper with Comparator and Sine Plate

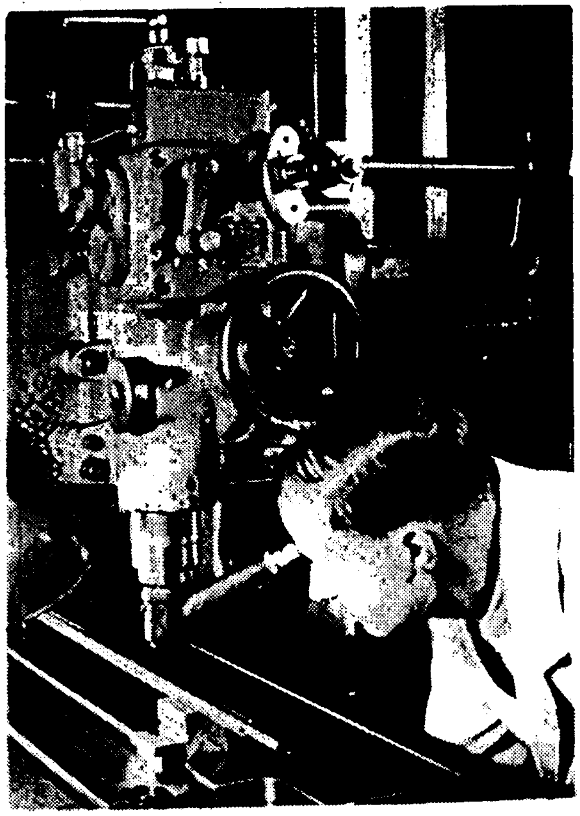


Fig. 80. Testing Accuracy of a Lead Screw

The manufacture of Boxford Lathe components demands high quality Jigs and fixtures together with strict and thorough inspection.
Jigs and fixtures are manufactured in the firm's up-to-date tool room, and the fine accuracy of the lathe components is assured by the wide use of Societe Genevoise Jig Boring Machines and precision Grinding Machines.
Some of the inspection equipment is shown in the illustrations on this page. Fig. 79 shows an Inspector checking the taper on a component by means of a .0025 mm (·0001") Comparator held in a Double Column Stand. The work is mounted on a mandrel resting in two V-Blocks on a precision Sine Plate.
In Fig. 80, an Inspector is checking the pitch accuracy of a Lathe Lead Screw. This is done on a Jig Boring Machine using a locating microscope.
Thread forms and tool angles, etc. are checked on the Universal Measuring Projectors. These projectors have a range of magnification from 10:1 to 100:1.
Continued improvement in metallurgy and application of methods to the production of Lathes brings new reliability and precision into all modern Boxford Lathes.

Plain Turning

Chapter 5

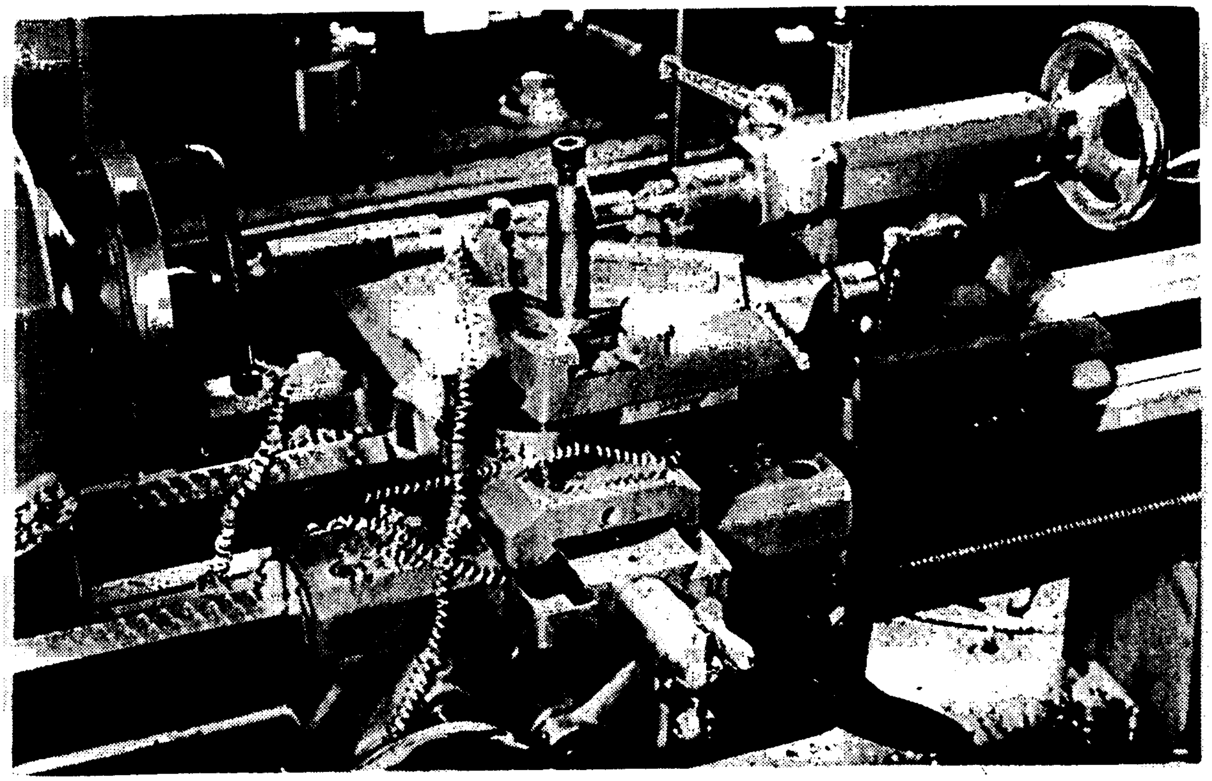


Fig. 82. Turning Between Centres

The above illustration shows a shaft being machined between the lathe centres. Whenever circumstances permit, work should be mounted in this way. It is supported at both ends and thus allows heavier cuts to be taken.

Locating Centre Holes

Centre holes must be drilled in each end of the work before it can be mounted on the lathe centres for machining. There are several reliable methods of locating the position of these centre holes accurately. They are described below.

Divider Method

Chalk the shaft ends, set the dividers to approximately one-half of the shaft's diameter and scribe four lines across each end.

Combination Square Method

The centre head of a combination square should be held firmly against the shaft, as shown in Fig. 83 while two lines are scribed close to the blade across each end of the shaft.

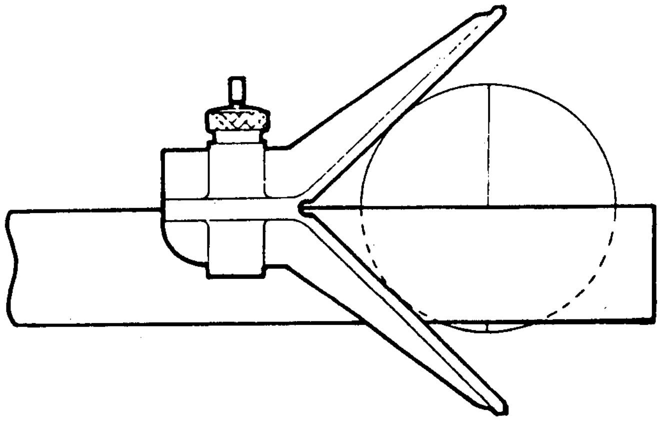


Fig. 83. Combination Square Centre Head to Locate Centres

Hermaphrodite Caliper Method
Each end of the work should be chalked. The hemaphrodite caliper should then be set at slightly more than half the diameter and four lines scribed as shown in Fig. 86.

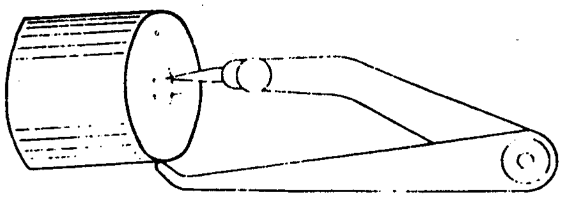


Fig. 86. Centring a Shaft with Hermaphrodite Caliper

Centring Irregular Shapes
To centre irregular-shaped work use a surface gauge and V-block as illustrated in Fig. 87.

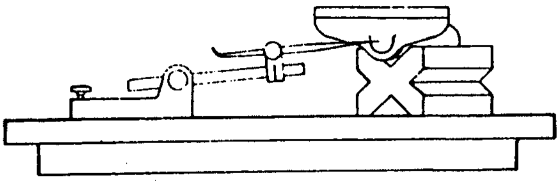


Fig. 87. Centring an Irregular Shape

Drilling the Centre Holes
When the centres have been accurately determined, drill and countersink the centre holes in each end of the shaft. The lathe can be used or a drilling machine can be employed. Use a combination centre drill and countersink as shown in Fig. 88 or a small twist drill followed by a 60 degree countersink as shown in Fig. 89.

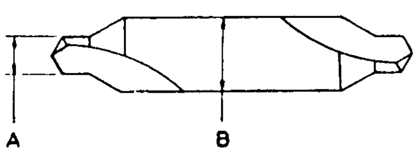
Fig. 88. Combination Centre Drill

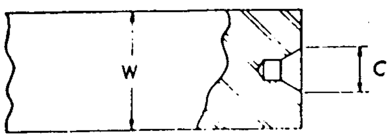
Fig. 89. 60° Countersink

Centre Drill and Countersink
Normally the combination centre drill and countersink is used to drill centre holes. There are available several standard sizes suitable for various sizes of work. These are listed below.

Use great care when drilling the centre holes. Have the speed of the spindle at about 150 revs. per minute and do not crowd the drill. If it is crowded the point may break off and remain embedded in the work. It will then probably be necessary to heat the shaft end to a cherry red and allow it to cool slowly so that the drill point will be annealed and can then be drilled out.

Table 2. Centre Holes for 5 mm to 100 mm ($\frac{3}{16}$" to 4") dia. shafts.
(Using standard combination centre drills)





Dia. of Work W (mm)	Dia. of Countersink C (mm)	Dia. of Drill Point A (mm)	Dia. of Body B (mm)	Number of Drill
5 to 8 ($\frac{3}{16}$" $\frac{5}{16}$")	3	1.6	5	016 (B.S.2)
9 to 25 ($\frac{3}{8}$" 1")	5	2.0	6.3	020 (B.S.3)
30 to 50 ($1\frac{1}{8}$" 2")	6	3.15	10	031 (B.S.4)
55 to 100 ($2\frac{1}{8}$" 4")	10	4	12.5	040 (B.S.5)

Drilling Centre Holes with a Lathe Chuck

Short shafts and rods of a diameter small enough to pass through the headstock spindle can easily be centred by using a universal three jaw chuck, as illustrated in Fig. 90. When this method is used the end of the shaft must be faced smooth before drilling the centre hole.

The shaft's unsupported end should not project more than 150 mm (6") beyond the chuck jaws. If a shaft is too large to pass through the headstock spindle and is too long to be held firmly by the chuck jaws alone, it can be supported on the outer end in a fixed steady, (see Fig. 204).

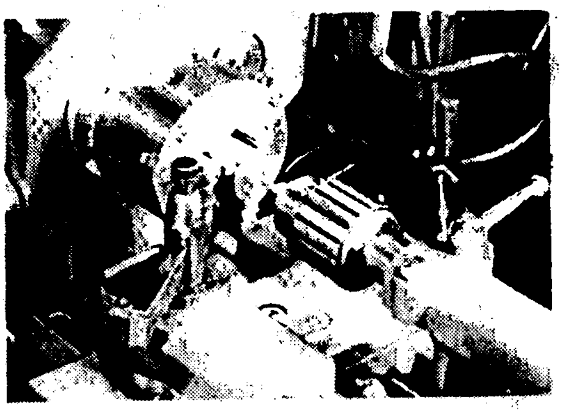


Fig. 90. Centring in the Lathe

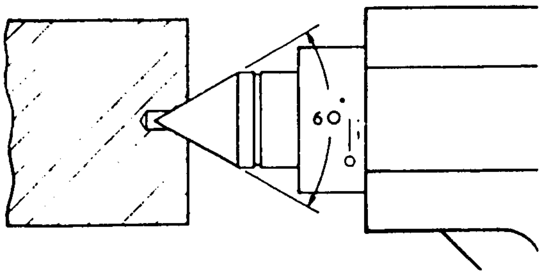


Fig. 91. Correctly drilled Centre Hole

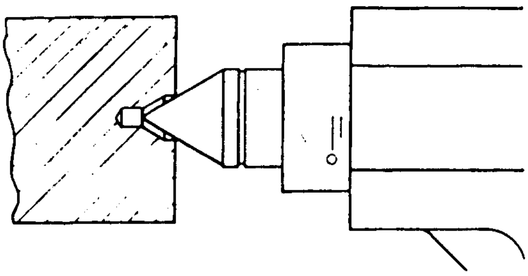


Fig. 92. Centre Hole Drilled too Deep

Correct Centre Hole

The centre hole, to be correct, ought to be the size required for the shaft's diameter, as listed in Table 2, and the countersink must fit the centre point perfectly, as shown in Fig. 91. Enough clearance must also be given at the bottom of the countersink.

Allow for the thickness of metal that will be faced off at the end when drilling centre holes. If this is not done the centre holes will be too small to support the shaft when the ends have been faced.

Badly Drilled Centre Holes

Badly drilled centre holes are frequently responsible for unsatisfactory work. A shallow centre hole with an incorrect angle and no clearance for the tip of the centre point or a centre hole that has been drilled too deeply as shown in Fig. 92. If the centre holes are poorly made it is impossible to expect accurate work, apart from the risk of damaging the lathe centres.

Fig. 93. Cranked Lathe Dog

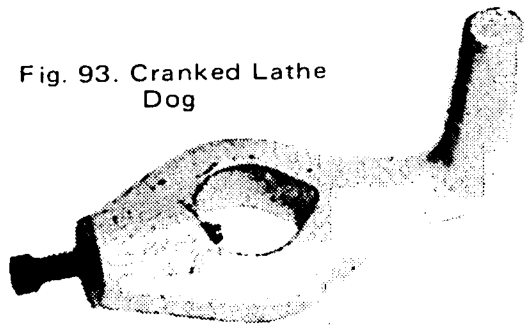


Fig. 94. Straight Lathe Dog

Lathe Dogs for Driving Work Between Centres

Most popular among the types of lathe dog is the common one shown in Fig. 93. The safety lathe dog equipped with a headless set screw is not likely to catch the operator's sleeve. The set screw must always be securely tightened when the lathe dog is being attached to the work.

For small lathes the acme type lathe carrier is recommended for shafts over 25 mm (1") dia. This requires a driving pin fixed to the catch or driving plate (see Boxford accessory leaflet for details).

Mounting Centres in the Lathe Spindle
Never mount the lathe centres in the headstock or tailstock until you have thoroughly cleaned the centres, the tapered holes and the spindle sleeve. The smallest quantity of dirt or a very tiny chip can cause the centre to run out of true. Use a cloth and stick to clean the taper holes, and never insert a finger in the revolving spindle.

Removing the Lathe Centres
Hold the sharp point of the headstock centre in a piece of rag in the right hand. With the left hand use a rod to give the centre a sharp tap through the spindle hole.
When you want to remove the tailstock centre, turn the tailstock handwheel to the left until the end of the tailstock screw ejects the centre.

Alignment of Centres
The lathe centres should always be checked for alignment, as depicted in Fig. 95 before work is mounted between them. Should the tailstock centre not line up, loosen the tailstock clamp lever and adjust the tailstock body in the proper direction by means of the tailstock centring screws.

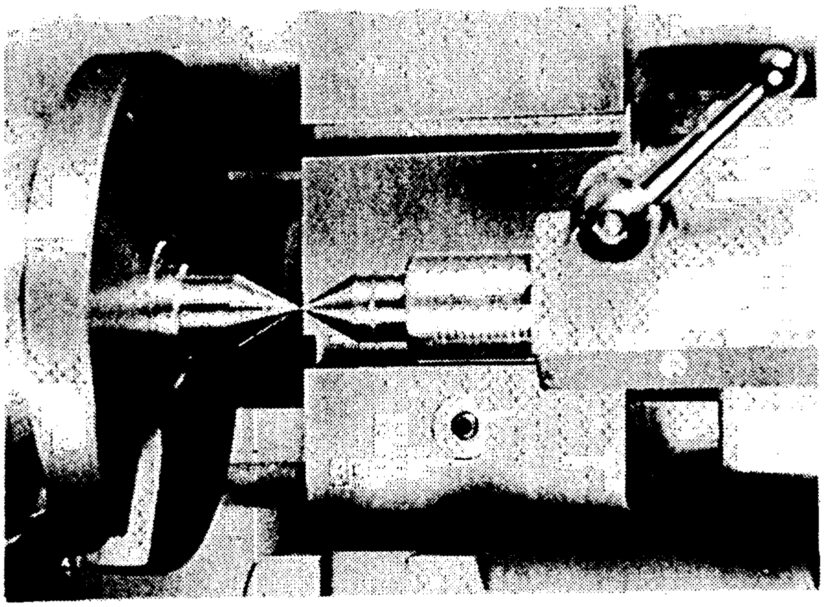


Fig. 95. Check for Alignment of Centres

Mounting Work Between Centres
Before any work is mounted between centres, a drop of oil must be placed in the centre hole for the tailstock centre point. See that the tail of the lathe dog fits freely into the catch plate slot so that the work rests securely both on the headstock centre and the tailstock centre. Be certain that there is no binding of the lathe dog in the slot of the catch plate. The tailstock centre should not be too tight against the work.

Expansion
There is always a danger of heating and consequent expansion of work while it is being machined in the lathe. Such expansion of work mounted between centres will probably cause binding. If this happens, stop the lathe and re-adjust the tailstock centre. When a long shaft is being machined it may be necessary to make several such adjustments.

Facing the Ends

Face the ends of a shaft square before the diameter is turned. The tool bit should be ground as indicated in Fig. 57 and the cutting edge must be set exactly on the centre as shown in Fig. 96. Take care not to break the point of the tool against the tailstock centre. The tool should be fed out to face the end in the manner shown in Fig. 97.

Position of Tool for Turning

Fig. 47 shows the proper way of grinding the tool bit for turning. The tool bit's cutting edge and the edge of the tool holder must not extend further than is necessary over the edge of the compound rest.

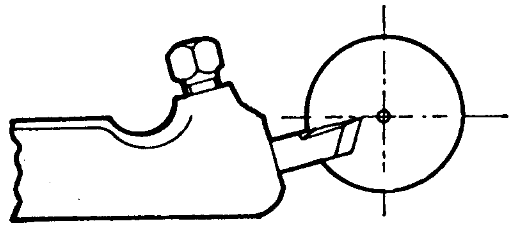


Fig. 96. Tool Bit Position for Facing End of Shaft

Fig. 98 shows how to set the tool so that it will not dig into the work if it happens to slip in the tool post. Instead, it will move in the direction of the arrow away from the work.

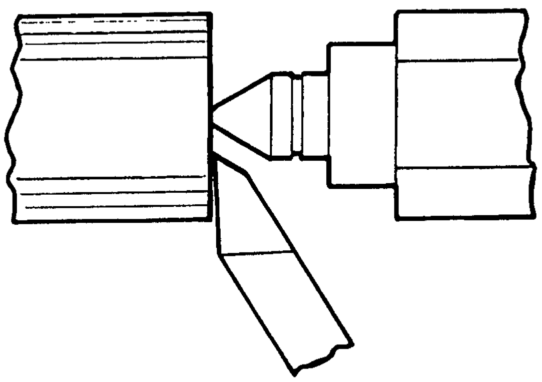


Fig. 97. Facing the End of a Shaft

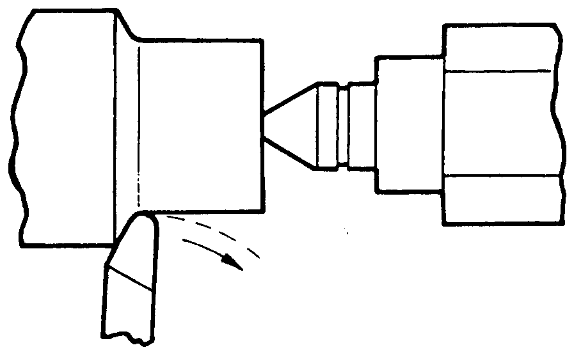


Fig. 98. Position of a Tool Bit for Turning

Direction of Power Feed

Feed the tool towards the headstock if possible. This applies the pressure of the cut to the headstock centre which revolves with the work.

Rate of Power Feed

The size of the lathe, the nature of the work and the amount of stock to be removed are the factors governing the rate of the power feed.

A feed of 0.2 mm (·008") per revolution can be used on a small lathe, but feeds as coarse as 0.5 mm (·020") are not infrequent for rough turning on larger lathes. When turning long, slender shafts care must always be used because a heavy cut with a coarse feed can bend the shaft and ruin the work.

Cutting Speeds for Turning

The cutting speed to achieve the most efficient results depends on the metal being machined, the depth of the cut, the feed and the type of tool bit employed. Too slow a cutting speed may result in lost time while if it is too fast the tool quickly dulls. The following table shows the cutting speeds which are recommended for high speed tool bits:

Table 3. Cutting Speeds

Material	Roughing Cuts 0.25 to 0.75 mm (0·010" to 0·030")		Finishing Cuts 0.05 to 0.25 mm (0·002" to 0·010")		Screwcutting	
	metres/min.	feet/min.	metres/min.	feet/min.	metres/min.	feet/min.
Alloy Steel ...	20	60	25	80	10	30
Aluminium ...	105	350	150	500	20	60
Brass	90	300	120	400	20	60
Bronze ...	30	100	45	150	10	30
Cast Iron ...	25	80	30	100	10	30
Mild Steel ...	35	110	45	150	12	40

These speeds can be increased by 25 to 50 per cent. when a cutting lubricant is used. With tungsten-carbide tipped cutting tools in use the cutting speeds can be increased from 100 to 800 per cent.

To find the number of revolutions per minute for a given cutting speed in metres per minute, multiply this given cutting speed by 1000 and divide the product by the circumference (in millimetres) of the turned part.

Example: Find the number of revolutions per minute for a 50 mm shaft for a cutting speed of 40 metres per minute.

$$\frac{40 \times 1000}{3{\cdot}1416 \times 50} \text{ equals } 254{\cdot}65 \text{ R.P.M.}$$

If the shaft to be turned and the recommended cutting speeds are in different units (one in Imperial and one in metric) the general rule is to convert to one standard (using 1 inch = 25.4 mm), then as above dividing the cutting speed by the circumference and thus obtaining the R.P.M.

Testing Alignment of Centres

When the first roughing cut across a shaft has been taken, check the diameter at each end of the cut with calipers or micrometer to make sure that the lathe is turning straight. When the position of the tailstock is changed for a different length of work it may require adjustment. This is particularly true of old lathes which may have worn spots or burrs on the bed.

Fig. 99 shows an excellent method for testing the alignment of the lathe centres. Two collars turned on a shaft about 40 mm ($1\frac{1}{2}$") in diameter and 250 mm (10") in length are machined with a fine finishing cut without changing the adjustment of the cutting tool, both collars are measured, and, if their diameters do not agree, the adjustment of the centres is not accurate, and the tailstock should be adjusted in the required direction.

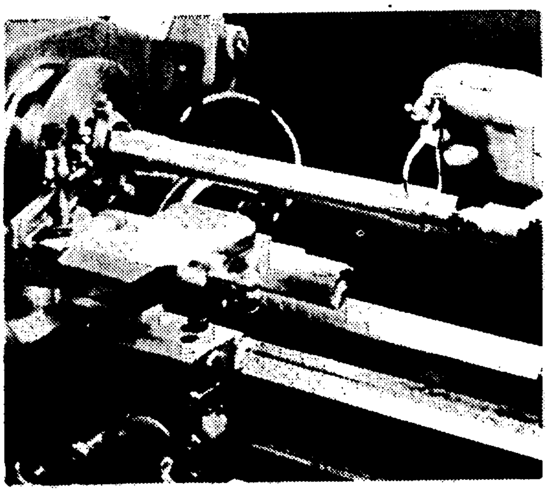


Fig. 99. Testing Alignment of Centres

The following table lists the spindle speeds for various diameters and metals and saves the trouble of making calculations.

Table 4. Spindle Speeds for Turning and Boring (in r.p.m.)

High Speed Tool Bits – Average Cuts

DIAMETER		ALLOY STEEL		CAST IRON		MILD STEEL		BRASS		ALUMINIUM	
Millimetres	Inches	20 metres/min.	60 ft./min.	25 metres/min.	80 ft./min.	40 metres/min.	120 ft./min.	100 metres/min.	300 ft./min.	120 metres/min.	400 ft./min.
	¾	335	305	420	405	665	610	1665	1530	2000	2040
20		320	290	400	380	640	575	1590	1435	1915	1910
25	1	250	230	320	305	510	460	1275	1150	1530	1530
	1½	165	155	210	205	330	305	835	765	1000	1020
40		160	145	200	190	320	285	795	715	955	955
50	2	125	115	160	150	255	230	635	575	765	765
75	3	85	76	105	100	170	155	420	380	510	510
100	4	63	57	80	76	125	115	320	285	380	380
125	5	51	46	64	61	100	92	255	230	305	305
150	6	42	38	53	51	85	76	210	190	255	255
175	7	36	33	45	44	73	65	180	165	220	220
200	8	32	29	40	38	64	57	160	145	190	190
250	10	25	23	32	31	51	46	125	115	155	155

Adjustment of Tailstock Body
To adjust the tailstock body release one of the adjusting screws (A or B) and tighten the opposite one for a similar distance. Take a further test cut on the collars and measure it. Repeat this operation until the necessary degree of accuracy has been secured.
A mark on the end of the tailstock at the junction of the base and body shows their relative positions. Do not rely upon this mark for fine accurate work, but rather make the alignment test described above to make certain that the centres are in line.

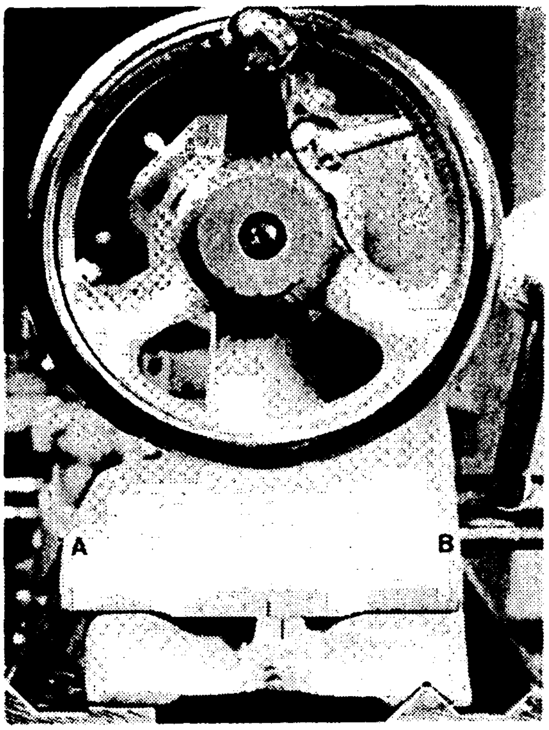


Fig. 100. Tailstock Body Offset

Machining to a Shoulder
Fig. 101 shows a good method of locating a shoulder on a shaft. Chalk the shaft, set the hermaphrodite calipers to the required dimension and scribe a line around the revolving shaft with the sharp point of the caliper.
Fig. 102 shows the way in which a round-nosed turning tool is used to finish a shoulder which has a fillet corner. (See Fig. 50.)

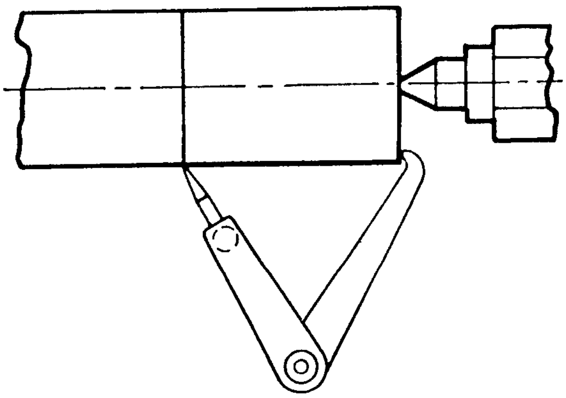


Fig. 101. Locating a Shoulder with Hermaphrodite Caliper

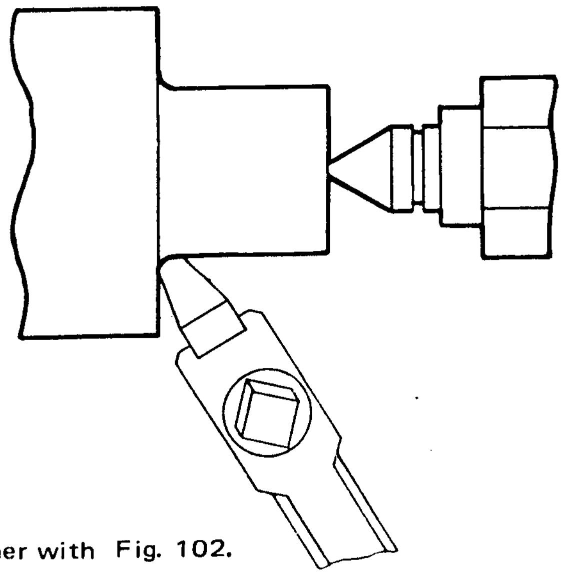


Fig. 102. Machining a Fillet Corner with Round-Nosed Tool

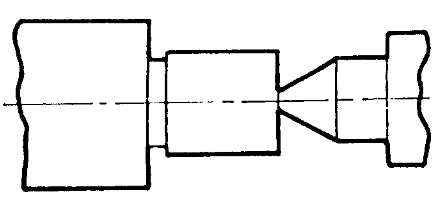


Fig. 103. Undercut for Square Corners

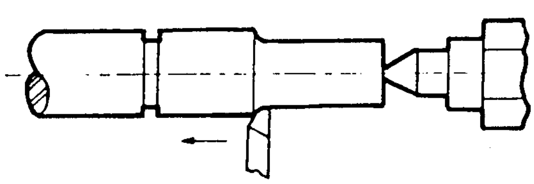


Fig. 104. Locating Shoulder with Parting-off Tool

Locating Shoulders with a Parting Tool

When production of a quantity of pieces is required, shoulders are generally located with a parting tool, as shown in Fig. 104 before the diameter is machined.

It is usual to neck or undercut the shoulder slightly, as shown in Fig. 103 when a square corner is needed, as for a bearing.

When measuring between two shoulders or facing shaft ends to length, a firm joint caliper is useful for measuring.

Taper Turning and Boring

Chapter 6

Taper Turning and Boring

Three methods can be used to turn and bore tapers in the lathe: by setting over the tailstock; by using the compound rest; and by using the taper attachment on the lathe. The length of the taper, the taper angle and the number of pieces to be machined governs the choice of method to be used.

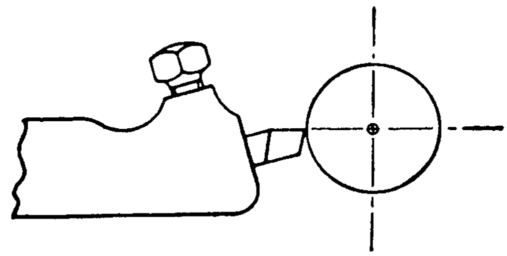


Fig. 105. Tool Bit on Centre for Taper Turning

Tool Bit Must be on Centre

Fig. 105 shows how the tool's cutting edge must be set exactly on the centre if accurate tapers are to be turned or bored. Another way of expressing it is to say that the cutting edge of the lathe tool must be exactly at the same height as the point of the tailstock centre. The tool must be in this position whatever method of turning and boring tapers is used.

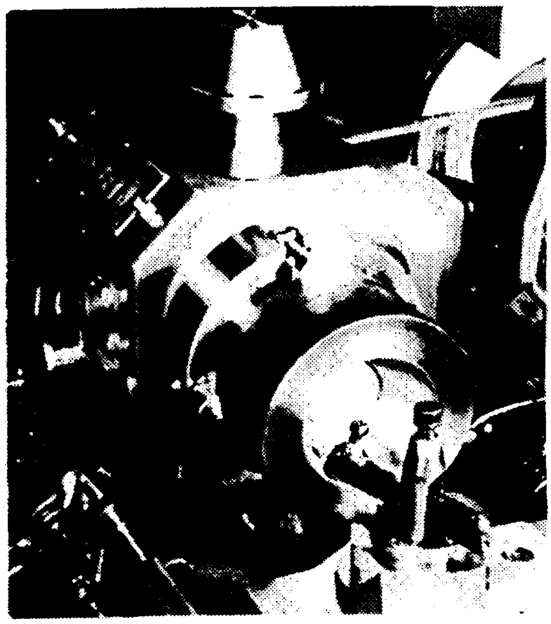


Fig. 106. Boring a Taper with Compound Rest

Taper Turning with Compound Rest

It is standard practice to use the compound rest of the lathe when turning and boring short tapers and bevels. In particular it is used for bevel gear blanks and for die and pattern work, etc. First, the compound rest swivel is set at the required angle and then the taper is machined by turning the compound rest feed screw by hand. See Figs. 106 and 107.

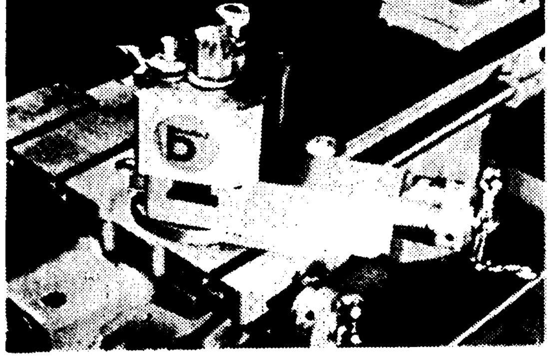
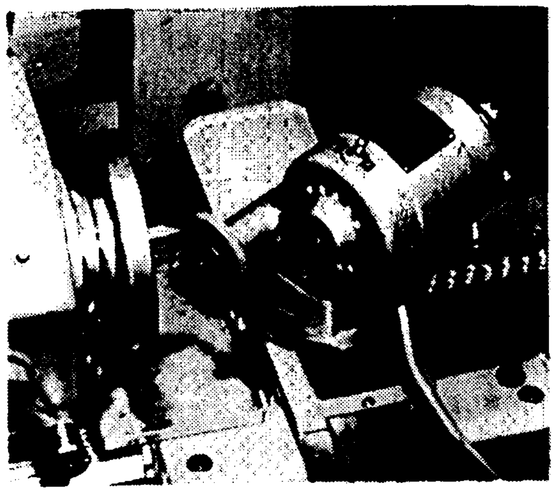


Fig. 107. Compound Rest Swivelled for Taper Turning



To True a 60 Degree Centre

Fig. 108 illustrates the use of the compound rest for short tapers. An electric grinding attachment is mounted on the lathe in place of the tool post. The compound rest is swivelled to the proper angle to permit the grinding wheel to be fed across the 60 degree centre point to grind it true. Lathe centres are usually too hard for a cutting tool to be used to machine them.

Fig. 108. Grinding a 60° Centre in the Lathe

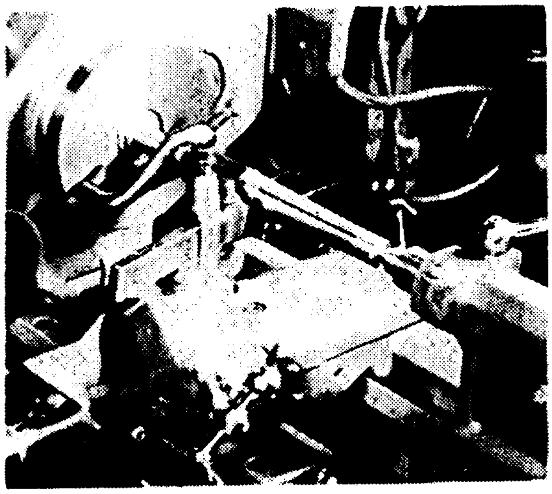


Fig. 109. Turning Taper with Tailstock Offset

Taper Turning with Tailstock Set Over

If work can be machined between centres taper turning can be done by setting over the tailstock body, as illustrated in Figs. 110, 111 and 112. Boring tapers, however, cannot be done by this method.

The tailstock body must be set over by an amount which depends upon the difference in the diameters of the taper and the overall length of the work. Fig. 110 shows how pieces of different lengths will be machined with different tapers with the amount of setover remaining the same. Observe that the tailstock centre is set over one-half of the total amount of the taper for the entire length of the work.

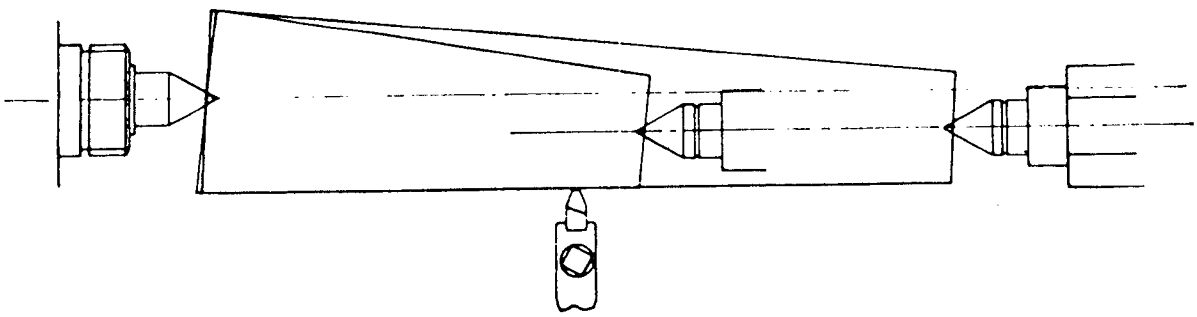


Fig. 110. Diag. to show Different Taper on different lengths with same Tailstock Offset

How to Calculate Amount of Setover for Tailstock

"Millimetres per centimetre" is a way of expressing tapers as is "Inches per foot" when using Imperial measure. Assume for example, that a metric taper has a taper of 1 mm in 2 centimetres (or 1 in 20); then if this taper has to be machined on a shaft 20 centimetres long the centre must be set over 5 mm

(half of 10 mm). Metric tapers can be expressed as ratios or percentages as in the above example the taper is 1 in 20 or 5%.
The following may be useful in calculating the amount of set over.
Given a taper in Millimetres per Centimetre: Take the overall length of the work piece in centimetres and multiply this by one-half the taper given in millimetres per centimetre and the answer will be the set over in mm.
Alternatively: Convert the taper in millimetres per centimetre to a ratio by multiplying the centimetres by 10. A taper of 1 mm in 2 cms becomes 1 in 20 and if the overall length of the component is 20 cm, the taper is therefore 1 cm (or 10 mm) and the setover is half this amount (5 mm).
Given diameters at end of taper: Take the total length of the work piece and divide by the length of the part to be tapered. Multiply this quotient by one-half the difference in diameters to obtain the amount of the set over.

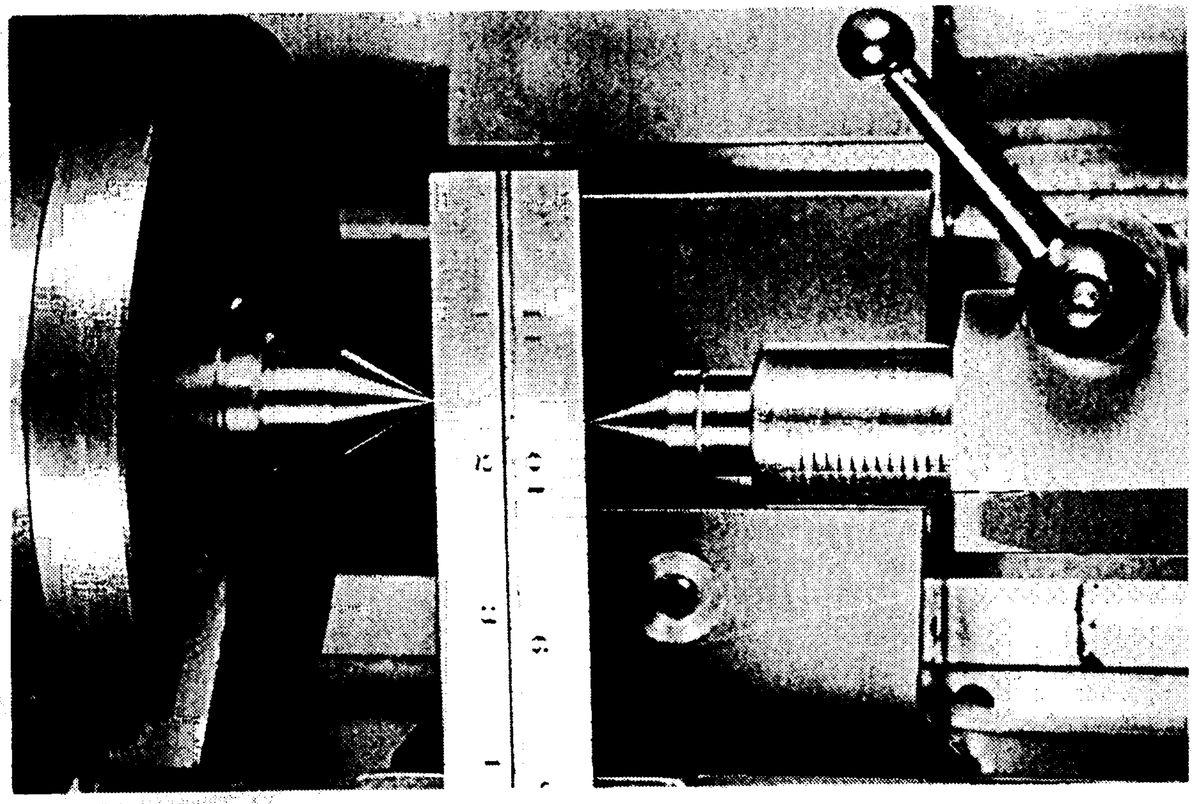


Fig. 112. Measuring the Amount of Offset

Adjusting the Tailstock Centre

When the tailstock must be set over for taper turning, loosen the tailstock locking lever and undo set screw "A" Fig. 111, the distance required. Screw in the set screw "B" for a similar distance until it is tight. Finally, clamp the tailstock to the lathe bed.

Measuring the Setover

In order to measure the setover of the tailstock centre, take a scale with graduations on both sides and place it between the two centres, as shown in Fig. 112. This will give an approximate measurement.

Taper Turning with Taper Attachment

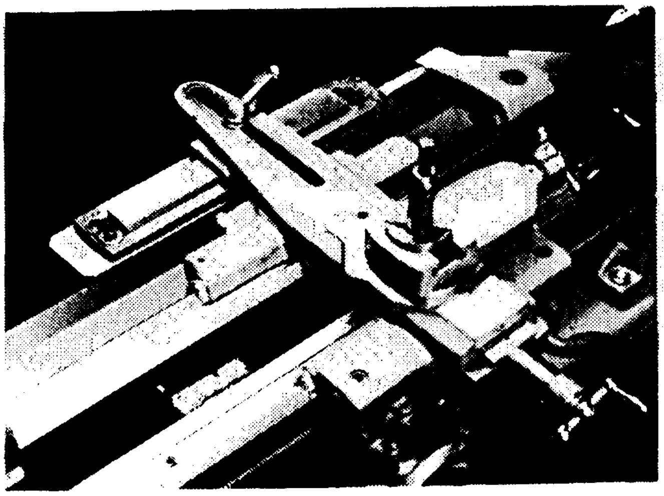
The taper attachment, which is used for turning and boring tapers in the lathe, removes the need to set over the tailstock. If desired, it may be set permanently at a standard taper. The taper attachment does not interfere with the use of the lathe for straight turning.

The taper attachment is especially valuable for boring tapered holes. Should the lathe not be equipped with a taper attachment, the compound rest top swivel can be set for the desired taper. The taper length is, however, limited to the comparatively short angular feed of the compound rest top when this method is employed.

Graduations on one end of the taper attachment swivel bar indicate the total taper in millimetres per centimetre (or inches per foot on Imperial machines), and on the other end the included angle of the taper is shown in degrees. See Fig. 114.

Plain Taper Attachment

Fig. 113 shows the plain taper attachment which is supplied for Boxford Lathes. It consists of a bracket attached to the rear of the lathe saddle, a compound slide with bracket and clamp for locking slide to lathe bed, and a cross-slide with slotted arm for adjusting its position when in use.



Using the Plain Taper Attachment

To use the plain taper attachment, disconnect the cross-feed leadscrew by removing the two screws which lock the leadscrew nut to the cross-slide of the lathe. The cross-slide is then free to move so that it can be controlled by the taper attachment. To engage the taper attachment the bracket must be clamped to the lathe bed, and the cross-slide clamped to the taper attachment's compound slide by means of the ball handle.

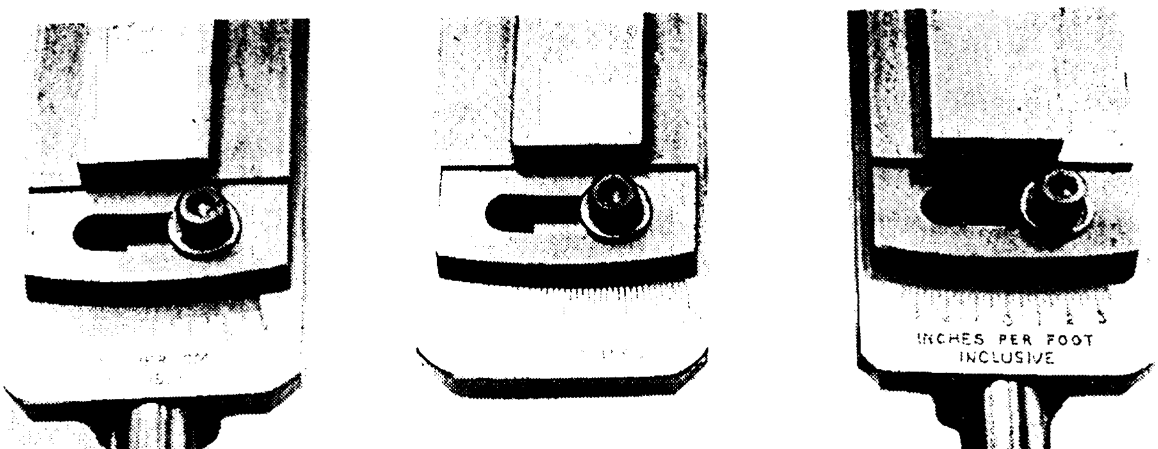


Fig. 114. Graduations on Taper Turning Swivel Slide

Setting the Taper Attachment Swivel Slide

Tapers can be specified in several ways: millimetres per centimetre (inches per foot on Imperial machines), in degrees or as a ratio. Lacking this information, it is necessary to calculate the taper to suit the scale engraved on the taper turning attachment.

To calculate the taper in millimetres per centimetre; the diameter in millimetres at the taper small end must be subtracted from the diameter in millimetres of the tapers large end, the result being divided by the length of the tapered portion in centimetres. This then gives the taper in millimetres per centimetre which can be set on the graduated scale at the end of the swivel slide (see fig. 114).

Another method for calculating metric tapers is to take the difference in the diameters of the taper in millimetres and divide this by the length of the taper also in millimetres; this will give a ratio (say 1:20 or $\frac{1}{20}$) which when multiplied by 100 gives a percentage which can also be read-off on the "MM PER CM" scale of the taper attachment since each graduation is in fact equal to a 1 percent inclusive taper (1 mm per cm equals 10% taper).

The method of calculating inches per foot on Imperial machines, is to take the difference in diameters of the taper in inches and divide this by the length of the taper portion in inches and the quotient multiplied by 12 gives the taper in inches per foot. A total taper of $\frac{1}{8}$"per foot is shown by each graduation on the "inches per foot" end of the swivel slide. If the calculation is expressed in decimal fractions instead of common fractions, reference to the table of decimal equivalents on page 95 will give the nearest fraction of an inch.

Be careful to remember that when setting the swivel slide for taper turning that the total (or inclusive) taper is indicated by the graduations. When the swivel slide is set at 5 degrees the machined taper will have a total included angle of 5 degrees, i.e. $2\frac{1}{2}$ degrees and not 5 degrees on either side of the centre line.

Once the taper attachment swivel slide has been set to the required angle, a trial cut must be made and the taper tested with a taper gauge or micrometers. It may be necessary to readjust the swivel slide because to make the graduations on the swivel slide coincide perfectly with the witness mark is a very difficult operation.

Standard tapered holes can be reamed by hand after boring to standardise the taper and size of the hole.

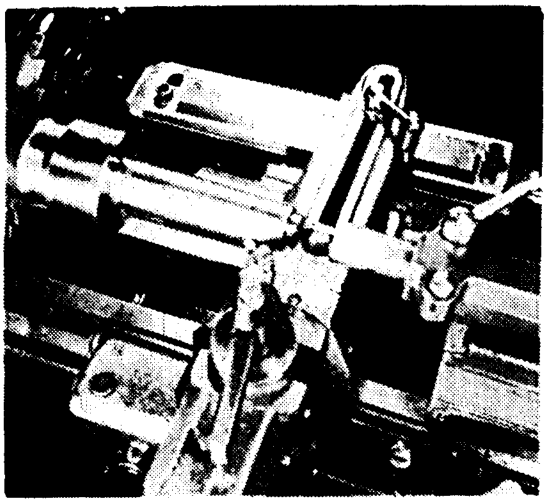


Fig. 115. Turning an Outside Taper with the Attachment

Morse Standard Tapers

Most manufacturers use Morse Standard Tapers in their lathes and drilling machine spindles. These tapers are fitted to both head and tailstock spindles of Boxford Lathes. Over is a list of the dimensions of various sizes of Morse Standard Tapers which have been accepted as part of the ISO recommendations.

Table 5. ISO Morse Taper Shanks

Dimensions in Millimetres

Number of Taper	Dia. of Plug at Small End	Dia. at End of Socket	Shank		Depth of Hole	Standard Plug Depth	Tongue		Keyway		End of Socket to Keyway	Percentage Taper	Taper per Inch (Inches)
			Whole Length	Depth			Thickness	Length	Width	Length			
	D	A	B	S	H	P	t	T	W	L	K		
0	6.4	9.045	59.5	56.5	52	50	3.9	6.5	4.2	15	49	5·2	·05205"
1	9.4	12.065	65.5	62	56	53.5	5.2	8.5	5.5	19	52	5	·04988"
2	14.6	17.780	80	75	67	64	6.3	10	6.6	22	62	5	·04995"
3	19.8	23.825	99	94	84	81	7.9	13	8.2	27	78	5	·0502"
4	25.9	31.267	124	117.5	107	102.5	11.9	16	12.2	32	98	5.2	·05194"
5	37.6	44.399	156	149.5	135	129.5	15.9	19	16.2	38	125	5.3	·05263"
6	53.9	63.348	218	210	188	182	19	27	19.3	47	177	5.2	·05214"

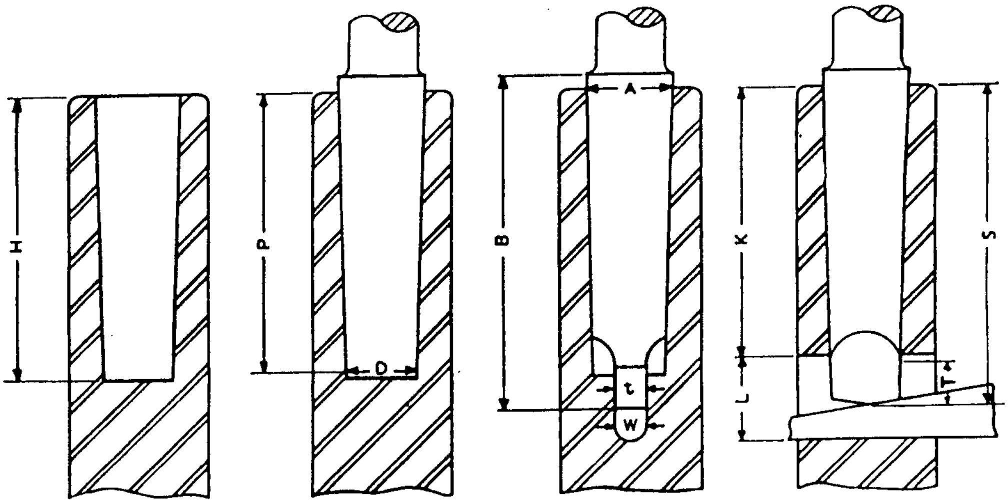


Fig. 116. Standard Morse Tapers (See Table Opposite)

Brown & Sharpe, Jarno and International Tapers

These are the names of three other systems of tapers which are in use. Brown & Sharpe and International Tapers are employed in milling machine spindles; Jarno and International Tapers are used for some makes of lathe spindles. Technical handbooks give the specifications of these three types of tapers, or are supplied by the manufacturers who use them.

Chuck Work

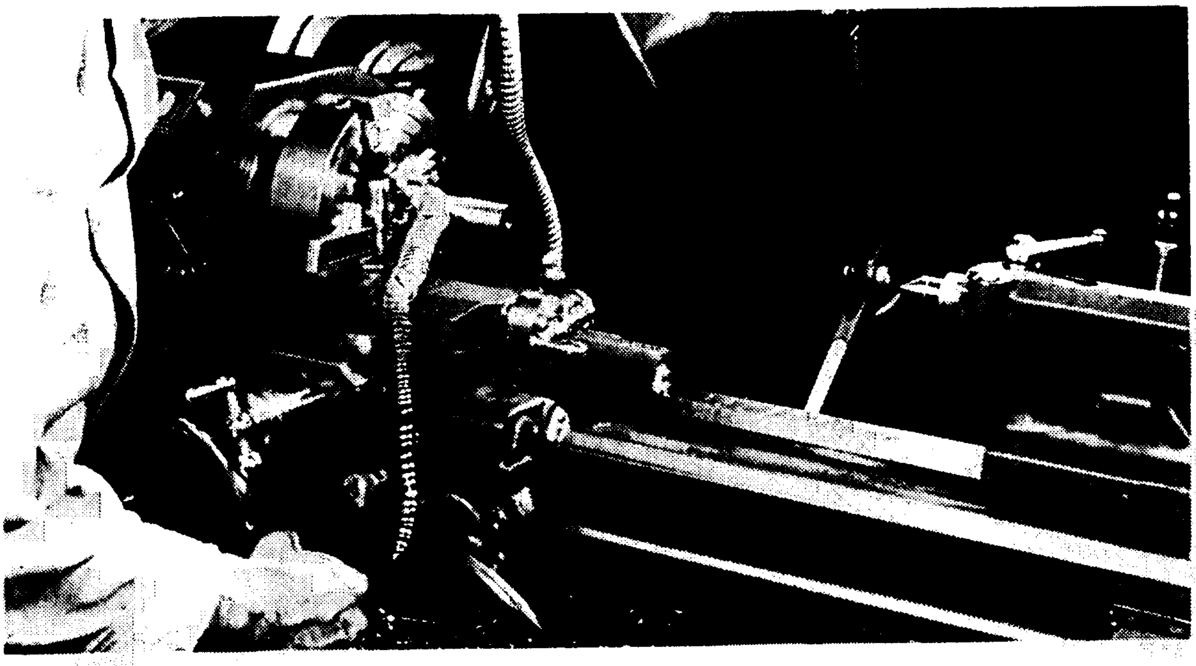


Fig. 117. Machining Work in an Independent Chuck

Chuck Work

There is some work which cannot be mounted between the lathe centres in order to machine it. The difficulty is overcome by using a chuck to hold such work. The above illustration shows the way in which this is done. Of the several types of chuck in use the most popular are the 4-jaw Independent chuck and the 3-jaw Self-centring chuck which are illustrated below.

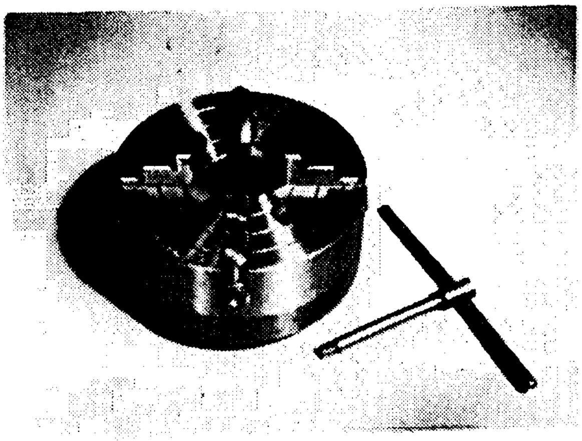


Fig. 118. 4-Jaw Independent Chuck

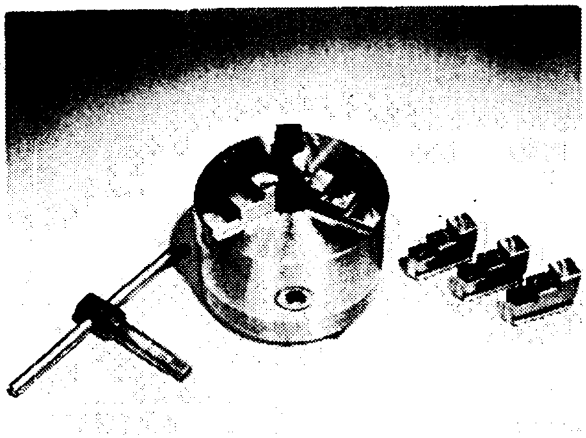


Fig. 119. 3-Jaw Self-Centring Chuck

Each of the four reversible jaws on the 4-jaw Independent chuck can be adjusted independently of the others. Where a lathe has only to have one chuck this type is recommended because it is capable of holding square, round or irregular shapes in either a concentric or an eccentric position.
The 3-jaw Self-Centring chuck will grip round or hexagonal work quickly,

since the three jaws move simultaneously and centre the work automatically. It is therefore widely used for this type of work. Two sets of jaws are needed, one set for external gripping and the other for internal gripping.
Soft jaws may be obtained for special classes of work.

Mounting Chuck on Spindle (Screwed Spindle Nose See Note p.54.)

Always clean and oil the threads of the lathe spindle and the chuck back plate very thoroughly before a chuck or a face plate is mounted on the lathe spindle. In addition, clean the shoulder ofthe spindle where the chuck back plate fits against it. If there is the slightest chip or burr at this point the chuck will be prevented from running true.
Never run the lathe with power while you are screwing the chuck on to the spindle, and do not spin the chuck up to the shoulder since that may make it very difficult to remove.

A chuck locking ring is available as an extra safety feature to prevent the chuck, faceplate, or driving plate unscrewing when extreme turning conditions are encountered in reverse.

The Independent Chuck

The reason for the widespread use of the independent chuck is its capacity to hold almost every shape and the fact that it can be adjusted to whatever degree of accuracy is required.
With concentric rings scribed on the chuck face, round work may be centred approximately as it is inserted in the chuck. When more accurate centring is necessary, start the lathe and hold a piece of chalk lightly against the revolving work. Then stop the lathe, loosen the jaw opposite the chalk mark slightly and tighten the opposite jaw. Repeat this test until the work has been centred with the required degree of accuracy. It is necessary to tighten all the four jaws securely before machining of the work is commenced.

Centring Work with Dial Test Indicator

When work with a smooth surface has to be centred accurately use a sensitive dial indicator. Almost any degree of accuracy can be obtained because the indicators dial is graduated to read off in 0.02 mm (or ·001").

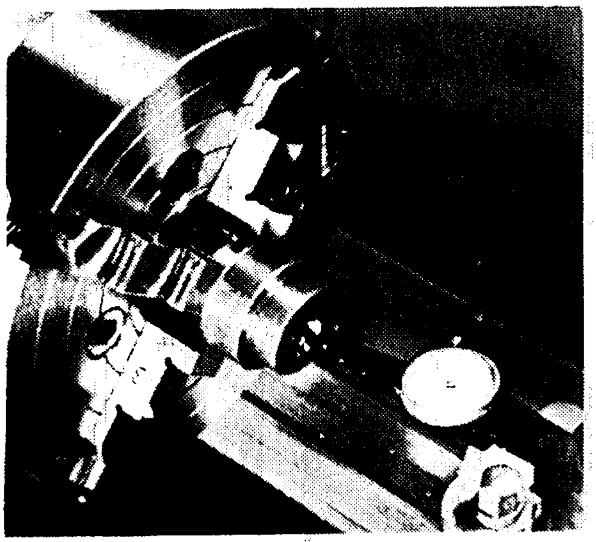


Fig. 120. Centring Work in Chuck with Dial Indicator

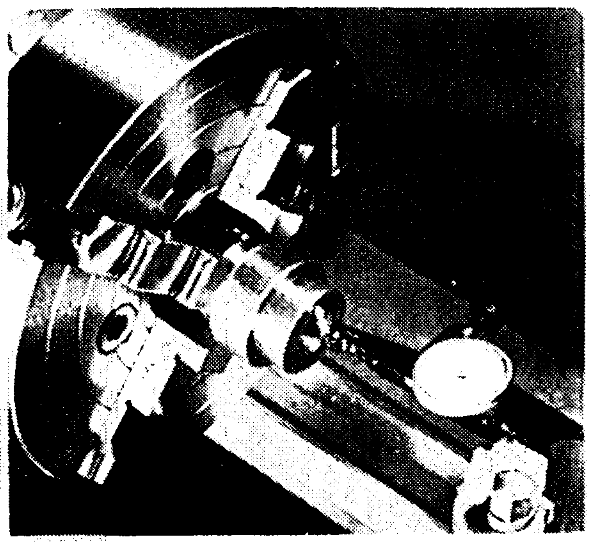


Fig. 121. Testing Face of Work for Wobble

Fig. 120 shows how to place the indicator in contact with the part to be centred. Revolve the lathe spindle slowly by hand and watch the indicator dial. Adjust the chuck jaws in the way described in the previous paragraph, until the degree of accuracy needed has been obtained.
Fig. 121 shows how to test the face of the work to be centred for wobble.

To Remove Chuck from Lathe Spindle
To loosen a chuck so that it can be removed from a bench lathe spindle, the back gear lever should be pulled forward on back geared machines, or the spindle locking pin fitted to lathes without back gearing. On Mark II underneath drive machines and TUD machines a spring-loaded spindle lock is provided at the front of the headstock. With the chuck key in position, a sharp blow with the hand to the chuck key should be sufficient to loosen the chuck. Excessive force should not be necessary and be avoided by making sure that the register and thread on the spindle and backplate are clean and oiled before fitting the chuck.
BEFORE FITTING ANY ACCESSORY TO SPINDLE NOSE ENSURE THREADS AND REGISTERS ARE CLEAN AND LIGHTLY OILED.

Size of Chucks
Use great care when selecting lathe chucks for the size of the lathe in use and the work for which the chucks are needed. Too small a chuck will restrict the lathe's capacity, whilst if it is too large the jaws may strike the lathe bed and do considerable damage.

The Self-Centring Chuck
Because all its three jaws move together and centre work automatically to within a few tenths of a millimetre, the self-centring chuck will quickly grip round or hexagonal work. As a general rule it will also centre work to within 0.075 mm (·003") in a new condition. Naturally the same degree of accuracy cannot be expected when the scroll has become worn.

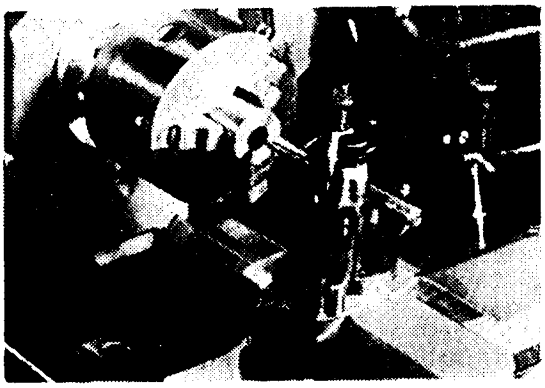


Fig. 123. Round Work Held in Self-Centring Chuck

An alternative type of 3-Jaw self-centring chuck whereby the 3 jaws can be centred very accurately is the 'Griptru', but for work which must be centred dead true the 4-Jaw independent chuck must always be used. When an independent or 'Griptru' type chuck is not available it is possible to place shims between the chuck jaws and the work. This will compensate for the lack of accuracy of the self-centring chuck.

Draw-In Collet Chuck
For Precision work such as making small tools and manufacturing small parts for watches, typewriters, wireless sets, etc., the draw-in collet chuck should be used. It is the most accurate of all the many types of chuck produced. Fig. 126 shows how collets are made in round, square and other shapes. Work which has to be held in a collet should not be more than 0.07 to 0.1 mm (·003" to ·004") smaller or larger than the nominal collet size. Heat-treated steel is the material from which collets are usually made.

Drill Chuck

There are several types of drill chuck on the market. They are used to hold drills, reamers, taps, etc. in both the headstock spindle and the tailstock spindle of the lathe. Some types, however, have neither the necessary accuracy nor holding power to prove satisfactory in use on the lathe. A good drill chuck may be defined as one that will hold drills concentric within 0.05 mm to 0.075 mm (·002"/·003"). It should also have a wrench or pinion key for tightening.

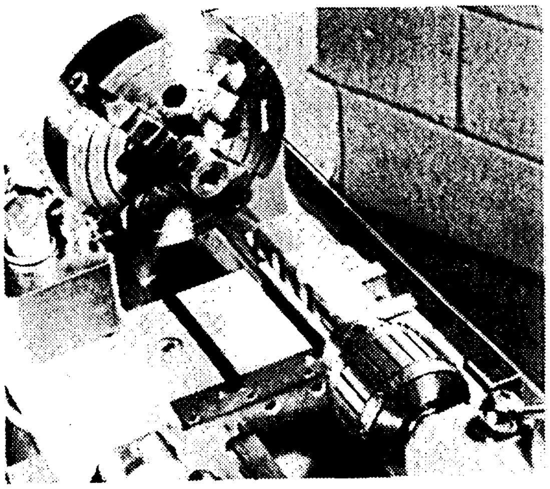


Fig. 124. Drill Chuck located in Tailstock

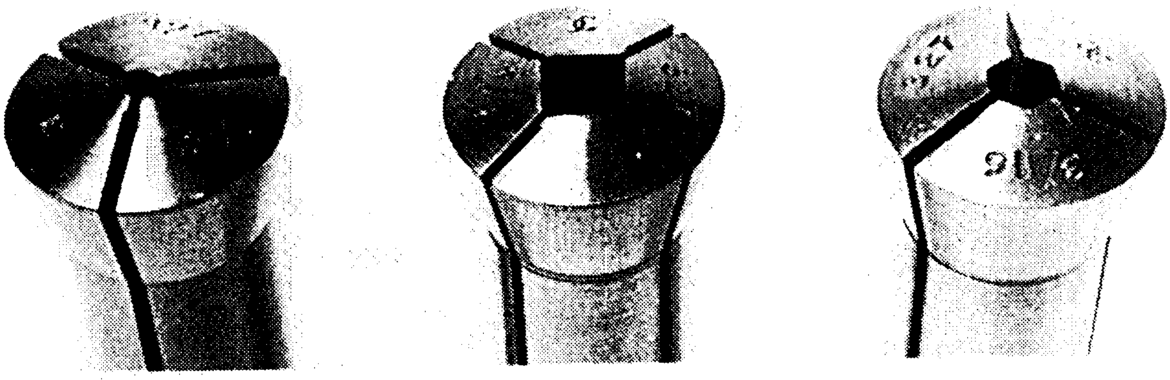


Fig. 126. Round Hexagon and Square Collets

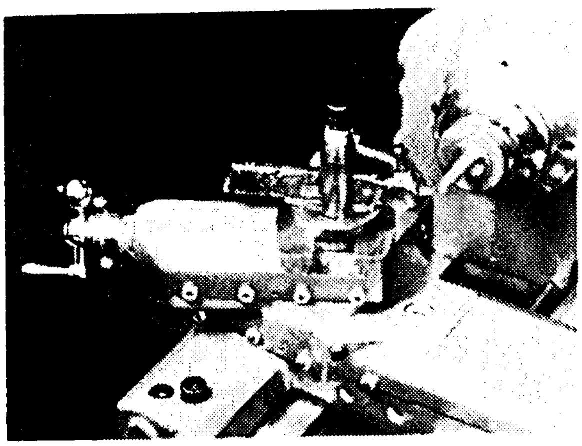


Fig. 125. Machine Work Held in a Draw-in Collet

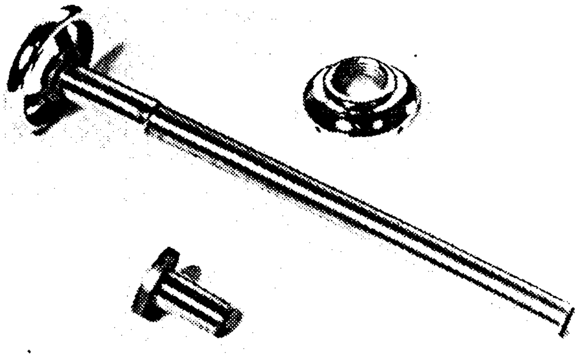


Fig. 127. Draw-Bar, Collet Adaptor and Nose Cap

Step Chuck and Closer
Fig. 128 illustrates the way in which a spring collet may be replaced by a step chuck and closer to hold discs such as gear blanks, etc. Step chucks for both internal and external gripping are illustrated, these being also known as ring chucks and disc chucks respectively.

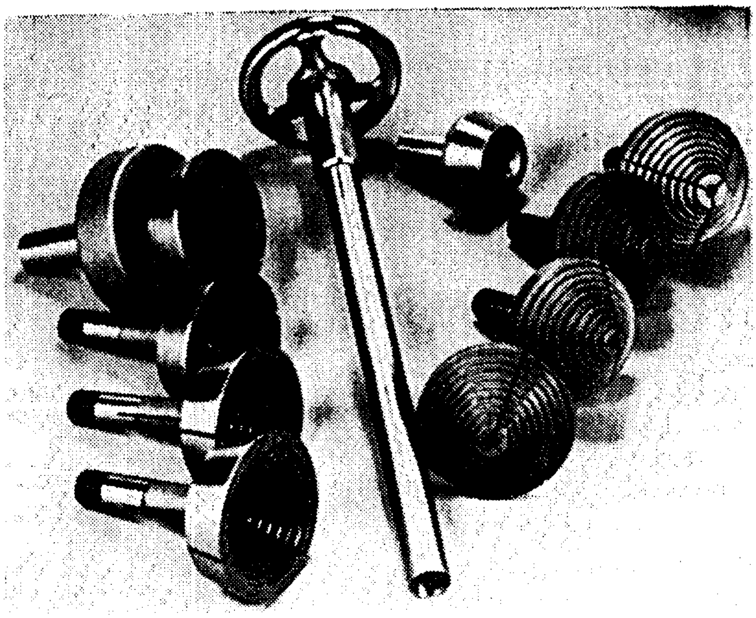


Fig. 128. Step Chucks and Closers with Draw Bar

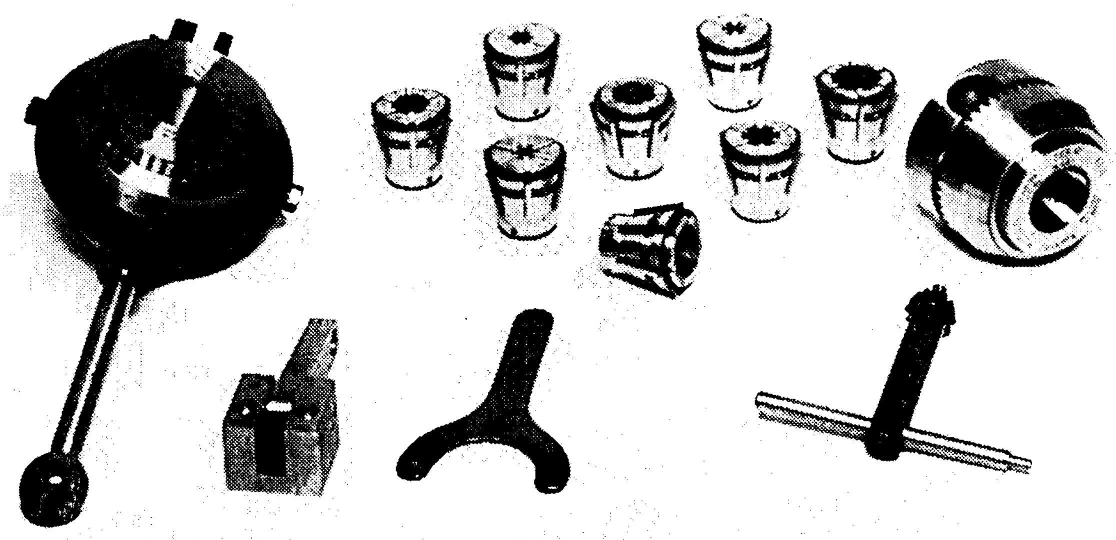


Fig. 129. Multisize Collet Chucks

Fig. 129 illustrates two types of multisize collet chuck. On the left is the lever operated dead length chuck (LC 10) with special mounting bracket and front cover, and on the right is the key operated chuck (KC 10). Only 8 collets are required to cover any size from 1.5 mm to 25 mm ($\frac{1}{16}$" to 1") round (5 of these will cover 3 mm to 19 mm—$\frac{1}{8}$" to $\frac{3}{4}$" A/F Hexagon).

NOTE
Direct Mounting Chucks
Ensure mounting face on chucks and spindle are clean and securely fixed.

Drilling, Reaming and Tapping

Chapter 8

Many drilling, reaming and tapping jobs are done more accurately and quickly in the lathe than by any other method.

A method of using a lathe for drilling operations is shown in Fig. 132. The work is supported by a drill pad placed in the tailstock spindle of the lathe. The tailstock handwheel is turned as the hole is drilled through the work. If it is thought desirable, the end of the work can rest on the lathe bed.

The hole's position should be centre punched to start the drill. High spindle speeds should be used when small diameter holes are being drilled.

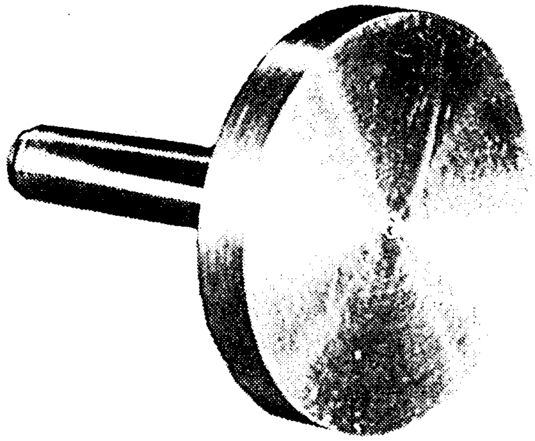


Fig. 133. Drill Pad for Use in Tailstock

Drill Pad for Tailstock

Fig. 133 illustrates a drill pad for the tailstock spindle of the lathe. This drill pad is substituted for the tailstock centre and supports the work being drilled.

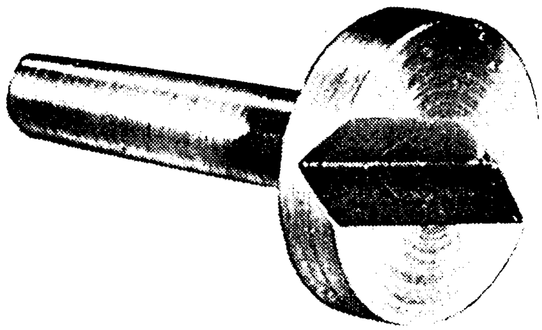


Fig. 134. Vee Centre Pad

Vee Centre Pad

The vee centre pad illustrated in Fig. 134 is similar to the drill pad except that it has a "V" to allow round work to be cross-drilled accurately. This is very useful when oil holes are being drilled in bushes, pin holes in shafts and similar work.

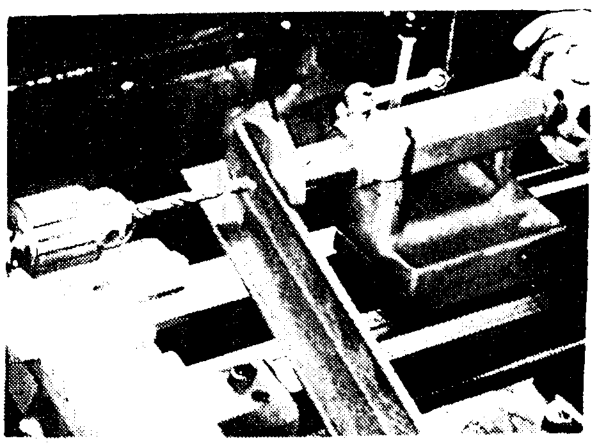


Fig. 132. Drilling in the Lathe

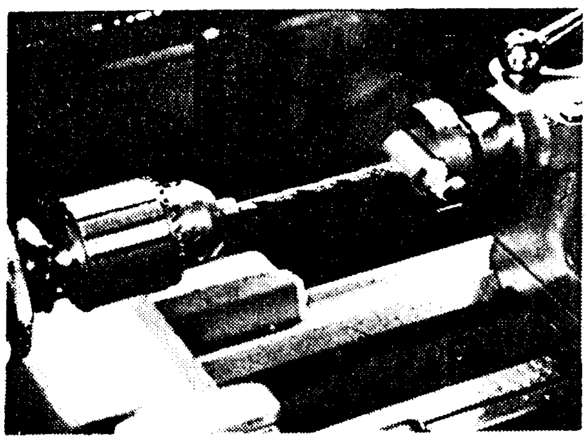


Fig. 135. Cross Drilling a Bush Located by Vee Centre Pad

Drilling Work Held in the Chuck

When the lathe is being used for drilling, most of the work is mounted in the lathe chuck, as shown in Fig. 136. Otherwise it is clamped to the face plate of the lathe. It is important to remember, when this method is in use, that the drill must be so started that it will run true, and that the hole will be drilled concentric with the outside diameter of the work.

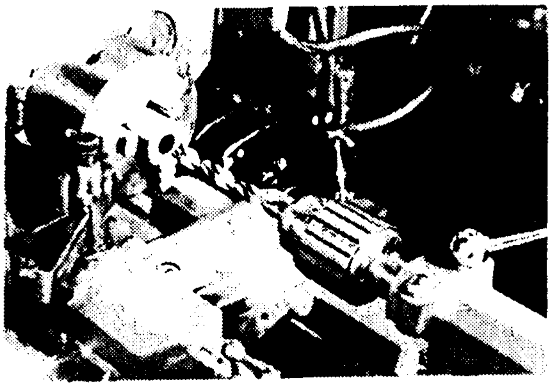


Fig. 136. Drilling Work in Chuck

One way of ensuring that the drill point is true is for the butt end of a lathe tool holder to be just touching the side of the drill. This prevents the drill from bending and makes it start approximately true in the centre of the work.

Centre Drilling

Should greater accuracy be needed a true starting point must be provided for the drill. To achieve this, the work must be centre drilled first of all by the use of a combination centre drill and countersink. See page 35.

Drilling in Steel

Always use plenty of oil on the drill point when drilling in steel. Any good cutting oil or even machine oil can be used.

Drilling a Cored Hole

Castings with cored holes are generally drilled with a four-lip drill. Fig. 137 shows how the hole in the casting should be chamfered to start the drill true. If this is not done the drill will follow the cored hole and may be thrown out of centre. It is advisable, for accurate drilling, to counterbore the hole for a short distance to give the drill point a perfectly concentric starting point.

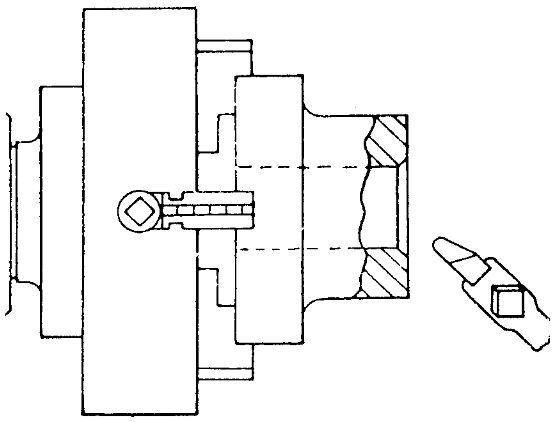


Fig. 137. Chamfering Cored Hole to start drill true

How to Sharpen Drills

To obtain accurate and efficient results in all drilling operations it is essential to have a correctly-ground drill point. To grind the point use a medium-grain grinding wheel that has been dressed true. Do not allow the grinding to overheat the drill point or the temper may be drawn.

Whenever possible drills should be re-ground in a drill-sharpening machine. If such a machine is not available the drills may be re-ground by hand.

Before a drill is ground by hand, the point of a new and unused drill should be studied and every effort made to duplicate it. This can be done by holding the drill at the correct angle with the grinding wheel and giving the drill point a wiping motion as it is ground, lowering the shank end of the drill and at the same time giving the drill itself a slight twist to the right. It is most important

to have both lips of the drill ground exactly alike.
Fig. 138 shows how the angle of the point should be from 120 to 130 degrees. The cutting lips, should be of exactly the same length and angle or the drill will cut oversize. 59 degrees is the best angle for general work.

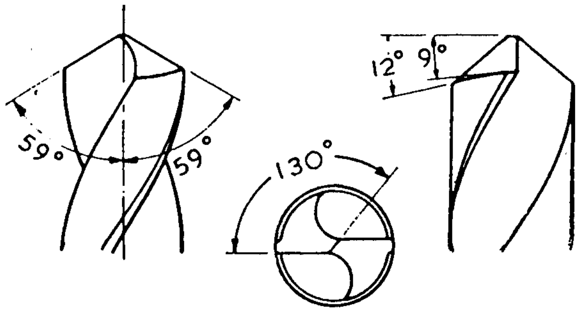


Fig. 138. Detail of Drill Point

Reaming in the Lathe
When a number of holes have to be finished quickly and accurately to the same diameter reamers are used in the lathe. Normally, the hole is first drilled or bored roughly to the required size, with enough stock allowed for reaming. The rose reamer and the fluted reamer are the two types in use.
Rose reamers are ground for cutting on the end only. They are designed for rough reaming because they do not produce a good finish nor an accurate diameter.

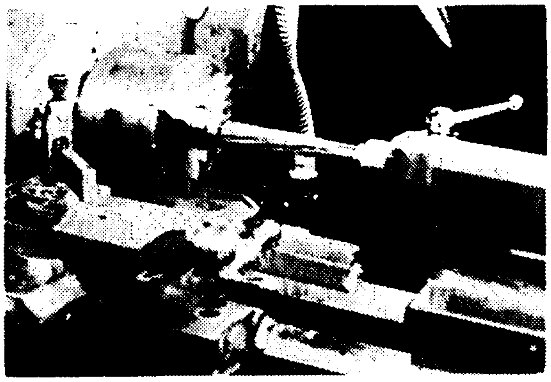


Fig. 139. Reaming in the Lathe

Fluted reamers are ground to cut on both sides and ends of the blades. They are usually used to produce an exact size and a good smooth surface after the rose reamer has been used for the initial reaming. Fluted reamers, only intended for light cuts, should not remove more than 0.25 mm (·010") from the hole.

Reamer in Drill Chuck
A drill chuck is usually used to hold straight shank reamers. Taper shank reamers can be inserted direct into the tailstock spindle. The reamer is fed carefully through the hole by turning the tailstock handwheel. A slow spindle speed must always be used and, when steel is being reamed, the reamer must, as previously mentioned for the drilling of steel, be kept flooded with oil.

Tapping Threads
Fig. 140 shows how a tap can be used so that threads can be tapped in the lathe. Operate the lathe spindle at slow speed and feed the tap to the work either by turning the tailstock handwheel or by sliding the entire tailstock on the lathe bed. A drill chuck can also be used to hold taps or a self-release tap holder made especially for these machines.

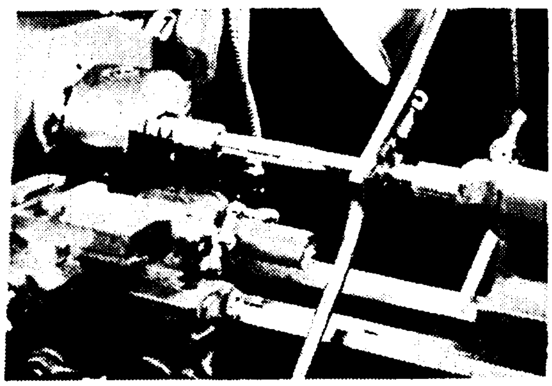


Fig. 140. Tapping in the Lathe

Screwcutting — Chapter 9

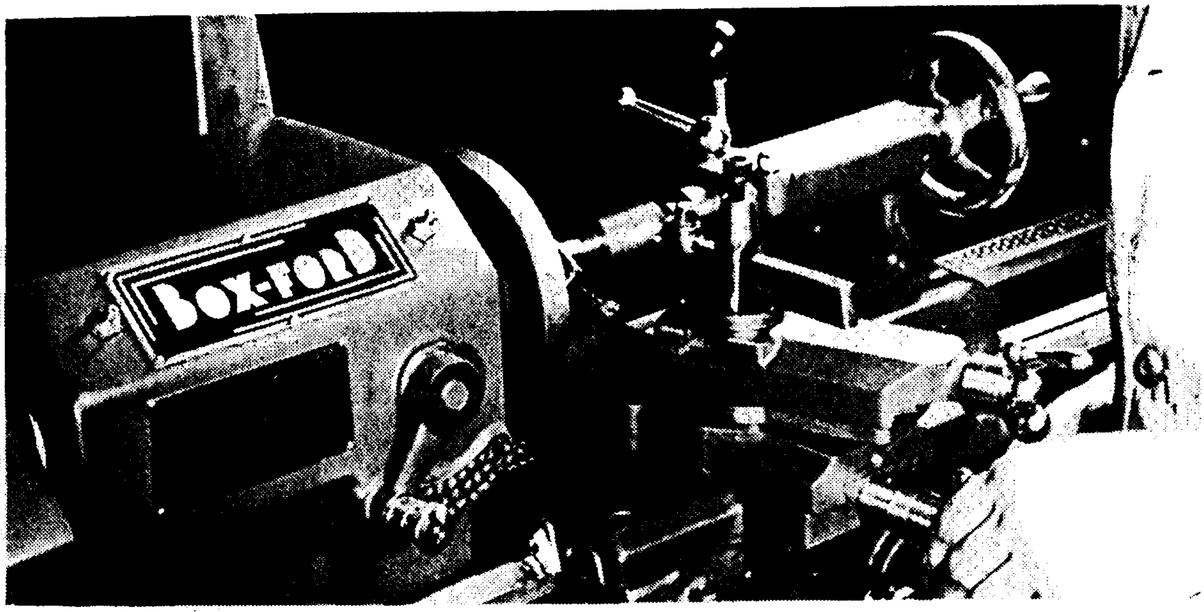


Fig. 141. Cutting a Screw Thread in the Lathe

To cut screw threads in the lathe the headstock spindle of the lathe is connected to the lead screw by a series of gears so that a positive carriage feed is obtained and the lead screw is driven at the required speed in relation to the headstock spindle.

It is possible to arrange the gearing between the headstock spindle and the lead screw so that any desired pitch of thread can be cut. If the lead screw has a 3 mm pitch thread, for example, and the gears are so arranged that the headstock spindle revolves three times whilst the leadscrew revolves once, the thread which is cut will be three times as fine as the thread on the lead screw, or 1.0 mm pitch.

The form of the thread to be cut, which can be ISO Metric, Whitworth (B.A.), American National, Acme, Square etc. will govern the shape to which the cutting tool must be ground.

If the reverse lever on the headstock is shifted it will reverse the direction of the lead screw's rotation and thus enable either right- or left-hand threads to be cut.

Fig. 142. Whitworth Form Thread

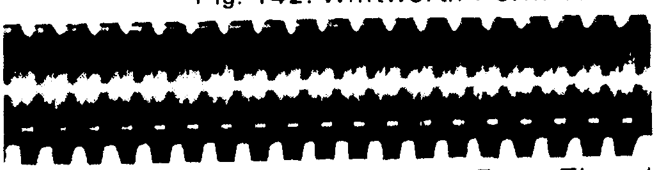


Fig. 143. Acme Form Thread

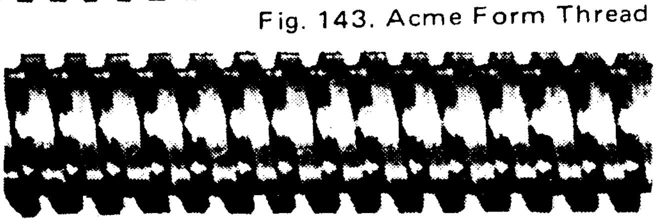


Fig. 144. 2-start Square Thread

ISO Metric Thread Form

ISO (International Organisation for Standardization) has adopted a thread form which is very similar to the Unified Thread form. The ISO Metric Thread will ultimately replace the various Whitworth and B.A. threads etc., but this will of course take several years. The British Standard Whitworth thread form is still however to be continued in the ISO recommendations for Pipe Threads.

P = PITCH in mm

H = 0.86603P

$\frac{H}{4}$ = 0.21651P

$\frac{H}{6}$ = 0.14434P $\frac{H}{8}$ = 0.10825P

R_R = 0.14434P

Depth of thread in screw = $\frac{17}{24}$ H = 0.61343P

Depth of thread in nut = $\frac{5}{8}$H = 0.54127P

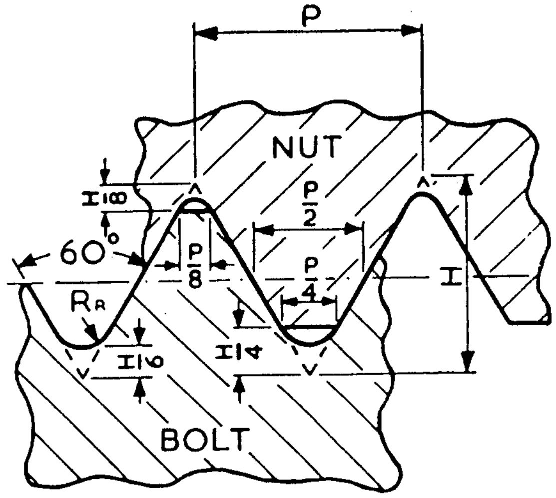


Fig. 146. ISO Metric Thread Form

Definitions

EFFECTIVE DIAMETER:—The effective or pitch diameter of a screw is the length of a line drawn through the axis at right angles to it, measured between the points where the line cuts the flanks of the thread.

MINOR, ROOT or CORE DIAMETER:—The minor or root diameter of a screw is the full diameter less twice the depth of thread and is measured at right angles to the axis.

FULL, OUTSIDE or MAJOR DIAMETER:—The full or Major diameter of a screw is the diameter by which most screws, bolts, etc., are recognised and is usually a nominal size such as M6, which indicates that the outside diameter of the thread is 6 mm diameter nominally, but in normal commercial practice may be a few hundredths of a millimetre smaller than 6 mm.

DEPTH OF THREAD:—The actual depth of thread is the length of a perpendicular line drawn from the centre of the crest or flat at the top of the actual thread to a base line touching the root of the actual thread.

PITCH:—The pitch of a screw thread is the distance from the centre of one thread to a centre of the next thread and is measured parallel to the axis.

LEAD:—The lead of a screw is the distance the screw advances along its axis in one complete turn. If the screw is a single start then the lead and the pitch are equal. With a two-start or double thread screw the lead is twice the pitch.

HELIX ANGLE:—The angle made by the helix of the thread at the pitch diameter with a plane perpendicular to the axis.

B.S. Whitworth Thread Form

P = PITCH = $\frac{1}{\text{No. of Threads per Inch}}$

D = DEPTH = ·6403 X P

R = RADIUS = ·1373 X P

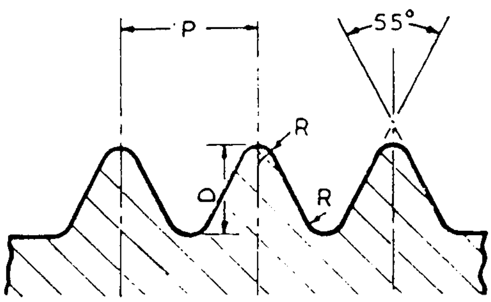


Fig. 145. B.S. Whitworth Thread Form

Table 6. ISO Metric Thread—Coarse Series

Diameter	2	2.5	3	4	5	6	8	10
Pitch	0.4	0.45	0.5	0.7	0.8	1.0	1.25	1.5
Basic effective diameter ...	1.740	2.208	2.675	3.545	4.480	5.350	7.188	9.026
Depth of thread in screw	0.25	0.28	0.31	0.43	0.49	0.61	0.77	0.92
Area of Root dia. (mm²) ...	1.79	2.98	4.47	7.75	12.7	17.9	32.8	52.3
Diameter of tapping drill	1.6	2.05	2.5	3.3	4.2	5.0	6.8	8.5

Diameter	12	16	20	24	30	36	42	48
Pitch	1.75	2.0	2.5	3.0	3.5	4.0	4.5	5.0
Basic effective diameter ...	10.863	14.701	18.376	22.051	27.727	33.402	39.077	44.752
Depth of thread in screw	1.07	1.23	1.53	1.84	2.15	2.45	2.76	3.07
Area of Root dia. (mm²) ...	76.2	144	225	324	519	759	1050	1380
Diameter of tapping drill	10.2	14.0	17.5	21.0	26.5	32.0	37.5	43.0

Table 6a. ISO Metric Thread—Fine Series

Diameter	8	10	12	14	16	18	20
Pitch	1.0	1.25	1.25	1.5	1.5	1.5	1.5
Basic effective dia. ...	7.350	9.188	11.188	13.026	15.026	17.026	19.026
Depth of thread in screw	0.61	0.77	0.77	0.92	0.92	0.92	0.92
Area of Root dia. (mm²) ...	36.0	56.3	86.0	116	157	205	259
Diameter of tapping drill	7.0	8.8	10.8	12.5	14.5	16.5	18.5

Diameter	22	24	30	36	42	48
Pitch	1.5	2.0	2.0	3.0	3.0	3.0
Basic effective diameter ...	21.026	22.701	28.701	34.051	40.051	46.051
Depth of thread in screw	0.92	1.23	1.23	1.84	1.84	1.84
Area of Root dia. (mm²) ...	319	365	586	820	1210	1540
Diameter of tapping drill	20.5	22.0	28.0	33.0	39.0	45.0

Table 7. British Standard Whitworth Thread

Diameter in Inches ...		1/8	3/16	1/4	5/16	3/8	7/16	1/2
Threads per inch ...		40	24	20	18	16	14	12
Pitch	mm in.	0.635 ·0250	1.058 ·04167	1.270 ·0500	1.411 ·05556	1.588 ·0625	1.814 ·07143	2.117 ·08333
Depth of Thread ...	mm in.	0.406 ·016	0.678 ·0267	0.813 ·0320	0.904 ·0356	1.016 ·040	1.161 ·0457	1.356 ·0534
Full Diameter ...	mm	3.175	4.763	6.350	7.938	9.525	11.113	12.700
Effective Diameter	mm in.	2.769 ·1090	4.084 ·1608	5.537 ·2180	7.033 ·2769	8.509 ·3350	9.952 ·3918	11.344 ·4466
Cross Sectional Area at bottom of thread	mm² in.²	4.387 ·0068	9.097 ·0141	17.548 ·0272	29.484 ·0457	44.064 ·0683	60.710 ·0941	78.322 ·1214
Diameter of Tap Drill	mm	2.5	3.7	5.1	6.5	8.0	9.3	10.5

Table 7. British Standard Whitworth Thread

Diameter in Inches ...		$\frac{9}{16}$	$\frac{5}{8}$	$\frac{3}{4}$	$\frac{7}{8}$	1	$1\frac{1}{8}$	$1\frac{1}{4}$
Threads per inch ...		12	11	10	9	8	7	7
Pitch	mm	2.117	2.309	2.540	2.822	3.175	3.629	3.629
	in.	·08333	·09091	·1000	·11111	·125	·14286	·14286
Depth of Thread ...	mm	1.356	1.478	1.626	1.806	2.032	2.324	2.324
	in.	·0534	·0582	·0640	·0711	·0800	·0915	·0915
Full Diameter ...	mm	14.288	15.875	19.050	22.225	25.400	28.575	31.750
Effective Diameter	mm	12.931	14.397	17.424	20.419	23.368	26.251	29.426
	in.	·5091	·5668	·6860	·8039	·9200	1·0335	1·1585
Cross Sectional Area at bottom of thread	mm^2	105.23	131.10	195.06	272.13	357.55	449.61	576.90
	$in.^2$	·1631	·2032	·3039	·4218	·5542	·6969	·8942
Diameter of Tap Drill	mm	12.0	13.5	16.25	19.25	22.0	24.5	28.0

Table 9. British Association Screw Thread

Number		0	1	2	3	4	5	6	7
Diameter	mm.	6.0	5.3	4.7	4.1	3.6	3.2	2.8	2.5
	In.	·2362	·2087	·1850	·1614	·1417	·1260	·1102	·0984
Pitch	mm.	1.0	.90	.81	.73	.66	.59	.53	.48
	In.	·0394	·0354	·0319	·0287	·0260	·0232	·0209	·0189
Threads per Inch ...		25·4	28·2	31·4	34·8	38·5	43·1	47·9	52·9
Depth of Thread ...	mm.	.6	.54	.485	.440	.395	.355	.320	.290
	In.	·0236	·0213	·0191	·0173	·0156	·0140	·0126	·0114
Effective Diameter	mm.	5.4	4.760	4.215	3.660	3.205	2.845	2.480	2.210
	In.	·2126	·1874	·1659	·1441	·1262	·1120	·0976	·0870
Minor Diameter ...	mm.	4.8	4.220	3.730	3.220	2.810	2.490	2.160	1.920
	In.	·1890	·1661	·1468	·1268	·1106	·0980	·0850	·0756
Do. B.S. Tolerance	mm.	.375	.340	.305	.275	.250	.220	.200	.180
for Nuts	In.	·0147	·0134	·0121	·0108	·0099	·0087	·0079	·0071
Radius	In.	·0071	·0064	·0058	·0052	·0047	·0042	·0038	·0034
Diameter of Tap Drill: Steel ...	mm.	5.1	4.5	4.0	3.4	3.0	2.65	2.3	2.05
Cast Iron, Brass or Ebonite	mm.	5.0	4.0	3.9	3.3	2.95	2.6	2.25	2.0

Number		8	9	10	11	12	13	14	15
Diameter	mm.	2.2	1.9	1.7	1.5	1.3	1.2	1.0	.90
	In.	·0866	·0748	·0669	·0591	·0512	·0472	·0394	·0354
Pitch	mm.	.43	.39	.35	.31	.28	.25	.23	.21
	In.	·0169	·0154	·0138	·0122	·0110	·0098	·0091	·0083
Threads per Inch ...		59·1	65·1	72·6	81·9	90·7	102	110	121
Depth of Thread ...	mm.	.260	.235	.210	.185	.17	.15	.14	.125
	In.	·0102	·0093	·0083	·0073	·0067	·0059	·0055	·0049
Effective Diameter	mm.	1.940	1.665	1.490	1.315	1.13	1.05	.86	.775
	In.	·0764	·0656	·0587	·0518	·0445	·0413	·0339	·0305
Minor Diameter ...	mm.	1.680	1.430	1.280	1.13	.96	.90	.72	.65
	In.	·0661	·0563	·0504	·0445	·0378	·0354	·0283	·0256
Do. B.S. Tolerance	mm.	.160	.145	.130	.115	.105	.095	.085	.080
for Nuts	In.	·0063	·0057	·0051	·0045	·0041	·0038	·0034	·0031
Radius	In.	·0031	·0028	·0025	·0022	·0020	·0018	·0016	·0015
Diameter of Tap Drill: Steel ...	mm.	1.8	1.55	1.4	1.2	1.05	0.98	0.8	0.7
Cast Iron, Brass or Ebonite	mm.	1.75	1.5	1.35	1.15	1.0	0.95	0.75	0.65

Table 8. British Standard Fine Thread

Diameter in Inches		3/16	7/32	1/4	9/32	5/16	3/8	7/16	1/2
Threads per Inch		32	28	26	26	22	20	18	16
Pitch	mm	0.794	0.907	0.977	0.977	1.154	1.270	1.411	1.588
	in.	·03125	·03571	·03846	·03846	·04545	·05000	·05556	·06250
Depth of Thread	mm	0.508	0.582	0.625	0.625	0.739	0.813	0.904	1.016
	in.	·0200	·0229	·0246	·0246	·0291	·03200	·0356	·0400
Full Diameter	mm	4.763	5.556	6.350	7.144	7.938	9.525	11.113	12.700
Effective Diameter	mm	4.255	4.976	5.725	6.518	7.198	8.712	10.208	11.684
	in.	·1675	·1959	·2254	·2566	·2834	·3430	·4019	·4600
Cross Sectional Area at bottom of thread	mm²	11.03	15.16	20.45	27.29	32.77	49.03	68.00	89.36
	in.²	·0171	·0235	·0317	·0423	·0508	·0760	·1054	·1385
Diameter of Tap Drill	mm	3.9	4.6	5.3	6.1	6.8	8.3	9.6	11

Diameter in Inches		9/16	5/8	11/16	3/4	13/16	7/8	1	1 1/8
Threads per Inch		16	14	14	12	12	11	10	9
Pitch	mm	1.583	1.814	1.814	2.117	2.117	2.309	2.540	2.822
	in.	·06250	·07143	·07143	·08333	·08333	·09091	·10000	·11111
Depth of Thread	mm	1.016	1.161	1.161	1.356	1.356	1.478	1.626	1.806
	in.	·0400	·0457	·0457	·0534	·0534	·0582	·0640	·0711
Full Diameter	mm	14.288	15.875	17.463	19.050	20.638	22.225	25.400	28.575
Effective Diameter	mm	13.272	14.714	16.302	17.694	19.281	20.747	23.774	26.769
	in.	·5225	·5793	·6418	·6966	·7591	·8168	·9360	1·0539
Cross Sectional Area at bottom of thread	mm²	117.94	114.26	180.09	209.61	252.32	291.61	385.29	489.42
	in.²	·1828	·2236	·2791	·3249	·3911	·4520	·5972	·7586
Diameter of Tap Drill	mm	12.6	14	15.5	16.75	18.5	19.75	22.75	25.5

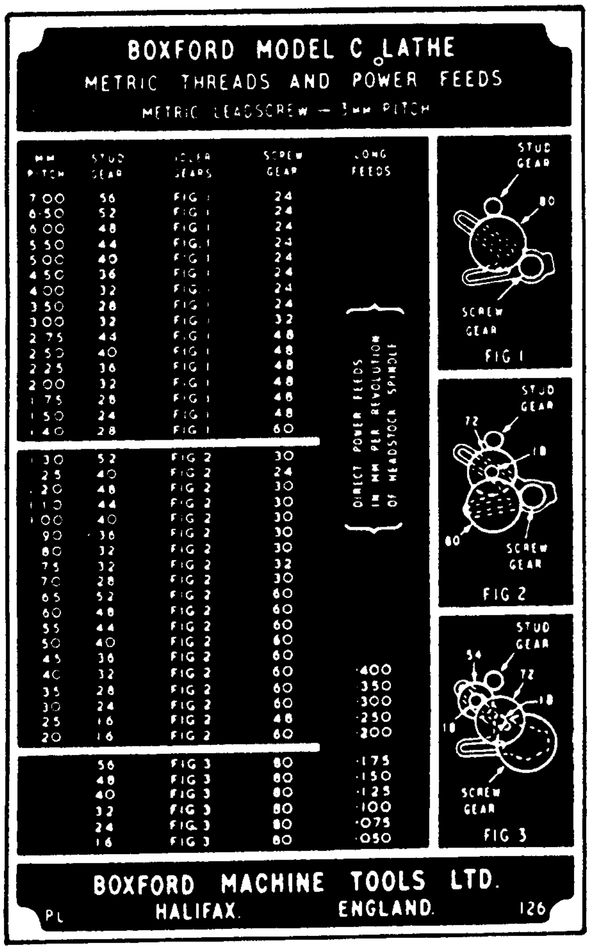
Cutting Threads on Standard Change Gear Lathes

To use the Standard Change Gear Lathe for cutting screw threads the apron half nuts are engaged with the lead screw. The number of teeth in the change gears used on the reverse stud and the lead screw, as well as on the compound gears in use, will determine the pitch of the thread to be cut.

When setting up the lathe to cut screw threads the pitch in mm, or the number of threads per inch to be cut must first be determined. Reference to the gear chart attached to the lathe (Fig. 148) will supply the necessary information. The first column headed 'MM Pitch' on Metric machines or 'Threads per Inch' on English machines will contain the thread to be cut. Under the heading 'Stud Gear' in the second column is given the number of teeth in the change gear which should be placed on the reverse stud of the lathe (see Fig. 149). The third column headed 'Idler Gear' list the figure number on the diagram of the index chart showing the arrangement of idler gear and compound gears. In column four headed 'Screw Gear' is a list of the number of teeth in the gear to be placed on the lead screw.

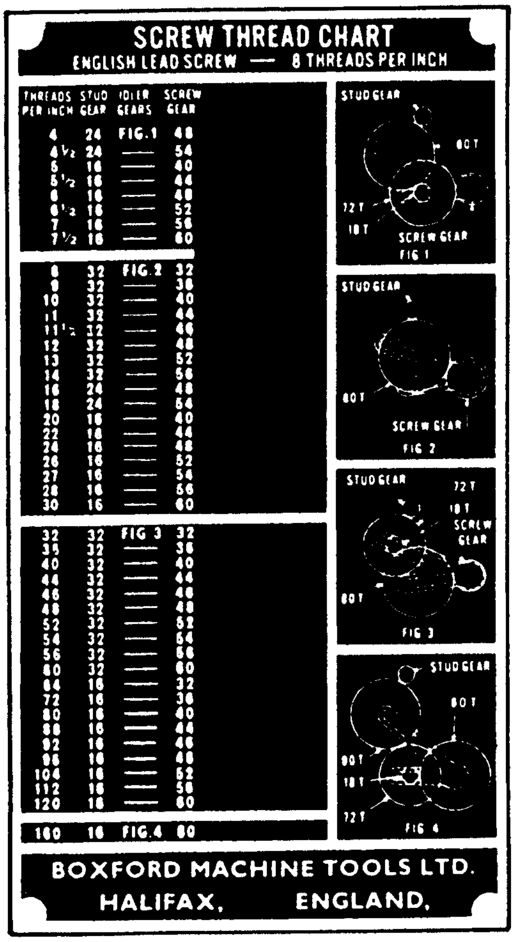
When the change gears required to cut the desired thread have been selected, they must be placed on the reverse stud and lead screw respectively and connected with the idler gear and compound gears, as shown on the change gear chart.

Metric Chart ("C" Model)



MM PITCH	STUD GEAR	IDLER GEARS	SCREW GEAR	LONG FEEDS
7·00	56	FIG 1	24	
6·50	52	FIG 1	24	
6·00	48	FIG 1	24	
5·50	44	FIG 1	24	
5·00	40	FIG 1	24	
4·50	36	FIG 1	24	
4·00	32	FIG 1	24	
3·50	28	FIG 1	24	
3·00	32	FIG 1	32	
2·75	44	FIG 1	48	
2·50	40	FIG 1	48	
2·25	36	FIG 1	48	
2·00	32	FIG 1	48	
1·75	28	FIG 1	48	
1·50	24	FIG 1	48	
1·40	28	FIG 1	60	
1·30	52	FIG 2	30	
1·25	40	FIG 2	24	
1·20	48	FIG 2	30	
1·10	44	FIG 2	30	
1·00	40	FIG 2	30	
·90	36	FIG 2	30	
·80	32	FIG 2	30	
·75	32	FIG 2	32	
·70	28	FIG 2	30	
·65	52	FIG 2	60	
·60	48	FIG 2	60	
·55	44	FIG 2	60	
·50	40	FIG 2	60	
·45	36	FIG 2	60	
·40	32	FIG 2	60	·400
·35	28	FIG 2	60	·350
·30	24	FIG 2	60	·300
·25	16	FIG 2	48	·250
·20	16	FIG 2	60	·200
	56	FIG 3	80	·175
	48	FIG 3	80	·150
	40	FIG 3	80	·125
	32	FIG 3	80	·100
	24	FIG 3	80	·075
	16	FIG 3	80	·050

English Chart ("C" Model)



THREADS PER INCH	STUD GEAR	IDLER GEARS	SCREW GEAR
4	24	FIG.1	48
4½	24	——	54
5	16	——	40
5½	16	——	44
6	16	——	48
6½	16	——	52
7	16	——	56
7½	16	——	60
8	32	FIG.2	32
9	32	——	36
10	32	——	40
11	32	——	44
11½	32	——	46
12	32	——	48
13	32	——	52
14	32	——	56
16	24	——	48
18	24	——	54
20	16	——	40
22	16	——	44
24	16	——	48
26	16	——	52
27	16	——	54
28	16	——	56
30	16	——	60
32	32	FIG 3	32
36	32	——	36
40	32	——	40
44	32	——	44
46	32	——	46
48	32	——	48
52	32	——	52
54	32	——	54
56	32	——	56
60	32	——	60
64	16	——	32
72	16	——	36
80	16	——	40
88	16	——	44
92	16	——	46
96	16	——	48
104	16	——	52
112	16	——	56
120	16	——	60
160	16	FIG.4	80

Fig. 148. Change Gear Charts for Standard Change Gear Lathes

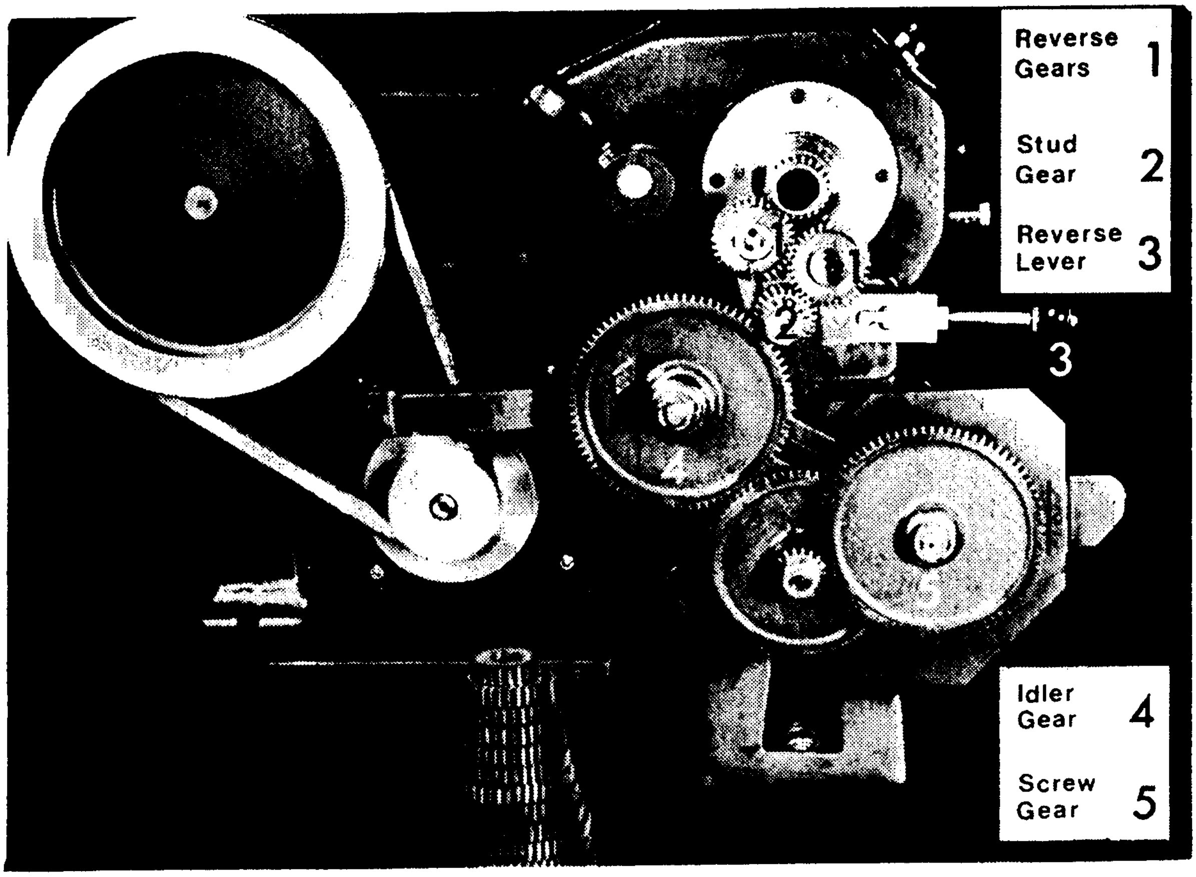


Fig. 149. Set up of Standard Change Gear Lathe for Cutting Screw Threads (Fig. 4 on gear chart)

Position of Spacing Collar

When simple gearing (Fig. 1 on metric chart) is used, the spacing collar on the lead screw must be placed outside the screw gear, as shown in Fig. 150. When compound gearing (Fig. 4 on gear chart) is used the collar must be inside the screw gear, as shown in Fig. 151.

Fig. 150. Position of Spacing Collar for Simple Gearing

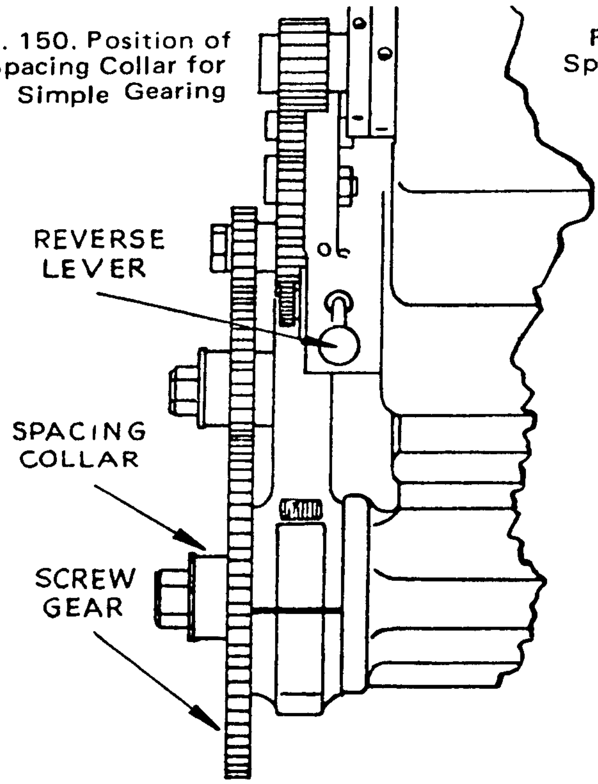
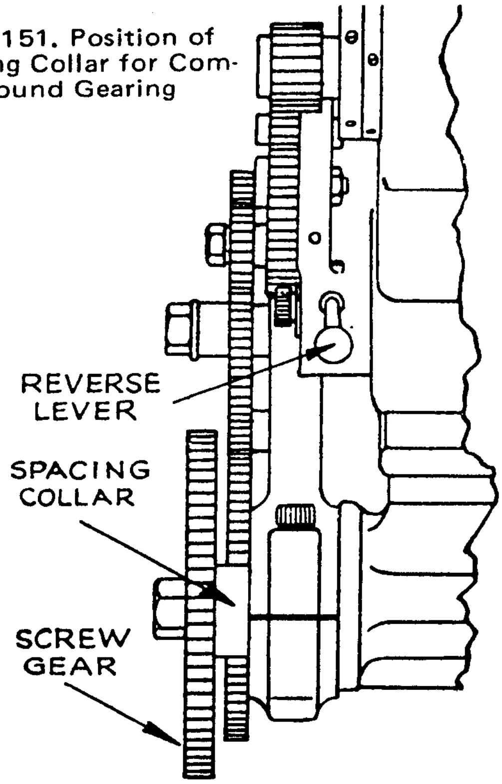


Fig. 151. Position of Spacing Collar for Compound Gearing



Cutting Screw Threads on Quick Change Lathes with Norton Type Gear Box

This which is illustrated in Fig. 152, permits various pitches of screw threads to be obtained without the use of loose change gears. The appropriate Metric or English chart is attached to the front face of the Gear Box. This chart reads directly the pitch in millimetres on Metric machines, or the threads per inch on English machines. To obtain various screw threads and feeds it is only necessary to arrange the levers of the gear box as indicated on the chart.

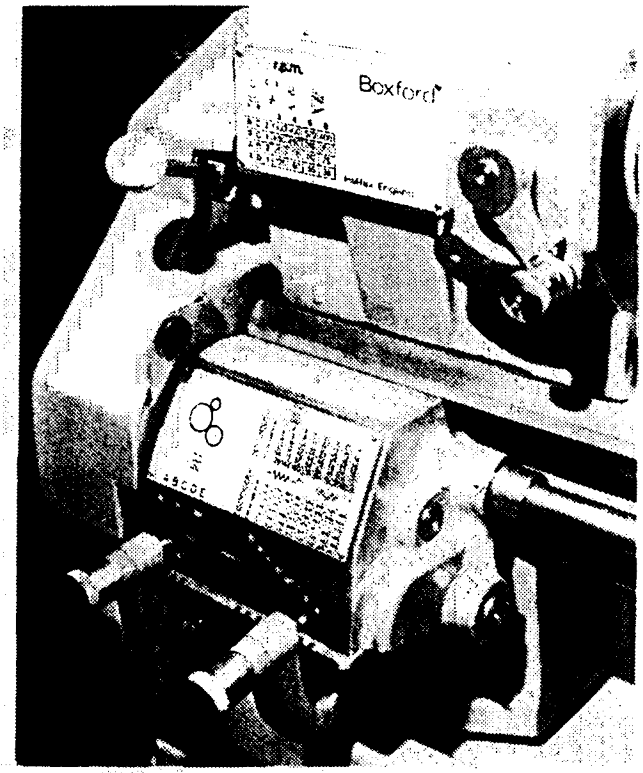


Fig. 152. Quick Change Lathe with Norton Type Gear Box

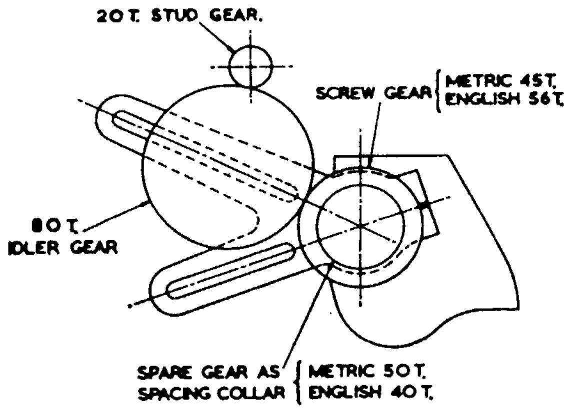


Fig. 153. Diagram of Standard Set Up on Model A Lathes

Bench Model Lathes with Gearboxes

Figs. 152, 154 and 155 illustrate the quick change mechanism used on bench model lathes. Changes of screw threads and power feeds are made by shifting the two levers on the front of the gear box, and by changing the stud gear.

On the English gear box any thread from 8 to 224 T.P.I. is readily obtainable merely by shifting the levers on the gear box. Coarse threads ranging from 4 to 7 T.P.I. can be secured by replacing the 20 T. Stud gear with the 40 T. stud gear. On metric gear box the coarser pitches are obtained by replacing the 20 T. stud gear with the 50 T. stud gear.

Tools for Cutting Screw Threads

The shape of the tool bit determines the shape or form of a screw thread cut on the lathe. It must be ground carefully and set properly if an accurate thread form is to be obtained. The most common thread forms are shown on pages 59, 73, and 74. A gauge must be used to grind the lathe tool to the required shape for any form of screw thread tool.

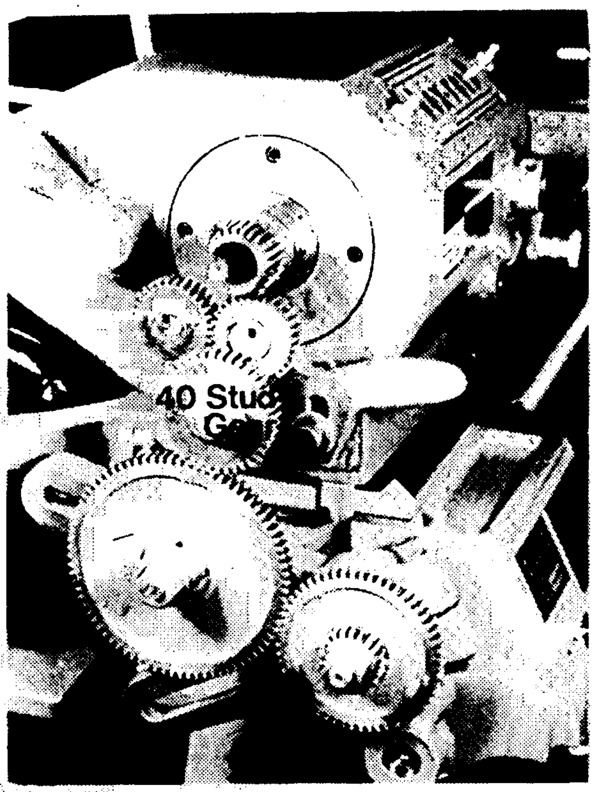


Fig. 154. Bench Model Lathe with Gearbox Set Up to Cut Coarse Pitches (e.g. 4 to 7 T.P.I.)

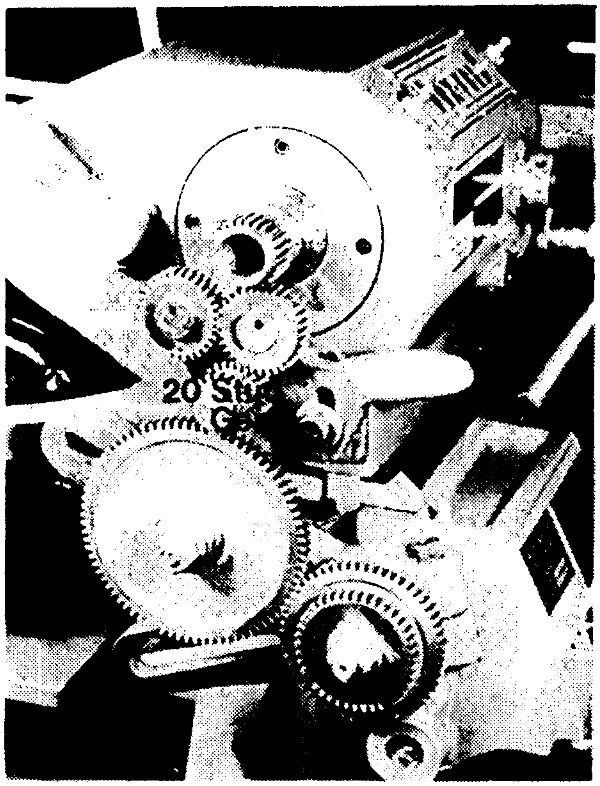


Fig. 155. Bench Model Lathe with Gearbox Set Up for Normal Feeds and Pitches (e.g. 8 to 224 T.P.I.)

Use of Thread Gauge

To cut ISO Metric threads in the lathe the point of the tool bit must be ground to an angle of 60 degrees (55° for Whitworth and $47\frac{1}{2}^\circ$ for B.A.). A thread tool gauge whose included angle is 60 degrees is used for grinding the tool to the exact angle required. It is usual to grind the top of the tool flat, without either side or back rake. Side rake must, however, be used on some occasions when threads are being cut in steel.

Fig. 156. Threading Tool Gauge

Front Clearance

Sufficient front clearance must be allowed on the tool bit to allow it to cut freely. As a rule, the front clearance is enough to prevent the tool from dragging in the helix angle of the thread. Except for very coarse pitches, therefore, the helix angle can be ignored.

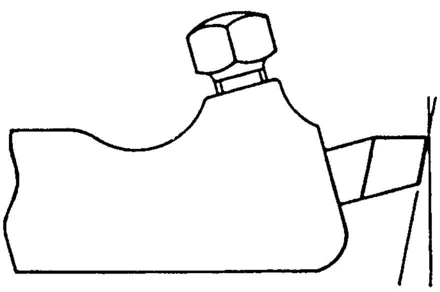


Fig. 157. Side View of Tool Bit showing Front Clearance

Setting Tool Bit

For cutting external screw threads the top of the threading tool must be placed exactly on the centre, as illustrated in Fig. 158. Make sure that the top of the tool is ground flat and is in exact alignment with the lathe centre.

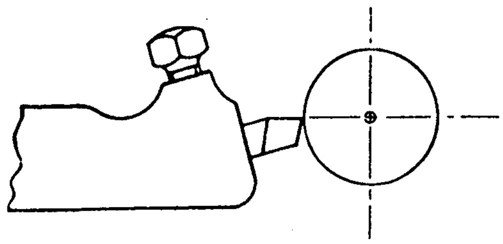


Fig. 158. Tool Bit Set on Centre for Cutting Screw Threads

Set the threading tool square with the work, as shown in Fig. 159. Use the tool gauge to adjust the point of the threading tool. If the tool is carefully set the result will be a perfect thread. Naturally, the angle of the thread will be incorrect if the threading tool is not set perfectly square with the work.

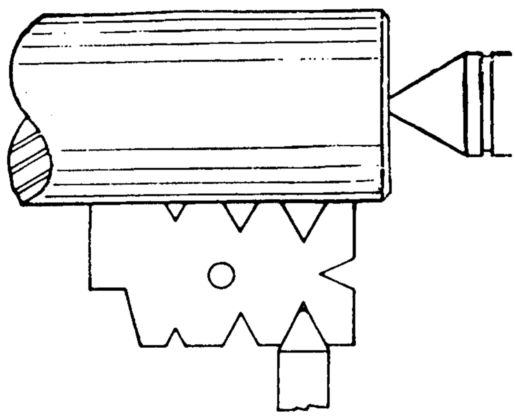


Fig. 159. Tool Bit Set Square for External Screw Thread

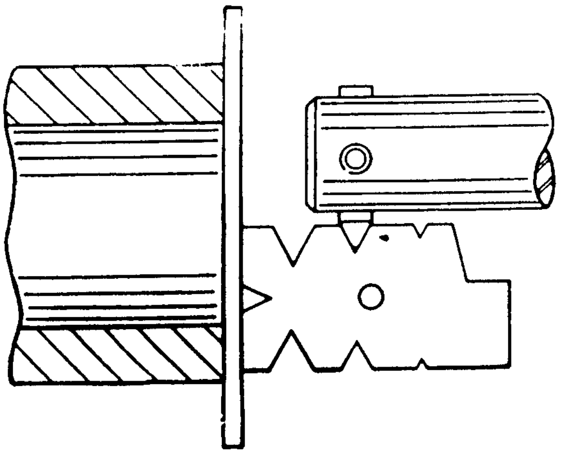


Fig. 161. Tool Bit Set Square for Internal Threading

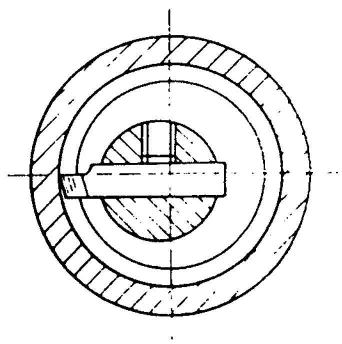


Fig. 160. Cutting Edge of Tool Bit Set on Centre for Internal Threading

Internal Threads

For cutting internal screw threads, also, the threading tool point is placed exactly on the centre. See Fig. 160. The tool point must be set perfectly square with the work. Fig. 161 shows how this can be done by fitting the point of the tool into the tool gauge.

To adjust the threading tool for cutting internal threads, always allow enough clearance between the tool and the inside diameter of the hole to permit backing out of the tool when the end of the cut has been reached. The boring bar, however, should be as large in diameter and as short as possible to prevent springing.

When cutting internal screw threads more front clearance is needed to prevent the heel of the tool from rubbing than when cutting external threads.

Engaging the Half-Nuts

The half-nuts may be engaged with the lead screw for cutting the screw thread only when the work has been mounted in the lathe and the cutting tool has been properly adjusted. Once the thread has been started, the half-nuts must not be disengaged from the lead screw unless a thread indicator dial is used. See page 72.

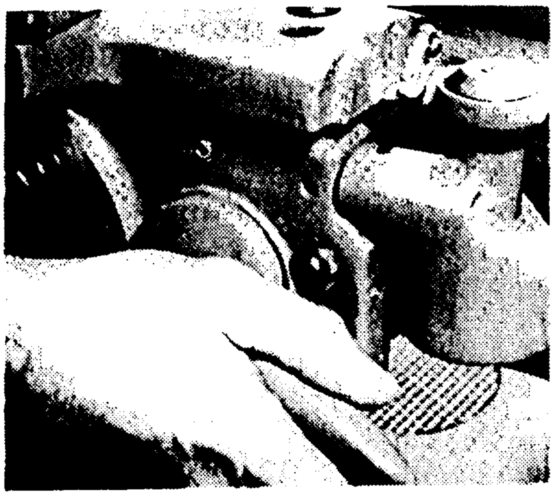


Fig. 162. Engaging the Half-Nuts with the Lead Screw

Position of Compound Rest for Cutting Screw Threads

It is usual, in those works which seek the maximum production, to place the compound rest of the lathe at an angle of 29 degrees for cutting 60 degree screw threads ($26\frac{1}{2}^\circ$ for 55° Whitworth etc).

Figs. 163 and 165 show how the compound rest is swung over to the right. The compound rest screw is used for adjusting the depth of cut. Most of the metal is removed by the left side of the threading tool. See Fig. 166. As a result, the chip curls more conveniently out of the way than it would if the tool were fed straight in.

The right side of the tool will shave the thread smooth and yield a good finish. It does not, however, remove enough metal to hamper the main chip taken by the left side of the tool.

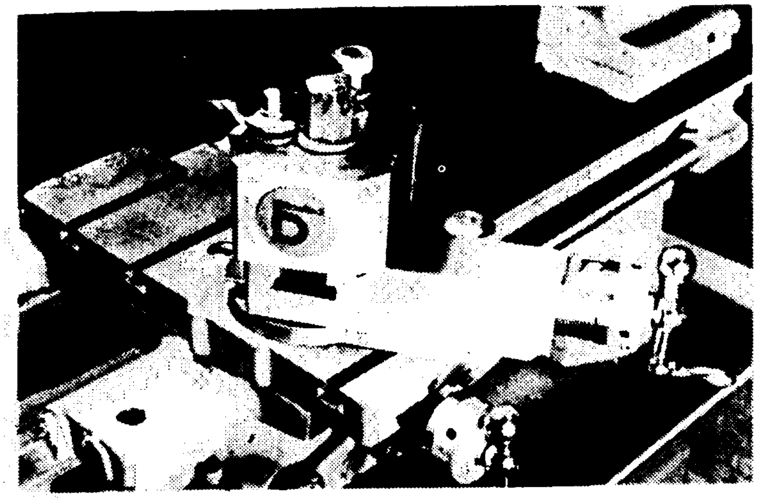


Fig. 163. Compound Rest Set at 29° for Cutting 60° ISO Metric Thread

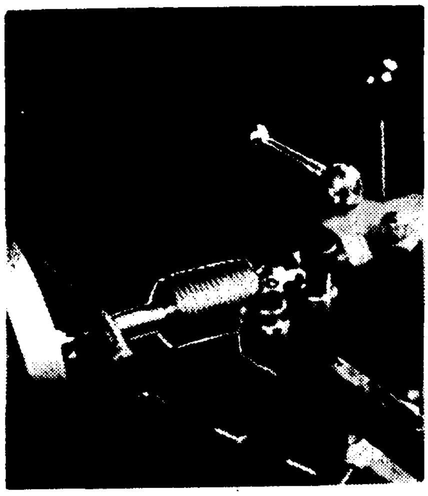


Fig. 164. Cutting a Whitworth Form Thread with Compound Rest Set at $26\frac{1}{2}^\circ$

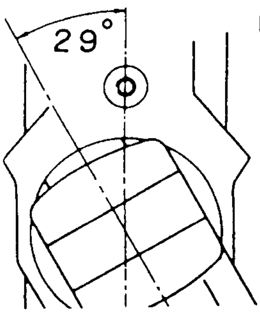


Fig. 165. Diagram showing Angle of Compound Rest for ISO Metric Threads

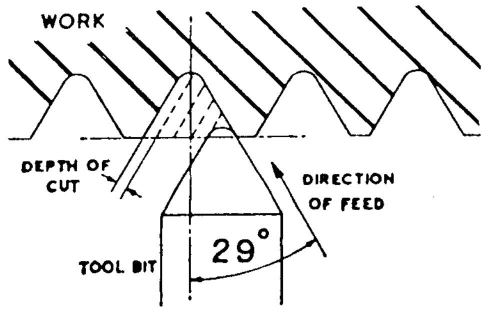


Fig. 166. Cutting Action of Tool Bit when Compound Rest is Set at Angle

Thread Cutting Stop

The play needed for the smooth operation of the change gears, lead screw, half-nuts, etc. causes a certain amount of lost motion. Therefore, the thread cutting tool must be withdrawn quickly at the end of each cut, prior to the lathe spindle being reversed to allow the return of the tool to the starting point. The tool point will dig into the thread and may break off if this precaution is not taken. To set the tool for each successive cut use the thread cutting stop.

First set the tool point so that it just touches the work. Next, lock the thread cutting stop to the saddle dovetail about a quarter of an inch from the cross slide. Then turn the thread cutting stop screw until the shoulder is tight against the stop. When ready to take the first cut, bring the tool rest back by turning the cross feed screw to the left for several turns and move the tool to the point where the thread is to start. Next, turn the cross feed screw to the right until the thread cutting stop screw strikes the thread cutting stop. The tool rest is now in the original position. If the compound rest feed screw is now turned in 0.05 mm to 0.07 mm (·002"/·003"), the tool will be in a position to take the first cut.

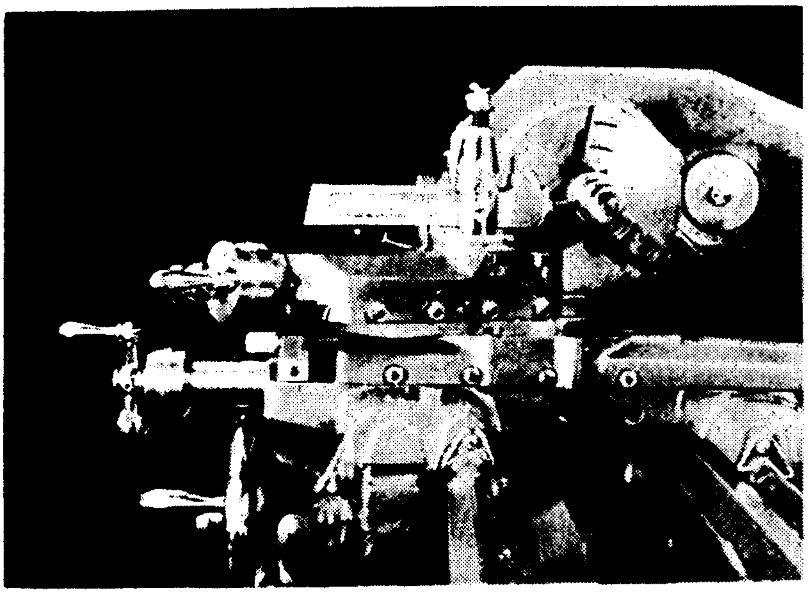


Fig. 167A. Thread Cutting Stop attached to Saddle Dovetail

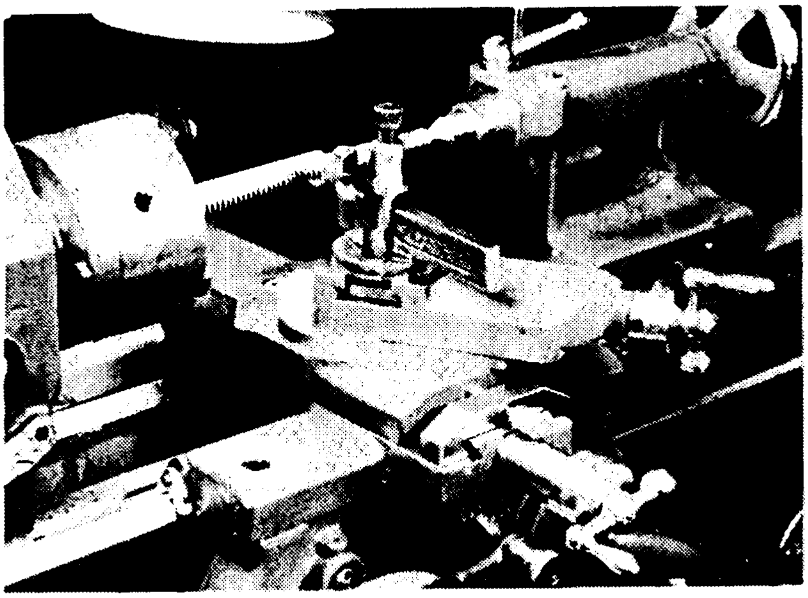


Fig. 167B. Thread Cutting Stop in Use

Micrometer Collar

Instead of using the thread cutting stop, the micrometer collar on the cross feed screw of the lathe can be used. In order to do so, the point of the threading tool, must be brought up so that it just touches the work. The micrometer collar should then be set at zero. On recent machines which have friction dials it is only necessary to rotate the micrometer collar by hand, but on previous machines the small grub screw must first be slackened, the micrometer turned to zero and the grub screw re-tightened. All adjustment to secure the required depth of cut must be done with the compound rest screw. The tool must be withdrawn at the end of each cut by giving the cross feed screw a complete turn to the left. The tool must then be returned to the starting point and the cross feed screw given a complete turn to the right, stopping at zero. The compound rest feed screw can then be adjusted for any depth of cut.

Taking the First Cut

When the lathe has been set up in accordance with the method described, a very light trial cut must be taken, just deep enough to leave a witness on the work's surface, as shown in Fig. 169. This trial cut is made to ensure that the lathe is arranged for cutting the desired pitch of the thread.

Measuring Screw Threads

The pitch in millimetres or the number of threads per inch can be checked by using a rule as shown in Fig. 170. Observe that the end of the rule rests either on the thread point or on one of the witness lines of the trial cut. The number of spaces between the end of the scale and the 20 mm mark on a Metric rule, or first inch mark on an imperial rule will give the number of millimetre pitches in 20 mm or the number of threads per inch respectively. Eight threads per inch are shown

Screw Pitch Gauge

Fig. 171 illustrates a screw thread gauge which is very useful for checking the finer pitches of screw threads. Such a gauge comprises several sheet metal plates which have been cut to the exact form of the various pitches of threads.

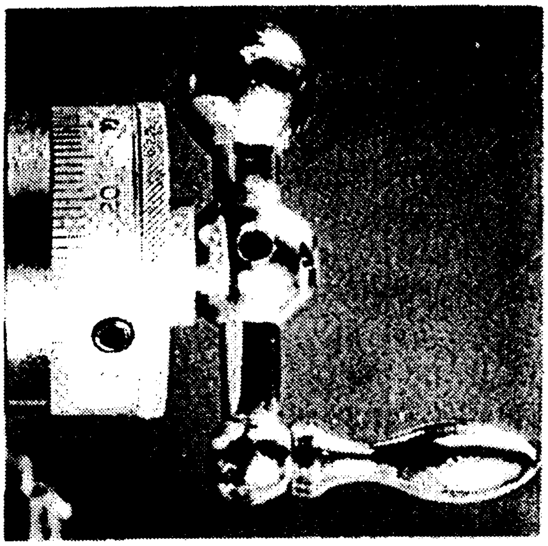


Fig. 168. Micrometer Collar

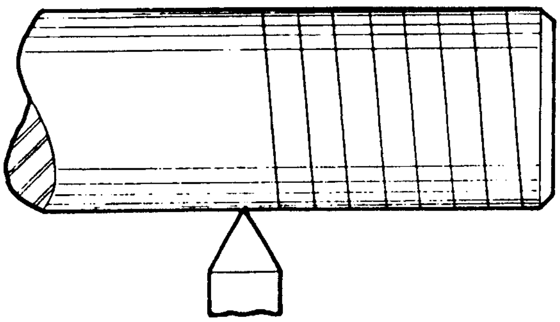


Fig. 169. Taking a Trial Cut

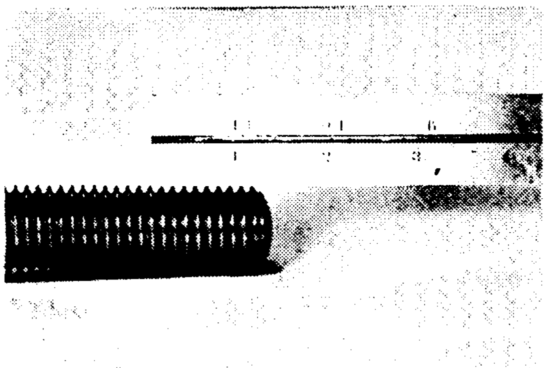


Fig. 170. Measuring the Number of Threads per Inch

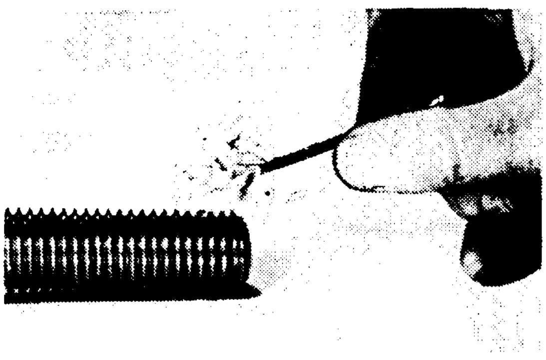


Fig. 171. Using a Screw Thread Gauge

Fitting and Testing Threads

Both the diameter and pitch of the thread can be finally checked with the nut to be used or by means of a thread ring gauge if one is available. There should be no shake or play on the nut. It should fit snugly without binding on the thread at any point.

The nut will fit perfectly if the angle of the thread is correct and the thread is cut to the proper depth. It is possible that the thread will appear to fit the nut even if the thread angle is wrong, but in reality it will only touch it at a few points. Consequently, the thread should also be checked by other methods instead of relying only on the nut or ring gauge as a check.

Resetting Tool after Cut has been Started

There may be occasions when the thread cutting tool has to be removed before the thread has been completed. If this happens, carefully readjust the tool so that it will follow its original groove when it is replaced in the lathe. Turn the lathe forward by hand to take up all the backlash before the tool is adjusted.

Set the compound rest top at an angle and adjust the cross feed screw and the compound rest feed screw simultaneously. By this means the tool point can be brought exactly into the original groove.

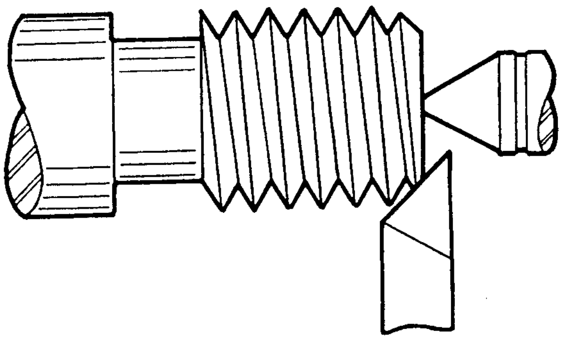


Fig. 172. 45° Chamfer at End of Thread

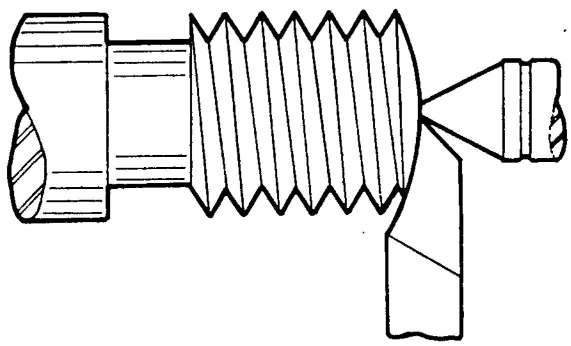


Fig. 173. Finishing End with Form Tool

Finishing the End of a Thread

There are several methods of finishing the end of the thread. Fig. 172 shows the 45 degree chamfer commonly used for bolts, cap screws, etc. For special screws and machine parts a forming tool is often used for doming the ends, as shown in Fig. 173.

It is difficult to stop the threading tool abruptly and as a consequence some provision is generally made for clearance at the end of a cut. Sometimes a hole is drilled in the shaft. More commonly, however, a recess or groove is cut around it. The groove is better since the lathe must be run very slowly if satisfactory results are to be secured with the drilled hole.

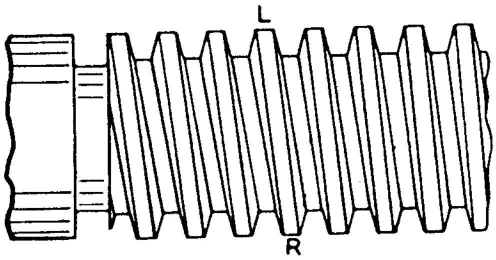


Fig. 174. Right-Hand Thread
Thread goes from Left to Right (L. to R. in illustration)

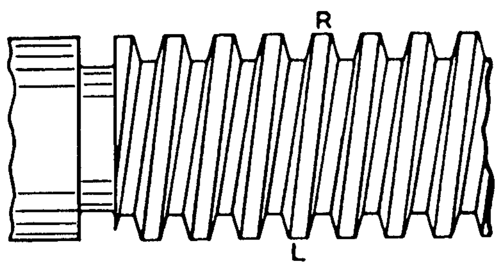


Fig. 175. Left-Hand Thread
Thread goes from Right to Left (R. to L. in illustration)

Thread Dial Indicator

A Thread Dial Indicator is used to save time, particularly when cutting long screw threads. When the lathe is set up for cutting screw threads, the thread dial indicates the relative positions of the lead screw, spindle and carriage of the lathe. This permits the half-nuts to be disengaged from the lead screw at the end of the cut, returning the carriage quickly to the starting point by hand without reversal of the lathe spindle, and re-engaging the half-nuts with the lead screw at a point to ensure that the tool follows exactly in the original path. A gear on the lower end of the thread dial shaft meshes with the lead screw and any movement of the carriage or lead screw is shown by a corresponding movement of the graduated dial at the top.

Thread Dial Indicators are available for both Metric and English machines but the metric one can only be used on a machine with a metric lead screw, and similarly the English one can only be used on a machine having an Imperial lead screw.

Fig. 176. Thread Dial Indicator

METRIC Thread Dial Indicator uses two gears in order to assist the cutting of 20 of the most popular ISO metric pitches. An adaptor block is provided to give the correct position for meshing either of the gears as indicated on the chart supplied. The block is secured in the apron and adjusted until the zero mark and the position "A" on the dial are in line when the half-nuts are engaged. The dial has nine marks which are lettered A, B, C, D or E the letter "A" being the one position common to all pitches. The first cut should be taken with the dial at the "A" position and then subsequent cuts can be taken at any of three positions as indicated on the chart.

ENGLISH Thread Dial Indicator uses a single gear and a dial with 8 equally spaced divisions. The half-nuts can be engaged as follows:

For all even-numbered threads at any line on the dial or each one-eighth revolution.

For all odd-numbered threads at any numbered line on the dial or each quarter revolution.

For all threads involving half threads per inch such as $11\frac{1}{2}$, at any odd numbered line or each half revolution.

For all threads involving one-fourth threads per inch (as $4\frac{3}{4}$), return to the original mark each time.

Use Oil When Cutting Threads in Steel

A good machine oil should be used when screw threads are being cut in steel so that a smooth thread may be produced. If oil is not used, the tearing of the steel by the cutting tool will result in a very rough finish.

Oil should be applied generously before each cut. It is best applied with a small paint brush when cutting external screw threads.

British Association Thread

P = PITCH (M.M.) = $(0.9)^n$ where n = the designating number of the screw.

D = DEPTH = 0.6 X P
R = RADIUS = $^2/11$ X P
} approx.

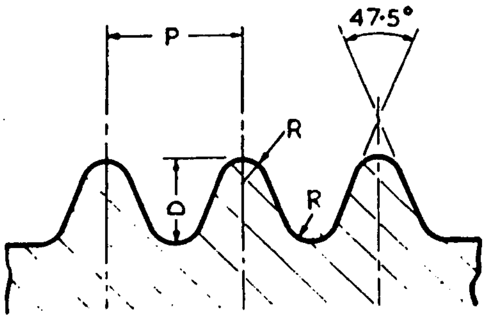


Fig. 178. British Association (B.A.) Screw Thread Form

Square Screw Threads

Square threads are used for vice screws, jack screws, etc. The sides of the tools for cutting square threads should be ground at an angle to conform to the helix angle of the thread, as shown in Fig. 179.

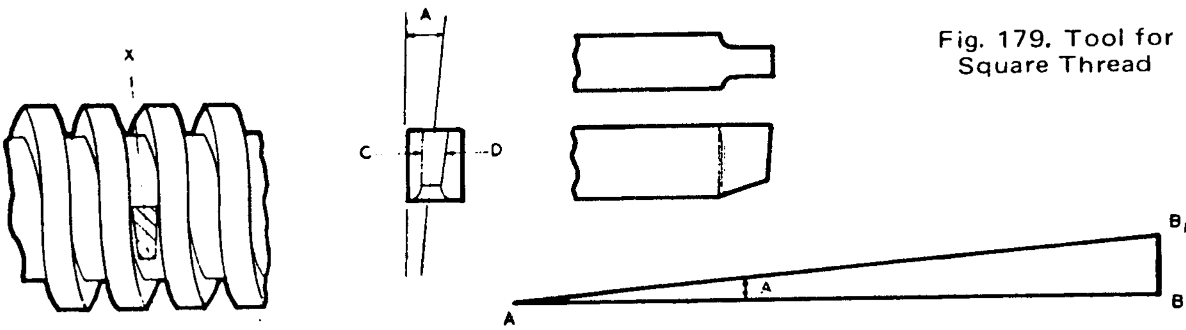


Fig. 179. Tool for Square Thread

To determine the helix angle of a screw thread, draw a line A-B equal to the circumference of the thread to be cut. Draw a line B-B1 equal to the lead of the thread and at right angles to the line A-B. Complete the triangle by drawing a line A-B1. Angle A in the triangle is the helix angle of the thread. The sides of the tool C and D should have a little clearance.

The width of the cutting edge of the tool for cutting square screw threads is exactly one-half the pitch, but the width of the tool for threading the nut should be from 0.02 to 0.07 mm (·001"/·003") larger, to allow a free fit on the screw.

P = PITCH in mm or $\frac{1}{\text{No. of Threads per Inch}}$

D = ·500 P

F = ·500 P

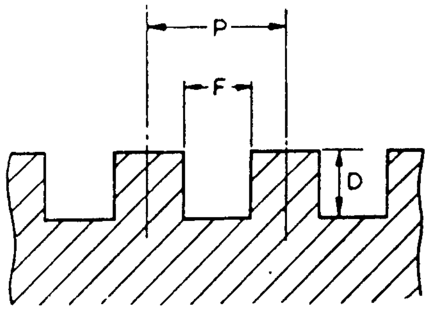


Fig. 180. Square Thread Form

Acme Screw Threads

P = PITCH in mm or $\frac{1}{\text{No. of Threads per Inch}}$

D = DEPTH + 0.25 mm (·010")

F = FLAT = .3707 P

C = FLAT = .3707 P – 0.13 mm (·0052")

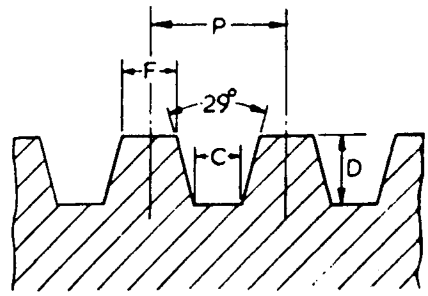


Fig. 181. Acme Thread Form

Acme screw threads are used for the feed screws and adjusting screws of machine tools and machinery of every kind. Acme threads are preferable to square threads because they are easier to cut.
Although the top and bottom of the threads are similar to a square thread in being flat, the sides of the thread have an included angle of 29 degrees, as shown in Fig. 181.
Fig. 182 shows the method of setting a threading tool for cutting an Acme thread.

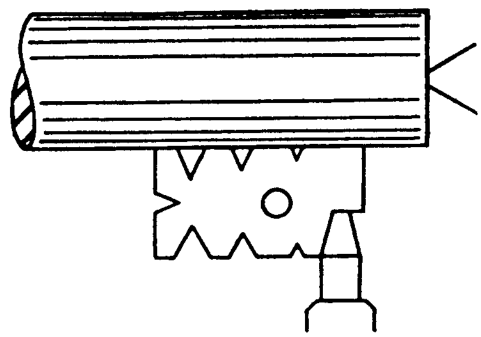


Fig. 182. Setting Tool for Cutting Acme Thread

29 degree Worm Thread

P = PITCH in mm or $\frac{1}{\text{No. of Threads per Inch}}$
D = DEPTH = ·6866 P
C = FLAT = ·31 P
F = FLAT = ·335 P
E = WIDTH = ·500 P
T = DEPTH TO PITCH LINE = ·3184 P

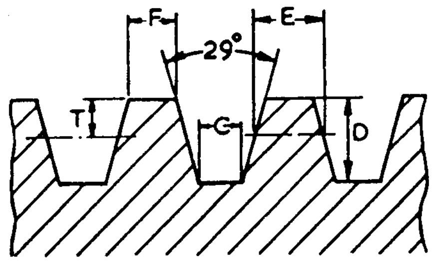


Fig. 183. Worm Thread Form

A 29 degree Worm Thread must not be confused with the Acme Standard Thread because it differs in the depth of the thread, the width of the top of the tooth and the width of the bottom of the tooth, as illustrated above.

American National Thread

P = Pitch = $\frac{1}{\text{No. of Threads per Inch}}$
D = Depth = 0·6495 X P
F = Flat = $\frac{P}{8}$

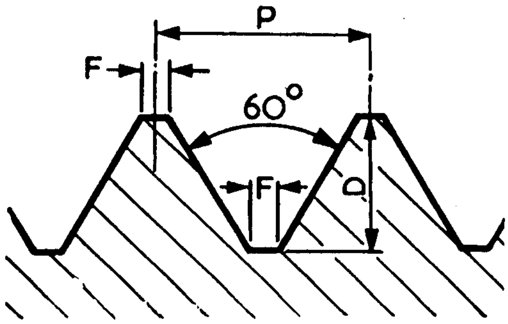


Fig. 184. American National Screw Thread Form

Metric and English Conversion Gears

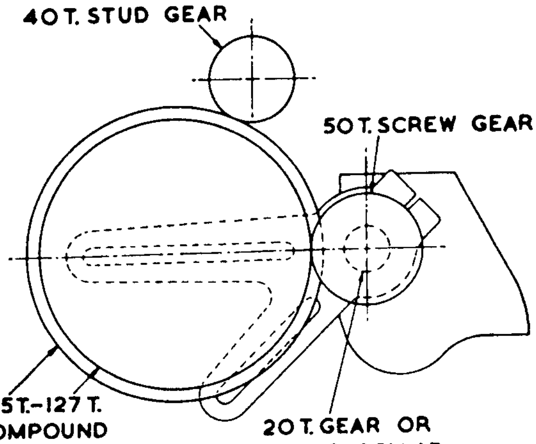


Fig. 186. Diagram showing standard Metric Change Gear Lathe Set Up for Cutting 10 T.P.I.

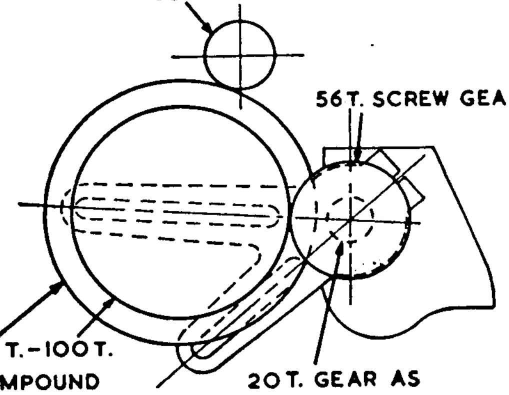
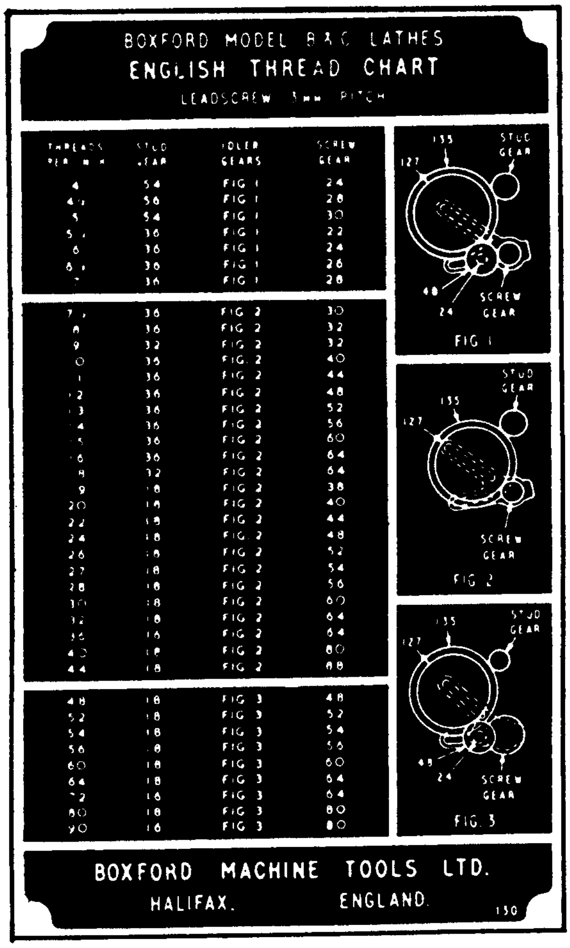


Fig. 185. Diagram showing standard English Change Gear Lathe Set Up for Cutting 4 mm Pitch Thread

Transposing gears are required when both English and metric screw threads are to be cut on the same lathe.
On lathes which have metric lead screws English transposing gears are used for cutting English screw threads. Metric transposing gears are used on lathes having English lead screws when it is desired to cut metric screw threads.

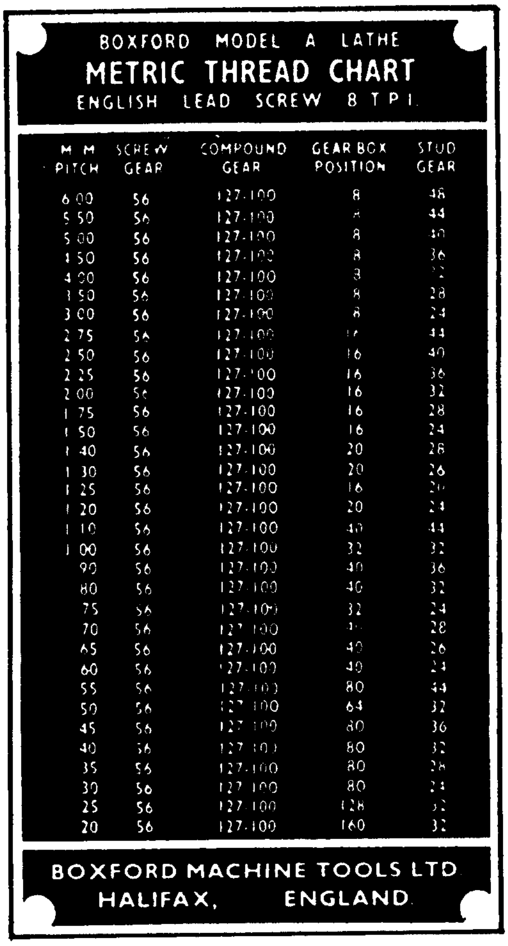


BOXFORD MODEL B & C LATHES
ENGLISH THREAD CHART
LEADSCREW 3mm PITCH

THREADS PER INCH	STUD GEAR	IDLER GEARS	SCREW GEAR
4	54	FIG 1	24
4½	56	FIG 1	28
5	54	FIG 1	30
5½	36	FIG 1	22
6	36	FIG 1	24
6½	36	FIG 1	26
7	36	FIG 1	28
7½	36	FIG 2	30
8	36	FIG 2	32
9	32	FIG 2	32
10	36	FIG 2	40
11	36	FIG 2	44
12	36	FIG 2	48
13	36	FIG 2	52
14	36	FIG 2	56
15	36	FIG 2	60
16	36	FIG 2	64
18	32	FIG 2	64
19	18	FIG 2	38
20	18	FIG 2	40
22	18	FIG 2	44
24	18	FIG 2	48
26	18	FIG 2	52
27	18	FIG 2	54
28	18	FIG 2	56
30	18	FIG 2	60
32	18	FIG 2	64
36	16	FIG 2	64
40	18	FIG 2	80
44	18	FIG 2	88
48	18	FIG 3	48
52	18	FIG 3	52
54	18	FIG 3	54
56	18	FIG 3	56
60	18	FIG 3	60
64	18	FIG 3	64
72	16	FIG 3	64
80	18	FIG 3	80
90	16	FIG 3	80

BOXFORD MACHINE TOOLS LTD.
HALIFAX. ENGLAND 130

Fig. 187. English Thread Chart for Standard Change Gear Lathes with Metric Leadscrew



BOXFORD MODEL A LATHE
METRIC THREAD CHART
ENGLISH LEAD SCREW 8 T P I.

M M PITCH	SCREW GEAR	COMPOUND GEAR	GEAR BOX POSITION	STUD GEAR
6.00	56	127-100	8	48
5.50	56	127-100	8	44
5.00	56	127-100	8	40
4.50	56	127-100	8	36
4.00	56	127-100	8	[illegible]
3.50	56	127-100	8	28
3.00	56	127-100	8	24
2.75	56	127-100	16	44
2.50	56	127-100	16	40
2.25	56	127-100	16	36
2.00	56	127-100	16	32
1.75	56	127-100	16	28
1.50	56	127-100	16	24
1.40	56	127-100	20	28
1.30	56	127-100	20	26
1.25	56	127-100	16	20
1.20	56	127-100	20	24
1.10	56	127-100	40	44
1.00	56	127-100	32	32
.90	56	127-100	40	36
.80	56	127-100	40	32
.75	56	127-100	32	24
.70	56	127-100	[illegible]	28
.65	56	127-100	40	26
.60	56	127-100	40	24
.55	56	127-100	80	44
.50	56	127-100	64	32
.45	56	127-100	80	36
.40	56	127-100	80	32
.35	56	127-100	80	28
.30	56	127-100	80	24
.25	56	127-100	128	32
.20	56	127-100	160	32

BOXFORD MACHINE TOOLS LTD
HALIFAX, ENGLAND

Fig. 187A. Metric Thread Chart for Standard Gearbox Lathes with English Leadscrew

The internationally agreed conversion for Imperial to Metric is 1 inch = 25.4 mm, and the use of the 127 T gear in the conversion gears on the Boxford standard lathes gives an exact conversion.
The conversion gears used are as follows:

Standard models (Screwed Nose Spindle).
- Metric to English–135/127 T Compound Gear.
- English to Metric–127/100 T Compound Gear.

VSL/LOO model (Tapered Nose Spindle).
- Metric to English–76/65 T Compound Gear.
- English to Metric–64/54 T Compound Gear.

N.B. The compact conversion gears on the VSL/LOO model are made possible by the use of the ratios in the respective gear boxes; a gear box being a standard fitment on this model.

Cutting Multiple Screw Threads

If a multiple thread has two grooves it is known as a double thread, if it has three it is called a triple thread and so on. See Fig. 189. Do not confuse the pitch and lead of a multiple thread. The pitch is the distance from a point on one thread to the corresponding point on the next thread. The lead is the distance a screw thread advances in one turn.

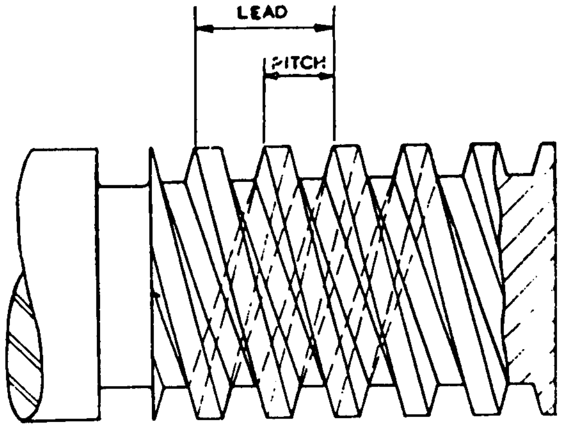


Fig. 189. A Double (or Two-Start) Thread

When multiple threads are being cut in the lathe, the first thread is cut to the desired depth. The work is then revolved part of a turn, and the second thread cut, etc. To obtain the exact spacing it is advisable to mill as many equally-spaced slots in the face plate for the lathe dog as there are multiple threads to be cut. Two slots should be milled for a double thread, three for a triple thread, etc. If slots cannot conveniently be cut in the face plate, equally-spaced studs may be attached to the face plate and a straight tail lathe dog used.

When cutting multiple threads work can also be indexed by disengaging the change gears after completing the first thread, and turning the spindle round to the required position for starting the next cut. Alternatively if the tool slide is set parallel to the bed, the position for the next cut can be obtained by accurately moving the tool slide the necessary pitch (taking care that any backlash is removed).

Cutting Threads with Die in Tailstock

A die can be mounted in the lathe tailstock to cut screw threads as shown in Fig. 190. Special self-releasing die holders to take either $\frac{13}{16}$" or 1" diameter dies can be supplied for use with "Boxford" machines.

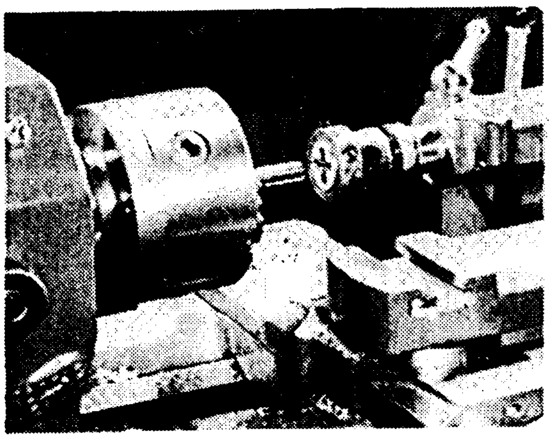


Fig. 190. Cutting Thread with Die in Self-release Die Holder in Tailstock

Special Types of Work

Chapter 10

The following pages give brief descriptions and illustrations of some of the most important of the many special types of work done on a lathe, such as knurling, filing and polishing.

Knurling

Sometimes the surface of a piece of work in the lathe is embossed and the process is called knurling. The embossing is done by a knurling tool in the lathe's tool post.

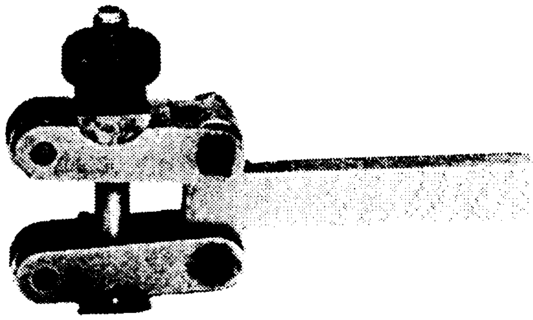


Fig. 191. Adjustable Knurling Tool

Fig. 192 illustrates three examples of knurling on the surface of a piece of steel. The knurl's pattern is similar for all three, but each grade is different, coarse, medium and fine.

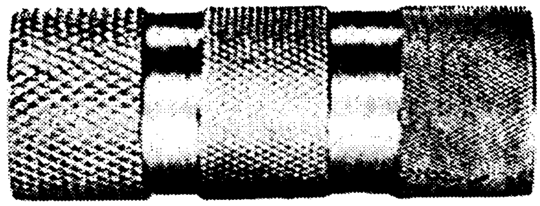


Fig. 192. Knurling

Fig. 191 illustrates an adjustable Knurling Tool in which the cut is applied by means of a nut which advances the two knurls towards each other when on the centre line of the work piece. This type of knurling tool greatly reduces the loading on the lathe cross-feed screw. With a conventional Knurling Tool (using a single or two fixed Knurles) the lathe should be arranged for the slowest back geared speed for all knurling operations. When the lathe has been started, the knurling tool must be forced slowly into the work at the right end until the knurl reaches a depth of approximately 0.4 mm ($\frac{1}{64}$ of an inch). The longitudinal feed of the carriage is then engaged and the knurling tool is fed across the face of the work. Plenty of oil should be used throughout the operation.

Once the left end of the knurl roller has reached the end of the work, the lathe spindle is reversed and the knurling tool fed back to the starting point. The knurling tool is not removed from the impression but is forced another 0.4 mm ($\frac{1}{64}$ of an inch) into the work and fed back across the work's face. This operation is repeated until the knurling has been completed.

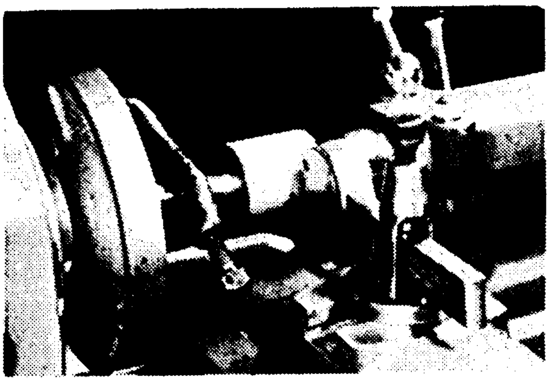


Fig. 193. Knurling a Steel Component in the Lathe

Machining Work on the Face Plate

Remove all dirt and chips from the threaded hole before mounting a face plate on the spindle nose of the lathe. The thread and shoulder on the spindle nose must also be cleaned because dirt, chips or burrs prevent the true running of the face plate.

The spindle threads must be oiled to ensure that the face plate screws on easily and can be removed without effort. Should there be any difficulty when screwing on the face plate, the plate must be unscrewed, any dirt, burrs or other obstructions removed and a further effort made. The hub of the face plate should screw tightly against the shoulder of the spindle, but the face plate must not be spun up to the shoulder suddenly because this makes it difficult to remove.

A face plate is invaluable for tool room work which involves the machining of holes in tools and jigs, etc. The holes must be spaced accurately for this class of work, with an allowance as a rule of not more than 0.025 mm (·001").

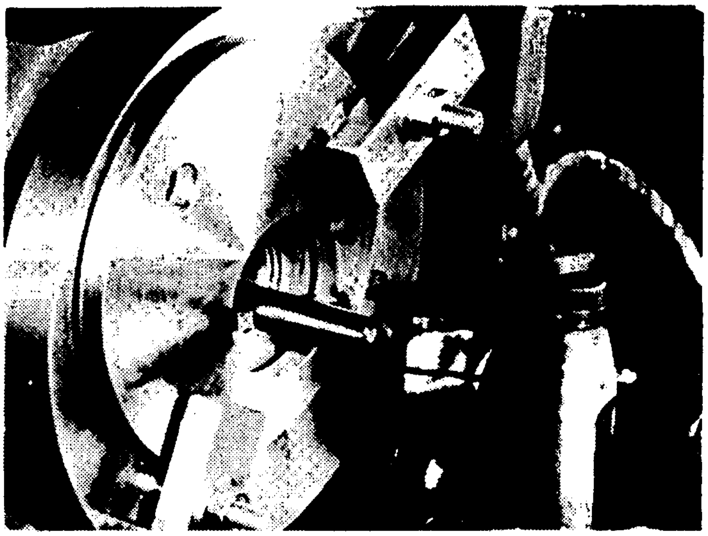


Fig. 194. Offset Hole Being Bored on the Face Plate

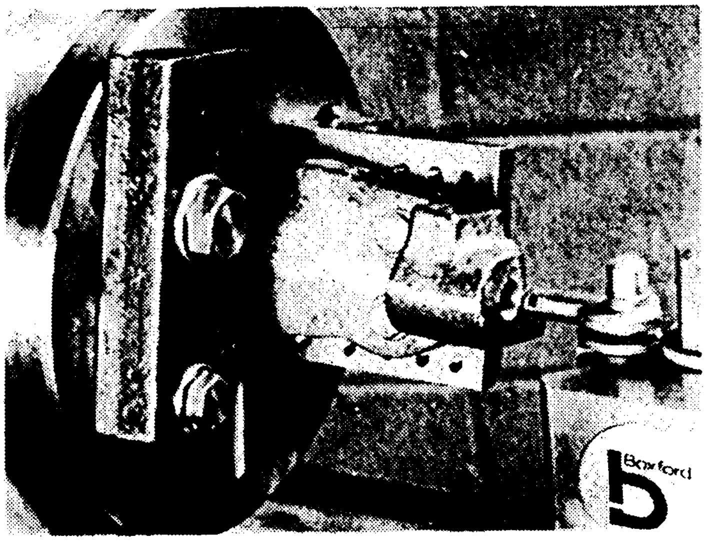


Fig. 195. Boring a Component on Angle Plate Attached to Face Plate

Clamping Work on Face Plate

Exercise care in clamping work on the face plate to avoid any springing either of the work or the face plate. To minimise the danger of the work slipping, insert a piece of paper between the face plate and the work. Balance weights should be used when necessary.

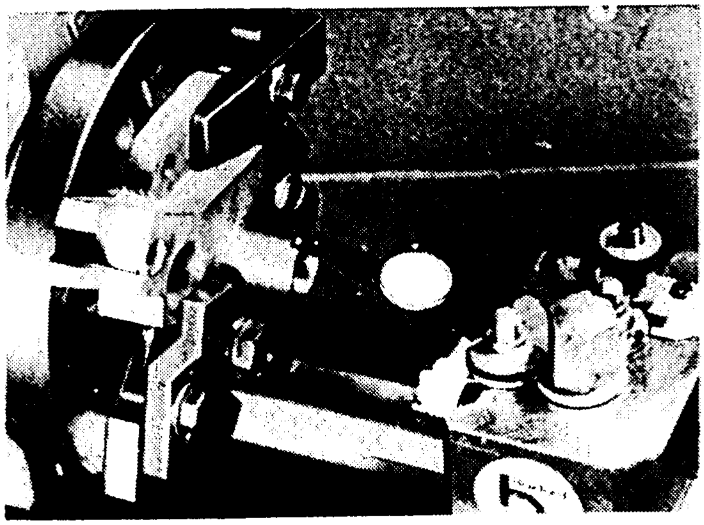


Fig. 196. Centring Work on the Face Plate with a Dial Indicator

Centring the Work

A dial indicator can be used, as shown in Fig. 196.

Lapping

Lapping is frequently used to finish hardened gauges, bushings and bearings in the lathe. (See Fig. 199). Diamond dust, emery dust and oil, emery cloth and other abrasives are used. It is usual to operate the lathe spindle at a high speed during lapping. See page 95.

The lap may be as primitive as a strip of emery cloth attached to a shaft, or it may be an elaborate construction of lead, copper, cast iron, etc. Careful lapping can result in much fine and precise work.

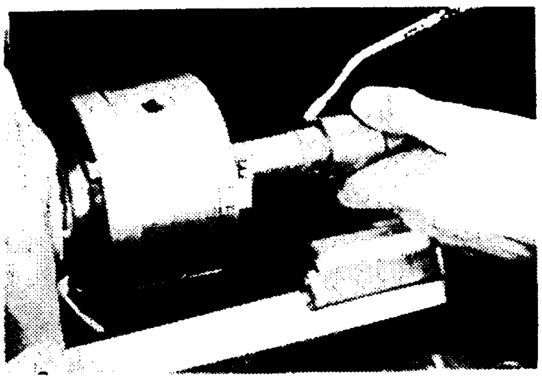


Fig. 199. Lapping a Hardened Bush in the Lathe

Machining Work on a Lathe Mandrel

When cylindrical work has been bored and reamed in a chuck, it is usually further machined on a mandrel between the lathe centres, as illustrated in Figs. 201 and 202. The slightly tapered mandrel is driven into the hole tightly enough to prevent the work from slipping on the mandrel whilst being machined.

Pulleys and similar work of large diameter should be driven with a pin or driver attached to the lathe face plate if this can be arranged. This will avoid any possibility of the work slipping on the mandrel.

Before the mandrel is driven into the hole, both the mandrel and the hole must be oiled to allow for the easy removal of the work from the mandrel. Without a lubricant the mandrel may seize in the work. Should that happen, it can only be driven out at the cost of ruining both the work and the mandrel.

When a mandrel is being driven out of a piece of work it must be driven in the opposite direction from that in which it entered.

Standard lathe mandrels, hardened and tempered, can be bought in various sizes. The surface to receive the work is usually ground to a taper of 1 in 2000 (·0005" per inch).

When special jobs have odd diameter holes, a soft mandrel can be used, turned to the proper diameter and tapered to secure a driving fit in the hole in the work. See page 87.

Special Mandrels

These are often employed for special classes of work. Fig. 202 shows a nut mandrel. This is used for finishing the outside diameter of gear blanks. There are also obtainable various types of expansion mandrels which are used where there is considerable variation in hole sizes.

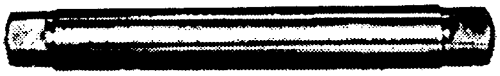


Fig. 200. A Lathe Mandrel

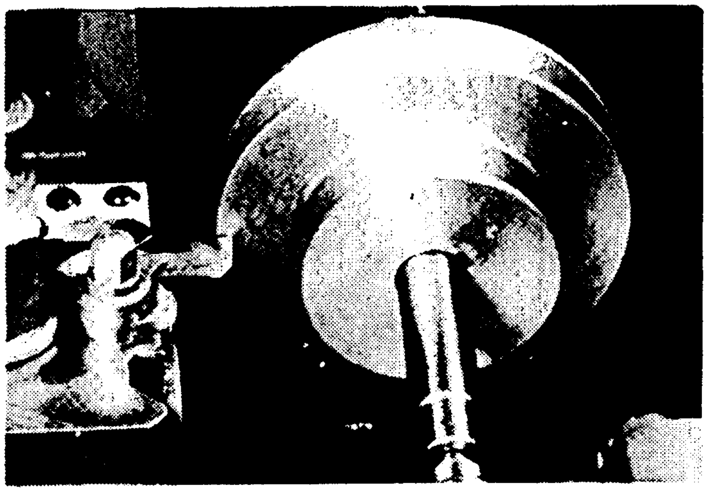


Fig. 201. Facing a V-Pulley on a Mandrel

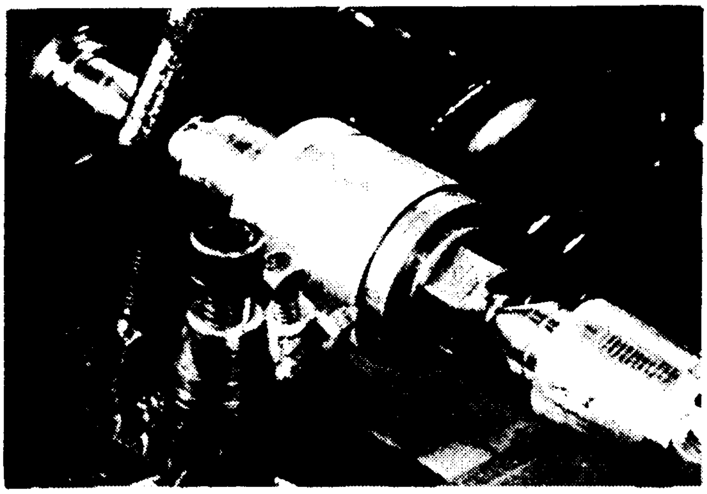


Fig. 202. Turning Several Gear Blanks on a special Mandrel

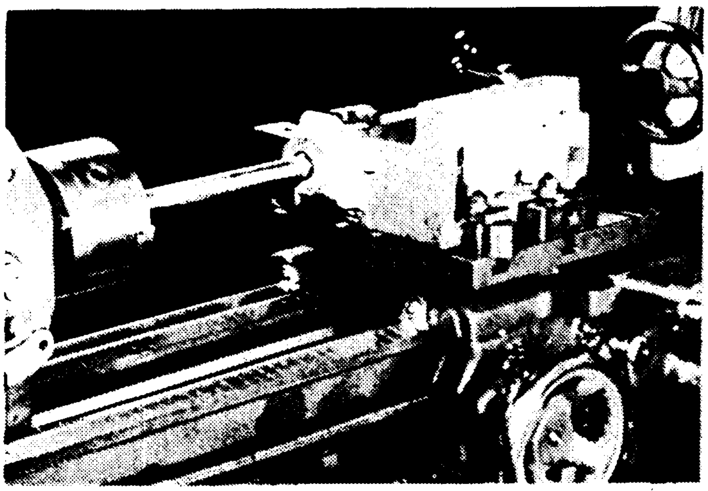


Fig. 203. Boring a Component Mounted on the Boring Table

Boring Work Mounted on a Lathe

Fig. 203 illustrates the method of mounting work on the lathe boring table for boring. The boring bar is held between centres and is driven by a lathe dog, or one end is held in the chuck (as in illustration). The work is clamped to the top of the boring table and is fed to the tool by the power longitudinal feed of the saddle.

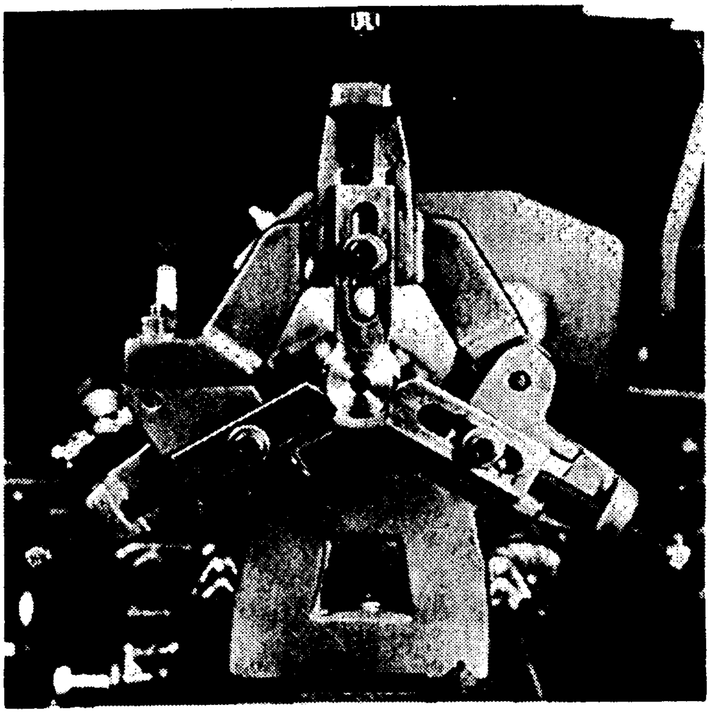


Fig. 204. Fixed Steady Attached to Lathe bed

The Use of the Fixed Steady

The purpose of the fixed steady is to support long shafts of small diameter when being turned, and for boring and threading spindles. Fig. 204 shows the end view of a fixed steady attached to the lathe bed.

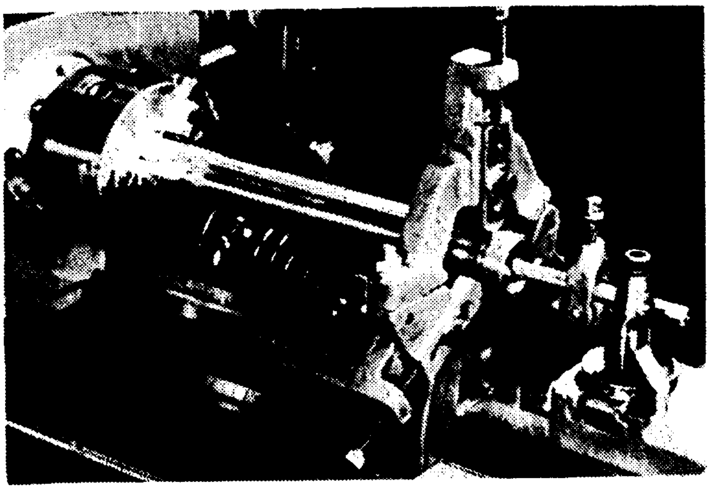


Fig. 205. Work Held in Chuck Supported at Free End by Fixed Steady

When work is mounted in the fixed steady, the steady itself is first placed on the lathe bed and the work put between the centres. Then the steady is moved into its proper position and the jaws are adjusted upon the work. This adjustment must be done most carefully because the work has to revolve in these jaws. When the jaws have been properly adjusted so that the work revolves freely, the jaws are clamped in position, the work fastened to the head spindle of the lathe and the tailstock moved out of the way. One end of the work can be held in a chuck, as shown in Fig. 205.

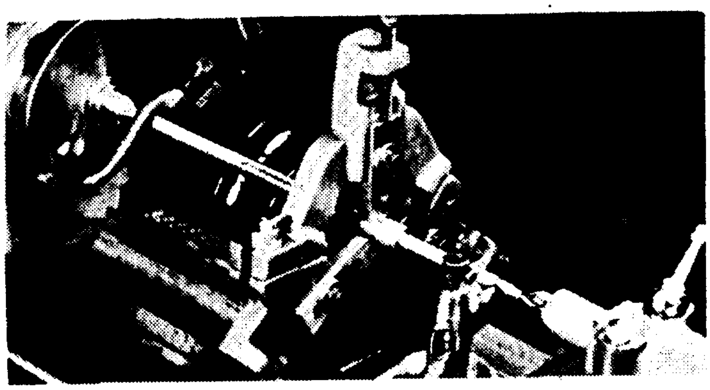


Fig. 206. Fixed Steady Supporting a Slender Shaft

The Use of the Travelling Steady

Support is needed for work of small diameter which might otherwise spring away from the cutting tool. This is provided by the travelling steady which is attached to the saddle of the lathe.

The travelling steady's adjustable jaws bear directly on the finished diameter of the work, as illustrated in Figs. 207 and 208. Since the travelling steady is attached to the saddle, it travels with the tool as the latter feeds along the work.

Small rollers are sometimes substituted for the rigid adjustable jaws when small shafts are being machined in quantity. The device is then known as the roller bearing travelling steady.

Fig. 209 shows how both the fixed steady and the travelling steady may be applied at the same time. The spindles or shafts to be machined, although of small diameter, are very long. To produce good quality work it is necessary to support the shaft with both the fixed steady and the travelling steady. The small, delicate spindles used in textile mills are machined by this method.

Milling in the Lathe

If a small shop has not enough milling work to justify the installation of a costly milling machine, it can undertake a good deal of milling work by using the milling attachment shown in Fig. 210. The attachment is mounted on the cross slide of the lathe, permitting either hand or power feeds to be used. The vertical slide is

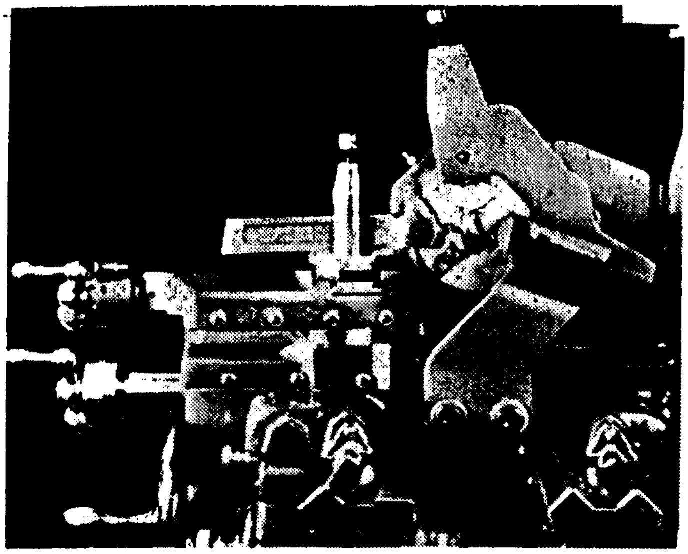


Fig. 207. Travelling Steady Attached to Lathe Saddle

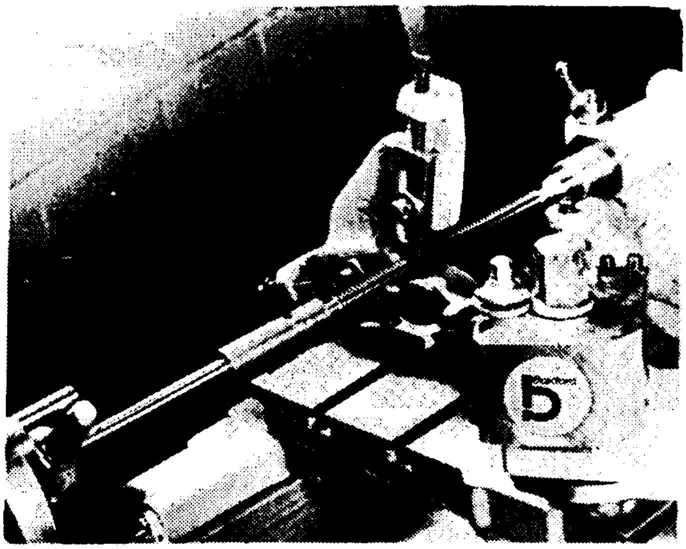


Fig. 208. Threading a Slender Shaft Supported by the Travelling Steady

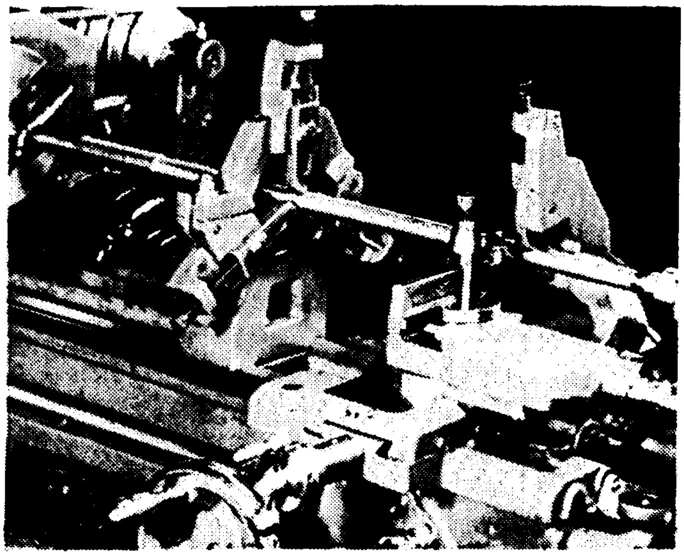


Fig. 209. Both Fixed and Travelling Steadies in Use

equipped with a friction dial reading in ·001" on English machines or .02 mm on metric.

All milling cuts should be taken with the rotation of the cutter against the direction of the feed as illustrated in Fig. 211.

Cutters up to 12.5 mm or 1" dia. shank can be held in the standard draw bar type collet attachment, but if the multisize collet is used shanks of up to 25.4 mm (1") dia. can be gripped. A 1" dia. arbor for direct mounting into the No. 3 Morse taper bore of the lathe spindle can also be used.

Standard Keyways

Fig. 212 and the table show the recognised standards for square keyways in shafts, pulleys, gears, etc. The key should always fit snugly without being too tight.

Cutting Gears on the Lathe

The dividing head attachment for the lathe, shown in Fig. 213 will cut, spur and bevel gears of all kinds. It will do graduating and milling, external key seating, cutting at angles, splining, slotting and all regular dividing head milling work. The dividing head construction is based on the Principle of a 40:1 worm and worm wheel, with various index plates.

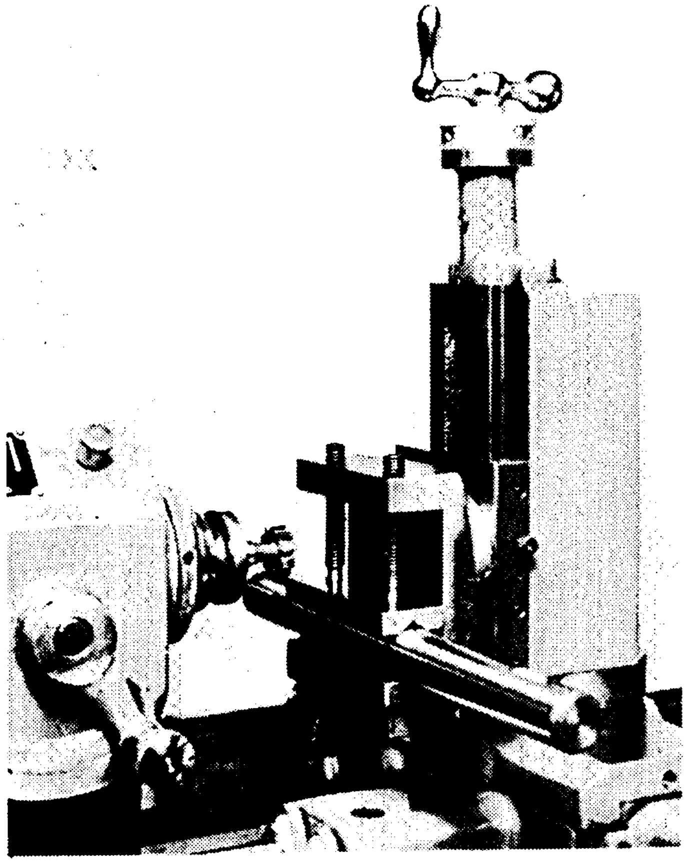


Fig. 210. Milling Attachment in Use

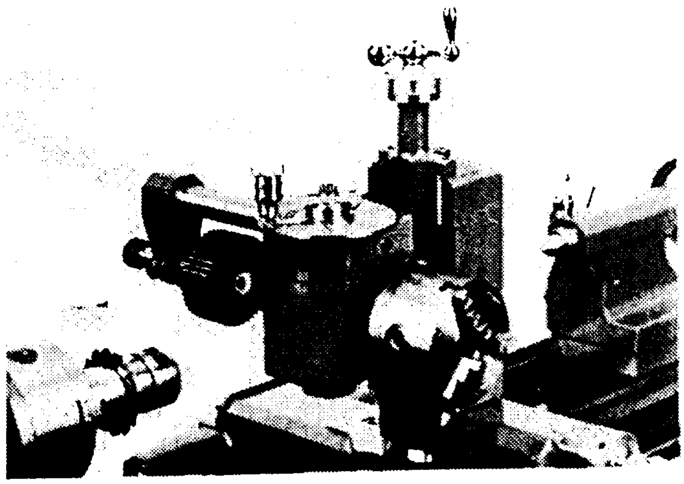


Fig. 213. Dividing Head Attachment

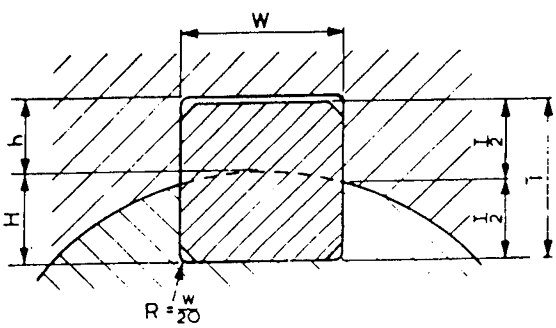


Fig. 212. Standard Keyways

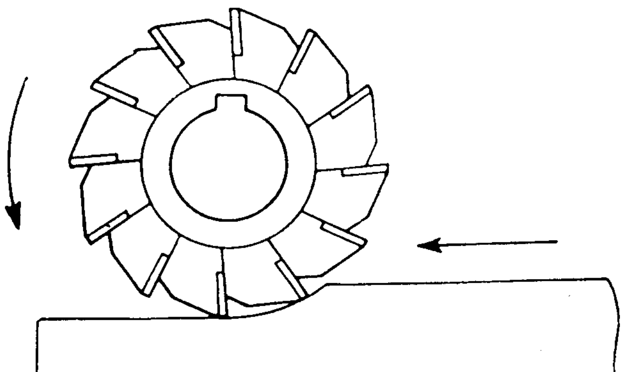


Fig. 211. Direction of Milling Cuts

Table 10. Square Keyways—British Standard and ISO Recommendations

Diameter of Shaft: Over	Diameter of Shaft: Up to and including	Key Designation: W · T	Key: Width W and Thickness T: Max.	Key: Width W and Thickness T: Min.	Keyway in shaft: Width W: Min.	Keyway in shaft: Width W: Max.	Keyway in shaft: Depth H: Min.	Keyway in shaft: Depth H: Max.	Keyway in hub: Width W: Min.	Keyway in hub: Width W: Max.	Keyway in hub: Depth h: Min.	Keyway in hub: Depth h: Max.	Bright Keybar: Width W and Thickness T: Max.	Bright Keybar: Width W and Thickness T: Min.
$\frac{3}{16}$"	$\frac{1}{4}$"	$\frac{3}{32}$" · $\frac{3}{32}$"	·096	·094	·093	·094	·056	·058	·094	·095	·044	·046	·096	·094
$\frac{1}{4}$"	$\frac{3}{8}$"	$\frac{1}{8}$" · $\frac{1}{8}$"	·127	·125	·124	·125	·073	·075	·125	·126	·058	·060	·127	·125
$\frac{3}{8}$"	$\frac{1}{2}$"	$\frac{5}{32}$" · $\frac{5}{32}$"	·158	·156	·155	·156	·091	·093	·156	·157	·072	·074	·158	·156
$\frac{1}{2}$"	$\frac{3}{4}$"	$\frac{3}{16}$" · $\frac{3}{16}$"	·190	·188	·187	·188	·106	·108	·188	·189	·088	·090	·190	·188
$\frac{3}{4}$"	1"	$\frac{1}{4}$" · $\frac{1}{4}$"	·252	·250	·249	·250	·142	·144	·250	·251	·114	·116	·252	·250
1"	$1\frac{1}{4}$"	$\frac{5}{16}$" · $\frac{5}{16}$"	·314	·312	·311	·312	·176	·178	·312	·313	·143	·145	·314	·312
$1\frac{1}{4}$"	$1\frac{1}{2}$"	$\frac{3}{8}$" · $\frac{3}{8}$"	·377	·375	·374	·375	·212	·214	·375	·376	·170	·172	·377	·375
$1\frac{1}{2}$"	$1\frac{3}{4}$"	$\frac{7}{16}$" · $\frac{7}{16}$"	·440	·438	·437	·438	·247	·249	·438	·439	·197	·199	·440	·438
$1\frac{3}{4}$"	2"	$\frac{1}{2}$" · $\frac{1}{2}$"	·502	·500	·499	·500	·282	·284	·500	·501	·224	·226	·502	·500
ALL DIMENSIONS IN MM. NORMAL CLASS FIT														
6	8	2 × 2	2.0	1.975	1.971	1.996	1.2	1.3	1.987	2.013	1.0	1.1	2.0	1.975
8	10	3 × 3	3.0	2.975	2.971	2.996	1.8	1.9	2.987	3.013	1.4	1.5	3.0	2.975
10	12	4 × 4	4.0	3.970	3.970	4.0	2.5	2.6	3.985	4.015	1.8	1.9	4.0	3.970
12	17	5 × 5	5.0	4.970	4.970	5.0	3.0	3.1	4.985	5.015	2.3	2.4	5.0	4.970
17	22	6 × 6	6.0	5.970	5.970	6.0	3.5	3.6	5.985	6.015	2.8	2.9	6.0	5.970

Fig. 215. Hand Rest Attachment

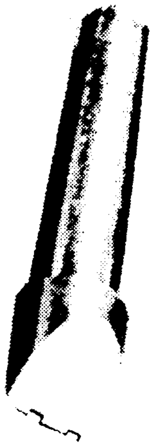


Fig. 216. Spur Centre

Fig. 217. Cup Centre

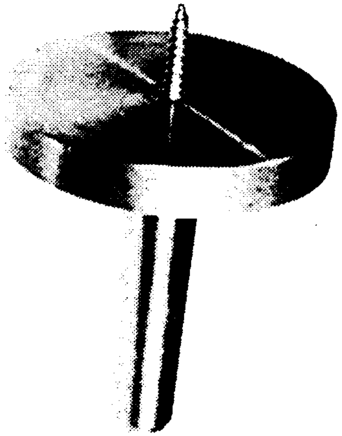


Fig. 216A. Screw Centre

Fig. 218. Hand Rest Attachment on the Lathe

Turning Wood, Fibre and Plastics

It is a very simple matter to turn wood in a metal-working lathe. Spur and cup centres are substituted for the 60 degree centres, a hand rest is attached and the lathe is ready for wood turning. Special pulleys may be used on the motor and countershaft to provide a series of high spindle speeds for wood turning, in addition to the regular speeds for metal work.

Machining of other materials can also be done. Bakelite, Alabaster, fibre and other plastics, synthetic resins and so forth may be turned and polished with completely satisfactory results.

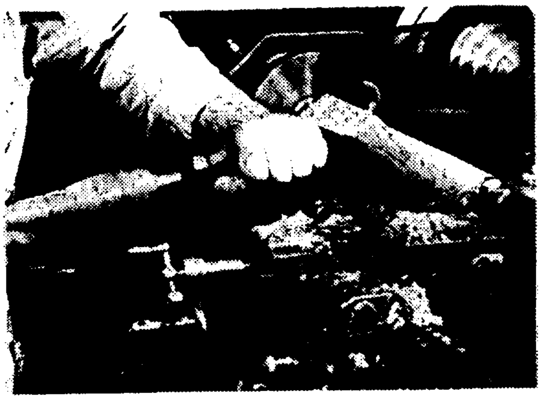


Fig. 214. Wood Turning in the Lathe

Micrometer Carriage Stop

The micrometer carriage stop is a micrometer spindle mounted in a clamp which is securely locked on the front V-way of the lathe bed in the manner shown in Fig. 219. A thumb screw is provided to lock the spindle at any point.

This micrometer carriage stop is used for facing shoulders to an exact length. It is usually included an the equipment of all toolroom lathes and is very convenient for many production operations.

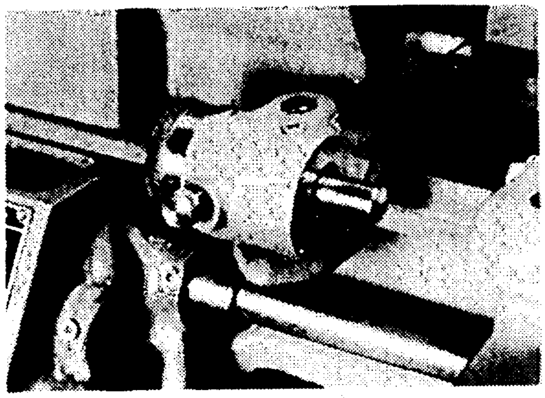


Fig. 219. Micrometer Carriage Stop

Four Position Carriage Stop

Fig. 220 shows the four position carriage stop fixed to the lathe bed. Each of the four adjustable screws may be set for a different tool position and can be turned into position to locate the saddle for each of four separate operations.

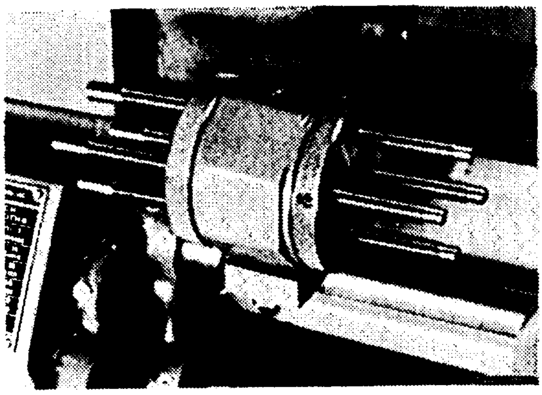


Fig. 220. 4-Position Carriage Stop

Metric Graduated Dials

When lathes are intended to be used solely for working in the metric system, they are equipped with all metric lead screws and metric graduated dials. The standard metric dial is graduated to read in .02 mm. An alternative "direct reading" cross slide dial is available, each division representing .05 mm reduction in the diameter of the work piece. All dials are adjustable so that they can be set at zero whenever desired.

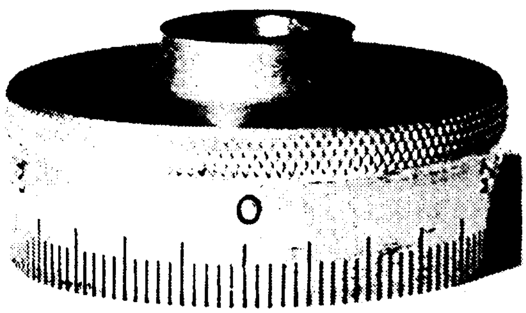


Fig. 221. Metric Dial

Metric Graduations on Tailstock Spindle
Fig. 222 shows how the tailstock spindle may be graduated in millimetres as a help to the operative in drilling accurately to the required depth.

Grinding in the Lathe
A lathe which is fitted with a good electric grinding attachment can be used for sharpening reamers and milling cutters, grinding hardened bushings and shafts, as well as many other grinding operations.
The lathe bed's V-ways must be protected by covering with a heavy cloth or canvas to keep from them the dust and grit associated with the grinding wheel. Most of the grit can easily be collected in a small vessel of oil or water placed just below the grinding wheel. In general, however, we do not recommend grinding in the lathe.

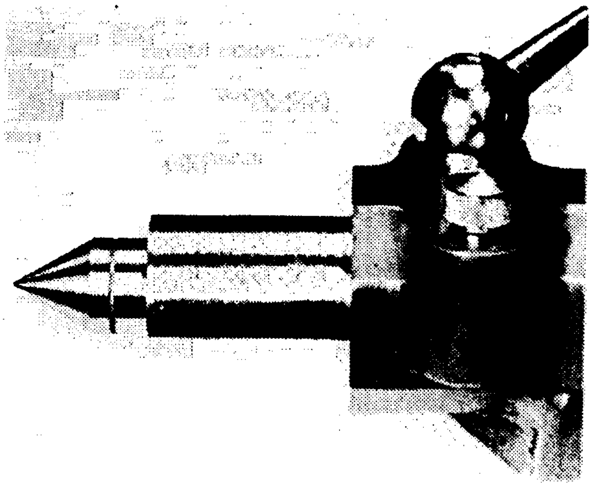


Fig. 222. Metric Graduations on Tailstock Spindle

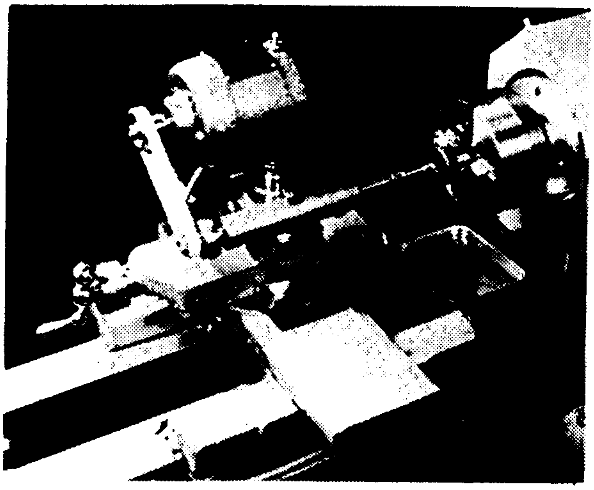


Fig. 223. Tool Post Grinder

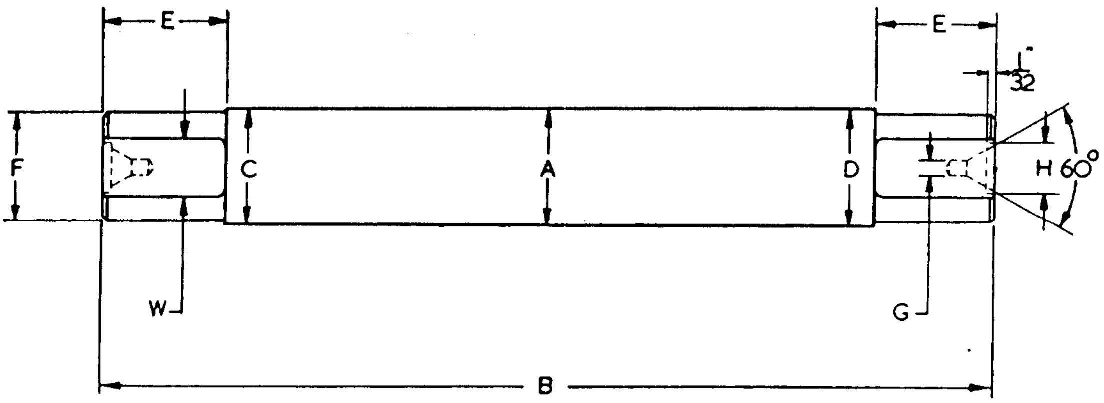


Fig. 224. Taper Mandrel for Machining Work between Centres

How to Make Lathe Mandrels
Lathe mandrels can be made from any good grade of machine steel. Old axles from motor cars are excellent for the purpose. There is no need to harden the mandrel when only a few parts are to be made.
The following table shows the dimensions recommended for standard lathe mandrels. There must be a slight taper so that the mandrel can be pressed tightly into the part. The mandrel size is always stamped on the large end.
It is important that centre holes in the mandrel ends should be large enough

to provide a good bearing and they must be perfectly concentric with the outside diameter of the mandrel.

It is desirable to case-harden the centre holes or even the entire mandrel when many parts have to be made. After hardening, the outside diameter of the mandrel must be finished. It will not run true unless finished, due to the warping of the steel during the hardening process.

Both the inside and outside of the part and the mandrel must be oiled before any part is mounted on the mandrel.

Precision Lathe mandrels are commercially available in Metric sizes from 4 to 50 mm and Imperial sizes from $\frac{3}{16}$" to 2".

Table 11. Sizes for Lathe Mandrels

Taper 1 in 2000 (·0005"/inch)

Nominal Diameter	Total Length	Small End	Large End	Undercut Length	Undercut Diameter	Centre Drill	Recess for Centre	Width of Flat
A	B	C	D	E	F	G	H	W
$\frac{1}{4}$"	$3\frac{3}{4}$"	·2495"	·2506"	$\frac{1}{2}$"	$\frac{15}{64}$"	$\frac{3}{64}$"	$\frac{1}{8}$"	$\frac{1}{8}$"
$\frac{3}{8}$"	$4\frac{1}{4}$"	·3745"	·3761"	$\frac{1}{2}$"	$\frac{11}{32}$"	$\frac{1}{16}$"	$\frac{7}{32}$"	$\frac{5}{32}$"
$\frac{1}{2}$"	5"	·4995"	·5015"	$\frac{1}{2}$"	$\frac{15}{32}$"	$\frac{3}{32}$"	$\frac{9}{32}$"	$\frac{1}{4}$"
$\frac{5}{8}$"	$5\frac{1}{2}$"	·6245"	·6266"	$\frac{5}{8}$"	$\frac{9}{16}$"	$\frac{3}{32}$"	$\frac{9}{32}$"	$\frac{5}{16}$"
$\frac{3}{4}$"	6"	·7495"	·7519"	$\frac{5}{8}$"	$\frac{11}{16}$"	$\frac{3}{32}$"	$\frac{5}{16}$"	$\frac{3}{8}$"
$\frac{7}{8}$"	$6\frac{1}{2}$"	·8740"	·8765"	$\frac{3}{4}$"	$\frac{25}{32}$"	$\frac{1}{8}$"	$\frac{7}{16}$"	$\frac{7}{16}$"
1"	7"	·9990"	1·0017"	$\frac{3}{4}$"	$\frac{7}{8}$"	$\frac{1}{8}$"	$\frac{7}{16}$"	$\frac{1}{2}$"
DIMENSIONS IN MM.								
6	70	5 990	6.015	10	5.5	1.25	3	3
8	85	7.990	8.022	10	7.25	1.25	4 5	3.5
10	95	9.990	10.025	12	9 0	1.6	6	4
12	115	11.985	12 028	14	11.0	2 0	7	6
14	115	13.985	14.028	14	13 0	2 0	8	7
16	130	15 980	16 028	17	15 0	2 5	8.5	8
18	130	17.980	18 028	17	17 0	2 5	8 5	9
20	160	19.980	20 042	18	18 0	2.5	8 5	10
22	160	21 980	22 042	18	20 0	3.15	11	11
25	170	24.975	24 040	20	23.0	3.15	11	12

Press Fits and Running Fits

The tables printed overleaf give the standard tolerances for press fits and running fits, etc. A standard size of hole is usually made and the shaft is made to the necessary size for the desired type of fit. In the tables the figures indicate the amount by which to increase or decrease the shaft diameter, provided that a standard hole size is maintained.

There is so much variation in working conditions that it is sometimes advisable to increase or decrease the allowances given in the tables. The length of the bearing, the material used and the speed are all relevant factors to be considered when the tolerance for a running fit is being calculated.

Table 12. Suggested Limits From ISO Recommendations

TOLERANCES IN STANDARD HOLES (IN MILLIMETRES) — **2 CLASSES**

	Nominal Diameters	0–3	3–6	6–10	10–18	18–30	30–50	50–80	80–120
H7	High Limit ...	+0.010	+0.012	+0.015	+0.018	+0.021	+0.025	+0.030	+0.035
	Low Limit ...	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
H8	High Limit ...	+0.014	+0.018	+0.022	+0.027	+0.033	+0.039	+0.046	+0.054
	Low Limit ...	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

INTERFERENCE (DRIVE FIT) ALLOWANCES ON SHAFTS FOR VARIOUS FITS

	Nominal Diameters	0–3	3–6	6–10	10–18	18–30	30–50	50–80	80–120
p6	High Limit ...	+0.012	+0.020	+0.024	+0.029	+0.035	+0.042	+0.051	+0.059
	Low Limit ...	+0.006	+0.012	+0.015	+0.018	+0.022	+0.026	+0.032	+0.037
	Tolerance ...	0.006	0.008	0.009	0.011	0.013	0.016	0.019	0.022

TRANSITION (PUSH FIT)

		0–3	3–6	6–10	10–18	18–30	30–50	50–80	80–120
k6	High Limit ...	+0.006	+0.009	+0.010	+0.012	+0.015	+0.018	+0.021	+0.025
	Low Limit ...	+0.0	+0.001	+0.001	+0.001	+0.002	+0.002	+0.002	+0.003
	Tolerance ...	0.006	0.008	0.009	0.011	0.013	0.016	0.019	0.022

CLEARANCE (CLOSE RUNNING FIT)

		0–3	3–6	6–10	10–18	18–30	30–50	50–80	80–120
g6	High Limit ...	—0·002	—0.004	—0.005	—0.006	—0.007	—0.009	—0.010	—0.012
	Low Limit ...	—0·008	—0.012	—0.014	—0.017	—0.020	—0.025	—0.029	—0.034
	Tolerance ...	0·006	0.008	0.009	0.011	0.013	0.016	0.019	0.022

Table 12a. Newall Limits

TOLERANCES IN STANDARD HOLES — **2 CLASSES**

	Nominal Diameters	0" to ½"	½"+to 1"	1"+to 2"	2"+to 3"	3"+to 4"	4"+to 5"	5"+to 6"
CLASS A	High Limit ...	+·00025	+·0005	+·00075	+·001	+·001	+·001	+·0015
	Low Limit ...	—·00025	—·00025	—·00025	—·0005	—·0005	—·0005	—·0005
	Tolerance ...	·0005	·00075	·001	·0015	·0015	·0015	·002
CLASS B	High Limit ...	+·0005	+·00075	+·001	+·00125	+·0015	+·00175	+·002
	Low Limit ...	—·0005	—·0005	—·0005	—·00075	—·00075	—·00075	—·001
	Tolerance ...	·001	·00125	·0015	·002	·00225	·0025	·003

FORCE FITS — **ALLOWANCES ON SHAFTS FOR VARIOUS FITS**

	Nominal Diameters	0" to ½"	½"+to 1"	1"+to 2"	2"+to 3"	3"+to 4"	4"+to 5"	5"+to 6"
CLASS F	High Limit ...	+·001	+·002	+·004	+·006	+·008	+·010	+·012
	Low Limit ...	+·0005	+·0015	+·003	+·0045	+·006	+·008	+·010
	Tolerance ...	·0005	·0005	·001	·0015	·002	·002	·002

DRIVE FITS

	Nominal Diameters	0" to ½"	½"+to 1"	1"+to 2"	2"+to 3"	3"+to 4"	4"+to 5"	5"+to 6"
CLASS D	High Limit ...	+·0005	+·001	+·0015	+·0025	+·003	+·0035	+·004
	Low Limit ...	+·00025	+·00075	+·001	+·0015	+·002	+·0025	+·003
	Tolerance ...	·00025	·00025	·0005	·001	·001	·001	·001

PUSH FITS

	Nominal Diameters	0" to ½"	½"+to 1"	1"+to 2"	2"+to 3"	3"+to 4"	4"+to 5"	5"+to 6"
CLASS P	High Limit ...	—·00025	—·00025	—·00025	—·0005	—·0005	—·0005	—·0005
	Low Limit ...	—·00075	—·00075	—·00075	—·001	—·001	—·001	—·001
	Tolerance ...	·0005	·0005	·0005	·0005	·0005	·0005	·0005

RUNNING FITS — **3 CLASSES**

	Nominal Diameters	0" to ½"	½"+to 1"	1"+to 2"	2"+to 3"	3"+to 4"	4"+to 5"	5"+to 6"
CLASS X	High Limit ...	—·001	—·00125	—·00175	—·002	—·0025	—·003	—·0035
	Low Limit ...	—·002	—·00275	—·0035	—·00425	—·005	—·00575	—·0065
	Tolerance ...	·001	·0015	·00175	·00225	·0025	·00275	·003
CLASS Y	High Limit ...	—·00075	—·001	—·00125	—·00150	—·00200	—·00225	—·00250
	Low Limit ...	—·00125	—·002	—·0025	—·00300	—·00350	—·00400	—·00450
	Tolerance ...	·0005	·001	·00125	·00150	·00150	·00175	·00200
CLASS Z	High Limit ...	—·0005	—·00075	—·00075	—·001	—·001	—·00125	—·00125
	Low Limit ...	—·00075	—·00125	—·0015	—·002	—·00225	—·0025	—·00275
	Tolerance ...	·00025	·0005	·00075	·001	·00125	·00125	·0015

Use of Coolant

As the name suggests, a coolant is an agent employed to cool the work and the cutting tool. It also facilitates production by lubricating the cutting tool, flushing away chips and preventing rust. Coolants are therefore used generously when machining steel parts to gain a higher cutting speed, produce a better finish and increase the tool's life.

Coolant Equipment

The simplest equipment is often effective to apply a coolant. For example, a small paint brush may be used, or a limited amount of coolant can also be applied with an ordinary oil can. When engaged on continuous high-speed work, however, it is desirable to equip the lathe with an oil tray, coolant pump and reservoir. See page 100.

Applying a Coolant

The coolant must be properly applied to the work if it is to cool it effectively. It is better to have a large stream running slowly than a small one running quickly. The coolant should reach the work exactly where the cutting action occurs, and not to one side or above the cutting tool.

Types of Coolants

Coolants vary in their properties. It is, therefore, necessary to choose the one most suitable for the particular type of work in hand. The characteristics of the most popular coolants are:—

Mineral Oils.—These are oils with a petroleum base mixed with chemicals to improve their lubricating and anti-welding qualities. They are less expensive than the usual lubricants recommended.

Soluble Oils.—These are mineral oils so treated that, when mixed with water, they yield an emulsion which serves as an excellent and inexpensive coolant. They carry the heat away better than mineral oils, but are deficient in lubricating qualities. They are generally used for rough turning operations only. Despite being mixed with water, they have the capacity for leaving a protective film on metal which is rust-resisting.

Information on Gears

ISO recommendations on Gearing include both the Diametral Pitch system and the metric Module system.

The following rules and formulas may be used for calculating the dimensions of involute spur gears.

Diametral Pitch. (DP)—Number of teeth divided by pitch diameter, or 3·1416 divided by circular pitch.

Example: When a gear has 60 teeth and the pitch diameter is 5", the diametral pitch is 60 divided by 5, and the diametral pitch is 12, or, in other words, there are 12 teeth to each inch of the pitch diameter, and the gear is 12 diametral pitch.

Circular Pitch. (CP)—Distance from centre to centre of two adjacent teeth along the pitch line. CP (inches) = 3·1416 divided by Diametral Pitch.

CP (mm) = 3·1416 × M where M is the module number.

Pitch Diameter. (PD)—Number of teeth divided by diametral pitch.

PD (mm) = Number of teeth × M.

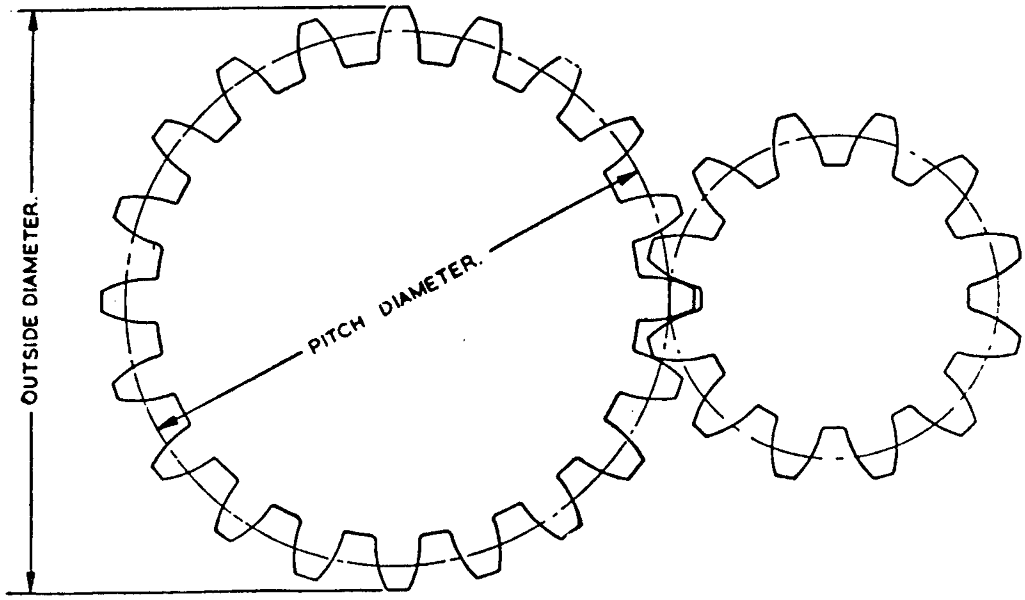


Fig. 225. Spur Gearing (Involute)

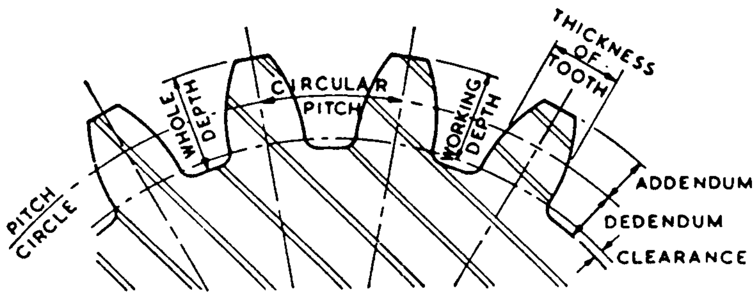


Fig. 226. Gear Tooth Names

Outside Diameter.—Number of teeth plus two divided by diametral pitch.

Example: If the number of teeth is 40 and the diametral pitch is 8, add 2 to the 40, making 42, and divide by 8. The quotient, 5·25, is the outside diameter of gear or blank.

Addendum.—1 divided by diametral pitch or equals M (mm).

Whole Depth of Tooth.—2·25 divided by diametral pitch or 2·25 X M (mm).

Thickness of Tooth.—1·5708 divided by diametral pitch or 1·5708 X M (mm).

Number of Teeth.—Pitch diameter multiplied by diametral pitch, or multiply outside diameter by diametral pitch and subtract 2.

Example: If the diameter of the pitch circle is 8", and the diametral pitch is 10, multiply 8 by 10 and the product, 80, will be the number of teeth in the gear.

Example: If the outside diameter is $5\frac{1}{4}$" and the diametral pitch is 8, multiply $5\frac{1}{4}$ by 8, and the product, 42, less 2, or 40, is the number of teeth.

Centre Distance.—Total number of teeth in both gears divided by twice the diametral pitch.

Example: If the two gears have 60 and 40 teeth respectively, and are 10 pitch, add 60 and 40, making 100, divide by 2 and then divide the quotient, 50, by the diametrical pitch, 10, and the result, 5 ins., is the centre distance.

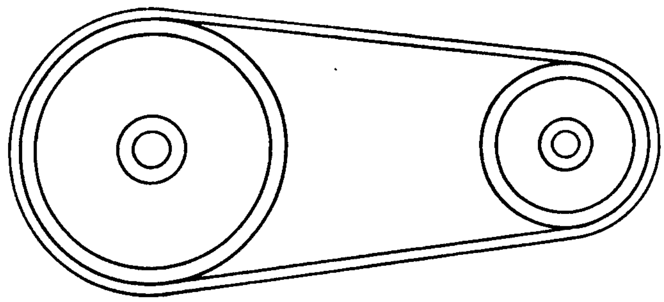


Fig. 227. Flat Belt Drive

Calculating the Speed and Size of Pulleys

Diameter of Driving Pulley.—Multiply the diameter of the driven pulley by its number of revolutions, and divide by the number of revolutions of the driver.

Diameter of Driven Pulley.—Multiply the diameter of the driving pulley by its number of revolutions, and divide the product by the number of revolutions of the driven pulley.

Speed of the Driven Pulley.—Multiply the diameter of the driving pulley by its number of revolutions, and divide by the diameter of the driven pulley.

Speed of Driving Pulley.—Multiply the diameter of the driven pulley by its number of revolutions, and divide by the diameter of the driving pulley.

The driving pulley is called the driver and the driven pulley is the driven or follower.

R.P.M. indicates the number of revolutions per minute.

Example: Problem 1.

Given: Speed of the driving pulley 250 R.P.M. Speed of the driven pulley 375 R.P.M. Diameter of the driven pulley 8 ins. (or 8 mm).

To find the diameter of the driving pulley.

375 X 8 equals 3,000.

Divided by 250 equals 12.

The diameter of the driving pulley is 12 ins (or 12 mm).

Width of Pulleys.—Pulleys for flat belts should be about 10% wider than the width of the belt used.

Types of Pulleys.—The two types of pulleys used for flat belts are the crowned face pulley and the flat face pulley. Crowned pulleys should be used whenever possible, since the crown keeps the belt on the pulley. Flat face pulleys should only be used when it is necessary to shift the belt from one position to another on the pulley, as in a drum pulley or a wide faced pulley on a machine used to match a fast and loose pulley on a countershaft.

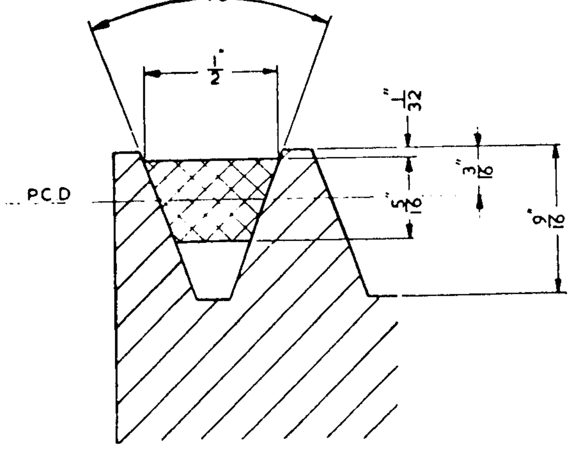


Fig. 228. Vee Pulley Dimensions for "A" Section V-Belts

VEE Pulleys are widely used for short centre drives, and the section of belt used on the Boxford $4\frac{1}{2}$" Screw Cutting Lathes is shown in Fig. 228.

Fitting a Backplate to a Chuck (Screwed Spindle Nose only)
When a chuck is to be used on a lathe it must first be fitted with a back plate that has been threaded to fit the spindle nose of the lathe. Back plates in a semi-machined state, which have been accurately threaded to fit the lathe spindle, can be obtained from the lathe manufacturer.

Mounting a Backplate on the Spindle
The threads of the back plate and the spindle nose must be cleaned thoroughly before the back plate is screwed on the spindle nose of the lathe. No chips, burrs or small particles of dirt must be allowed to lodge in the screw threads or on the nose face. The shoulder on the headstock spindle must equally be free from chips or burrs and kept perfectly clean.

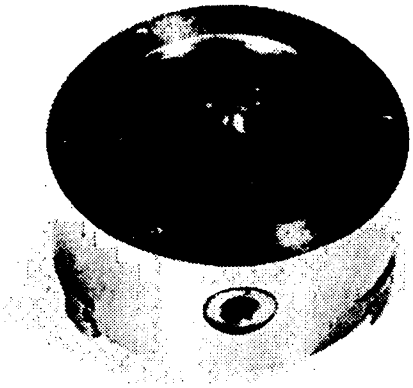


Fig. 230. View of Chuck showing Register

The threads of the headstock spindle and back plate must be oiled. Then the back plate should be screwed on to the spindle nose. The threads must never be jammed tightly or difficulty will be created when the back plate has to be removed when it is finished.

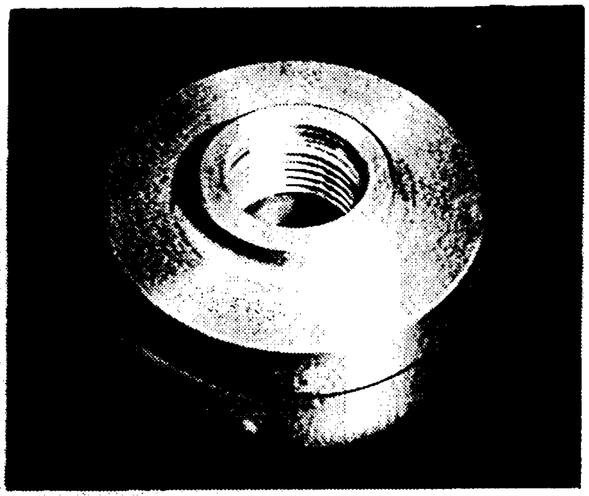


Fig. 229. Chuck Back Plate

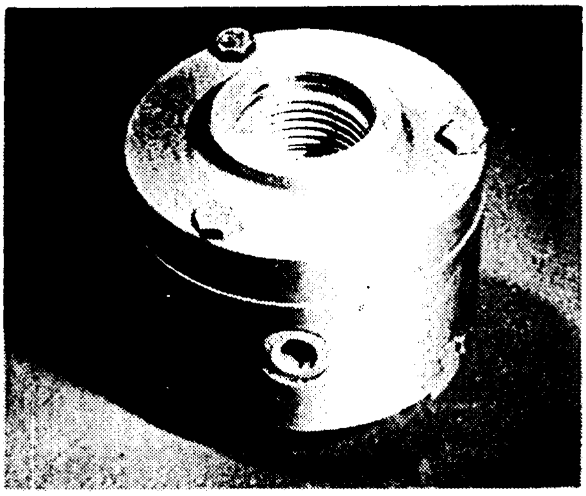


Fig. 231. Chuck and Back Plate Assembled

Finishing the Flange
The face of the back plate must first be machined, taking a roughing cut about 0.8 mm ($\frac{1}{32}$") in depth. This should be followed by one or two finishing cuts and not more than 0.025 mm (·001") should be removed in the final cut. The diameter of the recess in the back of the chuck must be measured carefully with inside calipers. The outside calipers must be set to correspond with the inside calipers. The diameter of the back plate flange must be carefully measured. Very light finishing cuts should be taken, and the chuck tried frequently on the back plate because the back plate must fit snugly into the recess in the back of the chuck.
When the back plate has been finished to fit this recess, it must be removed from the lathe spindle and the face of the flange chalked thoroughly. The

flange is then placed in the recess in the back of the chuck. The back plate is then tapped lightly so that the edge of the bolt holes in the chuck will mark the position of the bolt holes.
The holes should be drilled 1.5 mm ($\frac{1}{16}$") larger in diameter than the bolts used to secure the back plate to the chuck. To remove any possibility of the bolts binding, it is most important that the bolt holes should be large enough.

How to Calculate Change Gears for Thread Cutting
Whenever it is necessary to cut a special thread which does not appear on the gear chart of a lathe, or if no gear chart is available, it is easy to calculate the gears required. All Boxford Lathes are even geared; that is to say, the stud gear revolves the same number of times as the headstock spindle, and when gears of the same size are used on both the lead screw and the stud, the lead screw and spindle revolve the same number of times. It is therefore not necessary to consider the gearing between the headstock spindle and the stud gear when calculating change gears.
If simple gearing is to be used, the ratio of the number of teeth in the change gears used will be the same as the ratio between the thread to be cut and the thread on the lead screw. For example, if 10 threads per inch are to be cut on a lathe having a lead screw with 8 threads per inch, the ratio of the change gears would be 8 to 10. These numbers may be multiplied by any common multiplier to obtain the number of teeth in the change gears that should be used.
Rule.—To calculate change gears, multiply the pitch in millimetres (or T.P.I.) to be cut and the pitch in millimetres (or T.P.I.) of the lead screw by the same number.

Example: *Problem* – To cut 1 mm pitch thread on lathe having lead screw with 3 mm pitch.

Solution—1X20 equals 20—No. of teeth in gear on stud.
3 X 20 equals 60—No. of teeth in gear on lead screw.

If these gears are not to be found in the change gear set, any other number may be used as a common multiplier, such as 3, 5, 7, etc.
If compound gearing is used, the ratio of the compound idler gears must also be taken into consideration. This ratio is normally either 2:1 or 4:1, the calculation being the same as for simple gearing except that the number of threads actually cut will be twice or four times the number for simple gearing depending on which ratio of compound gears is used.

Allowance for Finish
When rough turning or boring a surface which must be finish turned, reamed, ground or otherwise finished, the amount of stock that should be left varies with the size of the work and the process used. Usually a minimum amount of stock should be left for the finishing cut, but it is important to leave enough stock to ensure that the work will "clean up". If parts have to be heat-treated there must be ample finish to allow for warping during the heat-treatment process. Allowance must also be made for any warping that may result from the removal of an outer surface in which strains are likely, such as castings, or steel bars that have not been normalised.

Finish Turning

An allowance of 0.4–0.8 mm ($\frac{1}{64}$" and $\frac{1}{32}$") has been found sufficient on the diameter, as a general rule, for finish turning work of normal lengths up to 50 mm (2") in diameter. When the diameter is larger than 50 mm (2") the allowance is usually between 0.8 and 1.5 mm ($\frac{1}{32}$" and $\frac{1}{16}$").

Grinding

0.25 to 0.3 mm (·010" to ·012") on the diameter is the normal allowance for grinding. Small parts require even less. Parts which have to be heat-treated, however, between the turning and grinding operations should have an allowance of from 0.4 to 0.5 mm (·015" to ·020") on the diameter to allow for warping.

Reaming

When holes have to be finished by reaming the drilling or boring must be undersize. Machine reaming in steel usually has an allowance of from 0.13 to 0.25 mm (·005" to ·010") and in cast iron 0.25 to 0.4 mm (·010" to ·015") but for hand reaming in both steel and cast iron 0.013 to 0.04 mm (·0005" to ·0015") is sufficient.

Lapping and Honing

The lapping and honing processes remove very little stock and it is usually enough to allow no more than 0.013 mm (·0005") on the diameter. See page 79.

Table 13. Decimal Equivalents of Fractional Parts of an Inch

$\frac{1}{64}$	·015625	$\frac{11}{32}$	·34375	$\frac{11}{16}$	·6875
$\frac{1}{32}$	·03125	$\frac{23}{64}$	·359375	$\frac{45}{64}$	·703125
$\frac{3}{64}$	·046875	$\frac{3}{8}$	·375	$\frac{23}{32}$	·71875
$\frac{1}{16}$	·0625	$\frac{25}{64}$	·390625	$\frac{47}{64}$	·734375
$\frac{5}{64}$	·078125	$\frac{13}{32}$	·40625	$\frac{3}{4}$	·75
$\frac{3}{32}$	·09375	$\frac{27}{64}$	·421875	$\frac{49}{64}$	·765625
$\frac{7}{64}$	·109375	$\frac{7}{16}$	·4375	$\frac{25}{32}$	·78125
$\frac{1}{8}$	·125	$\frac{29}{64}$	·453125	$\frac{51}{64}$	·796875
$\frac{9}{64}$	·140625	$\frac{15}{32}$	·46875	$\frac{13}{16}$	·8125
$\frac{5}{32}$	·15625	$\frac{31}{64}$	·484375	$\frac{53}{64}$	·828125
$\frac{11}{64}$	·171875	$\frac{1}{2}$	·5	$\frac{27}{32}$	·84375
$\frac{3}{16}$	·1875	$\frac{33}{64}$	·515625	$\frac{55}{64}$	·859375
$\frac{13}{64}$	·203125	$\frac{17}{32}$	·53125	$\frac{7}{8}$	·875
$\frac{7}{32}$	·21875	$\frac{35}{64}$	·546875	$\frac{57}{64}$	·890625
$\frac{15}{64}$	·234375	$\frac{9}{16}$	·5625	$\frac{29}{32}$	·90625
$\frac{1}{4}$	·25	$\frac{37}{64}$	·578125	$\frac{59}{64}$	·921875
$\frac{17}{64}$	·265625	$\frac{19}{32}$	·59375	$\frac{15}{16}$	·9375
$\frac{9}{32}$	·28125	$\frac{39}{64}$	·609375	$\frac{61}{64}$	·953125
$\frac{19}{64}$	·296875	$\frac{5}{8}$	·625	$\frac{31}{32}$	·96875
$\frac{5}{16}$	·3125	$\frac{41}{64}$	·640625	$\frac{63}{64}$	·984375
$\frac{21}{64}$	·328125	$\frac{21}{32}$	·65625	1	1·0
		$\frac{43}{64}$	·671875		

Table 14. Table of Metric Linear Measure

10 Millimetres	=	1 Centimetre	1 Centimetre	=	·3937 inch
10 Centimetres	=	1 Decimetre	1 Decimetre	=	3·937 inches
10 Decimetres	=	1 Metre	1 Metre	=	39·37 inches

Table 15. Drill Sizes with Decimal Equivalents

mm	Decimal Equiv.	Inch, No. or Letter	mm	Decimal Equiv.	Inch No. or Letter	mm	Decimal Equiv.	Inch, No. or Letter
1.0	·0394		2.75	·1083		5.4	·2126	
	·0400	60		·1094	$\frac{7}{64}$		·2130	3
	·0410	59		·1100	35	5.5	·2165	
1.05	·0413		2.8	·1102			·2188	$\frac{7}{32}$
	·0420	58		·1110	34	5.6	·2205	
	·0430	57	2.85	·1122			·2210	2
1.1	·0433			·1130	33	5.7	·2244	
1.15	·0453		2.9	·1142			·2280	1
	·0465	56		·1160	32	5.8	·2283	
	·0469	$\frac{3}{64}$	2.95	·1161		5.9	·2323	
1.2	·0472		3.0	·1181			·2340	A
1.25	·0492			·1200	31		·2344	$\frac{15}{64}$
1.3	·0512		3.1	·1220		6.0	·2362	
	·0520	55		·1250	$\frac{1}{8}$		·2380	B
1.35	·0532		3.2	·1260		6.1	·2402	
	·0550	54		·1285	30		·2420	C
1.4	·0551		3.3	·1299		6.2	·2441	
1.45	·0571		3.4	·1339			·2460	D
1.5	·0590			·1360	29	6.3	·2480	
	·0595	53	3.5	·1378			·2500	$\frac{1}{4}$, E
1.55	·0610			·1405	28	6.4	·2520	
	·0625	$\frac{1}{16}$		·1406	$\frac{9}{64}$	6.5	·2559	
1.6	·0630		3.6	·1417			·2570	F
	·0635	52		·1440	27	6.6	·2598	
1.65	·0650		3.7	·1457			·2610	G
1.7	·0669			·1470	26	6.7	·2638	
	·0670	51		·1495	25		·2656	$\frac{17}{64}$
1.75	·0689		3.8	·1496			·2660	H
	·0700	50		·1520	24	6.8	·2677	
1.8	·0709		3.9	·1535		6.9	·2717	
1.85	·0728			·1540	23		·2720	I
	·0730	49		·1562	$\frac{5}{32}$	7.0	·2756	
1.9	·0748			·1570	22		·2770	J
	·0760	48	4.0	·1575		7.1	·2795	
1.95	·0768			·1590	21		·2810	K
	·0781	$\frac{5}{64}$		·1610	20		·2812	$\frac{9}{32}$
	·0785	47	4.1	·1614		7.2	·2835	
2.0	·0787		4.2	·1654		7.3	·2874	
2.05	·0807			·1660	19		·2900	L
	·0810	46	4.3	·1693		7.4	·2913	
	·0820	45		·1695	18		·2950	M
2.1	·0827			·1719	$\frac{11}{64}$	7.5	·2953	
2.15	·0846			·1730	17		·2969	$\frac{19}{64}$
	·0860	44	4.4	·1732		7.6	·2992	
2.2	·0866			·1770	16		·3020	N
2.25	·0886		4.5	·1772		7.7	·3031	
	·0890	43		·1800	15	7.8	·3071	
2.3	·0906		4 6	·1811		7.9	·3110	
2.35	·0925			·1820	14		·3125	$\frac{5}{16}$
	·0935	42	4 7	·1850	13	8.0	·3150	
	·0938	$\frac{3}{32}$		·1875	$\frac{3}{16}$		·3160	O
2.4	·0945		4 8	·1890	12	8.1	·3189	
	·0960	41		·1910	11	8.2	·3228	
2.45	·0965		4.9	·1929			·3230	P
	·0980	40		·1935	10	8.3	·3268	
2.5	·0984			·1960	9		·3281	$\frac{21}{64}$
	·0995	39	5 0	·1968		8.4	·3307	
2 55	·1004			·1990	8		·3320	Q
	·1015	38	5.1	·2008		8 5	·3346	
2 6	·1024			·2010	7	8.6	·3386	
	·1040	37		·2031	$\frac{13}{64}$		·3390	R
2.65	·1043			·2040	6	8 7	·3425	
2.7	·1063		5 2	·2047			·3438	$\frac{11}{32}$
	·1065	36		·2055	5	8 8	·3465	
			5 3	·2087			·3480	S
				·2090	4	8 9	·3504	

Table 15. Drill Sizes with Decimal Equivalents continued

mm	Decimal Equiv.	Inch, No. or Letter	mm	Decimal Equiv.	Inch, No. or Letter	mm	Decimal Equiv.	Inch, No. or Letter
9.0	·3543		12.3	·4842		18.25	·7185	
	·3580	T		·4844	31/64		·7188	23/32
9.1	·3583		12.4	·4882		18.50	·7283	
	·3594	23/64	12.5	·4921			·7344	47/64
9.2	·3622		12.6	·4961		18.75	·7382	
9.3	·3661		12.7	·5000	1/2	19.00	·7480	
	·3680	U	12.8	·5039			·7500	3/4
9.4	·3701		12.9	·5079		19.25	·7579	
9.5	·3740		13.0	·5118			·7656	49/64
	·3750	3/8		·5156	33/64	19.50	·7677	
	·3770	V	13.1	·5157		19.75	·7776	
9.6	·3780		13.2	·5197			·7812	25/32
9.7	·3819		13.3	·5236		20.00	·7874	
9.8	·3858		13.4	·5276			·7969	51/64
	·3860	W		·5312	17/32	20.25	·7972	
9.9	·3898		13.5	·5315		20.50	·8071	
	·3906	25/64	13.6	·5354			·8125	13/16
10.0	·3937		13.7	·5394		20.75	·8169	
	·3970	X	13.8	·5433		21.00	·8268	
10.1	·3976			·5469	35/64		·8281	53/64
10.2	·4016		13.9	·5472		21.25	·8366	
	·4040	Y	14.0	·5512			·8438	27/32
10.3	·4055		14.25	·5610		21.50	·8465	
	·4062	13/32		·5625	9/16	21.75	·8563	
10.4	·4094		14.50	·5709			·8594	55/64
	·4130	Z		·5781	37/64	22.00	·8661	
10.5	·4134		14.75	·5807			·8750	7/8
10.6	·4173		15.00	·5906		22.25	·8760	
10.7	·4213			·5938	19/32	22.50	·8858	
	·4219	27/64	15.25	·6004			·8906	57/64
10.8	·4252			·6094	39/64	22.75	·8957	
10.9	·4291		15.50	·6102		23.00	·9055	
11.0	·4331		15.75	·6201			·9062	29/32
11.1	·4370			·625	5/8	23.25	·9154	
	·4375	7/16	16.00	·6299			·9219	59/64
11.2	·4409		16.25	·6398		23.50	·9252	
11.3	·4449			·6406	41/64	23.75	·9350	
11.4	·4488		16.50	·6496			·9375	15/16
11.5	·4528			·6562	21/32	24.00	·9449	
	·4531	29/64	16.75	·6595			·9531	61/64
11.6	·4567		17.00	·6693		24.25	·9547	
11.7	·4606			·6719	43/64	24.50	·9646	
11.8	·4646		17.25	·6791			·9688	31/32
11.9	·4685			·6875	11/16	24.75	·9744	
	·4688	15/32	17.50	·6890		25.00	·9842	
12.0	·4724		17.75	·6988			·9844	63/64
12.1	·4764			·7031	45/64	25.25	·9941	
12.2	·4803		18.00	·7087			1·0000	

The Surface Plate

The illustration shows a surface plate, a flat cast iron plate used to test plane surfaces in the building of fine machinery. A very thin film of such pigment as prussian blue or red lead is coated on the surface plate and then the surface to be tested is rubbed over it. The surface is then re-tested and re-scraped until it proves satisfactory. Toolmakers also use surface plates to locate and check holes in jig plates, laying out work, testing, inspecting and similar operations. A dial gauge and a set of slip gauges enable very accurate work to be done on surface plates.

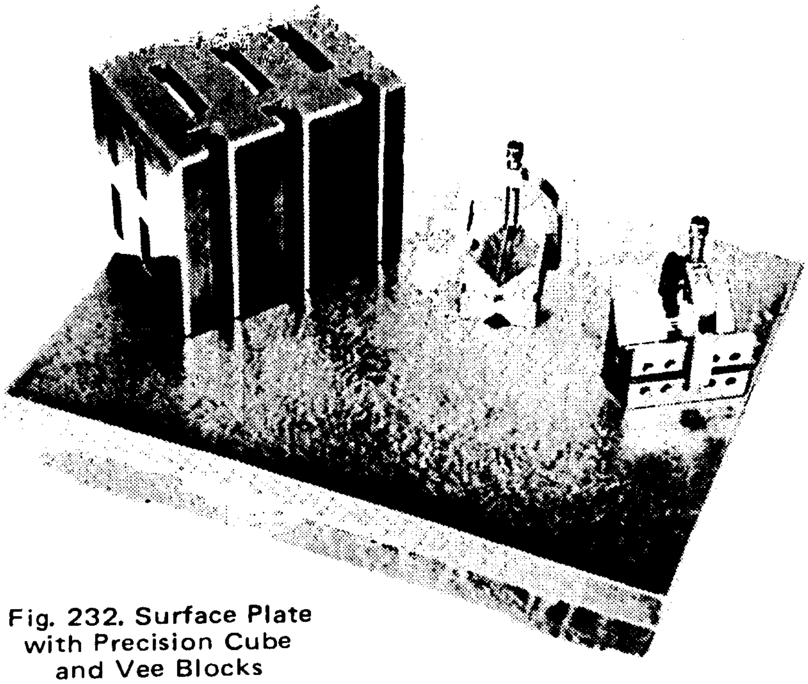


Fig. 232. Surface Plate with Precision Cube and Vee Blocks

Saddle Limit Switch

An adjustable saddle limit switch can be fitted to any Boxford machine equipped with No-volt release starter.

This can be set to provide both electrical cut-out through the No-volt release starter and a mechanical stop, so that the saddle cannot be traversed too close to the headstock. It is a very useful safety device which is proving very popular

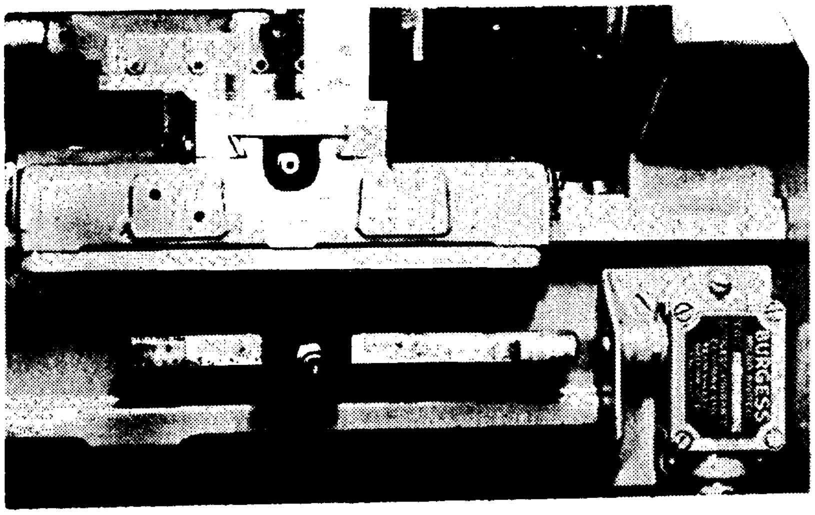


Fig. 233. Saddle Limit Switch

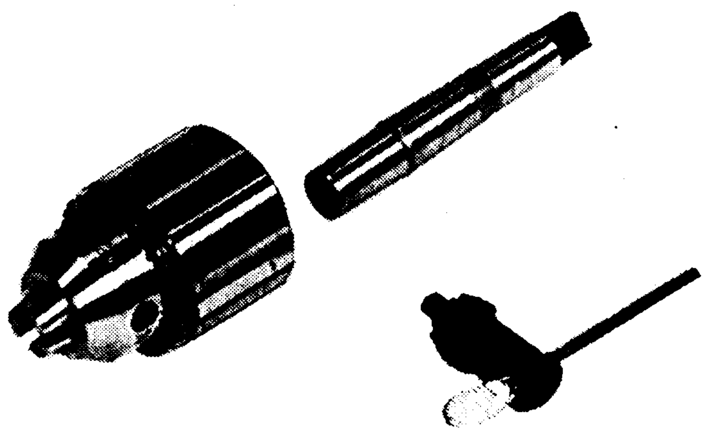


Fig. 236. Drill Chuck with Detachable No. 2 Morse Taper Shank

in training establishments.
Fig. 237 shows the Boxford Quick Change Toolpost. Toolholders may be clamped in any of the two positions by the eccentric clamping arrangement. Each toolholder is provided with a knurled adjusting sleeve and locking screw which enables rapid and simple setting of the tool height.

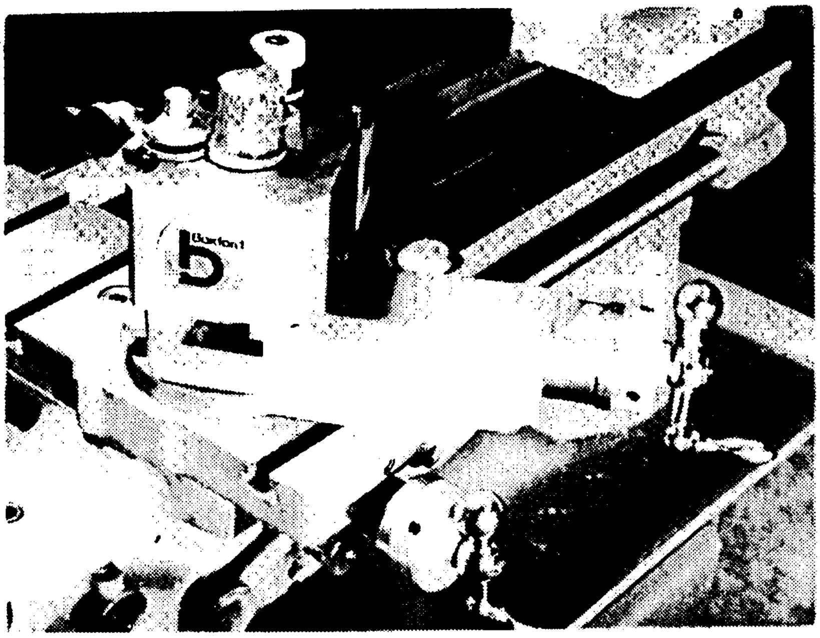


Fig. 237. "Boxford" Quick Change Toolpost

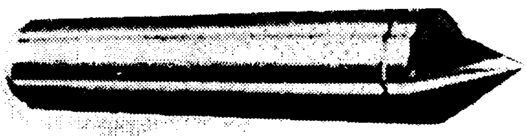


Fig. 239. A Half Centre

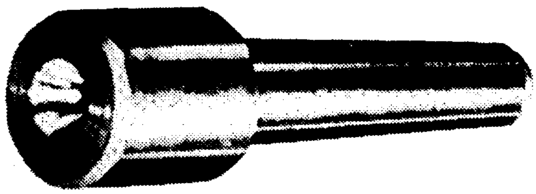


Fig. 240. A Female Centre

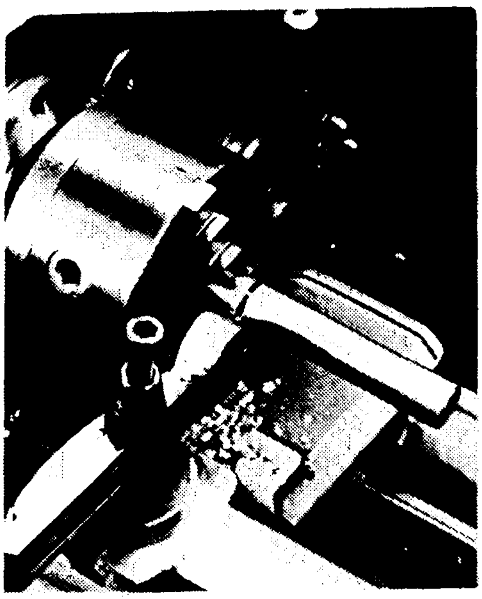


Fig. 238. Parting-off a Shaft in the Lathe

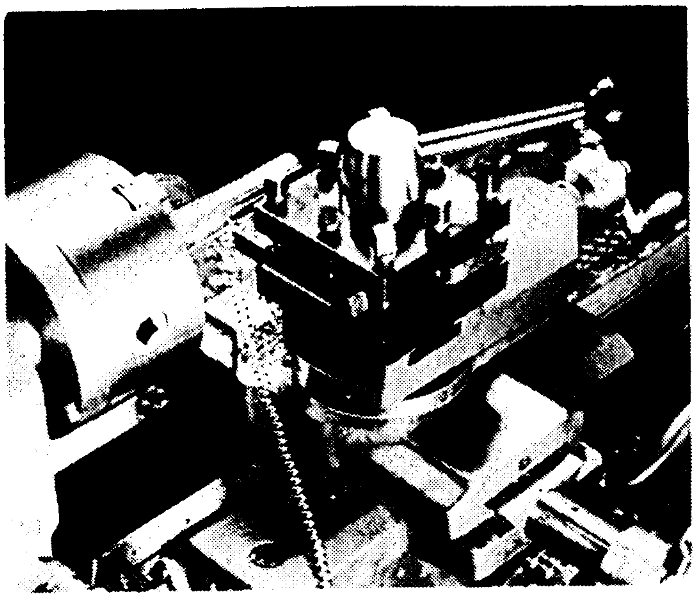


Fig. 241. A 4-Way Tool Post in use on the Lathe

A Six-station turret head, illustrated, can be supplied for use with Boxford Lathes.
This piece of equipment can with special tooling make a precision, high production, repetition machine and in conjunction with the cutting-off slide Fig. 243, a very useful and economic unit for large output of components in factories and workshops.

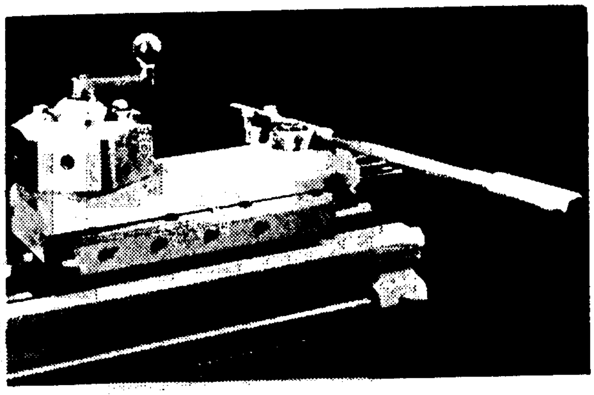


Fig. 242. Hand Lever Turret Head Attachment

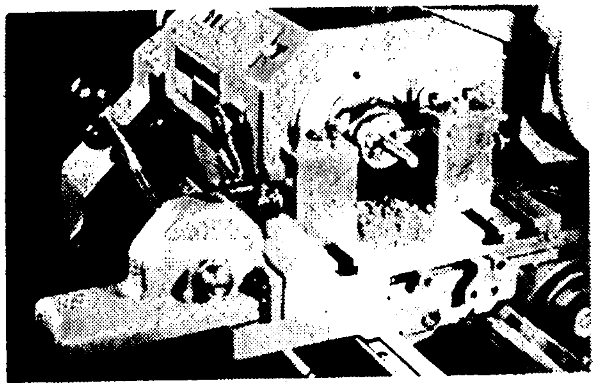


Fig. 243. Cutting-off Slide

Coolant Pump

The coolant tank on all Boxford cabinet bases is built into the centre section of the base. On recent machines access to this is from the rear, on earlier models access is from the front.

The electric pump is wired up to a switch on the control panel situated at the front of the machine, coolant flow and direction can be adjusted by the tap on the coolant fitting attached to the lathe saddle.

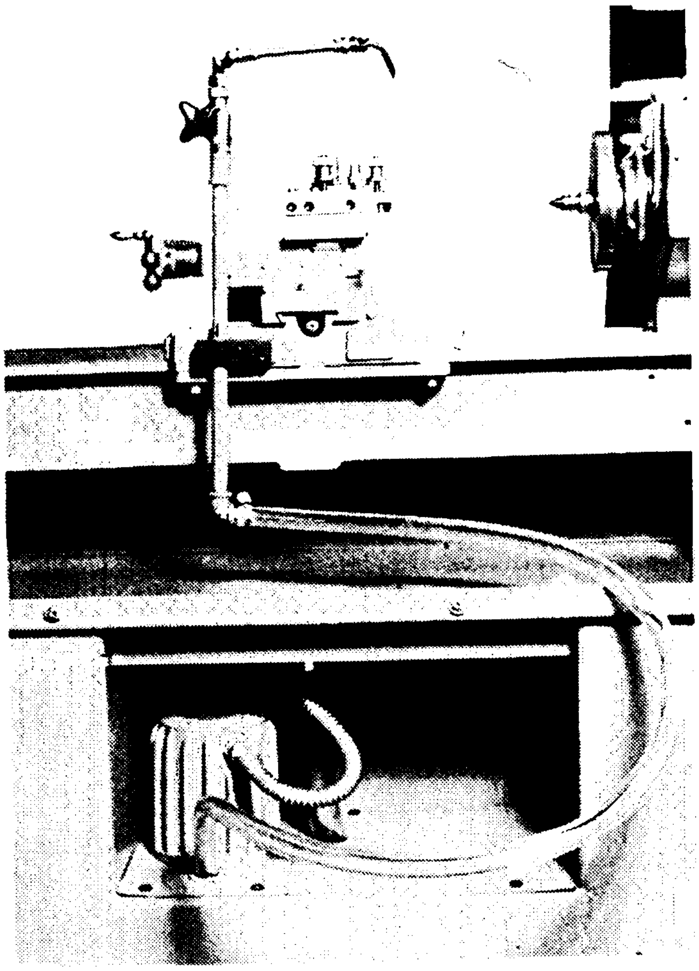


Fig. 244. Rear View of Cabinet showing pump and fittings (Rear splash guard removed)

Index

Index

Index

LIST OF TABLES