HOW TO RUN A LATHE

SOUTH BEND LATHE, INC.

PREFACE

ONE of the great needs of industry today is well trained workmen who are skillful with their hands and also trained to think about their work, diagnose troubles and suggest improvements. No man can hope to succeed in any line of work unless he is willing to study it and increase his own ability in it.

It is the purpose of this book to aid the beginner or apprentice in the machine shop and the student in the school shop to secure a better understanding of the fundamentals of the operation of a modern Screw Cutting Engine Lathe. In illustrating and describing the fundamental operations of modern lathe practice, we have made an effort to show only the best and most practical methods of machine shop practice in use in modern industries in the United States.

We are indebted to so many manufacturers, engineers, authors, educators, mechanics and friends for assistance in the preparation of this book that it would be impossible to give them individual mention here. However, we wish to express our appreciation for the cooperation that has made this work possible.

55th Edition of "How to Run a Lathe"

This 55th edition of the book "How to Run a Lathe" is printed in English. Other editions are printed in Spanish and Portuguese. The first edition, in English, was printed in 1907. Each succeeding edition has been revised and improved.

SOUTH BEND LATHE, INC.

How to Run a Lathe

REVISED EDITION 55

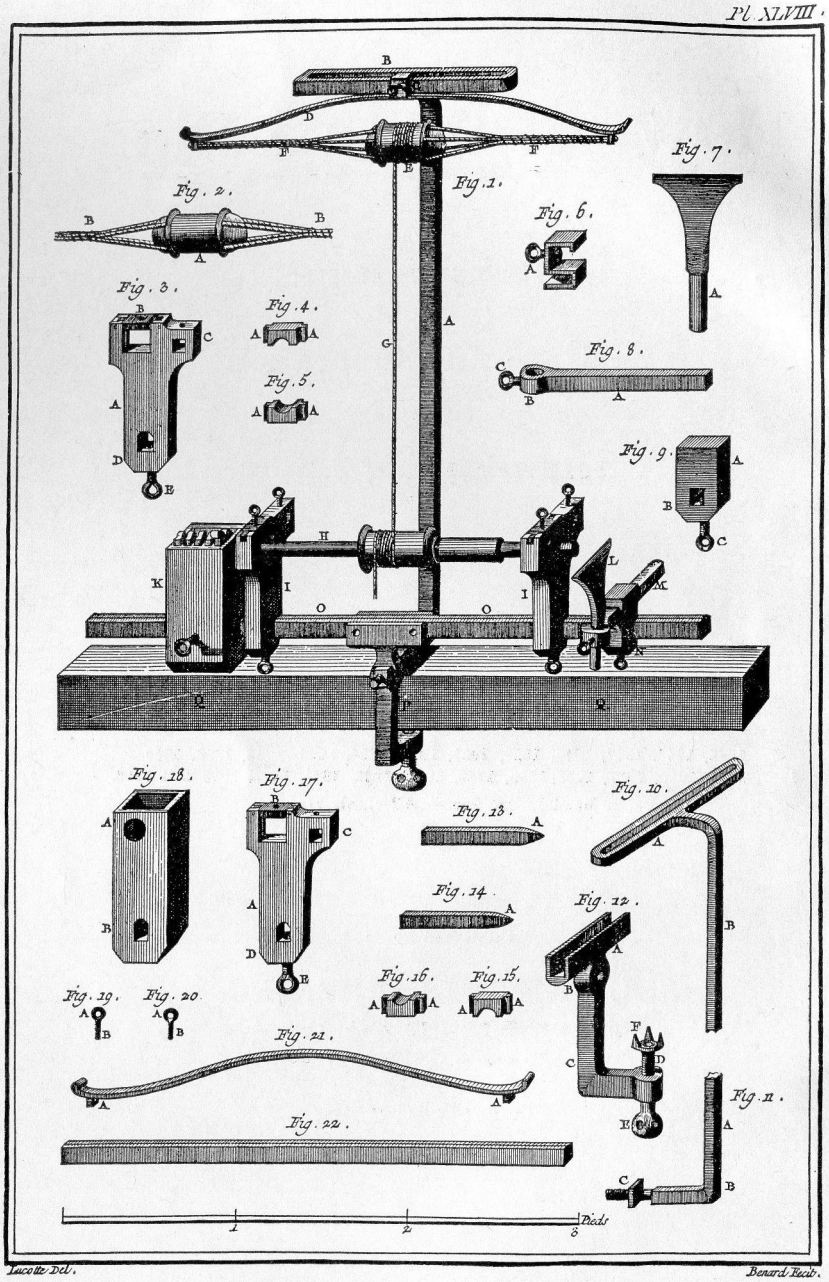
The Care and Operation of A Screw-Cutting Lathe

© 1914, 1916, 1917, 1919, 1920, 1921, 1922, 1924, 1926, 1928, 1929, 1930
1932, 1934, 1935, 1937, 1938, 1939, 1940, 1941, 1942, 1944, 1952, 1956 and 1958.
South Bend Lathe, Inc. — All rights reserved

Published by

SOUTH BEND LATHE, INC.
SOUTH BEND 22, INDIANA, U. S. A.
ESTABLISHED 1906 — INCORPORATED 1914

Printed in U.S.A.



Lucotte Del.

Benard Recit.

Tourneur, *Tours d'Horlogers.*

A small lathe built in France about 1770. From an old etching

Chapter I

HISTORY AND DEVELOPMENT OF THE SCREW CUTTING LATHE

The screw cutting engine lathe is the oldest and most important of machine tools and from it all other machine tools have been developed. It was the lathe that made possible the building of the steamboat, the locomotive, the electric motor, the automobile and all kinds of machinery used in industry. Without the lathe our great industrial progress of the last century would have been impossible.



Fig. 1. Early Tree Lathe

Early Screw Cutting Lathe

One of the earliest types of turning lathes was the tree lathe, shown in Fig. 1. A rope attached to a flexible branch overhead was passed around the work to revolve it. Later a strip of wood or "lath" was used to support the rope, and this is probably why the turning machine came to be known as a "lathe."

One of the earliest screw cutting lathes that we have record of was built in France about 1740. Fig. 2 shows this lathe as it was illustrated in a book published in 1741. A hand crank was attached direct to the headstock spindle. The spindle of this lathe was geared to the lead screw, but there was no provision for changing the gears for cutting various pitches of screw threads.

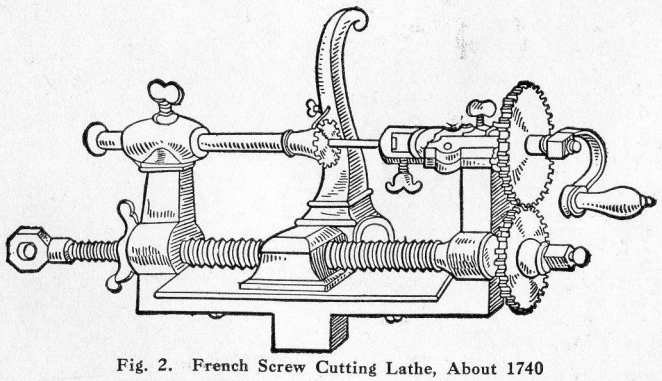


Fig. 2. French Screw Cutting Lathe, About 1740

(From Roe's ENGLISH AND AMERICAN TOOL BUILDERS; by permission of the publishers, McGraw-Hill Book Company, Inc.)

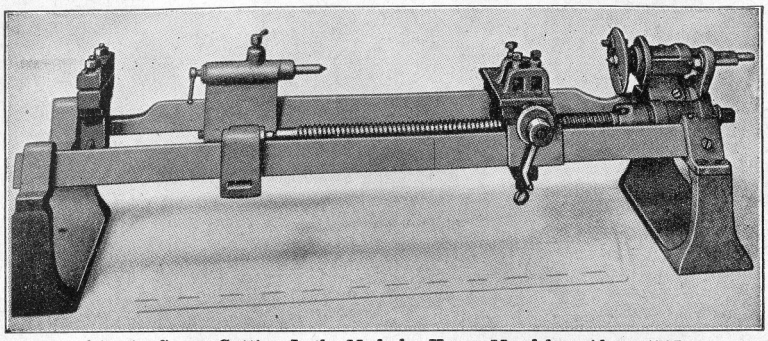


Fig. 3. Screw Cutting Lathe Made by Henry Maudslay, About 1797
(Courtesy Joseph Wickham Roe, Author of "English and American Tool Builders")

Henry Maudslay

Henry Maudslay, an Englishman, gave us the fundamental principles of the screw cutting engine lathe in a small lathe which he designed and built about 1797. On this lathe the gears used to connect the spindle with the lead screw could be changed, permitting the use of different gear ratios for cutting various pitches of screw threads.

Early American Lathes

Lathes were built in the United States between 1800 and 1830 with wood beds and iron ways. In 1836 Putnam of Fitchburg, Massachusetts built a small lathe with a lead screw. In 1850 iron bed lathes were made in New Haven, Connecticut, and in 1853 Freeland in New York City built a lathe, estimated 20 in. swing x 12 ft. bed, with iron bed and back geared head.

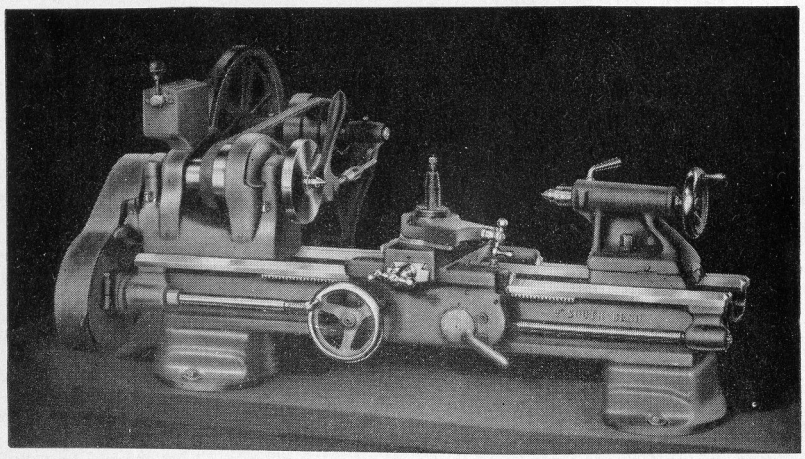


Fig. 4. A Modern Standard Change Gear Bench Lathe

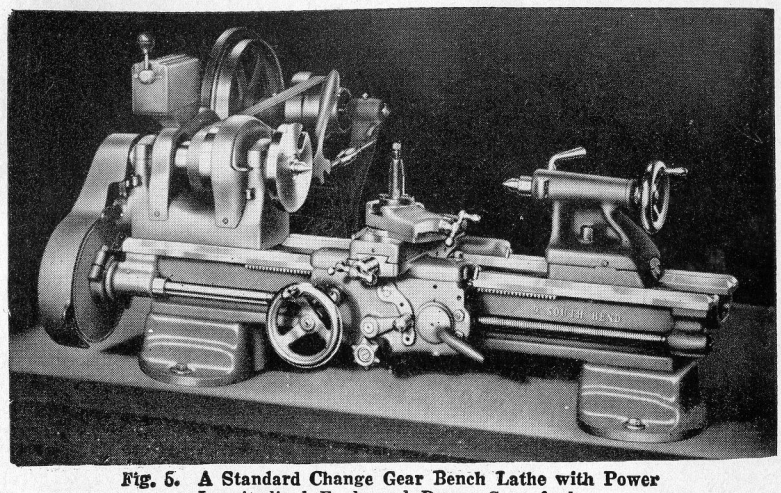


Fig. 5. A Standard Change Gear Bench Lathe with Power Longitudinal Feeds and Power Cross-feeds

The Modern Standard Change Gear Lathe

Modern bench lathes are shown in Figs. 4 and 5. These are known as standard change gear lathes because they have a set of independent change gears which are used to connect the headstock spindle with the lead screw for cutting various pitches of screw threads and for obtaining a series of power longitudinal turning feeds. The cross-feed for the lathe shown in Fig. 5 may also be operated by power through a friction clutch and series of gears in the apron.

The end view of a standard change gear bench lathe, and the gearing for connecting the headstock spindle with the lead screw of the lathe are shown in Fig. 6. The gears may be arranged so that practically any pitch of screw thread may be cut. The change gears are also used for obtaining a wide range of power cross feeds and power longitudinal feeds for turning and facing operations.

The standard change gear type of lathe is popular in the small shop, as it is less expensive than the quick change gear type of lathe. It is also widely used in industrial plants for production operations where few changes of threads and feeds are necessary. For this class of work the standard change gear lathe has an advantage in that when set up with the correct feeds for an operation the adjustments are not as easily tampered with and changed as they are on the quick change gear lathe.

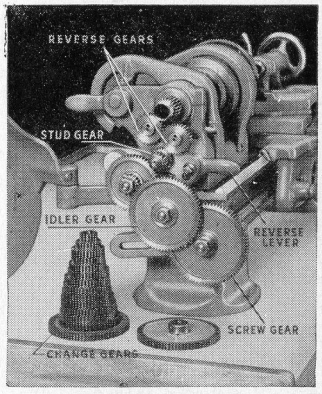


Fig. 6. End View of Standard Change Gear Lathe

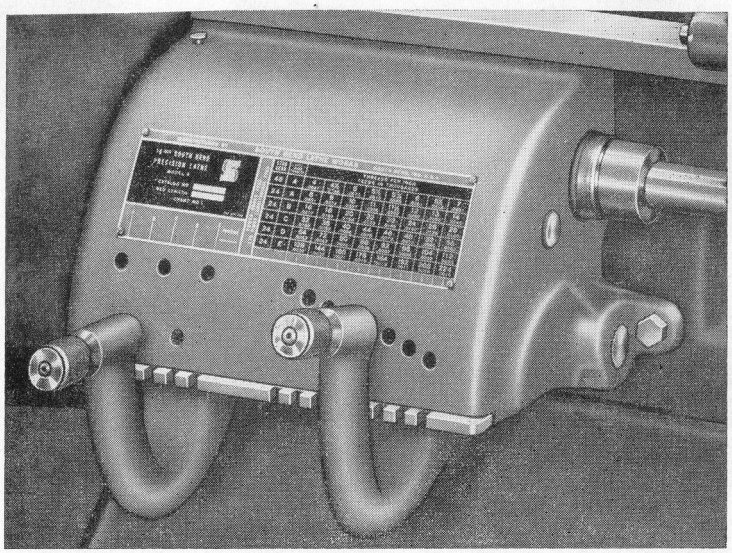


Fig. 7. A Modern Quick Change Gear Box

Quick Change Gear Lathe

A quick change gear lathe is one in which the gearing between the spindle and lead screw is so arranged that changes for obtaining various pitches of screw threads may be made through a quick change gear box without having to change loose gears.

Fig. 7 shows a modern quick change gear box. The quick change gear mechanism is attached to the left end of the lathe and provides a series of 48 changes for cutting screw threads from 4 to 224 per inch, also a wide range of power feeds for turning, boring, and facing. See page 25.

The quick change gear type of lathe is popular in busy shops where frequent changes of threads and feeds must be made such as in tool and die work, general repair and maintenance, and for some production operations.

Quick Change Gear Box

The interior of the quick change gear box is shown in Fig. 8. The gears in this gear box are shifted by levers operated from the front of the lathe and replace the independent change gears used on the standard change gear type of lathe.

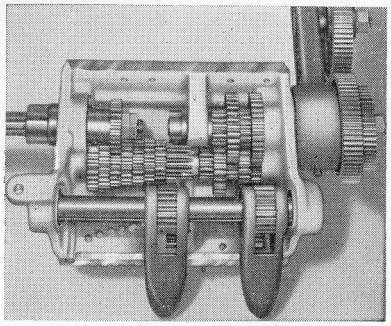
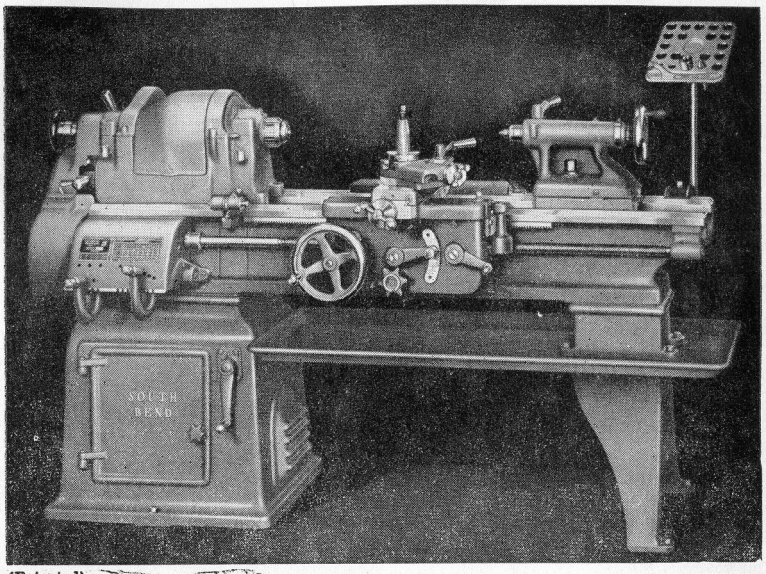


Fig. 8. Interior of Quick Change Gear Box



(Patented)

Fig. 9. A Modern Underneath Motor Driven Toolroom Lathe

Toolroom Lathe

The Toolroom Lathe is the most modern type of back-geared screw cutting lathe. It is supplied with underneath belt motor drive and quick change gear box, as shown in Fig. 9. Toolroom lathes are given special accuracy tests during the process of manufacture and are equipped with taper attachment, thread dial indicator, draw-in collet chuck attachment, collet rack, chip pan, and micrometer carriage stop. These attachments greatly increase the usefulness of the lathe.

The Toolroom Precision Lathe, as its name implies, is used in the toolrooms of industrial plants for making fine tools, test gauges and thread gauges, fixtures, jigs, etc., for making and testing the products manufactured.

A typical toolroom job, the making of a set of master thread gauges, is shown at right in Fig. 10. A threaded plug gauge for checking internal threads is being finished in the lathe and the round object in the lower left-hand corner is the threaded ring gauge for checking external threads.

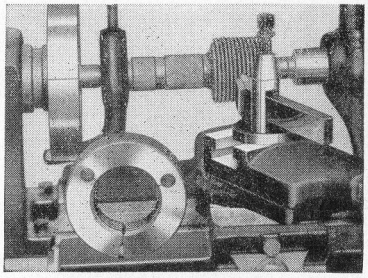
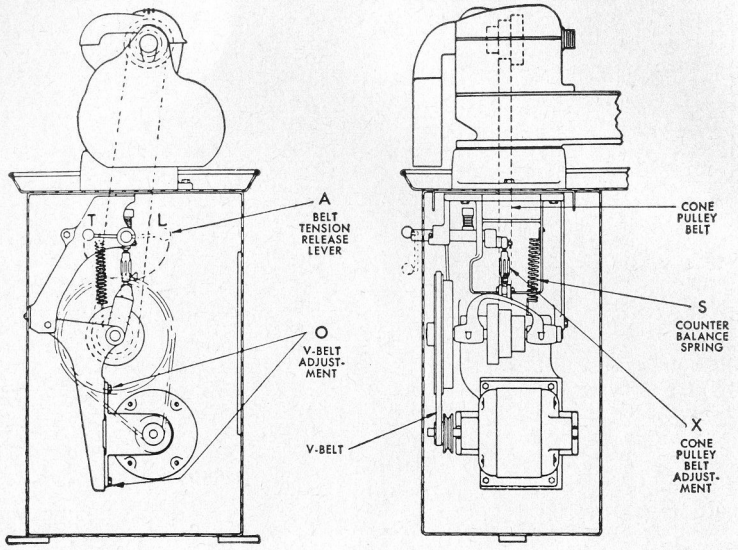
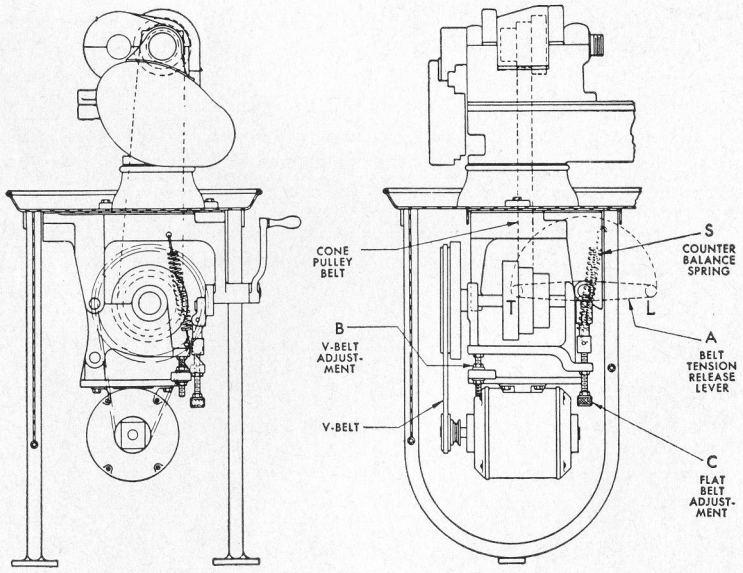


Fig. 10. A Typical Toolroom Job



End View Front View

Fig. 11. Underneath Belt Motor Drive for 9″ and Light Ten Lathes



End View Front View

Fig. 12. Underneath Belt Motor Drive for 10″ - 1″ Collet Lathe

The Underneath Belt Motor Drive

The modern underneath belt motor drive shown in Figs. 11, 12, 13, and 14 is an efficient and practical direct drive equipment for a back-geared screw cutting lathe. This drive is unusually compact and is silent, powerful and economical in operation.

The motor and driving mechanism are fully enclosed in the cabinet leg underneath the lathe headstock. There are no exposed pulleys, belts or gears and no overhead belts or pulleys to obstruct vision or cast shadows upon the work.

Power is transmitted from the motor to the countershaft by V-belt and from the countershaft up through the lathe bed to the headstock cone pulley by a flat leather belt.

Adjustments "B" and "C" are provided for taking up belt stretch and for obtaining any desired tension on the motor belt and cone pulley belt. A belt tension release lever "A" conveniently located on the front of the cabinet leg permits easy shifting of the cone pulley belt. A hinged cover encloses the headstock cone pulley when the lathe is in operation. See page 19.

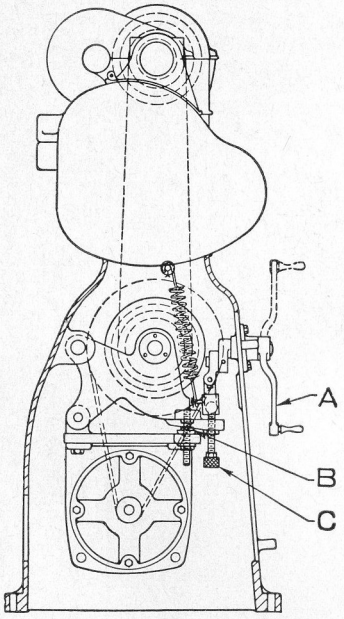


Fig. 13. Cross Section End View of Underneath Belt Motor Drive

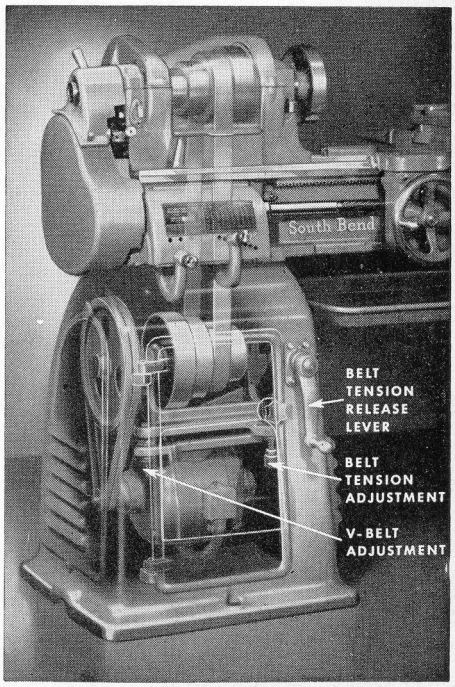


Fig. 14. Phantom View of Underneath Belt Motor Drive

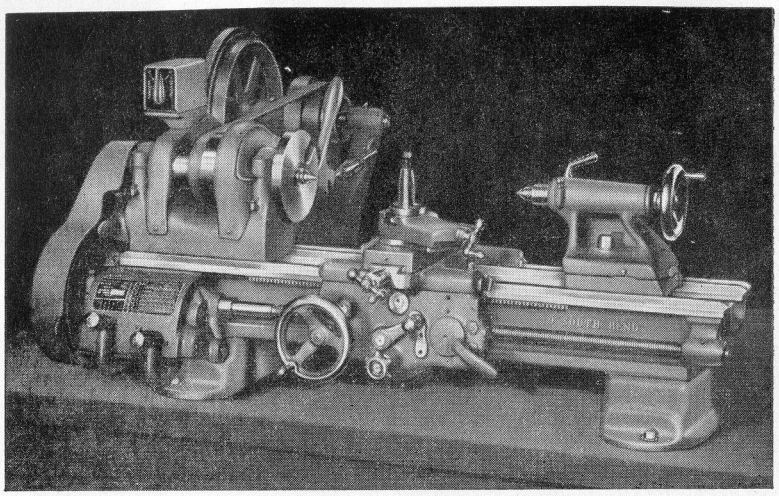


Fig. 15. A 9-inch Model A Bench Lathe with Adjustable Horizontal Motor Drive

Horizontal Motor Drive for Bench Lathes

The illustration above shows a 9-inch swing bench lathe equipped with an adjustable type horizontal motor drive unit. This is one of the most practical types of direct motor drive for a bench lathe.

The construction of the drive is shown below in Fig. 16. Belt tension adjustments "A" and "B" are provided for adjusting the tension of the cone pulley belt and the motor belt. A belt tension release lever "C" permits releasing the cone pulley belt tension so that the belt may easily be shifted from one step of the cone pulley to another. A flat leather belt is usually used between the cone pulleys and a V-belt is used between the motor pulley and the countershaft pulley.

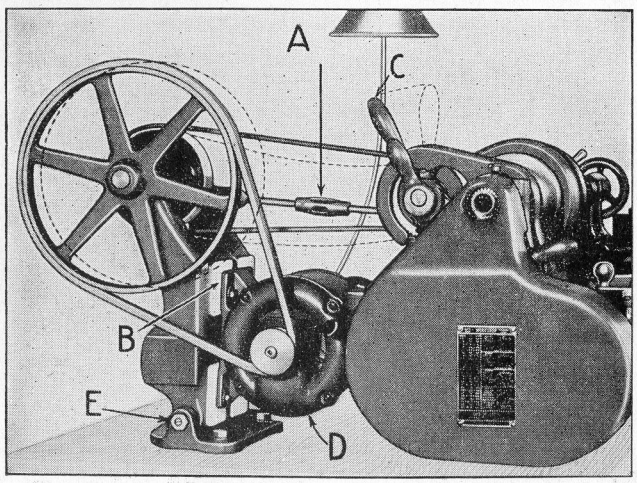


Fig. 16. End View of a Bench Lathe with Adjustable Horizontal Motor Drive

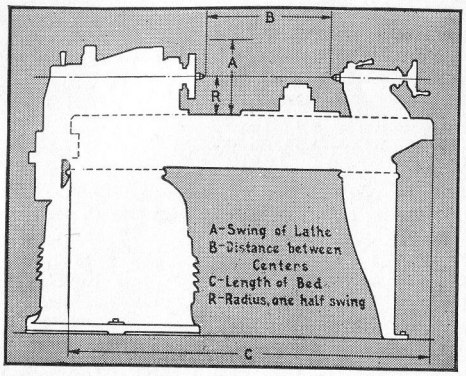


Fig. 17. Size and Capacity of a Lathe

Size and Capacity of the Lathe

In the United States the size of a Screw Cutting Lathe is designated by the swing over bed and the length of bed, as indicated in Fig. 17 above. For example, a 16 in. x 8 ft. lathe is one having a swing over the bed "A" sufficient to take work up to 16 in. in diameter and having a bed length "C" of eight feet.

European tool manufacturers designate the size of a lathe by its radius "R" or center height. For example, an 8-in. center lathe is a lathe having a radius of eight inches. What the European terms an 8-in. center lathe, the American calls a 16-in. swing lathe.

The swing over the tool rest of the lathe is less than the swing over the bed, and the maximum distance between centers "B" is less than the length of the bed. These figures must be considered carefully as they determine the size of work that can be machined between centers.

Selecting a Lathe for the Shop

When selecting a lathe, the most important point to consider is the size of the work. The lathe should be large enough to accommodate the various classes of work that will be handled. This is determined by the greatest diameter and length of work that will be machined in the lathe. The lathe selected should have a swing capacity and distance between centers at least 10% greater than the largest job that will be handled.

Types of Lathes for Various Classes of Work

If the lathe you require is a large one, 13-in. swing or more, the floor leg type is recommended. If the lathe needed is of 9-in. or 10-in. swing, either a bench lathe or a floor leg lathe may be selected. Floor leg lathes are usually more rigid than a lathe mounted on a bench because the heavy cast iron legs provide a sturdy, heavy support. If a bench lathe is used, the bench should be sturdy and rigid and should have a top of 2-in. lumber.

Type of Drive for the Lathe

The high operating efficiency of the modern electric motor has made the individual motor drive the most practical lathe drive in every respect. The lathe is more directly and easily controlled by the operator and, when an instant reversing motor is used, greater efficiency and convenience are obtained.

The overhead countershaft or group drive, which was formerly extensively used, is generally being replaced in modern industrial plants by the direct or individual motor drive because of lower installation and maintenance costs, greater safety for the operator, and the over-all improved shop conditions.

The individual drive makes possible the most efficient arrangement of shop equipment because it permits placing any machine at any angle in any location.

Change Gear Equipment

Quick change gear lathes are preferred in busy shops where frequent changes of threads and feeds are required. Standard change gear lathes are used in production shops on jobs that do not require many changes for threads or feeds, also in small shops that do not have a great deal of lathe work.

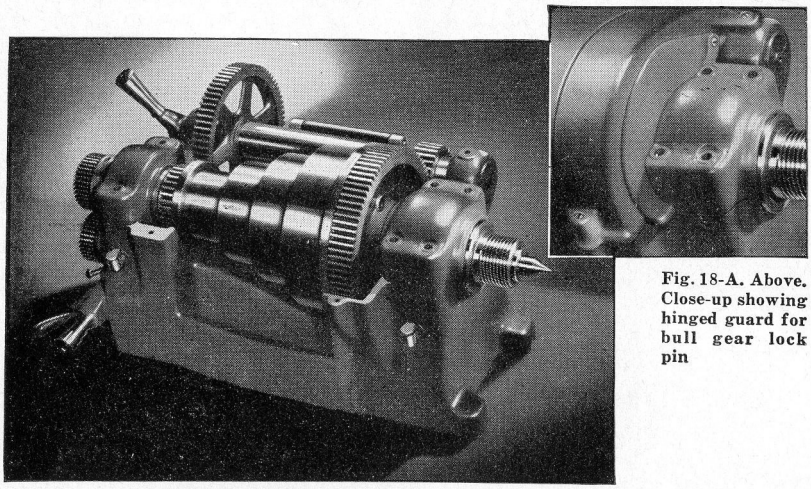


Fig. 18. Back-Geared Headstock, Gear Guards Removed

Fig. 18-A. Above. Close-up showing hinged guard for bull gear lock pin

Features the Lathe Should Have

In considering a metal working lathe for the shop it is well to bear in mind that the lathe will be used for many classes of work and that if carefully selected it should give years of satisfactory service.

The Headstock

The headstock is the most important unit of the lathe and should be back-geared, as shown in Fig. 18. The back gears provide the slow spindle speeds and power required for taking heavy cuts on large diameter work. Modern lathes are equipped with back gears having a quick acting bull gear lock which permits engaging or disengaging the back gears without using a wrench.

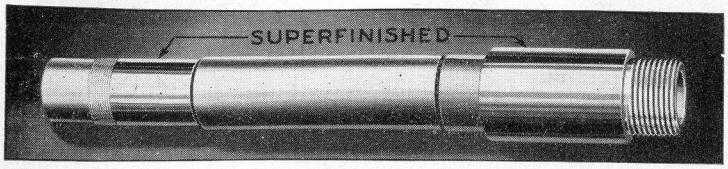


Fig. 19. Hardened Alloy Steel Headstock Spindle with Superfinished Spindle Bearing Surfaces

Headstock Spindle and Bearings

The lathe headstock spindle should be made of a good quality alloy spindle steel, and for best service should be heat treated after it is machined, and all bearing surfaces, including the taper hole, should be carburized, hardened and ground.

The journal bearing surfaces on the spindle should be "superfinished" to a smoothness of five microinches (.000005")*. When equipped with a superfinished spindle, precision bearing inserts, and proper lubrication facilities, the lathe can be operated at the high speeds essential for the efficient use of modern tungsten-carbide tipped cutter bits and the machining of plastics without danger of overheating or scoring the spindle bearings.

Lathe Bed Construction

The lathe bed is the foundation on which the lathe is built, so it must be substantially constructed and scientifically designed. Fig. 20 shows an end view of a lathe bed, which is an example of modern design.

Prismatic V-ways have been found to be the most accurate and serviceable type of ways for lathe beds and have been adopted by most of the leading machine tool builders. The two outer V-ways (1 and 4) guide the lathe carriage, while the inner V-way and flat way (2 and 3) align headstock and tailstock.

The V-ways of the lathe bed are carefully precision finished so that the headstock, carriage and tailstock are perfectly fitted and aligned parallel to the axis of spindle the entire length of bed.

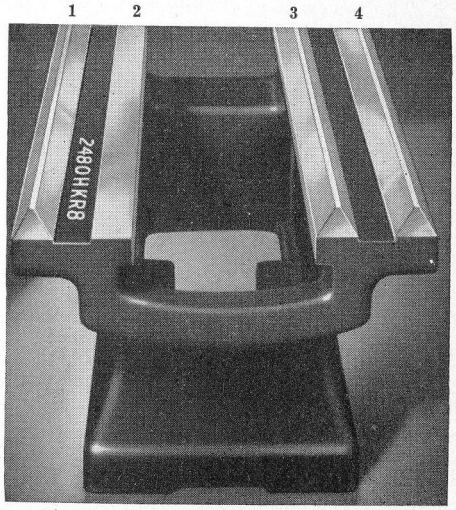


Fig. 20. End View of Lathe Bed Showing Prismatic V-Ways

*Measurements in microinches *rms*.

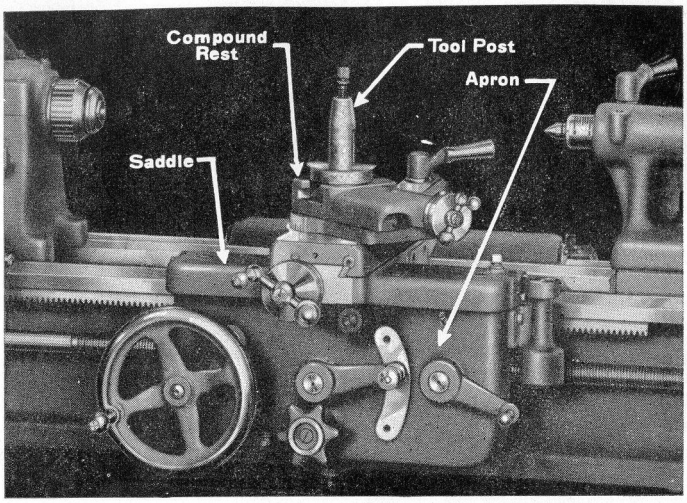


Fig. 21. A Well Designed Lathe Carriage

The Lathe Carriage

The lathe carriage includes the apron, saddle, compound rest and tool post. Since the carriage supports the cutting tool and controls its action, it is one of the most important units of the lathe. The carriage shown in Fig. 21 is modern and practical.

The apron is of double wall construction with all gears made of steel. A powerful multiple disc clutch is provided for driving the power feeds. An automatic safety device prevents the half nuts and automatic feeds from being engaged at the same time.

The threads of the lead screw are used only for thread cutting. A spline in the lead screw drives a worm in the apron which operates the power carriage feeds.

Interior of Apron

The interior of the apron is shown in Fig. 22 at right. The spline in the lead screw which drives the worm for operating the power longitudinal feeds and power cross feeds is clearly shown.

The half nuts for thread cutting are dovetailed into the back wall of the apron.

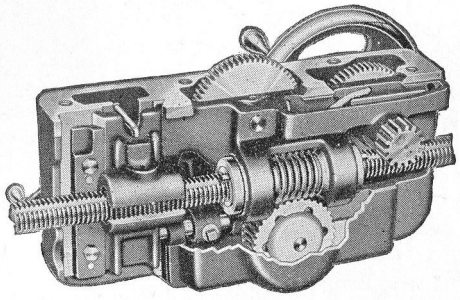


Fig. 22. Interior View of Double Wall Apron

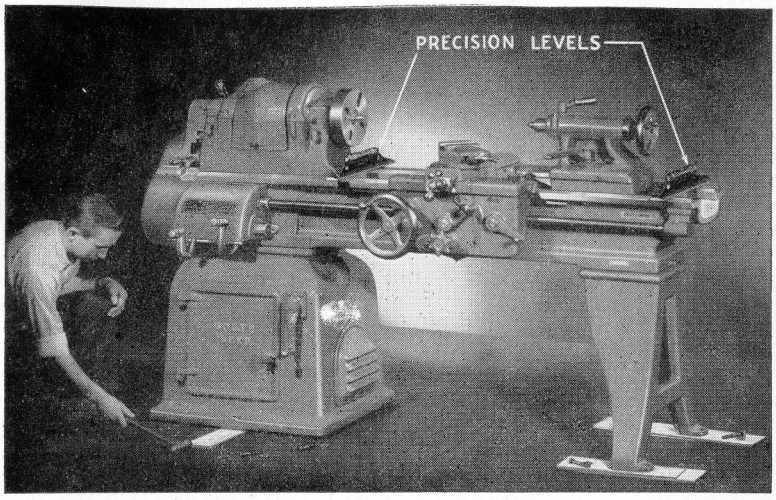


Fig. 23. Leveling the Lathe

Chapter II

SETTING UP AND LEVELING THE LATHE

A new lathe should be very carefully unpacked and installed so that all of the fine accuracy that has been built into the lathe by the manufacturer will be retained.

Do not allow a hammer or crow bar to strike the lathe while unpacking as this may cause serious damage. Look carefully in all packing material for small parts, instruction material, etc. Study all reference books and instruction sheets carefully before setting up the lathe.

Clean the new lathe thoroughly with a stiff brush and kerosene. Wipe with a clean cloth and then immediately cover all unpainted surfaces with a film of good machine oil to prevent rusting. Wipe off the old oil occasionally and do not allow dust, chips or dirt to accumulate. Cover the lathe with a plastic or canvas service cover when not in use. Keep the finished surfaces clean and well oiled and the lathe will retain its new appearance.

Solid Floor Required

It is very important that the lathe be set on a solid foundation and that it is carefully and accurately leveled. An erection plan showing how to set up and level the lathe is included in the shipment of the lathe. For best results the lathe should be set on a concrete foundation. A wood floor should be braced to prevent sagging and vibration if it is not substantially constructed.

The lathe may be leveled by placing shims of hard wood or metal under the legs, as shown in Fig. 23. If the lathe is not leveled it will not set evenly on all four legs and the weight of the lathe will cause the lathe bed to be twisted, throwing the headstock out of alignment with the V-ways of the bed and causing the lathe to turn and bore taper. If the lathe is not level it cannot turn out accurate work.

Use a Precision Level for Leveling the Lathe

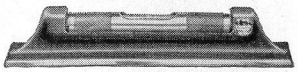


Fig. 24. A Precision Level

Use a precision level that is at least twelve inches long and sufficiently sensitive to show a distinct movement of the bubble when a .003 in. shim is placed under one end of the level. Level across the lathe bed at both the headstock end and the tailstock end, as shown in Fig. 23, page 15.

Bolt the Lathe to the Floor

Use lag screws or bolts to secure the lathe to the floor. If the lathe is set on a concrete floor or foundation, mark the location of the bolt holes and drill holes in the concrete with a star drill. Use expansion bolts or set bolts in melted lead or melted sulphur. Check the leveling of the lathe after bolting the lathe to the floor or bench.

Leveling Underneath Belt Motor Driven Lathes

When placing shims under the cabinet leg at the headstock end of the Underneath Motor Drive Lathe, use the shims only at the bolt pads. There should be clearance under the cabinet leg all the way round except at the two pads where the bolts go through the leg into the floor.

Bench Lathes

Bench Lathes should be mounted on a substantial bench, providing rigid support and leveled as outlined above. The bench top should be about 28 in. high and if made of wood should be of 2-in. lumber. The bench should be securely bolted to the floor so there will be no danger of the bench shifting and throwing the lathe out of level. Some bench lathes have leveling screws in the right leg which may be used for making the final leveling adjustments.

Readjust Shims

It may be necessary to readjust the shims under the lathe legs from time to time to compensate for settling of the building, even if the lathe is set on a concrete floor. For this reason the legs should not be bedded in concrete but should be bolted to the floor.

If at any time the lathe does not bore a straight hole, this is an indication that the lathe is no longer perfectly level and the shims should be readjusted.

Checking the Leveling of Lathe

After leveling the lathe, place a bar of steel one inch or larger in diameter in the chuck and machine two collars of equal diameter three or four inches apart, as shown in Fig. 25. Take a very light finishing cut across both of these collars without changing the adjustment of the cutter bit. Measure the diameter of each collar carefully with a micrometer.

If the collars are not the same diameter, this is an indication that the level used in setting up the lathe was not sufficiently sensitive. The leveling may be perfected by adjusting the shims under the front and back legs at the tailstock end of the lathe until the collars on the test piece are turned the same diameter.

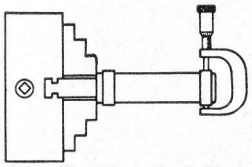
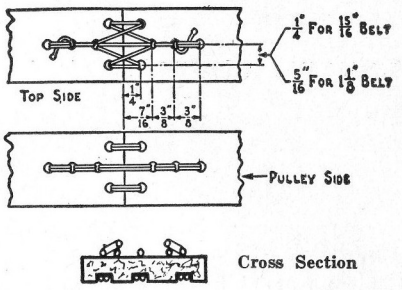


Fig. 25. Method of Testing Leveling of Lathe

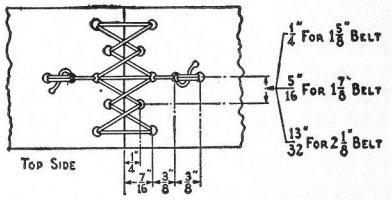
Belts for the Lathe Drive

Good quality flat leather belts are best for use on the lathe cone pulleys. They have sufficient elasticity to transmit power efficiently and will give excellent service without slipping if they are kept clean and the tension is properly adjusted as instructed on page 19. The smooth side of the belt should run next to the pulleys.

V-belts of correct specifications and of good quality should be used where required. Correct tension of V-belts is of greatest importance. Never adjust V-belts tighter than required to pull the load, or run so loose that they creep or slip. Either condition causes V-belt failure.



Spacing of Holes for 15/16" and 1-⅛" Belts



Spacing of Holes for 1-⅝", 1-⅞", and 2-⅛" Belts

Groove Pulley Side for Gut Lacing

Fig. 26. Specifications for Laced Belt Splice

Lacing Leather Belts

Leather belts may be joined by lacing with gut or rawhide thongs, as shown in Figs. 26 and 27.

When measuring for belt length the belt tension device must be adjusted so the cone pulleys are moved toward each other as far as possible. Determine the exact belt length by carefully measuring around the cone pulleys with a steel tape and then subtracting ⅜". This will give the belt tension enough to drive the lathe without requiring any initial adjustment of the belt tension device on the drive. Trim the ends of the belt square and to this length. Punch or drill ⅛" holes in the belt ends as shown. If a round gut lace is used, cut straight grooves on the pulley side, as shown by the cross section in Fig. 26, so the lace will lay flush with the belt surface and run smoothly over the pulleys.

Place the belt around the cone pulleys, mark the center of lace and place the marked midpoint at "A" as shown in Fig. 27. Being careful not to kink the lace, start one end toward "B" and thread this half of the lace in the direction of the arrowheads and finish at "K." Repeat with the other half of the lace, starting toward "C" and ending at "L." Fasten the ends as shown and burn close with a match to prevent them from pulling out. Fig. 27 shows 5 rows of holes, but the method of lacing remains the same whether there are 3, 5, or 7 rows of holes. Do not cross the lacing on the pulley side.

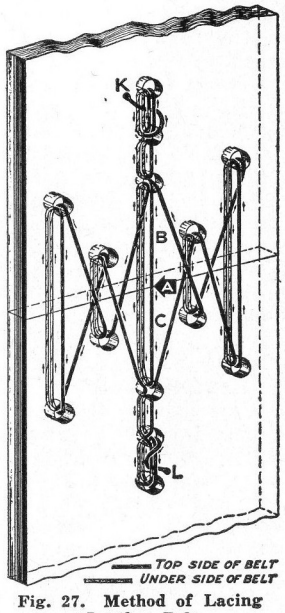


Fig. 27. Method of Lacing Leather Belt

Wire Belt Hooks

There are a number of good wire belt hooks on the market that can be used for splicing belts. Measure for the length of the belt in the same manner as outlined above and then deduct enough more, before cutting, to allow for the connecting pin (See Fig. 28). Wire belt hooks should never be used on any belt which is not completely guarded or which may be shifted while the machine is running.

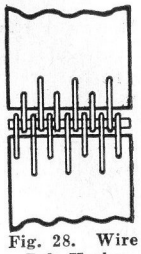


Fig. 28. Wire Belt Hooks

Glued Belt Splice

The cemented endless belt splice is preferred by many mechanics because, when properly made, it is very durable and will run over the pulleys more smoothly than a laced or wire hook splice. A good waterproof type of belt cement should be used. Experience has shown that a belt splice made with good acetone cement is practically permanent and is not subject to ordinary moisture or oil action.

Measure for the length of the belt to be used, in the same way as for the laced splice, then add five inches for the lap-over to be cemented. After the belt ends have been cut square and to length, the five inch overlapping ends are tapered as shown by the drawing, Fig. 29. Use a belt shave, small plane, or sharp knife to cut smooth, uniform tapers. Prepare two blocks of soft one inch wood, a little wider than the belt, and six inches long.

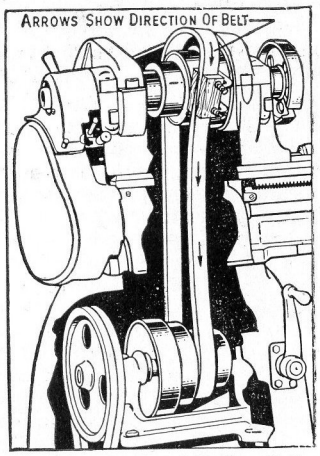


Fig. 29. Cemented Endless Belt Splice

Place the belt around the cone pulleys and apply cement according to directions of the cement manufacturer. Press the freshly cemented joint together and immediately lay on one of the wood blocks. Nail the belt, through the joint, to the block of wood with two or three small nails. This will prevent the splice from slipping or becoming crooked. Place the other block of wood on top of the joint and clamp firmly, using "C" clamps. Allow the cement to dry thoroughly before removing the clamps. Scrape any excess cement from the surfaces of the belt before using it.

Belt Tension on Lathe Drives

Maintaining the proper running tension for the flat leather belt and V-belts is of the greatest importance. Belts which are allowed to run loose will creep and slip and cause an over-all loss in the cutting efficiency of the lathe, and damage may result to fine work due to variation of cutting speeds. Belts which are run under too much tension overload the lathe and lathe drive. This results in loss of power, excessive bearing wear, causes the motor to run hot, and stretches the belts until they have to be shortened or replaced. Routine checking and adjustment of the belt tension, according to instructions given in the following paragraphs, will keep the lathe efficiency high and repair expenses low.

Horizontal Motor Driven Lathe Belts

The lathe should be stopped before the flat belt is shifted on the cone pulleys. Release the belt tension by pulling lever "C" forward, as shown in Fig. 30, and then shift belt to change spindle speeds.

To change flat belt tension, adjust turnbuckle "A" with lever "C" in the back (or down) position. Test tension by pressing down on belt midway between the cone pulleys. Belt should depress about one inch. If belt slips under power and tension is correct, it is probably oily and should be cleaned with naphtha or benzol. If the belt appears dry and stiff, a little neat's-foot oil will make it pliable.

Adjust V-belt tension by moving motor on its standard. Loosen the four screws at "B," Fig. 30. After moving motor, tighten screws. When properly adjusted V-belt should depress about one inch midway between pulleys, but must still be stretched tight enough to feel alive when tapped with finger tips. Check pivot screws at "E." If necessary, adjust so they are snug and tighten lock nuts.

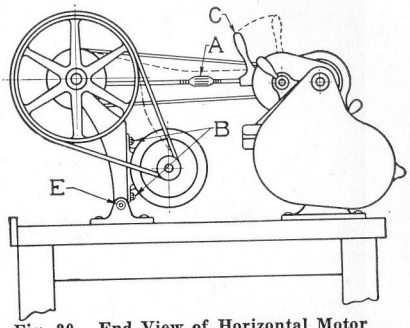


Fig. 30. End View of Horizontal Motor Driven Bench Lathe Showing Belt Tension Adjustment

Underneath Motor Driven Lathe Belts

Belt tension release lever "A" (Fig. 31) permits releasing cone pulley belt tension for shifting belt to change spindle speeds. See drawings of underneath motor drives, pages 8 and 9, Figs. 11, 12, 13, and 14.

Screw "C" adjusts tension of the cone pulley flat belt (see Figs. 12 and 31). This adjustment must be made with lever "A" in running position. The lathe shown in Fig. 11 has turnbuckle for making this adjustment.

Screw "B" adjusts tension of the motor V-belts. Turn nuts above and below motor mounting plate (Figs. 13 and 31). Adjust V-belt tension on lathe shown in Fig. 11, the same as shown in "Horizontal Motor Driven Lathe Belts," above.

Belts should be just tight enough to transmit the required power without slipping. Pressing the hand against a properly adjusted flat belt near the cone pulley should depress belt about ½". The V-belt, midway between pulleys, should depress about 1". Belts may be cleaned with naphtha or benzol and the flat belt treated with neat's-foot oil, as mentioned above on this page.

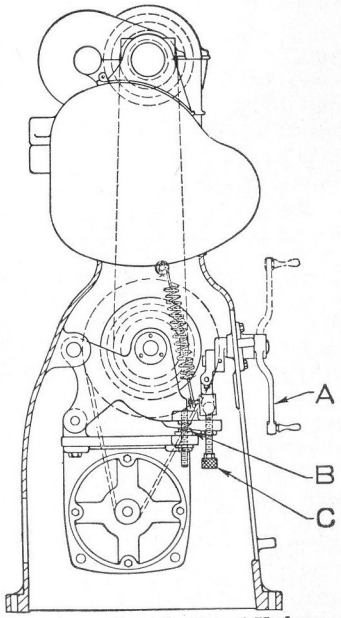


Fig. 31. Cross Section of Underneath Belt Motor Drive Showing Cone Pulley Belt and V-belts

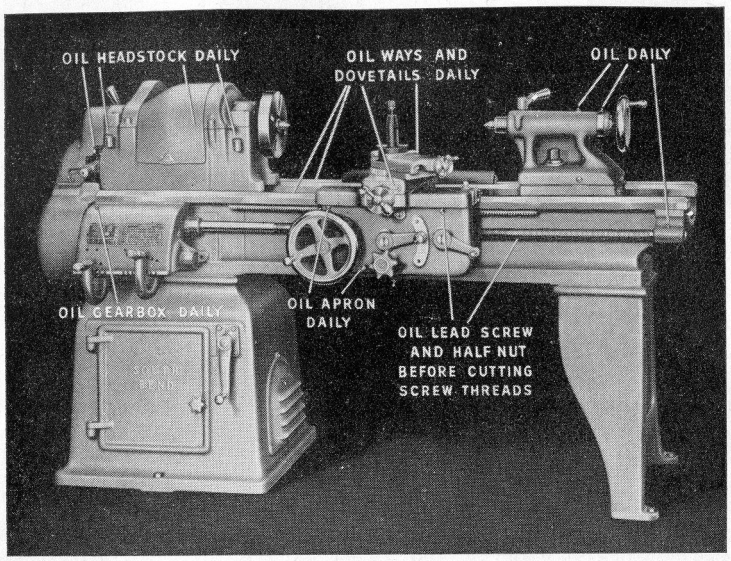


Fig. 32. Oiling Chart for a Lathe

Oiling the Lathe

Oil every bearing of a new lathe before starting it. Use high grade oils of proper viscosity as specified on the metal Lubrication Chart attached to each lathe. The lathe should be oiled twice daily for the first week it is used and once a day thereafter. Never oil the lathe while it is running and be certain that machine oil is used, not automobile engine oil.

Keeping the lathe well oiled has much to do with the length of its life and the quality of the work it will turn out. Follow the directions on the metal lubrication chart carefully if you wish to keep your lathe in first class condition.

Always oil in the same order so that no oil holes will be missed. If you do this the oiling will become a habit and will require only a very short time.

Do not use an excess of oil. A few drops in each oil hole is sufficient, and if more is applied it will only run out of the bearings and get on the lathe, making it necessary for you to clean it more frequently.

Oil the motor and drive bearings as specified in instructions supplied with the lathe. This is very important. Do not allow dust, chips or other debris to collect around the motor so it will overheat. An excessive rise in the motor running temperature will cause damage to its bearings and windings.

After you have completed the process of oiling the lathe and motor drive, wipe off the excess oil around the bearings with a clean cloth or waste. Keep the lathe clean. Do not allow oil, dirt, chips or rust to collect any place on the lathe.

Chapter III

OPERATION OF THE LATHE

Before starting a new lathe, the operator should carefully study the action of the various parts and become thoroughly familiar with the operation of all control levers and knobs.

The principal parts of the lathe are shown below in Fig. 33. Become familiar with the name of each part as they will be referred to frequently in the following pages where detailed information on the operation of the lathe is given.

Do not operate the lathe under power until it is properly set up and leveled, as outlined on page 15. Also make sure that all bearings have been oiled and that the belt tension is correct. Always pull the cone pulley belt by hand to make sure the lathe runs free before starting the lathe under power.

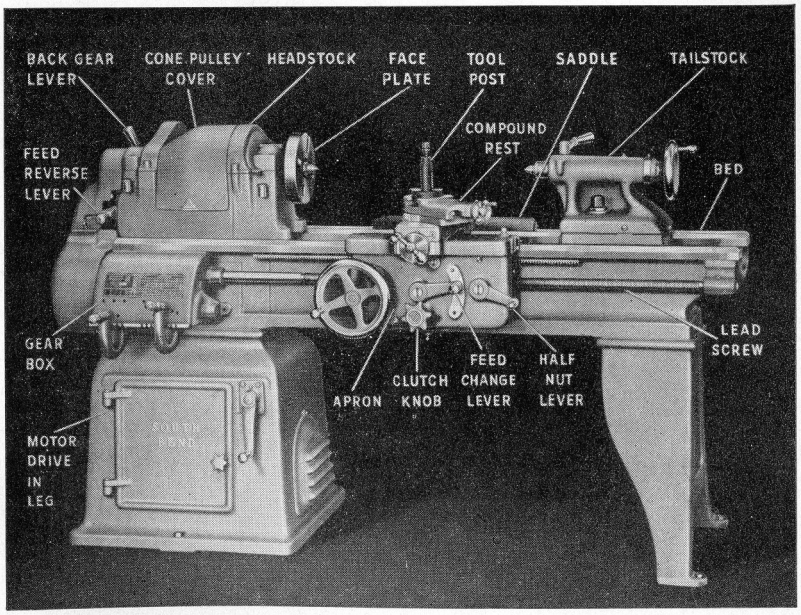


Fig. 33. Names of the Principal Parts of a Lathe

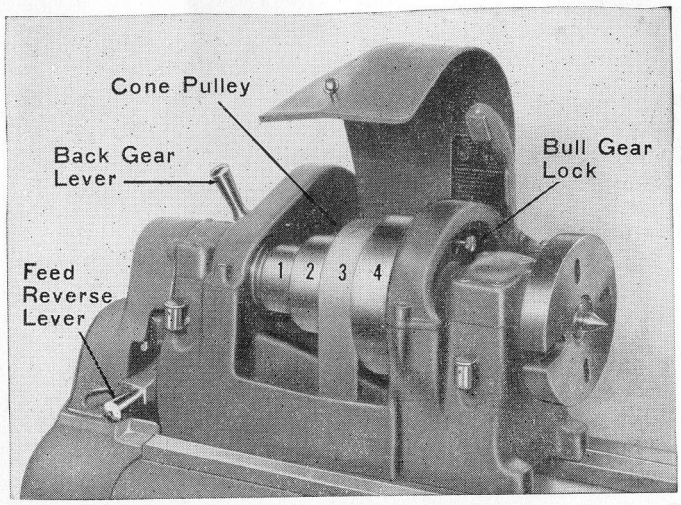


Fig. 34. Operating Parts of Lathe Headstock

Operation of Headstock

Spindle speeds are changed by shifting the belt from one step of the cone pulley to another and by engaging or disengaging the back gears. The cone pulley steps are numbered in the illustration above to correspond with the numbers in the tabulation on page 23, which shows the normal spindle speeds for various sizes of lathes.

Direct Belt Drive

To arrange the lathe headstock for direct belt drive, push the back gear lever back as far as it will go; then pull out and up on the bull gear lock pin and revolve the cone pulley slowly by hand until the bull gear lock slides into position and locks the cone pulley to the spindle.

Back Geared Drive

To engage the back gears for slow spindle speeds, pull the bull gear lock pin out and push it down to disconnect the cone pulley from the spindle; then pull the back gear lever forward. Revolve the cone pulley by hand to make sure the back gears are properly engaged. Do not engage the back gears while the lathe spindle is revolving.

Bull Gear Lock, Plunger Type

On some lathe headstocks the plunger type bull gear lock is used. For direct belt drive on these lathes the bull gear lock pin is pushed in, and for back-geared drive it is pulled out.

Feed Reverse Lever

The feed reverse lever on the left end of the headstock has three positions; up, central and down. The central position is neutral, and when in this position all power carriage feeds are disconnected. When the lever is in either the "up" position or "down" position the power carriage feeds will be in operation. The lathe should always be stopped before the position of the feed reverse lever is changed.

Spindle Speeds of Lathes

The spindle speeds for various sizes of lathes are listed in the tabulation below. The columns under which the speeds are listed are numbered 1, 2, 3, and 4 to correspond with the numbers on the cone pulley steps in Fig. 34, page 22. For example, those spindle speeds that are listed under Column 1 are obtained when the cone pulley belt is placed on the cone pulley step marked 1 in the illustration, Fig. 34.

SPINDLE SPEEDS OF LATHES

Size of Lathe	Type of Drive	Speed Range	Approximate Spindle Speeds in Revolutions Per Minute							
			Direct Belt Drive				Back-Gear Drive			
			1	2	3	4	1	2	3	4
9″ Bench	6-speed flat belt	....	680	392	235	...	130	78	50	...
9″ Bench & Self-Contained	12-speed flat belt	High Low	1270 692	750 410	446 244	... ...	250* 134	145 81	86 50	... ...
9″ Bench	8-speed V-belt	....	640	471	350	267	125	96	69	54
9″ Bench & Self-Contained	16-speed V-belt	High Low	1200 640	900 490	662 362	505 272	235* 130	179* 95	130 70	100 54
9″ & Light Ten	U.M.D. flat belt	High Low	1365 715	780 410	460 240	... ...	265* 135	155* 78	90 50	... ...
9″ & Light Ten	U.M.D. V-belt	High Low	1365 670	1010 495	760 370	570 285	265* 130	195* 95	150* 75	112 52
Light Ten Bench & Self-Contained	12-speed flat belt	High Low	1435 706	844 415	502 244	... ...	276* 137	165* 80	96 48	... ...
Light Ten Bench & Self-Contained	16-speed V-belt	High Low	1365 670	1010 495	760 370	570 285	265* 130	195* 95	150* 75	112 52
10-inch & 10-inch Turret	3-step pulley	High Low High† Low†	1400 740 700 370	898 470 449 235	585 304 292 152	... ...	250* 130 125 65	160* 85 80 42	105 55 52 27	... .
13-inch	4-step pulley	High Low†	940 470	628 314	418 209	270 135	135 67	90 45	60 30	40 20
13-inch & 13-inch Turret	3-step pulley	High Low†	940 470	497 248	270 135	... ...	135 67	71 35	40 20	... ...
14½-inch	4-step pulley	High Low†	875 437	545 272	350 175	215 107	130 65	80 40	50 25	30 15
14½-inch	3-step pulley	High Low†	875 437	428 214	215 107	... ...	130 65	61 30	30 15	... ...
16-inch	4-step pulley	High Low†	980 490	610 305	390 195	240 120	125 62	80 40	50 25	30 15
16-inch & 2-H	3-step pulley	High Low†	945 475	550 278	300 150	... ...	118 60	70 33	32 20	... ...
16-24-inch	6-speed	....	405	235	130	...	50	30	14	...
16-24-inch	8-speed	....	470	280	175	105	60	35	22	15
16-24-inch	12-speed†	High Low	790 400	460 230	250 125	... ...	100 50	60 29	27 15	... ...
16-24-inch	16-speed†	High Low	900 455	550 274	340 170	203 104	116 60	70 34	45 24	30 15

*It is not recommended that back-gears be engaged when drive is operated at high speed.
†Available only with 2-speed motor.

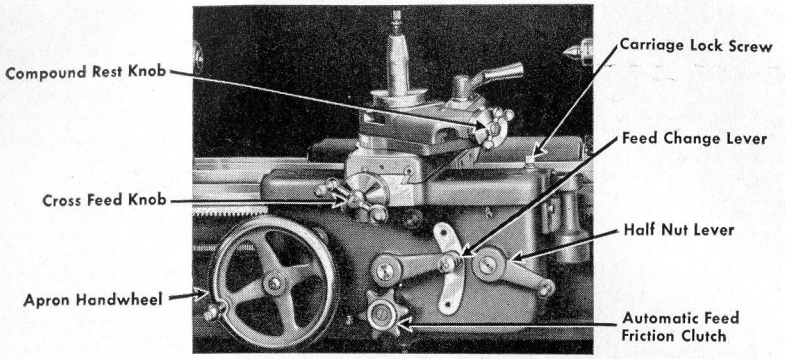


Fig. 35. Operating Parts of Lathe Carriage and Apron

Operation of Lathe Carriage and Apron

The principal operating parts of the lathe carriage and apron are shown above in Fig. 35. The apron hand wheel is turned to move the carriage along the lathe bed, and the cross feed knob and compound rest knob are turned to move the tool rest in and out. The carriage lock screw is used to lock the carriage to the lathe bed. This screw should never be tightened except for facing or cutting-off operations.

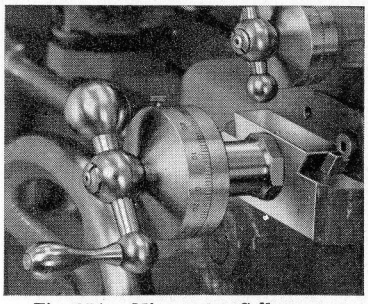


Fig. 35A. Micrometer Collar on Cross Feed Screw

Micrometer Collars

Each graduation of the micrometer collars on the cross feed knob and compound rest knob represents a movement of the compound rest of one-thousandth of an inch. The graduated collars may be set at zero by releasing the set screws which lock them in position.

Power Carriage Feeds

The power feed friction clutch controls the operation of both the power longitudinal feed and the power cross feed. To engage the clutch, turn the club knob to the right; to disengage, turn to the left. The direction of the feed is controlled by the position of the reverse lever on the headstock. (See page 22.)

The feed change lever has three positions: "up" for longitudinal feeds, "down" for cross feeds, and "center" for neutral.

The half nut lever is used only for thread cutting. The feed change lever must be in the "center" or neutral position before the half nuts can be engaged.

Operation of Tailstock

The tailstock may be locked on the lathe bed at any position by tightening the clamp bolt nut. To lock the tailstock spindle, tighten the binding lever.

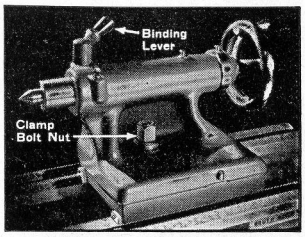


Fig. 36. Tailstock

Power Carriage Feeds on Quick Change Gear Lathes

A wide range of power longitudinal feeds and power cross feeds is available on all Quick Change Gear Lathes. To obtain any desired feed it is only necessary to arrange the levers on the gear box according to the direct reading index chart shown in Fig. 38. The threads per inch are shown in large figures on the index chart below. The smaller figures indicate the power longitudinal turning feeds in thousandths of an inch. See page 74.

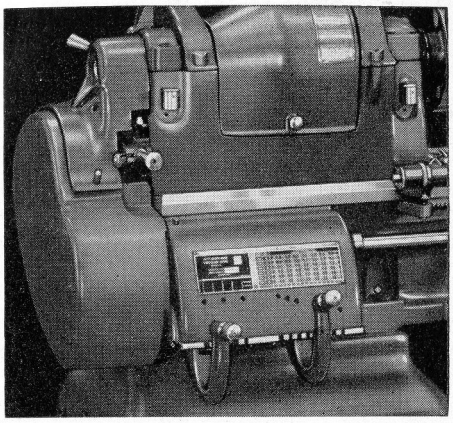


Fig. 37. Quick Change Gear Mechanism

MANUFACTURED BY SOUTH BEND LATHE WORKS SOUTH BEND, IND. U. S. A.

14½ & 16 INCH SOUTH BEND PRECISION LATHE
MODEL A
CATALOG NO.
BED LENGTH
CHART NO. 1

A B C D E POSITION

POWER CROSS FEED .375 TIMES LONGITUDINAL FEED

STUD GEAR	LEFT HAND TUMBLER	THREADS PER INCH FEEDS IN THOUSANDTHS							
48	A	4 .0841	4½ .0748	5 .0673	5½ .0612	5½ .0565	6 .0561	6½ .0518	7 .0481
24	A	8 .0421	9 .0374	10 .0337	11 .0306	11½ .0293	12 .0280	13 .0259	14 .0240
24	B	16 .0210	18 .0187	20 .0168	22 .0153	23 .0146	24 .0140	26 .0129	28 .0120
24	C	32 .0105	36 .0093	40 .0084	44 .0076	46 .0073	48 .0070	52 .0065	56 .0060
24	D	64 .0053	72 .0047	80 .0042	88 .0038	92 .0037	96 .0035	104 .0032	112 .0030
24	E	128 .0026	144 .0023	160 .0021	176 .0019	184 .0018	192 .0017	208 .0016	224 .0015

Fig. 38. Index Chart for Quick Change Gear Lathe

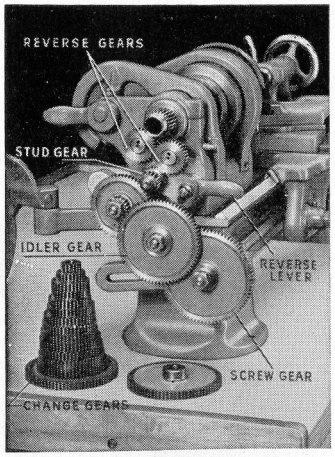


Fig. 39. Standard Change Gear Mechanism

Power Carriage Feeds on Standard Change Gear Lathes

Standard Change Gear Lathes are equipped with a set of independent change gears for cutting screw threads and obtaining various power longitudinal feeds and power cross feeds. Compound gearing is used for fine threads and feeds. See pages 72 and 114.

A large "screw gear" should be placed on the lead screw and a small "stud gear" on the reverse stud. These two gears should be connected with idler gears as shown. To obtain finer or coarser feeds, use a smaller or larger "stud gear."

Notes on Lathe Work

A mixture of red lead and machine oil is a good lubricant for the tailstock center of a lathe.

A precision level that will show an error of .003 in. per foot should be used to level the lathe when installing, as a level lathe will assure precision accuracy of the work.

Clean and oil the threads before screwing a chuck or face plate onto the lathe spindle.

After grinding a tool, hone it to a keen edge with an oil stone—the cutting edge will last longer.

Always make sure the spindle tapers of the lathe are clean and free from burrs and dirt before inserting the lathe centers.

If the face plate or chuck does not run true, examine the shoulder of the lathe spindle and face of hub on face plate or chuck back for burrs, dirt, etc.

When cutting screw threads in steel, use a small brush to spread oil on the work preceding each cut. Lard oil is preferable, but a good machine oil or cutting oil will do.

Use Flat Leather Belts

Flat leather belts are recommended for use on lathe cone pulleys.

Leather belts are better than canvas or rubber belts for use on lathe cone pulleys. Leather belts are more efficient, last longer, have more elasticity and give better service.

If a belt has a tendency to come off from the pulley there is something wrong. Usually the pulleys are out of alignment. Find out what the trouble is and remedy it. Do not try to hold the belt on the pulley with a brace.

Notes on Belts and Pulleys

To find the approximate length of a belt, multiply half the sum of the pulley diameters by 3-1/7 and add twice the distance between the pulley centers.

The smooth side of the belt should always run next to the pulley.

Keep belts clean and dry. Do not allow moisture, machine oil or dirt to collect on them.

A pulley should be about 10% wider than the belt.

Driving pulleys for shifting belts should have flat face, all other pulleys should be crowned.

For stepped or flanged pulleys double ply belting is better than single ply belting.

Don't shift a moving belt by hand; use a stick or belt shifter.

Never put a belt on a pulley while it is revolving rapidly.

A belt may run crooked if the ends are not cut square before lacing, or if laced unevenly.

Don't run belts too tight, or with the flesh side next to the pulley.

Chapter IV

LATHE TOOLS AND THEIR APPLICATION

In order to machine metal accurately and efficiently, it is necessary to have the correct type of lathe tool with a keen, well supported cutting edge, ground for the particular kind of metal being machined, and set at the correct height.

High speed steel cutter bits mounted in forged steel holders, as shown in Figs. 40, 46, 48 and 50, are the most popular type of lathe tools. The 10-in-1 Tool Block shown in Fig. 50A may be used in place of the individual tool holders if desired. See page 98.

The boring tool, cutting-off tool, threading tool and knurling tool are required for various classes of work that cannot be readily accomplished with the regular turning tool.

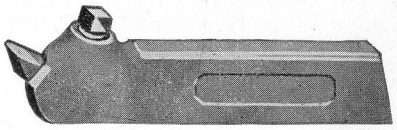


Fig. 40. Tool Holder with Cutter Bit



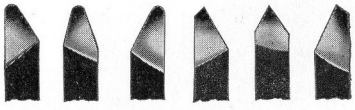
Fig. 41. Unground Cutter Bit



Fig. 41-A Holder for Grinding Cutter Bits



Fig. 42. Cutter Bit After Grinding



A L. H. Turning | B Round Nose | C R. H. Turning | D L. H. Side | E Threading | F R. H. Side

Fig. 44. Set of Ground Cutter Bits

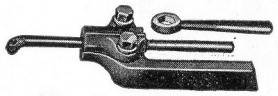


Fig. 43. Boring Tool

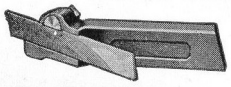


Fig. 45. Cutting-off Tool



Fig. 46. Straight Tool Holder

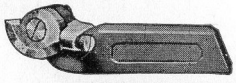


Fig. 47. Threading Tool

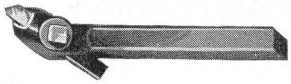


Fig. 48. Left Hand Tool Holder

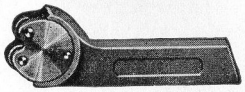


Fig. 49. Knurling Tool



Fig. 50. Right Hand Tool Holder

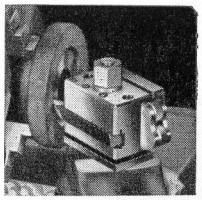


Fig. 50A. 10-in-1 Tool Block. See page 98.

Correct Height of Cutting Edge

The cutting edge of the cutter bit should be about 5° above center, or 3/64 in. per inch in diameter of the work, as shown in Fig. 51 at right, for ordinary straight turning. The position of the cutter bit must be taken into consideration when grinding the various angles, as the height of the cutter bit determines the amount of front clearance necessary to permit free cutting.

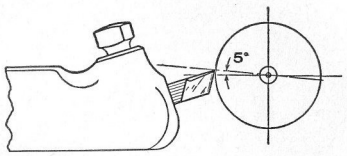


Fig. 51. Cutting Edge Above Center

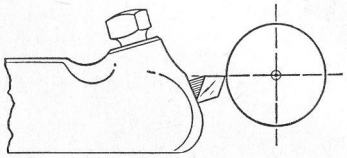


Fig. 52. Cutting Edge On Center

The cutting edge of the cutter bit should always be placed exactly on center, as shown in Fig. 52, for all types of taper turning and boring, and for cutting screw threads, also for turning brass, copper and other tenacious metals.

Tool Angle Varies

The included angle of the cutting edge of a cutter bit is known as the tool angle or angle of keenness and varies with the texture of the work to be machined. For example, when turning soft steel a rather acute angle should be used, but for machining hard steel or cast iron the cutting edge must be well supported and therefore the angle is less acute.

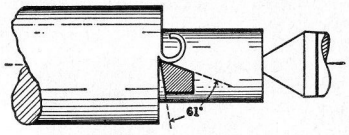


Fig. 53. Tool Angle for Steel

It has been found that an included angle of 61° is the most efficient tool angle for machining soft steel. This is the angle of the cutter as shown in Fig. 53. For machining ordinary cast iron, the included angle of the cutting edge should be approximately 71°. However, for machining chilled iron or very hard grades of cast iron, the tool angle may be as great as 85°.

Cutter Bit Grinding Gauge

A cutter bit grinding gauge, shown in Figs. 54-A, 54-B, and 54-C, is helpful for grinding the correct angles on the cutter bit.

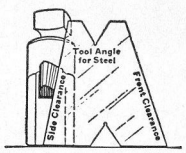


Fig. 54-A. Checking Side Clearance

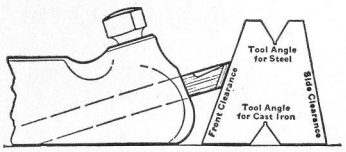


Fig. 54-B. Checking Front Clearance

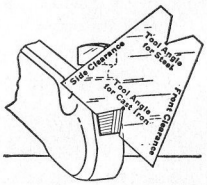


Fig. 54-C. Checking Tool Angle

Grinding Lathe Tool Cutter Bits

The angle of the cutter bit with the bottom of the tool holder must be taken into consideration when grinding cutter bits.

The side clearance (Fig. 55) is to permit the cutting edge to advance freely without the heel of the tool rubbing against the work.

The front clearance (Fig. 56) is to permit the cutting edge to cut freely as the tool is fed to the work.

Too much clearance will weaken the cutting edge so that it will break; but insufficient clearance will prevent the tool from cutting.

Side rake and back rake (Figs. 55 and 56) also facilitate free cutting. For cast iron, hard bronze and hard steel, very little side rake or back rake are required. (See page 28.)

The angle of keenness (Fig. 55) may vary from 60° for soft steel to nearly 90° for cast iron, hard steel, bronze, etc.

Figs. 57 to 61, inclusive, show the various steps in grinding a cutter bit for general machine work. Honing the cutting edge (Fig. 62) will improve the quality of the finish and lengthen the life of the tool.

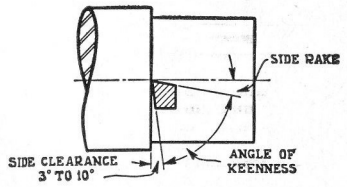


Fig. 55. Correct Side Clearance and Side Rake of Cutter Bit

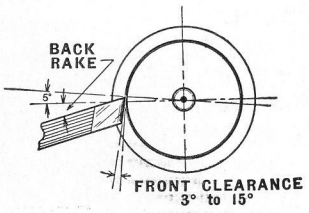


Fig. 56. Correct Front Clearance and Back Rake of Cutter Bit

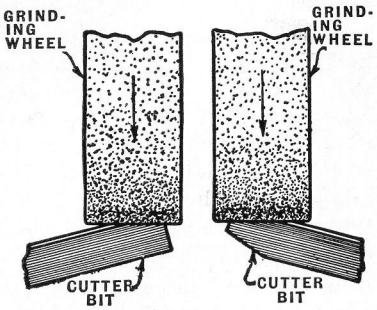


Fig. 57. Grinding Left Side of Cutter Bit

Fig. 58. Grinding Right Side of Cutter Bit

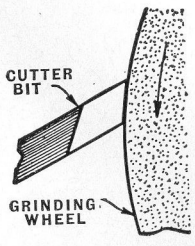


Fig. 59. Grinding Front of Cutter Bit

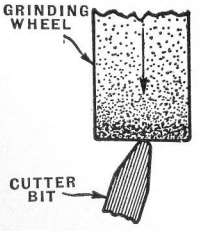


Fig. 60. Rounding End of Cutter Bit

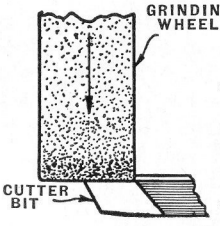


Fig. 61. Grinding Side Rake and Back Rake



Fig. 62. Honing the Cutting Edge of Cutter Bit with an Oil Stone

Cutter Bit for Rough Turning

Figs. 63 and 64 illustrate an excellent tool for taking heavy roughing cuts to reduce the diameter of a steel shaft to the approximate size desired. This tool will cut freely but does not produce a very smooth finish. When using this type of tool it is advisable to leave sufficient stock for a finishing cut with the round nosed tool shown at the bottom of the page.

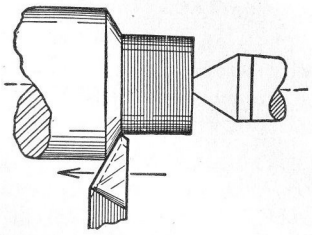


Fig. 63. Application of Roughing Tool

Grind the tool to the shape shown in Fig. 64, and see Figs. 55 and 56 on page 29 for information on grinding the correct front clearance, etc.

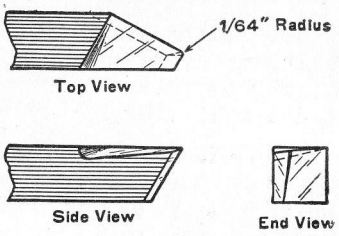


Fig. 64. Detail of Roughing Tool

The cutting edge of the tool is straight and the point is only slightly rounded. A very small radius at the point (approximately $\frac{1}{64}$ in.) will prevent the point of the tool from breaking down but will not impair the free cutting quality of the tool.

The tool angle or included angle of the cutting edge of this tool should be approximately 61° for ordinary machine steel. If a harder grade of alloy or tool steel is to be machined, the angle may be increased, and if free cutting Bessemer screw stock is to be machined, the angle may be slightly less than 61°.

Hone the cutting edge of the tool with a small oil stone. This will lengthen the life of the tool and it will cut better.

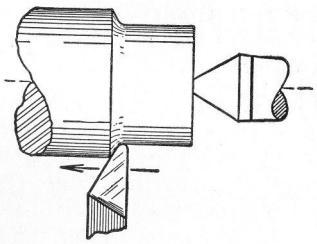


Fig. 65. Application of Finishing Tool

Cutter Bit for Finish Turning

Figs. 65 and 66 illustrate a round nosed turning tool for taking finishing cuts. The tool is very much the same shape as the more pointed tool for rough turning shown above, except that the point of the tool is rounded. (Approximately $\frac{1}{32}$ in. to $\frac{1}{16}$ in. radius.)

This tool will produce a very smooth finish if, after grinding, the cutting edge is well honed with an oil stone and a fine power carriage feed is used.

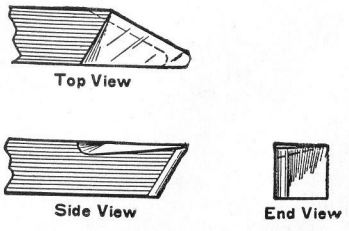


Fig. 66. Detail of Finishing Tool

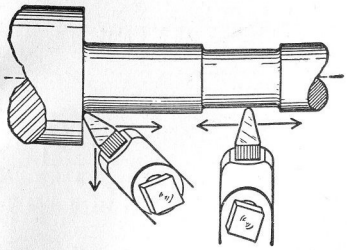


Fig. 67. Application of Round Nosed Tool

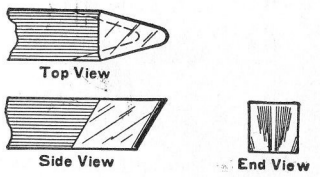


Fig. 68. Detail of Round Nosed Tool Bit

Round Nosed Turning Tool

The round nosed turning tool shown above is ground flat on top so that the tool may be fed in either direction, as indicated by the arrows in Fig. 67. This is a very convenient tool for reducing the diameter of a shaft in the center. The shape of the cutter bit is shown in Fig. 68, and the correct angle for the front clearance and side clearance can be obtained by referring to Figs. 55 and 56, page 29.

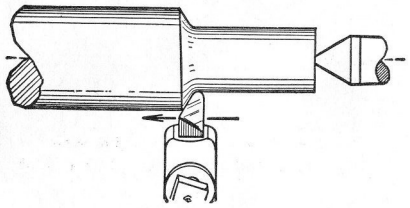


Fig. 69. Application of Right Hand Turning Tool

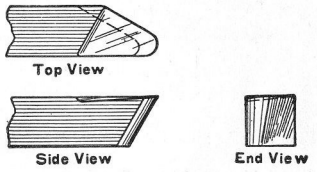


Fig. 70. Detail of Right Hand Turning Tool

Right Hand Turning Tool

The right hand turning tool shown above is the most common type of tool for general all around machine work. This tool is used for machining work from right to left, as indicated by the arrow in Fig. 69. The shape of the cutter bit is shown in Fig. 70. See page 29 for correct angles of clearance.

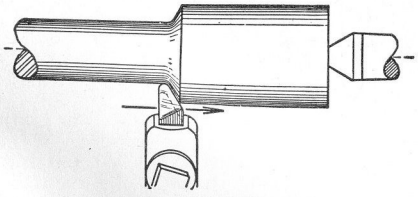


Fig. 71. Application of Left Hand Turning Tool

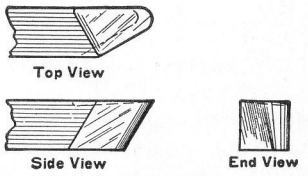


Fig. 72. Detail of Left Hand Turning Tool

Left Hand Turning Tool

The left hand turning tool illustrated in Figs. 71 and 72 is just the opposite of the right hand turning tool shown in Figs. 69 and 70. This tool is designed for machining work from left to right.

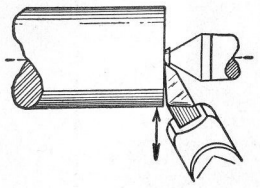


Fig. 73. Application of Right Hand Side Tool

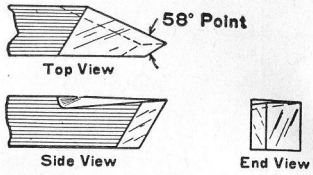


Fig. 74. Detail of Right Hand Side Tool

Right Hand Side Tool

The right hand side tool is intended for facing the ends of shafts and for machining work on the right side of a shoulder. This tool should be fed outward from the center, as indicated by the arrow in Fig. 73. The point of the tool is sharp and is ground to an angle of 58° to prevent interference with the tailstock center. When using this cutter bit care should be taken not to bump the end of the tool against the lathe center, as this will break off the point. See page 29 for correct angle of side clearance and front clearance.

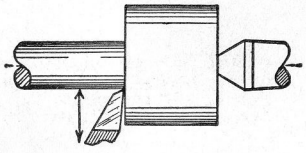


Fig. 75. Application of Left Hand Side Tool

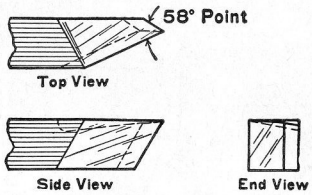


Fig. 76. Detail of Left Hand Side Tool

Left Hand Side Tool

The left hand side tool shown in Figs. 75 and 76 is just the reverse of the right hand side tool shown in Figs. 73 and 74. This tool is used for facing the left side of the work, as shown in Fig. 75.

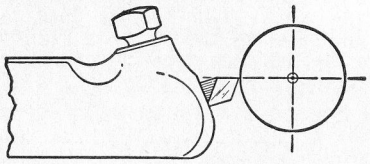


Fig. 77. Application of Screw Thread Cutting Tool

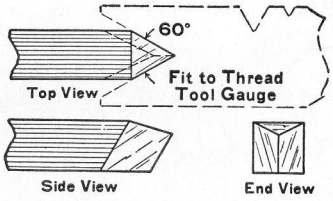


Fig. 78. Detail of Thread Cutting Tool

Thread Cutting Tool

Figs. 77 and 78 show the standard type of cutter bit for cutting United States or American National Form screw threads. The cutter bit is usually ground flat on top, as shown in Fig. 77, and the point of the tool must be ground to an included angle of 60°, as shown in Fig. 78. Careful grinding and setting of this cutter bit will result in perfectly formed screw threads. When using this type of cutter bit to cut screw threads in steel, always keep the work flooded with lard oil in order to obtain a smooth thread. Machine oil may be used if no lard oil is available.

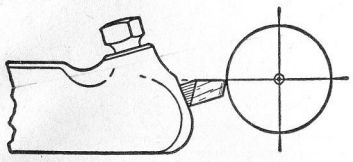


Fig. 79. Application of Brass Turning Tool

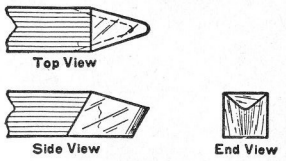


Fig. 80. Detail of Brass Turning Tool

Brass Turning Tool

The brass turning tool shown above is similar to the round nosed turning tool illustrated in Figs. 67 and 68, except that the top of the tool is ground flat so that there is no side rake or back rake. This is to prevent the tool from digging into the work and chattering.

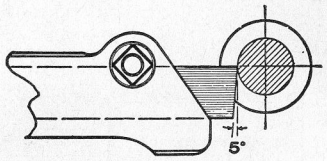


Fig. 81. Application of Cutting-off Tool

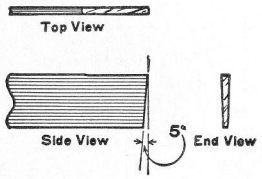


Fig. 82. Detail of Cutting-off Tool

Cutting-off Tool

The cutting-off tool should always be set exactly on center, as shown in Fig. 81. This type of tool may be sharpened by grinding the end of the cutter blade to an angle of 5° as shown in Fig. 82. The sides of the blade have sufficient taper to provide side clearance, so do not need to be ground. When cutting off steel always keep the work flooded with oil. No oil is necessary when cutting off cast iron.

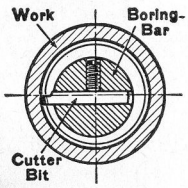


Fig. 83. Application of Boring Tool

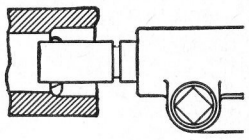


Fig. 84. Detail of Boring Tool

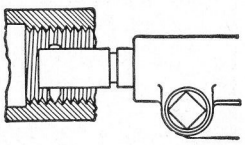


Fig. 85. Inside Threading Tool

Boring Tool and Inside Threading Tool

The boring tool is ground exactly the same as the left hand turning tool shown in Figs. 71 and 72 on page 31, except the front clearance of boring tool must be ground at a slightly greater angle so that the heel of the tool will not rub in the hole of the work. The inside threading tool is ground the same as the screw thread cutting tool shown in Figs. 77 and 78 on page 32, except that the front clearance must be increased for the same reason as for the boring tool.

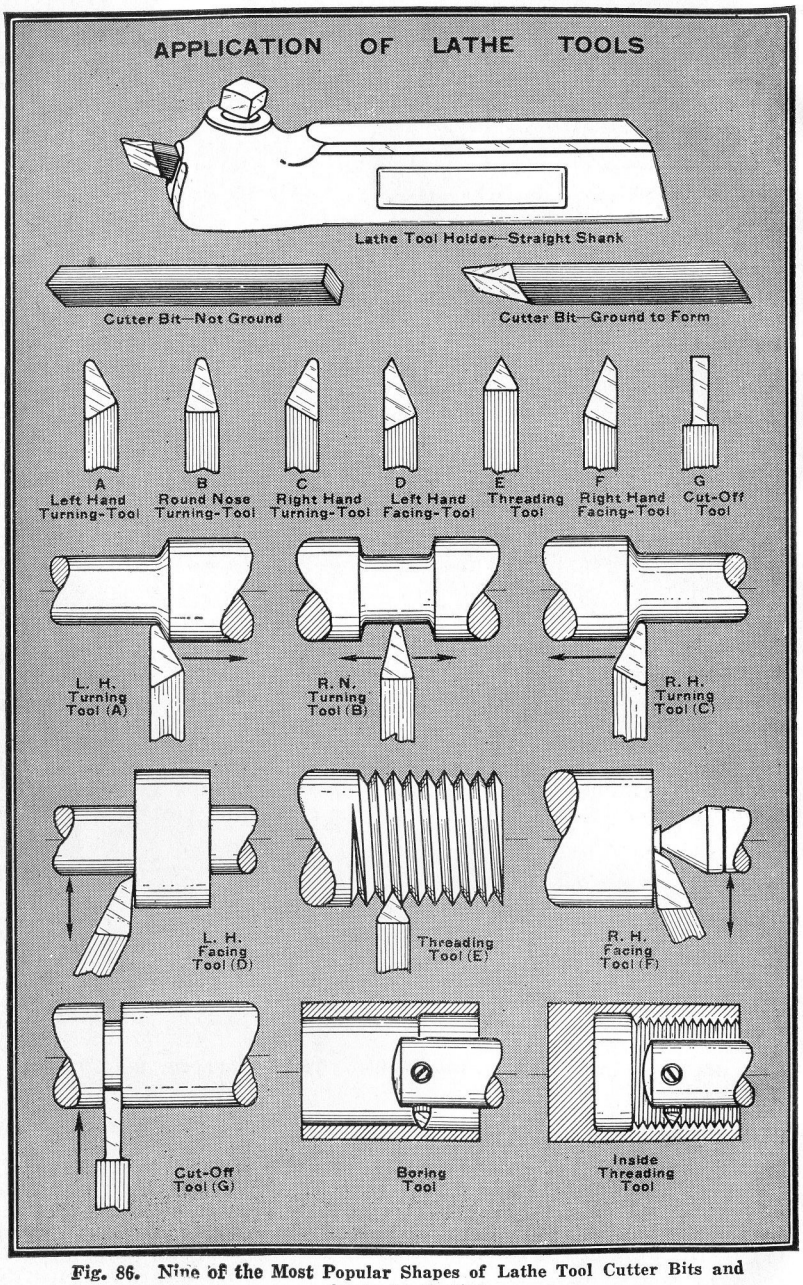


Fig. 86. Nine of the Most Popular Shapes of Lathe Tool Cutter Bits and Their Application

Stellite Cutter Bits

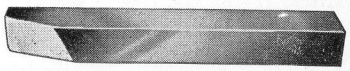


Fig. 87. Stellite Cutter Bit

Stellite cutter bits will stand higher cutting speeds than high speed steel cutter bits. Stellite is also used for machining hard steel, cast iron, bronze, etc.

Stellite is a non-magnetic alloy which is harder than ordinary high speed steel. It will stand very high cutting speeds and the tool will not lose its temper even though heated red hot from the friction generated by taking the cut.

Stellite is more brittle than high speed steel, and for this reason should have just enough clearance to permit the tool to cut freely, as the cutting edge must be well supported to prevent chipping and breaking.

Tungsten Carbide Cutting Tools

Tungsten carbide tipped cutting tools are used for manufacturing operations where maximum cutting speeds are desired, and are highly efficient for machining cast iron, alloyed cast iron, copper, brass, bronze, aluminum, babbitt and abrasive non-metallic materials such as fibre, hard rubber and plastics. Cutting speeds may vary from 110 to 650 surface feet per minute, depending on the depth of cut and the feed.

Tungsten carbide tipped cutter bits must be ground on a special grade of grinding wheel, as they are so hard they cannot be satisfactorily ground on the ordinary grinding wheel. The cutting edge must be well supported to prevent chipping and should have just enough clearance to permit the tool to cut freely.

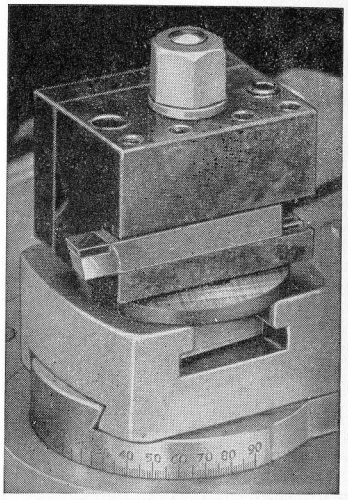


Fig. 88. Carbide Tipped Cutting Tool Mounted in 10-in-1 Tool Holder for Rigid Support

Tantalum Carbide Cutting Tools

Tantalum carbide is a term applied to a combination of tungsten carbide and tantalum carbide. Tantalum carbide tipped cutting tools are similar to tungsten carbide tools, but are used mostly for machining steel.

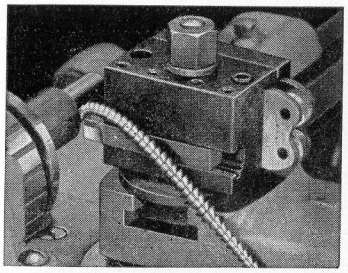


Fig. 88A. Machining a Steel Shaft at High Speed with Carbide Tipped Cutting Tool

Titanium Carbide Cutting Tools

Titanium carbide is a term applied to a combination of tungsten carbide and titanium carbide. Titanium carbide is interchangeable with tantalum carbide in its uses.

Cutting Tools for Non-Ferrous Materials

The machinability of non-ferrous materials such as copper, bronze, aluminum, and plastics varies greatly, and such materials often require specially ground cutting tools. The cutting speeds also vary greatly and have an important effect on tool life and surface finish. Suggested cutting speeds and tool angles for machining the most common non-ferrous materials are given in the table below. Also see page 50.

Machining Soft Metals

Aluminum, magnesium alloys, and other comparatively soft metals require keen edged tools with more clearance, front rake, and side rake than the harder metals. To increase the amount of back rake, the cutting edge of the tool is sometimes placed high above center. This, of course, cannot be done when turning tapers or facing, and frequent readjustment is necessary when the diameter of the work varies.

When machining tenacious metals such as pure copper the cutting tool should be honed to a very keen cutting edge to prevent tearing the work and producing a rough finish. Light cuts at medium feeds with a round nose tool having 1/32″ to 1/16″ nose radius usually produce best results.

Machining Plastics

Because of the great variety of plastic materials, many problems are encountered which require some thought and resourcefulness in grinding cutting tools. Hot set molded plastics often contain abrasive fillers which dull the cutting edge quickly unless it is well supported by minimum clearance angles. However, other plastic materials may require very large clearance angles to prevent the tool from dragging. Laminated plastics and vulcanized fibre should be machined at high speeds, as the cutting edge of the tool will dull quickly at lower speeds.

CUTTING SPEEDS AND TOOL ANGLES FOR NON-FERROUS MATERIALS

See Figs. 55 and 56 page 29 for tool angle diagrams

Material	Cutting Speed f. p. m.	Front Clearance Degrees	Side Clearance Degrees	Back Rake Degrees	Side Rake Degrees
Aluminum*	300–400	7	8	30	20
Brass — Leaded	300–700	6	7	0	0
Bronze — Free Cutting	300–700	6	5	0	1½
Bronze — Readily Machinable	150–300	10	8	7	7
Bronze — Tough	75–150	12	15	15	25
Copper — Pure	75–150	7	7	10	25
Die Castings	225–350	7	7	8	10
Magnesium Alloy	275–400	8	8	6	4
Monel Metal	50–170	6	6	7	12
Plastics — Cast Resin	200–600	10	12	30	25
Plastics — Cold Set Molded	200–600	10	10	0	0
Plastics — Hot Set Molded	200–600	6	6	10	20
Plastics — Laminated	200–600	8	8	30	30
Stainless Steel	50–150	8	8	9	0
Wood	400–800	20	20	30	30

*Kerosene lubricant used.

These are suggested starting angles for general work. Slightly smaller or larger angles may prove more efficient, depending on the texture of the material machined and the type of cutting tool used.

Chapter V

HOW TO TAKE ACCURATE MEASUREMENTS

The ability to take accurate measurements can be acquired only by practice and experience. Careful and accurate measurements are essential to good machine work. All measurements should be made with an accurately graduated steel scale or a micrometer. Never use a cheap steel scale or a wood ruler, as they are likely to be inaccurate and may cause spoiled work.

An experienced mechanic can take measurements with a steel scale and calipers to a surprising degree of accuracy. This is accomplished by developing a sensitive "caliper feel" and by carefully setting the calipers so that they "split the line" graduated on the scale.

Setting an Outside Caliper

A good method for setting an outside caliper to a steel scale is shown in Fig. 96. The scale is held in the left hand and the caliper in the right hand. One leg of the caliper is held against the end of the scale and is supported by the finger of the left hand while the adjustment is made with the thumb and first finger of the right hand.

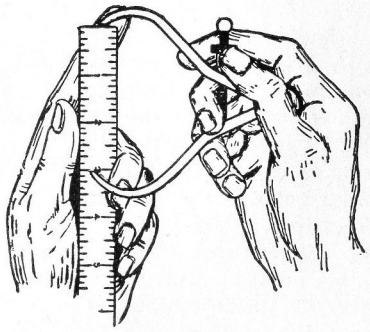


Fig. 96. Setting an Outside Caliper

Measuring with Calipers

The proper application of the outside caliper when measuring the diameter of a cylinder or a shaft is shown in Fig. 97. The caliper is held exactly at right angles to the center line of the work and is pushed gently back and forth across the diameter of the cylinder to be measured. When the caliper is adjusted properly, it should easily slip over the shaft of its own weight. Never force a caliper or it will spring and the measurement will not be accurate.

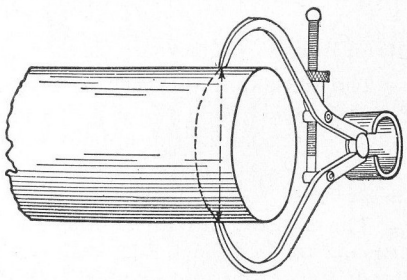


Fig. 97. Measuring with an Outside Caliper

Setting Inside Calipers

To set an inside caliper for a definite dimension, place the end of the scale against a flat surface and the end of the caliper at the edge and end of the scale. Hold the scale square with the flat surface. Adjust the other end of the caliper to the required dimension.

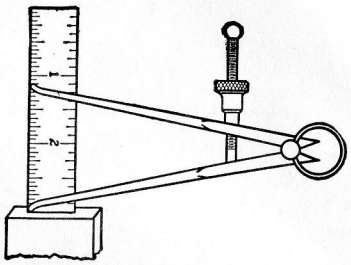


Fig. 98. Setting an Inside Caliper

Measuring Inside Diameters

To measure an inside diameter, place the caliper in the hole as shown on the dotted line and raise the hand slowly. Adjust the caliper until it will slip into the hole with a very slight drag. Be sure to hold the caliper square across the diameter of the hole.

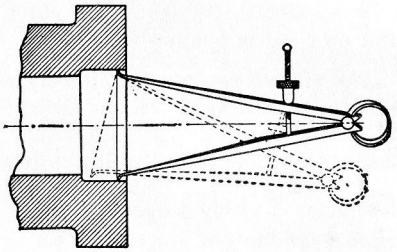


Fig. 99. Measuring with Inside Caliper

Transferring Measurements

In transferring measurement from an outside caliper to an inside caliper, the point of one leg of the inside caliper rests on a similar point of the outside caliper, as shown in Fig. 100. Using this contact point as a pivot, move the inside caliper along the dotted line shown in illustration, and adjust with the thumb screw until you feel your measurement is just right.

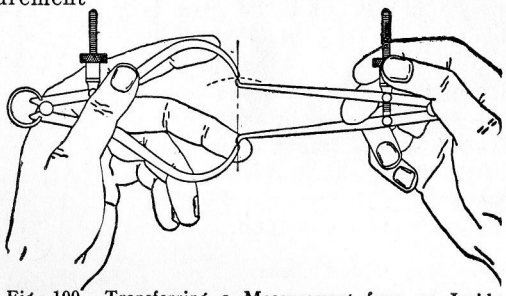


Fig. 100. Transferring a Measurement from an Inside Caliper to an Outside Caliper

Hermaphrodite Caliper

The hermaphrodite caliper shown in Fig. 101 is set from the end of the scale exactly the same as the outside caliper.

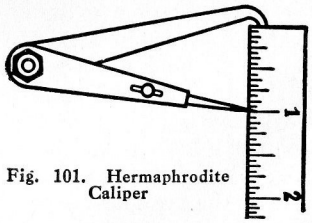


Fig. 101. Hermaphrodite Caliper

Caliper Feel

The accuracy of all contact measurements is dependent upon the sense of touch or feel. The caliper should be delicately and lightly held in the finger tips, not gripped tightly. If the caliper is gripped tightly, the sense of touch is very much impaired.

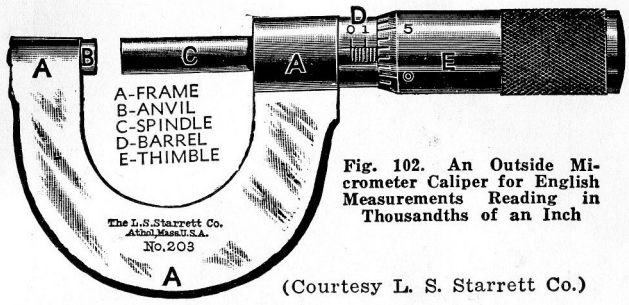


Fig. 102. An Outside Micrometer Caliper for English Measurements Reading in Thousandths of an Inch

(Courtesy L. S. Starrett Co.)

How to Read a Micrometer (English Measurement)

Each graduation on the micrometer barrel "D" represents one turn of the spindle or .025 in. Every fourth graduation is numbered and the figures represent tenths of an inch since 4x.025 in. = .100 in. or $\frac{1}{10}$ of an inch.

The thimble "E" has twenty-five graduations, each of which represents one-thousandth of an inch. Every fifth graduation is numbered, from five to 25.

The micrometer reading is the sum of the readings of the graduations on the barrel and the thimble. For example, there are seven graduations visible on the barrel in the illustration above. Since each graduation represents .025 in., the reading on the barrel is 7x.025 in. or .175 in. To this must be added the reading on the thimble which is .003 in. The correct reading is the sum of these two figures or .175 in. + .003 in. = .178 in. Therefore this micrometer is set for a diameter of .178 in.

Metric System Micrometer

Micrometers for measuring in the Metric system are graduated to read in hundredths of a millimeter as shown at right in Fig. 103. For each complete revolution the spindle travels ½ mm or .50 mm, and two complete revolutions are required for 1.00 mm. Each of the upper set of graduations on the barrel represent 1 mm. (two revolutions of the spindle) and every fifth graduation is numbered 0, 5, 10, 15, etc. The lower set of graduations subdivides each millimeter division into two parts.

The beveled edge of the thimble is divided into 50 graduations, each of which represents .01 mm.

The micrometer reading is the sum of the readings on the barrel and the thimble. For example, in Fig. 103 there are three millimeter graduations visible on the barrel, also a ½ mm graduation. The reading on the thimble is .36 mm. Therefore, the reading is 3.00 mm + .50 mm + .36 mm = 3.86 mm.

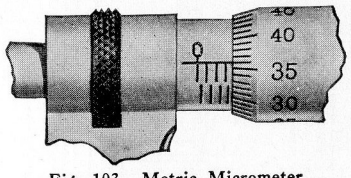


Fig. 103. Metric Micrometer
(Courtesy Brown & Sharpe Mfg. Co.)

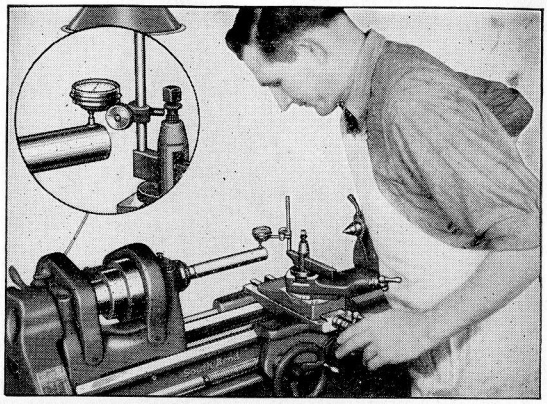


Fig. 104. Testing Headstock Spindle with Test Bar and Test Indicator

The Accuracy of a Screw Cutting Lathe

In manufacturing the back-geared screw cutting lathe, accuracy is given the most careful attention. A few of the accuracy tests are shown below. The illustration above shows the method of testing the headstock spindle of a lathe to see that the taper of the spindle runs true and that the axis of the spindle is parallel to the ways of the lathe.

The test bar is made of steel and may be from 10 in. to 12 in. long, depending on the size of the lathe. It is machined between centers and ground on the taper shank and also on the two larger diameters where the indicator readings are taken, as shown below. A dial test indicator used with this bar, as shown above, will disclose an error of one ten-thousandth of an inch.

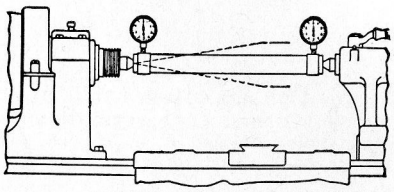


Fig. 105. Testing Alignment of Tailstock Spindle and Headstock Spindle

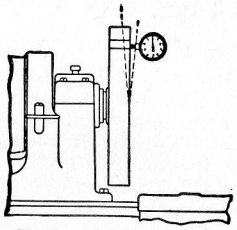


Fig. 106. Testing Amount of Concavity of Face Plate with Dial Indicator

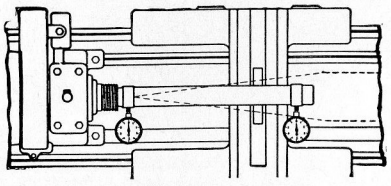


Fig. 107. Testing Alignment of Headstock Spindle with Ways of Lathe Bed

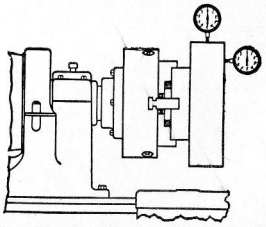


Fig. 108. Testing Accuracy of Chuck Jaws on Diameter and Face

Use of Toolmaker's Buttons

Toolmaker's buttons are small bushings used for accurately locating drill jig plates and similar work on the face plate of the lathe for boring holes to given center-to-center distances.

The locations of the holes to be bored are first carefully laid out on the work in the usual way, using scale measurements. Each point is then center punched, drilled and tapped for the small screw used to attach the toolmaker's button to the work. The hole through the button is much larger than the screw which passes through it permitting the button to be adjusted in any direction to conform with the exact location in which the hole is to be bored in the work. See Fig. 109.

The screws holding the buttons on the work are adjusted so that a light tap with a very small brass or lead hammer will move the button a fraction of a thousandth. Measuring with micrometer or vernier caliper or with precision gauge blocks, the buttons can be adjusted with extreme accuracy. See Fig. 110. Also see Fig. 359 page 121.

After all buttons are properly located on the work, the screws are tightened to lock them securely in position. Locations are checked to be sure all buttons are still in exactly the correct position and the work is then lightly clamped on the face plate with the button indicating the position of the first hole to be bored approximately centered. A dial indicator is used to check the centering as shown in Fig. 111. The lathe spindle is slowly revolved by hand and necessary re-adjustments are made until the button runs dead true. The work may then be clamped tightly on the face plate and the hole bored.

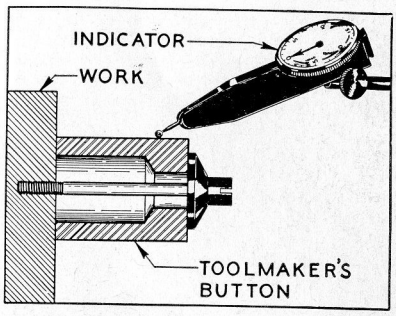


Fig. 109. Cross-section of Toolmaker's Button as used for Locating Work on Face Plate of Lathe

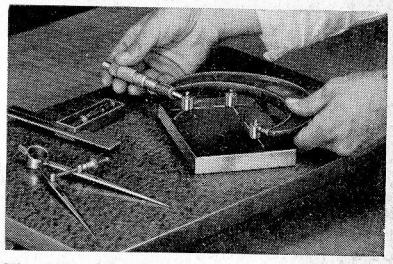


Fig. 110. Measuring Across Toolmaker's Buttons Mounted on Jig Plate

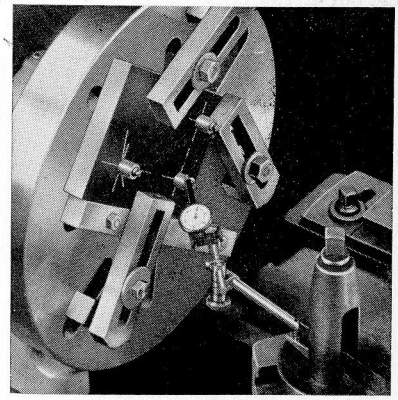
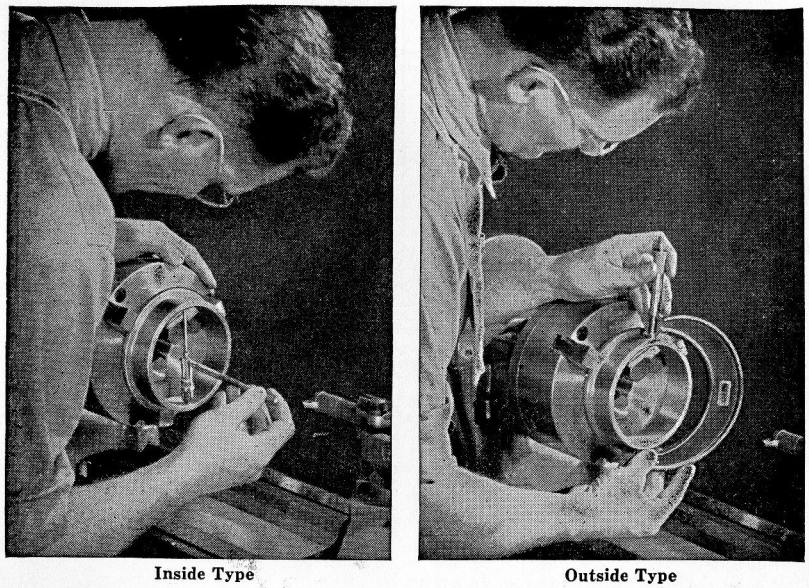


Fig. 111. Locating Work on Face Plate with Toolmaker's Button and Indicator

After the first hole is bored, each additional hole is located and bored in turn by the same process. When this method is carefully followed, the center distances between the holes can be held to extremely close tolerances.



Inside Type Outside Type

Fig. 111. Using Micrometers to Measure Work in the Lathe

Using Micrometers

To obtain accurate and consistent measurements with micrometers, it is necessary for the user to develop a sense of "feel" for the correct tension of the micrometer against the work and then exercise due care in taking measurements and readings. The inside micrometer is used to measure internal diameters and is read in the same manner as the outside micrometer, as described on page 39. Micrometers should be checked frequently with master gauge blocks to insure against any inaccuracy developing in the instruments themselves.

Compound Rest Adjustment for Fine Cuts

When the compound rest is set at approximately 84° (84° 16′ to be exact) each graduation on the compound rest graduated collar represents an angular movement of .001 in., and a cross-feed movement of .0001 in. or a reduction in the diameter of the work of .0002 in.

This method of adjusting the cutting tool will be of help when taking fine precision finishing cuts fractional thousandths in depth. See page 100.

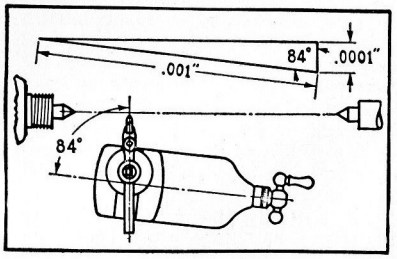


Fig. 112. Compound Rest Set for Precision Cross-feed Adjustment

This method can also be used to advantage for final grinding operations performed with the tool post grinding attachment on the lathe.

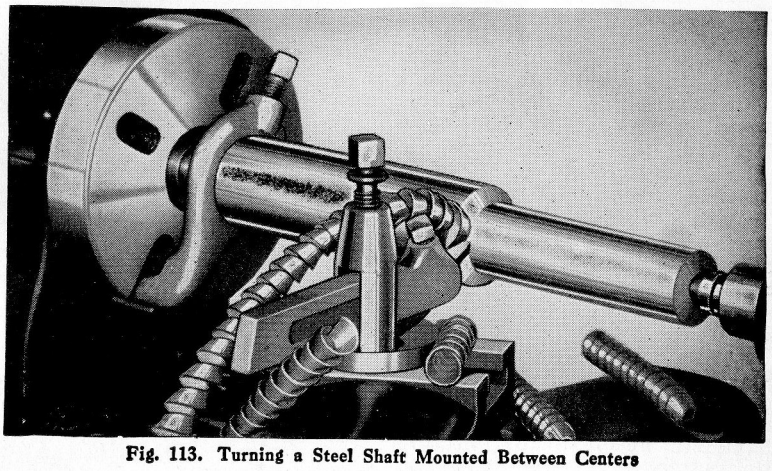


Fig. 113. Turning a Steel Shaft Mounted Between Centers

Chapter VI

PLAIN TURNING

The illustration above shows the lathe in operation machining a shaft between the lathe centers. Whenever possible, work should be mounted in this way for machining as heavier cuts may be taken because the work is supported on both ends.

Locating Center Holes

There are several good methods for accurately locating the center holes which must be drilled in each end of the work before it can be mounted on the lathe centers for machining.

Divider Method

Chalk the ends of the shaft, set the dividers to approximately one-half the diameter of the shaft, and scribe four lines across each end, as shown in Fig. 114.

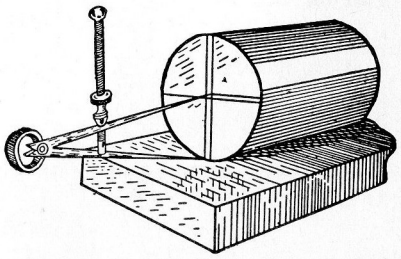


Fig. 114. Locating Centers with Dividers

Combination Square Method

Hold the center head of a combination square firmly against the shaft, as shown in Fig. 115, and scribe two lines close to the blade across each end of the shaft.

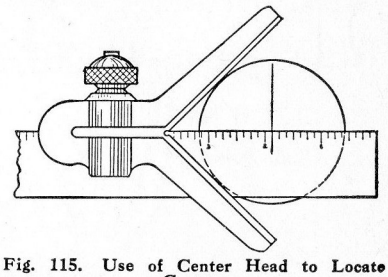


Fig. 115. Use of Center Head to Locate Centers

Hermaphrodite Caliper Method

Chalk each end of the work, set the hermaphrodite caliper a little over half the diameter, and scribe four lines as shown in Fig. 116.

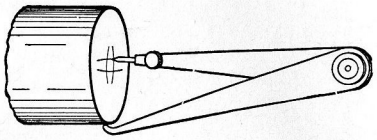


Fig. 116. Centering with Hermaphrodite Calipers

Centering Irregular Shapes

Work that is irregular in shape may be centered with a surface gauge and V-block, as shown in Fig. 117.

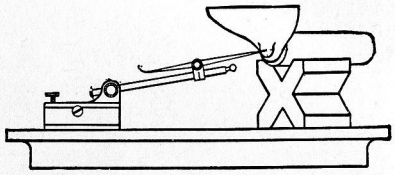


Fig. 117. Centering an Irregular Shape

Bell Center Punch

The bell centering cup is placed over the end of the work and the center punch or plunger is struck a sharp blow with the hammer, automatically locating the center.

Punching the Center

Place the center punch vertically at the center point and tap with a hammer, making a mark sufficiently deep so that the work will revolve on the center points when placed in the lathe.

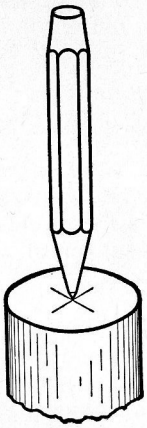


Fig. 118. Punching the Center

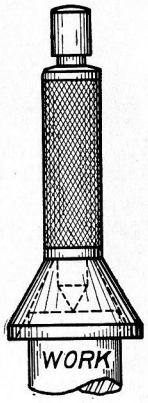


Fig. 119. Bell Center Punch

Test on Centers

After a piece has been center punched it should be tested on centers, as shown in Fig. 120, to make sure that the centers are accurately located. Spin the work with the left hand and mark the high spots on each end of the cylinder with a piece of chalk in the right hand.

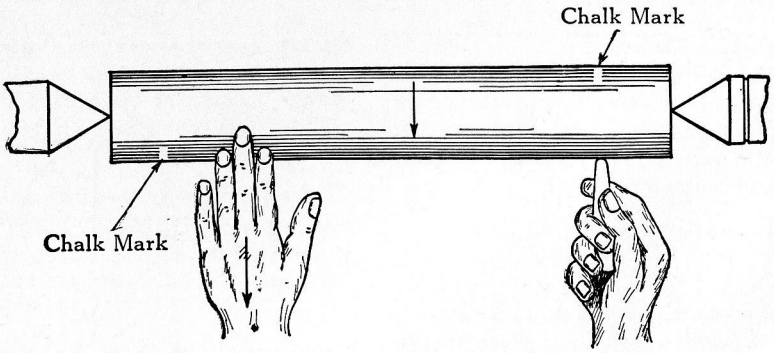


Fig. 120. Testing the Accuracy of Center Punch Marks

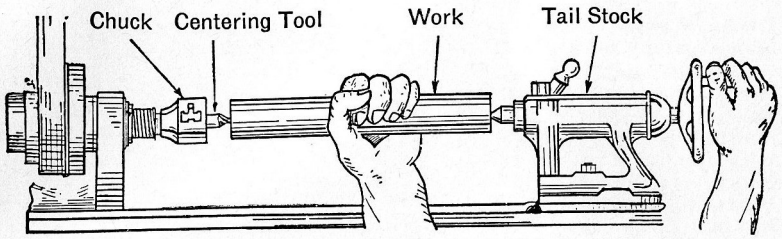


Fig. 121. Drilling the Center Hole in the End of a Shaft

Changing Location of Center Holes

If the centers have not been accurately located, the position of the center punch mark can be changed by placing the center punch at an angle, as shown in Fig. 122, and driving the center over. The shaft should be securely clamped in a vise while this is done.

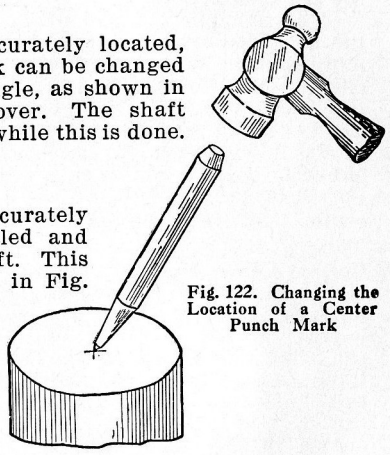


Fig. 122. Changing the Location of a Center Punch Mark

Drilling the Center Holes

After the centers have been accurately located, center holes should be drilled and countersunk in each end of the shaft. This may be done in the lathe, as shown in Fig. 121, or in a drill press. A combination center drill and countersink, shown in Fig. 124, or a small twist drill followed by a 60° countersink, shown in Fig. 123, may be used.

Center Drill and Countersink

The combination center drill and countersink is usually used for drilling center holes. Several standard sizes suitable for various sizes of work are available, as listed in the tabulation below.

Some care should be exercised in drilling the center holes. The spindle speed should be about 150 r. p. m. and the drill should not be crowded. If the drill is crowded and the point is broken off in the work it may be necessary to heat the end of the shaft to a cherry red and allow it to cool slowly so that the drill point will be annealed and can be drilled out.

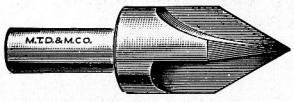
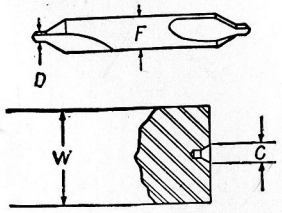


Fig. 123. A 60° Countersink



Fig. 124. Combination Center Drill and Countersink

SIZE OF CENTER HOLE FOR $\frac{3}{16}$ IN. TO 4 IN. DIAMETER SHAFTS



Diameter of Work W	Large Diameter of Countersunk Hole C	Diameter of Drill D	Diameter of Body F
$\frac{3}{16}$ in. to $\frac{5}{16}$ in.	$\frac{1}{8}$ in.	$\frac{5}{64}$ in.	$\frac{3}{16}$ in.
$\frac{3}{8}$ in. to 1 in.	$\frac{3}{16}$ in.	$\frac{7}{64}$ in.	$\frac{1}{4}$ in.
1 $\frac{1}{4}$ in. to 2 in.	$\frac{1}{4}$ in.	$\frac{1}{8}$ in.	$\frac{5}{16}$ in.
2 $\frac{1}{4}$ in. to 4 in.	$\frac{5}{16}$ in.	$\frac{3}{16}$ in.	$\frac{7}{16}$ in.

Drilling Center Holes with a Lathe Chuck

Small diameter rods that can be passed through the headstock spindle and short shafts are easily centered with the aid of a universal chuck, as shown in Fig. 125. When this method is used, the end of the shaft should be faced smooth before drilling the center hole.

The unsupported end of the shaft should not extend more than 10 in. beyond the chuck jaws. Shafts that are too large to pass through the headstock and too long to be held firmly by the chuck jaws alone can be supported on the outer end in a center rest (See pages 66 and 92.)

Correct Center Hole

To be correct, the center hole must be the size required for the diameter of the shaft, as listed in the tabulation on page 45, and the countersink must fit the center point perfectly, as shown in Fig. 126. There must also be sufficient clearance at the bottom of the countersink.

When drilling center holes, allow for the thickness of the metal that will be faced off of the end; otherwise, the center holes will be too small to support the shaft after the ends are faced.

Poorly Drilled Center Holes

One of the most common causes of unsatisfactory lathe work is poorly drilled center holes. Fig. 127 shows a shallow center hole with incorrect angle and no clearance for the tip of the center point. Fig. 128 shows a center hole that has been drilled too deep. Accuracy cannot be expected when center holes are poorly made, and the lathe centers may be damaged.

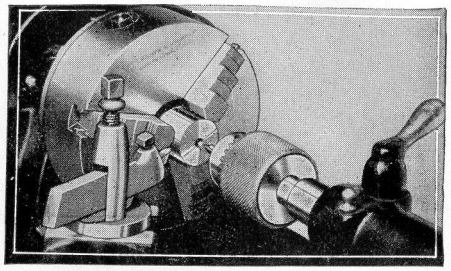


Fig. 125. Drilling a Center Hole with Stock Held in Lathe Chuck and Drill in Tailstock Spindle Chuck

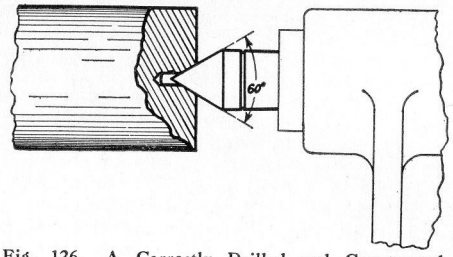


Fig. 126. A Correctly Drilled and Countersunk Hole Fits the Lathe Center Perfectly

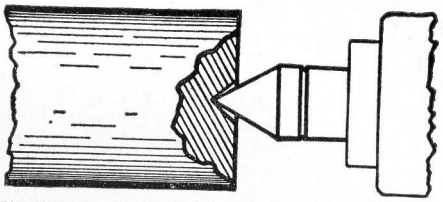


Fig. 127. A Poorly Drilled Center Hole, Too Shallow and Incorrect Angle

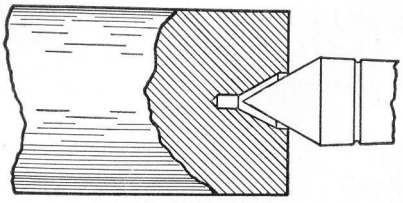


Fig. 128. An Incorrect Center Hole, Drilled Too Deep to Fit Center

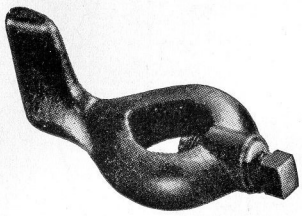


Fig. 129. Standard Lathe Dog

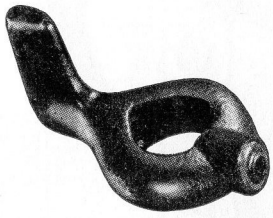


Fig. 130. Safety Lathe Dog

Lathe Dogs for Driving Work on Centers

The common lathe dog shown in Fig. 129 is the most popular type. Fig. 130 shows a safety lathe dog which has a headless set screw and is not likely to catch in the operator's sleeve. Fig. 132 shows a clamp lathe dog, used principally for rectangular work in the lathe. When attaching the lathe dog to the work, make sure that the set screw is securely tightened.

Mounting Centers in the Lathe Spindles

Before mounting the lathe centers in the headstock or tailstock spindle, thoroughly clean the centers, the tapered holes and the spindle sleeve "A," Fig. 131. A very small chip or a little dirt will cause the center to run out of true. Use cloth and stick to clean taper hole. Do not insert finger in revolving spindle.

Removing the Lathe Centers

With a piece of rag in your right hand, hold the sharp point of the headstock center, and with the left hand give the center a sharp tap with a rod through the spindle hole. Fig. 133 shows a steel rod with a small bushing attached for removing the headstock spindle center and taper sleeve.

To remove the tailstock center, turn the tailstock handwheel to the left until the end of the tailstock screw bumps the end of the center. This will loosen the center and it may be removed from the spindle.

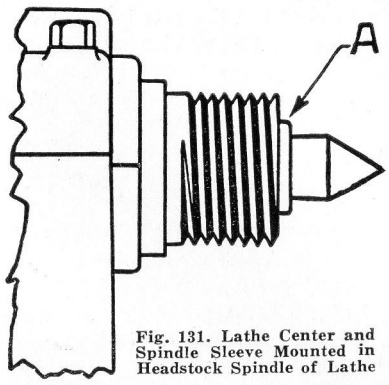


Fig. 131. Lathe Center and Spindle Sleeve Mounted in Headstock Spindle of Lathe

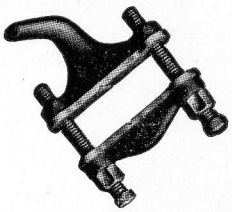


Fig. 132. Clamp Lathe Dog

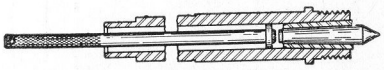


Fig. 133. Knock Out Bar for Removing Headstock Spindle Center

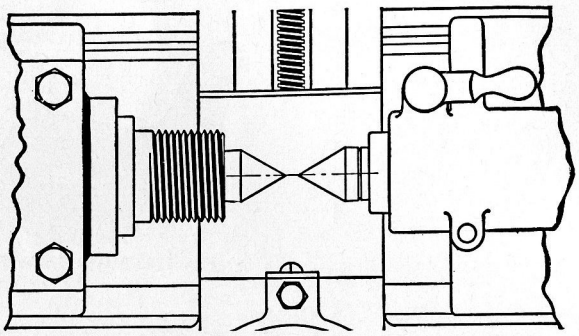


Fig. 134. Checking the Alignment of the Lathe Centers

Check Alignment of Centers

Before mounting work between the lathe centers they should be checked for alignment, as shown in Fig. 134. If the tailstock center does not line up, loosen the tailstock clamp bolt and set over the tailstock top in the proper direction by adjusting the tailstock set over screws. (See page 51.)

Mounting the Work Between Centers

Place a drop of oil in the center hole for the tailstock center point before mounting the work between centers. The tail of the lathe dog should fit freely into the slot of the face plate so that the work rests firmly on both the headstock center and the tailstock center, as shown in Fig. 136. Make sure that the lathe dog does not bind in the slot of the face plate, as shown in Fig. 135.

The tailstock center should not be tight against the work, but should not be too loose. The work must turn freely, for if the tail center is too tight it will stick and may be ruined.

Expansion of Work

When work is machined in the lathe it may become hot and expand. The expansion of work mounted between centers will cause it to bind, making it necessary to stop the lathe and readjust the tailstock center. When machining a long shaft, several readjustments of the tailstock center may be required.

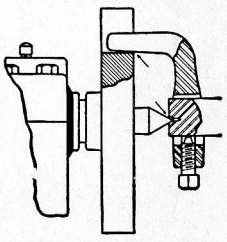


Fig. 135. Incorrect Mounting

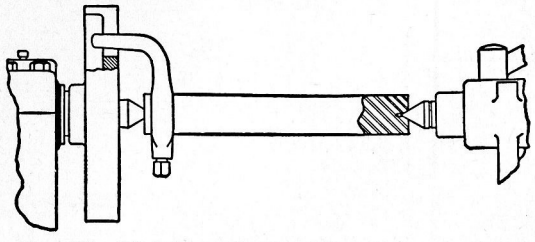


Fig. 136. Work Properly Mounted Between the Lathe Centers, Ready to Machine

Facing the Ends

Before turning the diameter of a shaft, the ends should be faced square. Grind the cutter bit as shown in Fig. 74, page 32, and set the cutting edge exactly on center, as shown in Fig. 137. Be careful not to break the point of the tool against the tailstock center. Feed the tool out to face the end, as shown in Fig. 138.

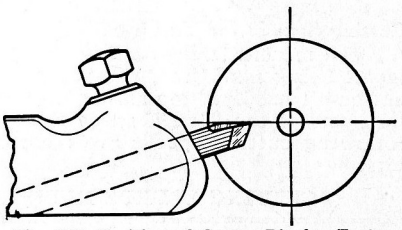


Fig. 137. Position of Cutter Bit for Facing the End of a Shaft

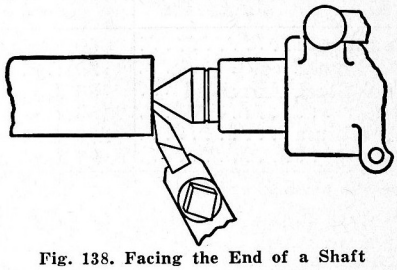


Fig. 138. Facing the End of a Shaft

Position of Tool for Turning

Grind the cutter bit for turning as shown in Fig. 64, page 30. The cutting edge of the cutter bit and the end of the tool holder should not extend over the edge of the compound rest any farther than necessary. (See "A" and "B," Fig. 139.)

The tool should be set as shown in Fig. 140 so that if the tool slips in the tool post it will not dig into the work, but instead it will move in the direction of the arrow away from the work.

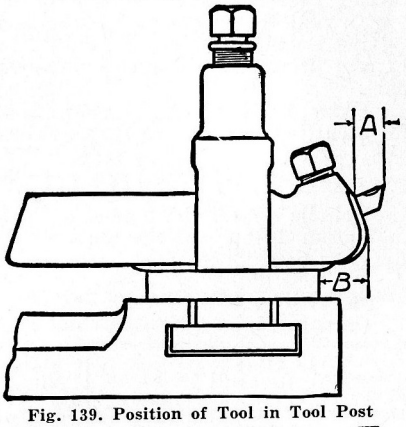


Fig. 139. Position of Tool in Tool Post

Direction of Power Feed

The feed of the tool should be toward the headstock, if possible, so that the pressure of the cut is on the head spindle center which revolves with the work.

Rate of Power Feed

The rate of the power feed depends on the size of the lathe, the nature of the work, and the amount of stock to be removed.

On a small lathe a feed of .008 in. per revolution of the spindle may be used, but on larger sizes of lathes feeds as coarse as .020 in. are often used for rough turning. Care must be taken when turning long slender shafts as a heavy cut with a coarse feed may bend the shaft and ruin the work.

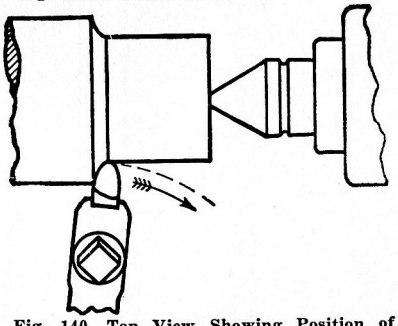


Fig. 140. Top View Showing Position of Tool for Turning

Cutting Speeds for Turning

The most efficient cutting speed for turning varies with the kind of metal being machined, the depth of the cut, the feed and the type of cutter bit used. If too slow a cutting speed is used, much time may be lost, and if too high a speed is used the tool will dull quickly. The following cutting speeds are recommended for high speed steel cutter bits:

CUTTING SPEEDS IN SURFACE FEET PER MINUTE

Kind of Metal	Roughing Cuts .010 in. to .020 in. Feed	Finishing Cuts .002 in. to .010 in. Feed	Cutting Screw Threads
Cast Iron	60 f. p. m.	80 f. p. m.	25 f. p. m.
Machine Steel	90 f. p. m.	100 f. p. m.	35 f. p. m.
Tool Steel, Annealed	50 f. p. m.	75 f. p. m.	20 f. p. m.
Brass	150 f. p. m.	200 f. p. m.	50 f. p. m.
Aluminum	200 f. p. m.	300 f. p. m.	50 f. p. m.
Bronze	90 f. p. m.	100 f. p. m.	25 f. p. m.

See also pages 98 and 108.

If a cutting lubricant is used, the above speeds may be increased 25% to 50%. When using tungsten-carbide tipped cutting tools, the cutting speeds may be increased from 100% to 800%. See page 108.

To find the number of revolutions per minute required for a given cutting speed, in feet per minute, multiply the given cutting speed by 12 and divide the product by the circumference (in inches) of turned part.

Example: Find the number of revolutions per minute for 1 in. shaft for a cutting speed of 90 ft. per minute.

$$\frac{90 \times 12}{3.1416 \times 1} = 343.77 \text{ r. p. m.}$$

Spindle speeds for various diameters and metals are listed in the tabulation below to eliminate the necessity of making calculations. Refer to page 23 for spindle speeds of various sizes of lathes.

SPINDLE SPEEDS IN R.P.M. FOR TURNING AND BORING
Calculated for Average Cuts with High Speed Steel Cutter Bits

Diameter in Inches	Alloy Steels 50 f. p. m.	Cast Iron 75 f. p. m.	Machine Steel 100 f. p. m.	Hard Brass 150 f. p. m.	Soft Brass 200 f. p. m.	Aluminum 300 f. p. m.
1	191	287	382	573	764	1146
2	95	143	191	287	382	573
3	64	95	127	191	254	381
4	48	72	95	143	190	285
5	38	57	76	115	152	228
6	32	48	64	95	128	192
7	27	41	55	82	110	165
8	24	36	48	72	96	144
9	21	32	42	64	84	126
10	19	29	38	57	76	114
11	17	26	35	52	70	105
12	16	24	32	48	64	96
13	15	22	29	44	58	87
14	14	20	27	41	54	81
15	13	19	25	38	50	75
16	12	18	24	36	48	72

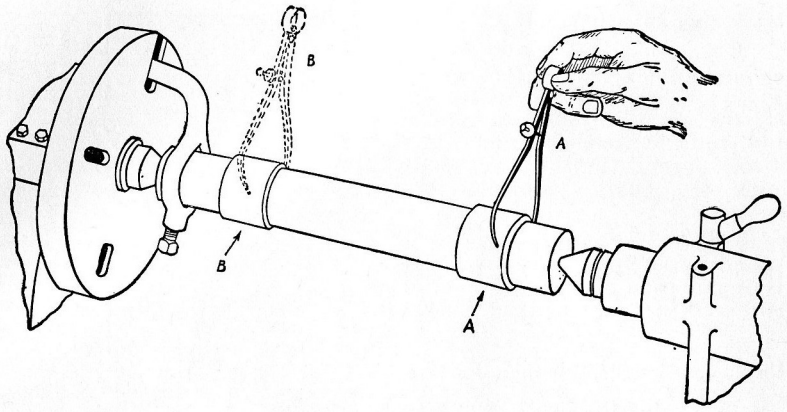


Fig. 141. Testing the Alignment of the Lathe Centers for Straight Turning

Testing Alignment of Centers

After taking the first roughing cut across a shaft, check the diameter at each end of the cut with calipers or micrometers to make sure the lathe is turning straight. Sometimes when the position of the tailstock is changed for a different length of work, it will require adjustment. This is especially true on old lathes which may have worn spots and burrs on the bed.

Fig. 141 shows a good method of testing for the alignment of the lathe centers. Two collars, A and B, turned on a shaft about 1½ in. in diameter and 10 in. long, are machined with a fine finishing cut without changing the adjustment of the cutting tool. Collar A is measured, and without making any adjustment on the caliper, collar B is tested to see how it compares with collar A. If collar A is not the same diameter as collar B, then the adjustment of centers is not correct, and the tailstock top should be adjusted in the direction required.

Adjustment of Tailstock Top

The tailstock top is adjusted by releasing one of the adjusting screws of the tailstock top and tightening the opposite screw a similar distance. Then take another test cut on the collars, measure and continue this operation until the desired degree of accuracy is obtained.

There is a mark on the end of the tailstock where the bottom and top join to show the relative position of the tailstock top and bottom. For fine, accurate work, this mark should not be depended upon, but the alignment test should be made as described above to be sure that the centers are in line.

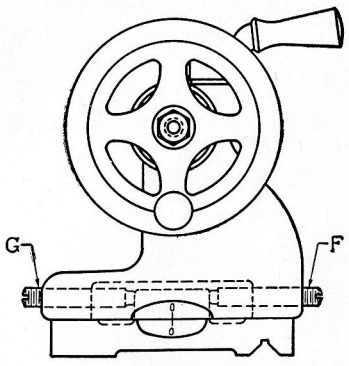


Fig. 142. Tailstock Set on Center

Machining to a Shoulder

A good method for locating a shoulder on a shaft is shown in Fig. 143. After chalking the shaft, set the hermaphrodite calipers to the required dimension and scribe a line around the revolving shaft with the sharp point on the caliper.

Fig. 144 shows the use of a round nosed turning tool for finishing a shoulder having a fillet corner. (See page 31, Fig. 67.)

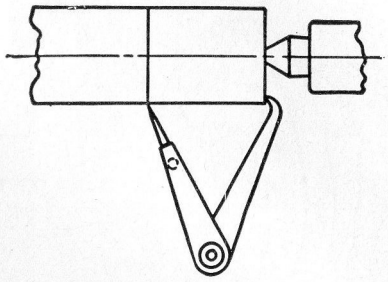


Fig. 143. Measuring with Hermaphrodite Caliper to Locate a Shoulder

Locating Shoulders with a Parting Tool

In production work where a quantity of pieces are required, shoulders are usually located with a parting tool, as shown in Fig. 146, before the diameter is machined.

When a square corner is required, as for a bearing, it is customary to neck or undercut the shoulder slightly, as shown in Fig. 145.

A firm joint caliper is convenient for measuring when facing the ends of a shaft to length or for measuring between two shoulders, as shown in Fig. 147.

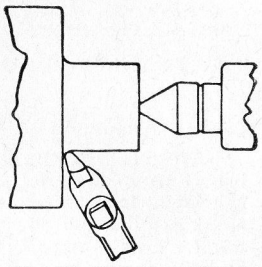


Fig. 144. Finishing a Shoulder with a Fillet

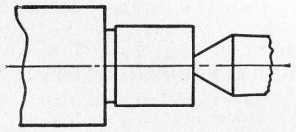


Fig. 145. Detail of a Shoulder with a Recess

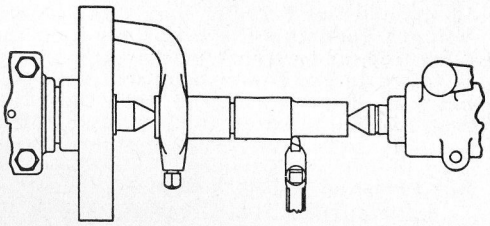


Fig. 146. Location of a Shoulder Marked with a Parting Tool Before Turning the Diameter

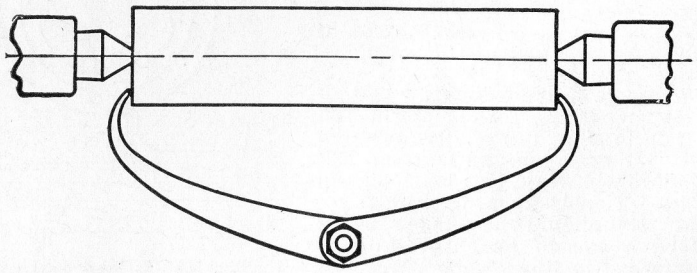


Fig. 147. Measuring the Length of a Shaft with a Firm Joint Caliper

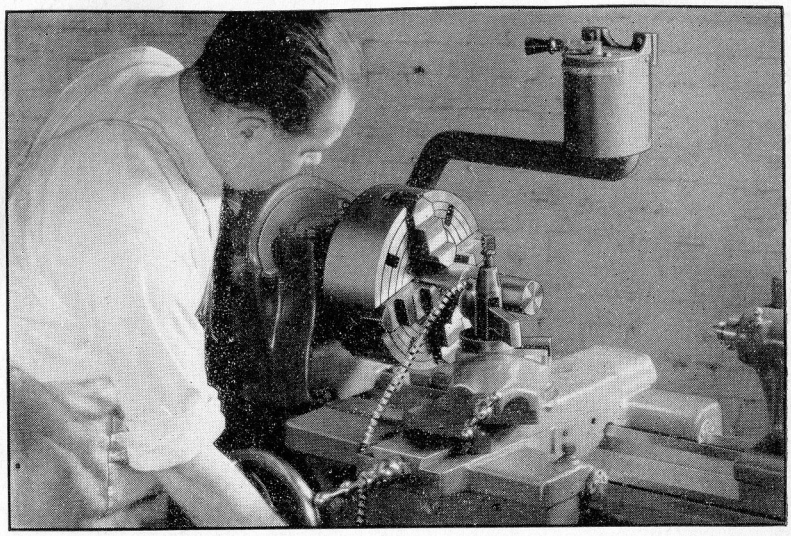


Fig. 148. Machining a Shaft in an Independent Chuck

Chapter VII

CHUCK WORK

Work that cannot readily be mounted between the lathe centers is usually held in a chuck, as shown above, for machining. Several types of chucks are used, but the most popular are the 4-jaw Independent chuck and the 3-jaw Universal chuck shown below.

A 4-jaw Independent chuck has four reversible jaws, each of which may be independently adjusted. This type of chuck is recommended if the lathe is to have but one chuck, as it will hold square, round and irregular shapes in either a concentric or an eccentric position.

The 3-jaw Universal chuck is used for chucking round and hexagonal work quickly, as the three jaws move simultaneously and automatically center the work. Two sets of jaws are required, one set for external chucking and the other for internal chucking.

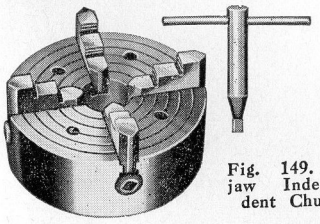


Fig. 149. 4-jaw Independent Chuck

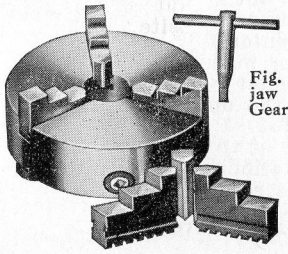


Fig. 150. 3-jaw Universal Geared Scroll Chuck

Mounting Chuck on Spindle

Before mounting a chuck or a face plate on the lathe spindle, thoroughly clean and oil the threads of the spindle and the chuck back. Also clean the shoulder of the spindle where the chuck back fits against it. A very small chip or burr at this point will prevent the chuck from running true.

Hold the chuck against the spindle nose with the right hand and arm and turn the lathe spindle cone with the left hand to screw the chuck onto the spindle just tight enough to hold it securely.

Do not run the lathe with power while screwing the chuck onto the spindle, and do not spin the chuck up to the shoulder or it may be very difficult to remove.

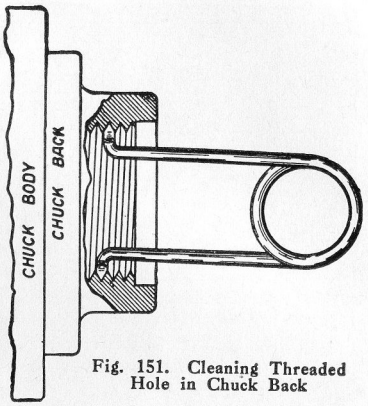


Fig. 151. Cleaning Threaded Hole in Chuck Back

The Independent Chuck

The independent chuck is used more than any other type of chuck because it will hold practically any shape and can be adjusted to any degree of accuracy required.

Concentric rings scribed on the face of the chuck permit centering round work approximately, as it is placed in the chuck. To center more accurately, the lathe is started and a piece of chalk is held lightly against the revolving work, as shown in Fig. 152. The lathe is then stopped and the jaw opposite the chalk mark is loosened slightly. The opposite jaw is then tightened. The test is repeated until the work is centered with the necessary accuracy. All four jaws must be securely tightened before starting to machine the work.

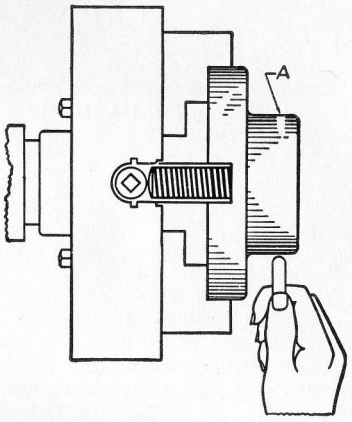


Fig. 152. Centering Work in an Independent Chuck

Use of Center Indicator

The center indicator is used for accurately centering work that has been laid out and center punched for drilling and boring. The short end of the center indicator is placed in the center punch mark and the tailstock center point is brought up close to the opposite end, as shown in Fig. 153. For accurate work, the long end of the indicator should remain stationary when the lathe spindle is revolved.

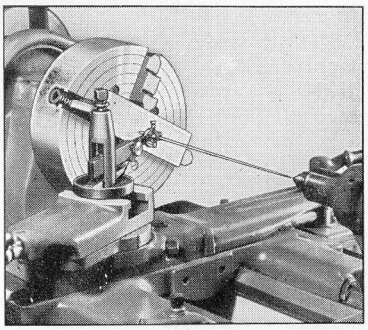


Fig. 153. Centering Work with a Center Indicator

Centering Work With Dial Test Indicator

A sensitive dial indicator may be used for accurately centering work having a smooth surface. The dial of the indicator is graduated to read in thousandths of an inch so that practically any required degree of accuracy may be obtained.

The indicator is placed in contact with the part to be centered, as shown in Fig. 154, and the hand on the indicator dial is watched as the lathe spindle is revolved slowly by hand. The chuck jaws are adjusted as described on page 54 until the required degree of accuracy is obtained.

The part to be centered should also be tested on the face for wobble as shown in Fig. 155.

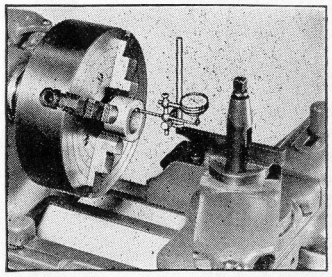


Fig. 154. Centering Work with a Dial Test Indicator

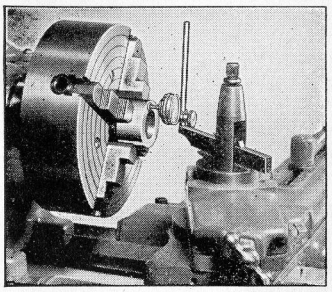


Fig. 155. Testing Face of Work with Dial Indicator for Wobble

Removing Chuck from Lathe Spindle

To start a chuck so it can be removed from the lathe spindle, engage the back gears, place a wood block between the chuck jaw and back ways of bed, as shown in Fig. 156, and turn cone pulley by hand. After starting the chuck, place a board across the bed ways to protect them from damage in case the chuck is dropped off the spindle. This procedure also applies to face plates.

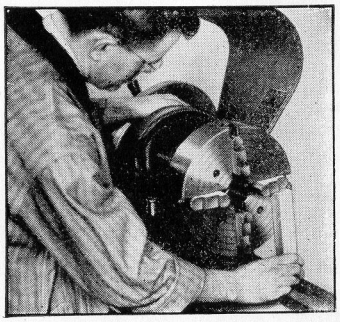


Fig. 156. Removing a Chuck from Lathe Spindle

Practical Sizes of Chucks

Lathe chucks should be carefully selected for the size of the lathe and the work for which they are to be used. If the chuck is too small, the capacity of the lathe is restricted, but if it is too large the jaws may strike the lathe bed and the chuck will be awkward to use and difficult to handle.

The most practical sizes of chucks for use with various sizes of lathes are listed in the tabulation below.

Practical Sizes of Chucks for Lathes

Size of Lathe	4-Jaw Independent Lathe Chuck	3-Jaw Universal Geared Scroll Chuck
9-in. lathe	6 in.	5 in.
10-in. lathe	6 in.	5 in.
13-in. lathe	7½ in.	6 in.
14½-in. lathe	9 in.	7½ in.
16-in. and 16/24-in. lathes	10 in.	7½ in.

The Universal Chuck

Round and hexagonal work may be chucked quickly in the universal chuck as all three jaws move simultaneously and automatically center the work within a few thousands of an inch. This type of chuck will usually center work within .003 in. when new, but when the scroll becomes worn this degree of accuracy cannot be expected.

Since there is no way to adjust the jaws independently on this type of chuck, it is not used where extreme accuracy is required. The 4-jaw independent chuck should always be used when work must be centered to run dead true. However, if no independent chuck is available shims may be placed between the chuck jaws and the work to compensate for the inaccuracy of universal chuck.

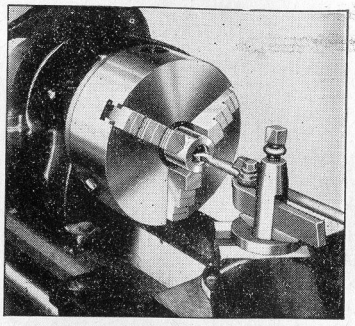


Fig. 157. Round Work Held in Universal Chuck

Headstock Spindle Chuck

The headstock spindle chuck, shown in Figs. 158 and 159, is similar to a drill chuck except that it is hollow and is threaded so that it may be screwed on to the spindle nose of the lathe.

This type of chuck is suitable for holding bars, rods and tubes that are passed through the headstock spindle of the lathe, also other small diameter work. It is more accurate than the average universal chuck and will usually center work within .002 in.

The headstock spindle chuck is inexpensive and for some classes of work it may be used instead of the more expensive draw-in collet chuck.

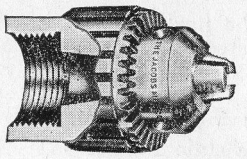


Fig. 158. Hollow Headstock Spindle Chuck

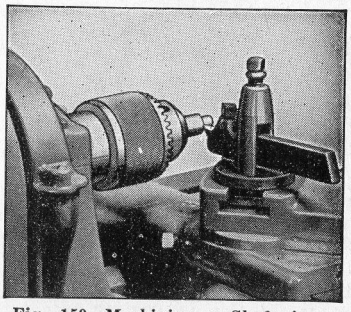


Fig. 159. Machining a Shaft in a Hollow Headstock Chuck

Drill Chuck

Drill chucks are used in both the headstock spindle and the tailstock spindle of the lathe for holding drills, reamers, taps, etc. There are several types of drill chucks on the market and some do not have sufficient accuracy and holding power for satisfactory use on the lathe. A good drill chuck will hold drills concentric within .002 in. or .003 in. and should have a wrench or a pinion key for tightening.

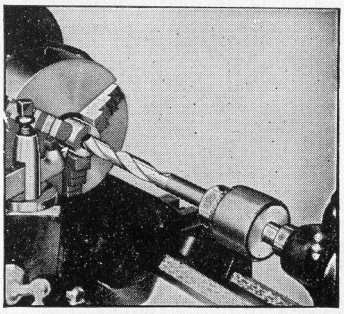


Fig. 160. Drill Chuck Mounted in Tailstock Spindle

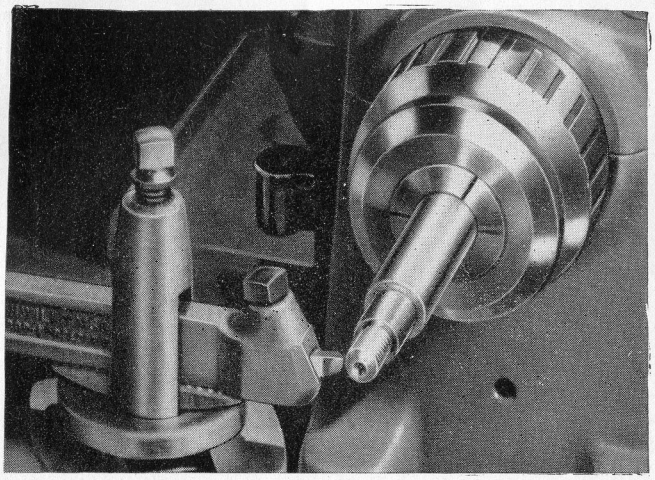


Fig. 161. Machining a Part Held in a Draw-in Collet Chuck

Draw-in Collet Chuck

The draw-in collet chuck is the most accurate of all types of chucks and is used for precision work, such as making small tools and manufacturing small parts for watches, typewriters, radios, etc. The collets are made for round, square and other shapes, as shown in Figs. 162, 164 and 165. The work held in the collet should not be more than .001 in. smaller or .001 in. larger than the collet size. Collets are usually made of heat-treated steel, but for some classes of work brass collets are often used.

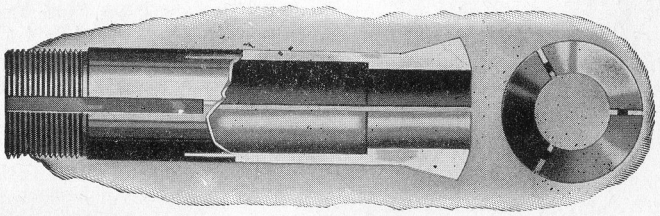


Fig. 162. Side View and End View of a Spring Collet for Round Work

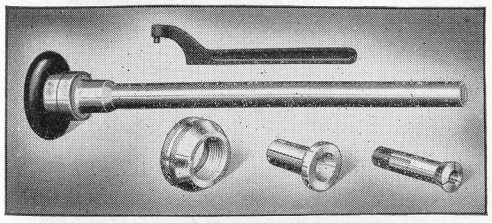


Fig. 163. Handwheel Draw-in Collet Chuck

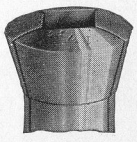


Fig. 164. Square Collet

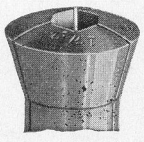


Fig. 165. Hexagon Collet

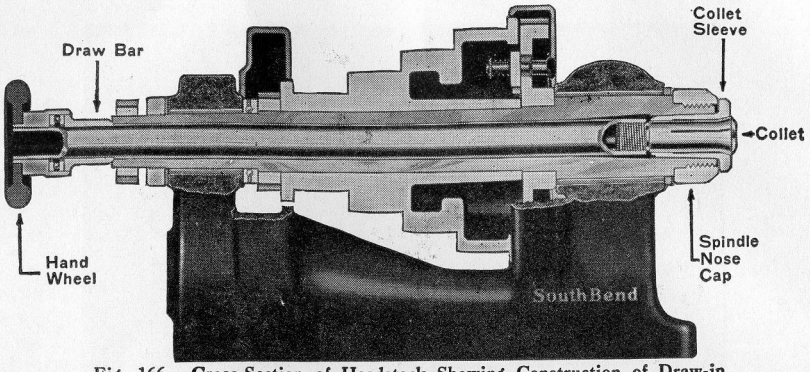


Fig. 166. Cross-Section of Headstock Showing Construction of Draw-in Collet Chuck Attachment

Construction of Draw-in Collet Chuck

The construction of the draw-in collet chuck is shown above in Fig. 166. The hollow draw-bar with handwheel attached extends through the headstock spindle of the lathe and is threaded on the right end to receive the spring collet. Turning the handwheel to the right draws the collet into the tapered closing sleeve and tightens the collet on the work. Turning the handwheel to the left releases the collet.

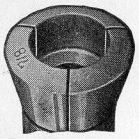


Fig. 167. Pot Collet

Step Chuck and Closer

The spring collet may be replaced with a step chuck and closer, as shown in Fig. 168, for holding discs such as gear blanks, etc. A pot collet, shown in Fig. 167, may be used in place of the step chuck for small diameters.

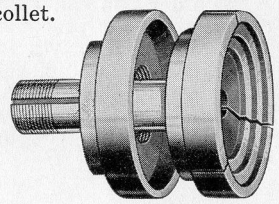


Fig. 168. Step Chuck and Closer

Handlever Draw-in Collect Chuck

The handlever draw-in collet chuck, shown in Fig. 169, is similar to the handwheel type draw-in collet chuck except that the collet is opened and closed by moving the hand lever to the right or left. This permits gripping or releasing the work without stopping the lathe spindle if desired.

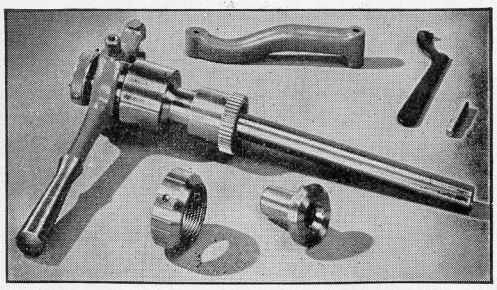


Fig. 169. Handlever Type Draw-in Collet Chuck Attachment

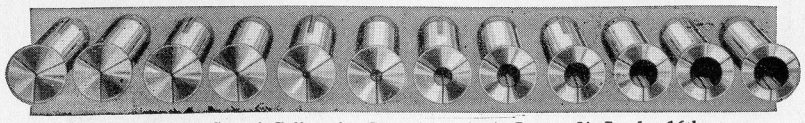


Fig. 170. A Set of Collets for Round Work $\frac{1}{16}$ In. to $\frac{3}{4}$ In. by 16ths

Chapter VIII

TAPER TURNING AND BORING

There are three methods of turning and boring tapers in the lathe: by setting over the tailstock; by using the compound rest; and by using the taper attachment of the lathe. The method used depends on the length of the taper, the angle of taper and the number of pieces to be machined.

Cutter Bit Must Be on Center

The cutting edge of the tool must be set exactly on center, as shown in Fig. 173, to turn or bore an accurate taper. That is, the cutting edge of the lathe tool must be exactly the same height as the point of the tailstock center. The position of the tool applies to all methods of turning and boring tapers.

Taper Turning with Compound Rest

The compound rest of the lathe is usually used for turning and boring short tapers and bevels, especially for bevel gear blanks and for die and pattern work, etc. The compound rest swivel is set at the required angle and the taper is machined by turning the compound rest feed screw by hand. See Figs. 171 and 172.

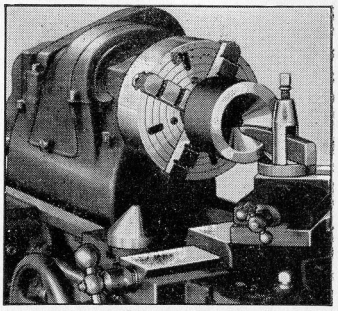


Fig. 171. Machining a Conical Punch and Die with Compound Rest

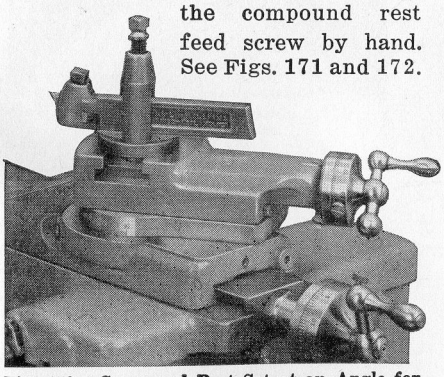


Fig. 172. Compound Rest Set at an Angle for Taper Turning and Boring

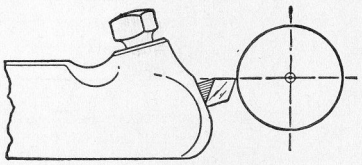


Fig. 173. Lathe Tool Cutter Bit Set on Center for Taper Turning

Truing a 60° Center

A good example of the use of the compound rest for short tapers is shown in Fig. 174. An electric grinding attachment is mounted on the lathe in place of the tool post and the compound rest is swiveled to the correct angle to permit feeding the grinding wheel across the 60° center point to grind it true. Lathe centers are usually too hard to be machined with a cutting tool. See pages 100 and 101.

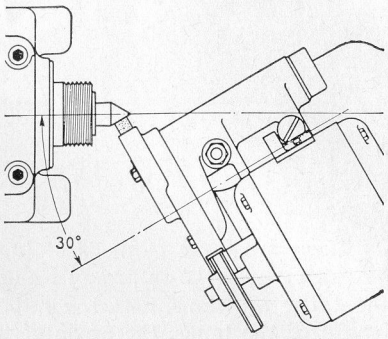


Fig. 174. Truing a 60° Lathe Center Point

Testing the Angle

All angular or bevel turning should be tested with a gauge of some kind, as it is difficult to read the graduations with sufficient accuracy to set the compound rest swivel for an exact taper. Fig. 175 shows the use of a center gauge for testing the 60° angle of a lathe center point.

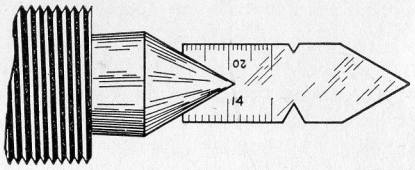


Fig. 175. Testing the Angle of a Lathe Center Point

Taper Turning with Tailstock Set Over

Work that can be machined between centers can be turned taper by setting over the tailstock top, as shown in Figs. 177, 178 and 179. This method cannot be used for boring tapers.

The amount the tailstock top must be set over depends on the amount of the taper per foot and the over-all length of the work. With the same amount of set over, pieces of different lengths will be machined with different tapers, as shown in Fig. 177. Notice that the tailstock center is set over one-half the total amount of the taper for the entire length of the work.

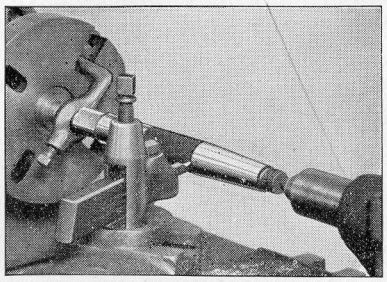


Fig. 176. Machining a Taper with the Tailstock Top Set Over

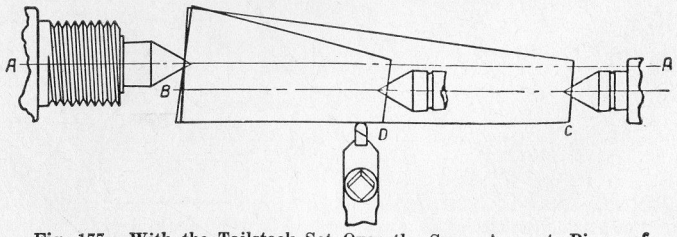


Fig. 177. With the Tailstock Set Over the Same Amount, Pieces of Different Lengths are Machined with Different Tapers

How to Calculate Amount of Setover for Tailstock

Tapers are usually specified in "inches per foot." For example, some Brown & Sharpe tapers are ½ in. per foot. To machine a Brown & Sharpe taper of ½ in. per foot on a shaft exactly one foot long, the center should be set over ¼ in. or .250 in. If the piece is only 10 in. long, then the amount of setover would be $\frac{10}{12}$ of .250 in. or .208 in. The following rules may be used for calculating the amount of setover:

TAPER IN INCHES PER FOOT GIVEN—Divide the total length of the stock in inches by twelve and multiply this quotient by one-half the amount of taper per foot specified. The result is the amount of setover in inches.

DIAMETERS AT ENDS OF TAPER GIVEN—Divide the total length of the stock by the length of the portion to be tapered and multiply this quotient by one-half the difference in diameters; the result is the amount of the setover.

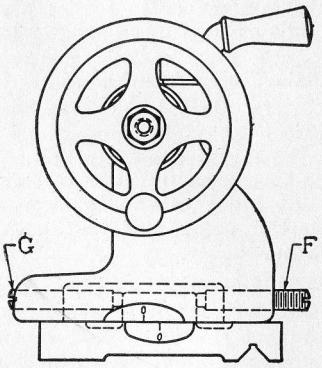


Fig. 178. Tailstock Top Set Off Center for Taper Turning

Adjusting the Tailstock Center

To set over the tailstock center for taper turning, loosen clamping nut of tailstock and back off set screw "F," Fig. 178, the distance required; then screw in set screw "G" a like distance until it is tight, and clamp the tailstock to the lathe bed.

Measuring the Setover

To measure the setover of the tailstock center, place a scale having graduations on both edges between the two centers, as shown in Fig. 179. This will give an approximate measurement.

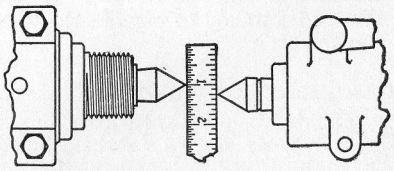


Fig. 179. Measuring the Setover

Fitting Tapers to Gauges

The best way to machine an accurate taper is to fit the taper to a standard gauge. To test the taper, make a chalk mark along the entire length of the taper, place the work in the taper it is to fit and turn carefully by hand. Then remove the work and the chalk mark will show where the taper is bearing.

If the taper is a perfect fit, it will show along the entire length of the chalk mark. If the taper is not perfect, make the necessary adjustment, take another light cut and test again. Be sure the taper is correct before turning to the finished diameter.

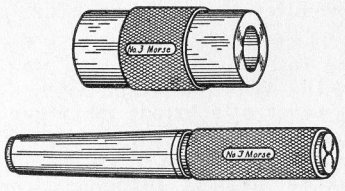


Fig. 180. Morse Standard Taper Plug Gauge and Socket Gauge

Taper Turning with Taper Attachment

The taper attachment is used for turning and boring tapers in tne lathe. It eliminates the necessity of setting over the tailstock, and if desired may be set permanently for a standard taper. The taper attachment does not interfere with using the lathe for straight turning.

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

Graduations on one end of the taper attachment swivel bar indicate the total taper in inches per foot, and on the other end, the included angle of the taper is shown in degrees. See Fig. 182-A.

Plain Taper Attachment

The plain taper attachment (used on all 9" South Bend Lathes) shown at right consists of a bracket attached to the back of the lathe carriage, a compound slide with clamp for locking slide to lathe bed and a connecting bar to connect the slide block of the taper attachment to the compound rest base of the lathe.

When the plain taper attachment is to be used it is necessary to disconnect the cross feed screw by removing the bolt "A" which locks the cross feed nut to the compound rest base of the lathe. This leaves the compound rest base free to slide so that it may be controlled by the taper attachment. Binding screws "B" and "C" are tightened to engage the taper attachment.

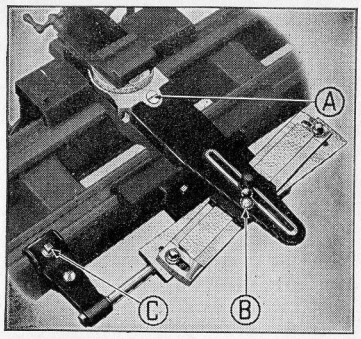


Fig. 181. Plain Taper Attachment

Telescopic Taper Attachment

The telescopic taper attachment (used on 10" to 16" swing South Bend Lathes) shown in the illustration below, Fig. 182, is similar to the plain taper attachment described above except that it is equipped with a telescopic cross feed screw. This feature eliminates the necessity of disconnecting the cross feed screw when the taper attachment is to be used.

The cross feed screw may be used to adjust the turning tool for the required diameter and the taper attachment may then be engaged by tightening binding screws "X" and "Y". To change back to straight turning, it is necessary to loosen binding screws "X" and "Y".

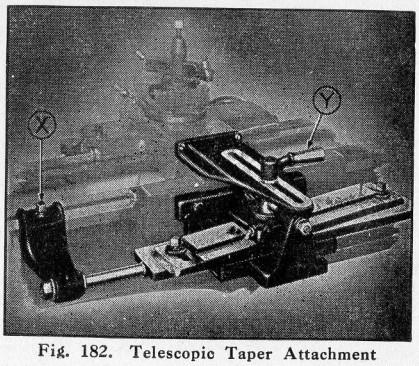


Fig. 182. Telescopic Taper Attachment

Setting the Taper Attachment Swivel Bar

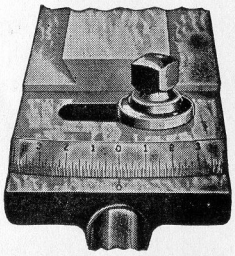


Fig. 182A. Graduations in Inches per Foot on Swivel Bar

Tapers are usually specified in inches per foot or in degrees. When this information is not available the taper in inches per foot should be calculated before setting the taper attachment for taper turning.

To calculate the taper in inches per foot, subtract the diameter in inches at the small end of taper (B, Fig. 183) from the diameter in inches at the large end of the taper (A); divide the result by the length of tapered portion (C) in inches, and multiply by 12. The answer is the taper in inches per foot and indicates the graduation on the "taper per foot" end of the swivel bar which should be set in line with the witness mark in order to machine the desired taper. See Fig. 182A.

Each graduation on the "taper per foot" end of swivel bar represents a total taper of 1/16" per foot. If the taper per foot has been calculated or specified in decimal fractions instead of common fractions, refer to the decimal equivalent table on page 115 for the nearest fractional part of an inch.

When setting the swivel bar for taper turning, remember that the total taper is indicated by the graduations, either in inches per foot or in degrees. For example, if the swivel bar is set at 5° the taper machined will have a total included angle of 5°; that is 2½° (not 5°) each side of center line.

After setting the taper attachment swivel bar to the required angle, take a trial cut and test the taper with a taper gage or micrometers. Some readjustment of the swivel bar will probably be necessary, as it is difficult to align the graduations on swivel bar perfectly with the witness mark. See page 61 for information on fitting and testing tapers with taper gauges.

Standard tapered holes may be hand reamed after boring to standardize the taper and size the hole.

Turning an outside taper is shown in Fig. 183. Boring an inside taper with the aid of a center rest is shown in Fig. 184. See page 92.

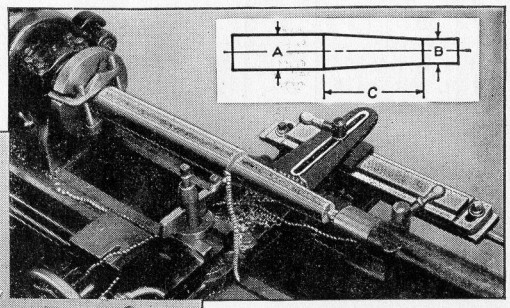


Fig. 183. Taper Turning with the Taper Attachment

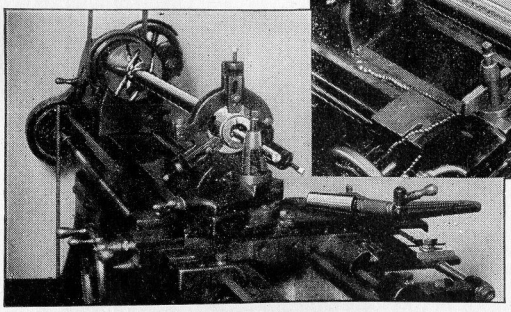


Fig. 184. Boring a Taper with the Taper Attachment

Morse Standard Tapers

Morse Standard Tapers are used for lathe and drill press spindles by most of the manufacturers of lathes and drill presses in the United States. South Bend Lathes have both head and tailstock spindles fitted for Morse Standard Tapers. The dimensions of various sizes of Morse Standard Tapers are listed in the tabulation below.

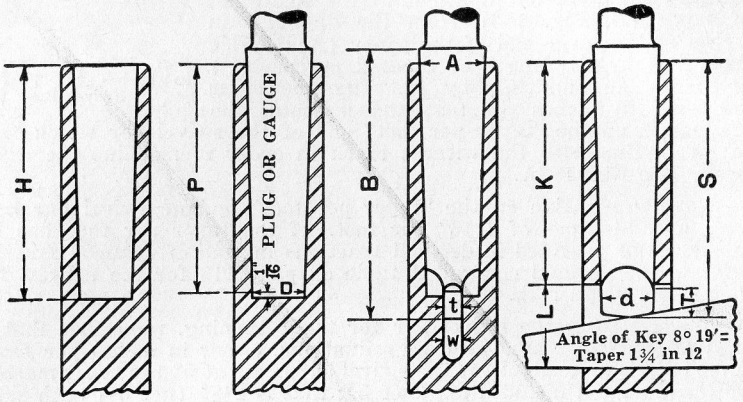


Fig. 185. Chart Showing Principal Dimensions of Morse Standard Tapers Which are Listed in Tabulation Below

DIMENSIONS OF MORSE STANDARD TAPERS

All Dimensions in Tabulations Below Are in Inches

Number of Taper	Diam. of Plug at Small End	Diam. at End of Socket	Shank		Depth of Hole	Standard Plug Depth	Tongue		Keyway		End of Socket to Keyway	Taper per Foot
			Whole Length	Depth			Thickness	Length	Width	Length		
	D	*A*	*B*	*S*	*H*	*P*	*t*	*T*	*W*	*L*	*K*	
0	0.252	0.3561	2 11/32	2 7/32	2 1/32	2	5/32	1/4	0.160	9/16	1 15/16	.6246
1	0.369	0.475	2 9/16	2 7/16	2 3/16	2 1/8	13/64	3/8	0.213	3/4	2 1/16	.5986
2	0.572	0.700	3 1/8	2 15/16	2 5/8	2 9/16	1/4	7/16	0.260	7/8	2 1/2	.5994
3	0.778	0.938	3 7/8	3 11/16	3 1/4	3 3/16	5/16	9/16	0.322	1 3/16	3 1/16	.6023
4	1.020	1.231	4 7/8	4 5/8	4 1/8	4 1/16	15/32	5/8	0.478	1 1/4	3 7/8	.6233
5	1.475	1.748	6 1/8	5 7/8	5 1/4	5 3/16	5/8	3/4	0.635	1 1/2	4 15/16	.6315
6	2.116	2.494	8 9/16	8 1/4	7 3/8	7 1/4	3/4	1 1/8	0.760	1 3/4	7	.6256
7	2.750	3.270	11 5/8	11 1/4	10 1/8	10	1 1/8	1 3/8	1.135	2 5/8	9 1/2	.6240

The figures in the "Taper per Foot" column have been revised to conform with the standard end diameters and lengths.

Brown & Sharpe and Jarno Tapers

Two other system of tapers are widely used. The Brown & Sharpe Tapers are used for milling machine spindles and the Jarno Tapers for some makes of lathe spindles. Specifications of these tapers can be obtained from technical hand books or from manufacturers using them.

Chapter IX

DRILLING, REAMING AND TAPPING

Many drilling, reaming and tapping jobs can be done more quickly and with greater accuracy in the lathe than by any other method.

Fig. 186 (at right) illustrates the use of the lathe as a drill press. A drill pad placed in the tailstock spindle of the lathe is used to support the work.

The tailstock hand wheel is turned as the hole is drilled through the work. The end of the work may rest on the lathe bed if desired.



Fig. 186. Using the Lathe as a Drill Press

The Location of the hole should be center punched to start the drill. The lathe should be operated at high speed when drilling small diameter holes.

Drill Pad for Tailstock

A drill pad for the tailstock spindle of the lathe is shown in Fig. 187. The drill pad replaces the tailstock center and supports the work for drilling.

Crotch Center

The crotch center shown in Figs. 187 and 188 is similar to the drill pad except that it has a "V" so that round work may be accurately cross drilled. This is very convenient for drilling oil holes in bushings, drilling pin holes in shafts, etc.



Fig. 187. (Right) Crotch Center for Use in Tailstock of Lathe

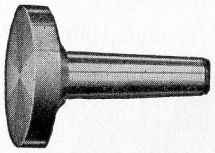


Fig. 187-A. (Left) Drill Pad for Use in Tailstock of Lathe

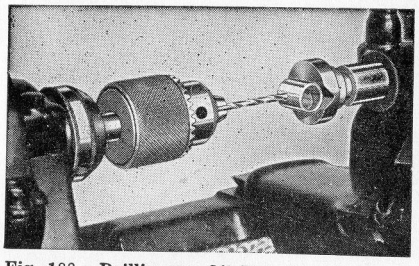


Fig. 188. Drilling an Oil Hole in a Bushing with Crotch Center in Tailstock

Drilling Work Held in the Chuck

Most of the drilling in the lathe is done with the work mounted in the lathe chuck (as shown in Fig. 189) or clamped to the face plate of the lathe. When this method is used it is important that the drill be started so that it will run true and the hole will be drilled concentric with the outside diameter of the work.

One method for starting the drill point true is illustrated in Fig. 190. The butt end of a lathe tool holder just touching the side of the drill will prevent the drill from bending and cause it to start approximately true in the center of the work.

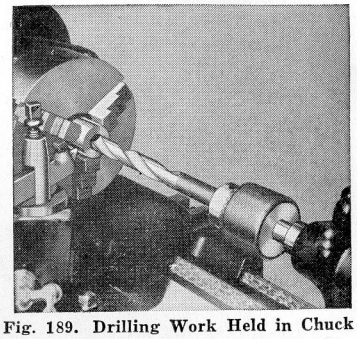


Fig. 189. Drilling Work Held in Chuck

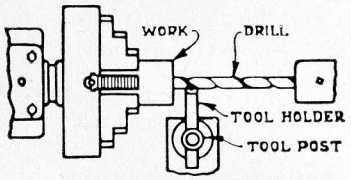


Fig. 190. Using Tool Holder to Steady Point of Drill

Center Drilling

When greater accuracy is required it is best to provide a true starting point for the drill. To do this the work should first be center drilled using a combination center drill and countersink, as shown in Fig. 191. The point of the center drill may be ground off as shown in Fig. 192 to prevent breaking.

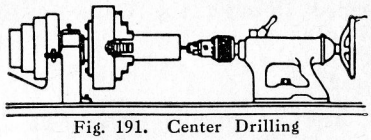


Fig. 191. Center Drilling

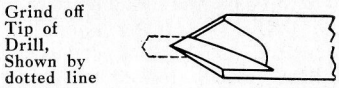


Fig. 192. Detail of Center Drill

Center Drill Holder

A special holder for mounting the combination center drill and countersink in the tailstock for drilling center holes is shown in Fig. 193. This provides an extremely rigid and accurate support for the drill, resulting in more accurately located center holes than can usually be obtained by mounting the center drill in a 3-jaw drill chuck. Large pieces that cannot be properly mounted in the chuck alone can be supported on the outer end by a steady rest, as shown in Fig. 194.

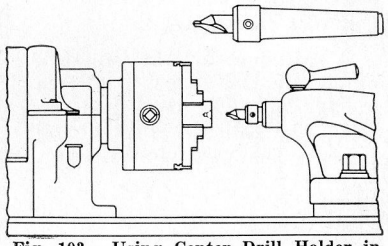


Fig. 193. Using Center Drill Holder in Tailstock to Drill Accurate Center Hole in Piece Held in Chuck

Drilling in Steel

When drilling in steel use plenty of lard oil on the point of the drill. If no lard oil is available, any good cutting oil or even machine oil may be used. However, lard oil is preferable, and for some deep hole drilling is the only satisfactory lubricant.

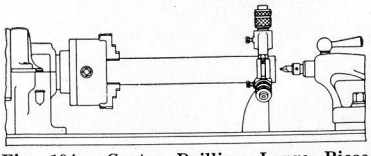


Fig. 194. Center Drilling Large Piece with the Aid of a Steady Rest

Drilling a Cored Hole

Castings having cored holes are usually drilled with a four lip drill. The hole in the casting should be beveled, as shown in Fig. 195, to start the drill true; otherwise, the drill will follow the cored hole and may be thrown off center. For accurate drilling it is advisable to counterbore the hole a short depth to give the drill point a perfectly concentric starting point.

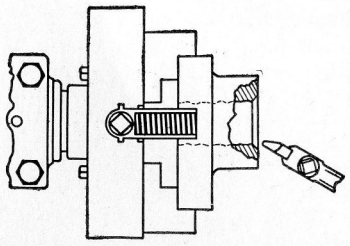


Fig. 195. Machining a Bevel in a Cored Hole to Start Drill True

How to Sharpen Drills

Correct grinding of the drill point is essential for accuracy and efficiency in all drilling operations. A medium grain grinding wheel that has been dressed true should be used for grinding drill points. The drill point should not be overheated by grinding or the temper may be drawn.

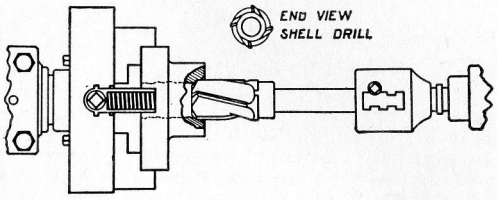


Fig. 196. Drilling a Cored Hole with a Four Lip Shell Drill



Fig. 197. Correct Point Angle

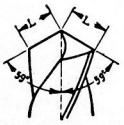


Fig. 198. Correct Lip Angle

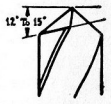


Fig. 199. Correct Clearance

Before grinding a drill, study the point of a new drill as received from the manufacturer; then try to duplicate it. This can be accomplished 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 giving the drill a slight twist to the right simultaneously. It is very important that both lips of the drill be ground exactly the same.

The angle of the chisel point or dead center should be from 120° to 130°, as shown in Fig. 197. The cutting lips "L" Fig. 198, should be exactly the same length and angle; otherwise the drill will cut oversize. The best angle for general work is 59°, as indicated.

The clearance back of the cutting edge should be from 12° to 15°, as shown in Fig. 199. Less clearance may prevent the drill from cutting freely, and more clearance will cause the cutting edge to dull quickly.

A drill grinding gauge similar to the one shown in Fig. 200 will aid in grinding the correct angle and length of cutting lip on the drill point.

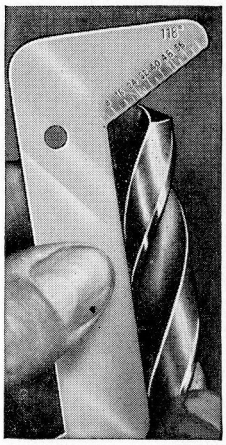


Fig. 200. Drill Grinding Gauge

Reaming in the Lathe

Reamers are used in the lathe to finish a number of holes quickly and accurately to the same diameter. Usually the hole is first drilled or bored roughly to size, allowing sufficient stock for reaming. Two types of reamers are used, the rose reamer and the fluted reamer.

Rose reamers are ground for cutting on the end only and are intended for rough reaming as they do not produce a good finish or an accurate diameter.

Fluted reamers are ground for cutting on both the ends and the sides of the blades and are usually used after the rose reamer to obtain an exact size and produce a good smooth finish. Fluted reamers should be used only for light cuts, removing not over .010 in. from the hole.

Reamer in Drill Chuck

Straight shank reamers are usually held in a drill chuck, as shown in Fig. 201. Taper shank reamers may be inserted direct in the tailstock spindle. The reamer is fed carefully through the hole by turning the tailstock handwheel. Always use a slow spindle speed and when reaming steel keep the reamer flooded with lard oil.

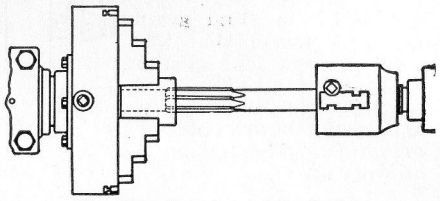


Fig. 201. Reaming in the Lathe

Floating Reamer Driver

For some reaming operations it is desirable for the reamer to follow a bored hole as accurately as possible, and for this type of work a floating reamer driver similar to the one shown in Fig. 202 is used.

Large reamers are sometimes supported on the tailstock center point. A lathe dog is attached to the reamer shank and a stick with one end resting against the lathe bed is placed between the reamer shank and the tail of the dog.

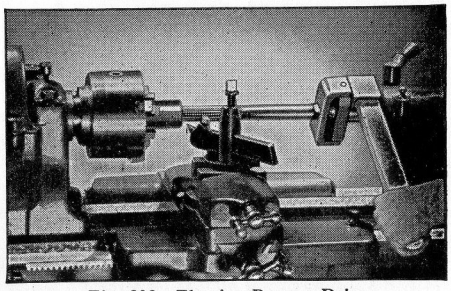


Fig. 202. Floating Reamer Driver

Tapping Threads

Threads may be tapped in the lathe, using a tap as shown in Fig. 203. The lathe spindle should be operated at slow speed and the tap fed to the work by turning the tailstock handwheel, or by sliding the entire tailstock on the lathe bed. Taps may also be held in a drill chuck.

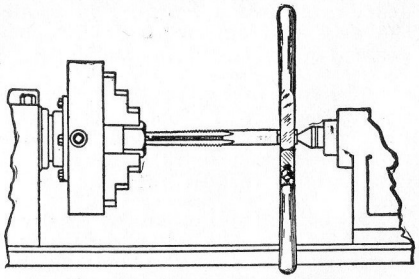


Fig. 203. Tapping in the Lathe

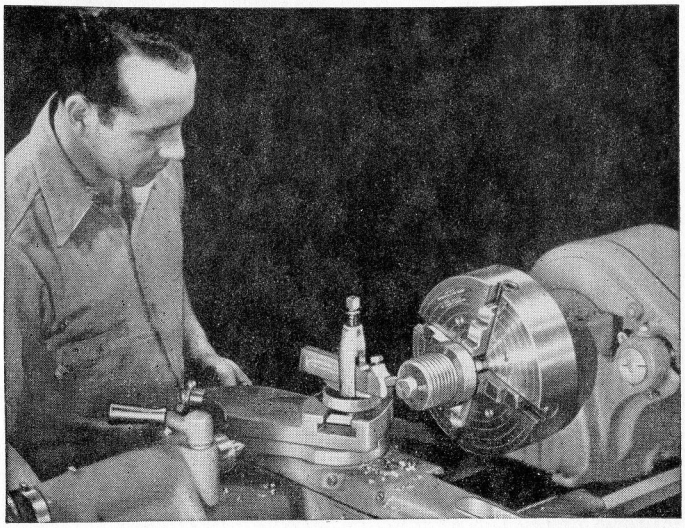


Fig. 204. Cutting Screw Threads in the Lathe

Chapter X

CUTTING SCREW THREADS

Cutting screw threads in the lathe is accomplished by connecting the headstock spindle of the lathe with 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 with relation to the headstock spindle.

The gearing between the headstock spindle and lead screw may be arranged so that any desired pitch of the thread may be cut. For example, if the lead screw has eight threads per inch and the gears are arranged so that the headstock spindle revolves four times while the lead screw revolves once, the thread cut will be four times as fine as the thread on the lead screw or 32 threads per inch.

The cutting tool is ground to the shape required for the form of the thread to be cut, that is, American National Form, "V," Acme, Square, Whitworth, International Metric, etc.

Either right hand or left hand threads may be cut by reversing the direction of rotation of the lead screw. This may be accomplished by shifting the reverse lever on the headstock.

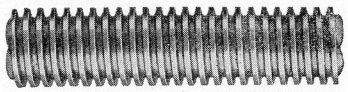


Fig. 205. Acme Screw Thread



Fig. 206. National Coarse Thread

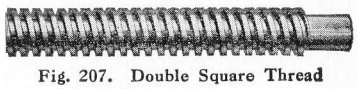


Fig. 207. Double Square Thread

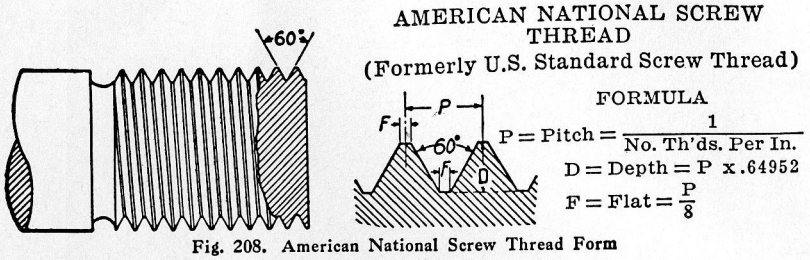


Fig. 208. American National Screw Thread Form

American National Screw Threads

The National Screw Thread Commission in 1918 was authorized by Congress to establish a standard system of screw threads for use in the United States. As a result this commission established the American National Screw Thread System which has been approved by the Secretary of War, Secretary of the Navy, and Congress, and is now generally used by all shops in the United States.

The form of the thread adopted is shown above and tables for both the Fine Thread Series and Coarse Thread Series are given on page 71. A report of the National Screw Thread Commission defines the following terms:

Screw Thread. A ridge of uniform section in the form of a helix on the surface of a cylinder or cone.

External and Internal Threads. An external thread is a thread on the outside of a member. Example: A threaded plug. An internal thread is a thread on the inside of a member. Example: A threaded hole.

Major Diameter (formerly known as "outside diameter"). The largest diameter of the thread of the screw or nut. The term "major diameter" replaces the term "outside diameter" as applied to the thread of a screw and also the term "full diameter" as applied to the thread of a nut.

Minor Diameter (formerly known as "core diameter"). The smallest diameter of the thread of the screw or nut. The term "minor diameter" replaces the term "core diameter" as applied to the thread of a screw and also the term "inside diameter" as applied to the thread of a nut.

Pitch Diameter. On a straight screw thread, the diameter of an imaginary cylinder, the surface of which would pass through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder.

Pitch. The distance from a point on a screw thread to a corresponding point on the next thread measured parallel to the axis.

Lead. The distance a screw thread advances axially in one turn. On a single-thread screw, the lead and pitch are identical; on a double-thread screw the lead is twice the pitch; on a triple-thread screw, the lead is three times the pitch, etc.

International Screw Thread Form

The need for an international screw thread form has been recognized for many years. Although several international missions have been made and hundreds of conferences held, little was accomplished until the summer of 1948 when United States and British representatives agreed on a 60° thread form with flat top and rounded bottom as shown in Fig. 208A. This thread form is a compromise between the 60° American National thread form with flat top and bottom shown in Fig. 208, and the British Whitworth 55° thread form with rounded top and bottom shown in Fig. 246. It is similar to the International Standard Metric thread form shown in Fig. 250.

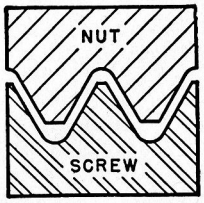


Fig. 208A. Diagram Showing International Screw Thread Form.

TABLES OF AMERICAN NATIONAL STANDARD SCREW THREAD PITCHES AND RECOMMENDED TAP DRILL SIZES

American National Coarse Standard Thread (N.C.)

Formerly U. S. Standard

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
1	64	.073	53	0.0595
2	56	.086	50	0.0700
3	48	.099	46	0.0810
4	40	.112	43	0.0890
5	40	.125	38	0.1015
6	32	.138	33	0.1130
8	32	.164	29	0.1360
10	24	.190	25	0.1495
12	24	.216	16	0.1770
1/4	20	.250	7	0.2010
5/16	18	.3125	F	0.2570
3/8	16	.375	5/16	0.3125
7/16	14	.4375	U	0.3680
1/2	13	.500	27/64	0.4219
9/16	12	.5625	31/64	0.4843
5/8	11	.625	17/32	0.5312
3/4	10	.750	21/32	0.6562
7/8	9	.875	49/64	0.7656
1	8	1.000	7/8	0.875
1 1/8	7	1.125	63/64	0.9843
1 1/4	7	1.250	1 7/64	1.1093

American National Fine Standard Thread (N.F.)

Formerly S. A. E. Thread

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
0	80	.060	3/64	0.0469
1	72	.073	53	0.0595
2	64	.086	49	0.0730
3	56	.099	44	0.0860
4	48	.112	42	0.0935
5	44	.125	37	0.1040
6	40	.138	32	0.1160
8	36	.164	29	0.1360
10	32	.190	21	0.1590
12	28	.216	14	0.1820
1/4	28	.250	7/32	0.2187
5/16	24	.3125	I	0.2720
3/8	24	.375	R	0.3390
7/16	20	.4375	25/64	0.3906
1/2	20	.500	29/64	0.4531
9/16	18	.5625	33/64	0.5156
5/8	18	.625	37/64	0.5781
3/4	16	.750	11/16	0.6875
7/8	14	.875	13/16	0.8125
1	14	1.000	15/16	0.9375
1 1/8	12	1.125	1 3/64	1.0468
1 1/4	12	1.250	1 11/64	1.1718

TABLES OF AMERICAN NATIONAL SPECIAL SCREW THREAD PITCHES (N.S.) AND RECOMMENDED TAP DRILL SIZES

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
1	56	.0730	54	0.0550
4	32	.1120	45	0.0820
4	36	.1120	44	0.0860
6	36	.1380	34	0.1110
8	40	.1640	28	0.1405
10	30	.1900	22	0.1570
12	32	.2160	13	0.1850
14	20	.2420	10	0.1935
14	24	.2420	7	0.2010
1/16	64	.0625	3/64	0.0469
3/32	48	.0938	49	0.0730
1/8	40	.1250	38	0.1015
5/32	32	.1563	1/8	0.1250
5/32	36	.1563	30	0.1285
5/16	24	.1875	26	0.1470
3/16	32	.1875	22	0.1570
7/32	24	.2188	16	0.1770
7/32	32	.2188	12	0.1890
1/4	24	.250	4	0.2090
	27		3	0.2130
	32		7/32	0.2187

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
5/16	20	.3125	17/64	0.2656
	27		J	0.2770
	32		9/32	0.2812
3/8	20	.375	21/64	0.3281
	27		R	0.3390
7/16	24	.4375	X	0.3970
	27		Y	0.4040
1/2	12	.500	27/64	0.4219
	24		29/64	0.4531
	27		15/32	0.4687
9/16	27	.5625	17/32	0.5312
5/8	12	.625	35/64	0.5469
	27		19/32	0.5937
11/16	11	.6875	19/32	0.5937
	16	.6875	5/8	0.6250
3/4	12	.750	43/64	0.6719
	27		23/32	0.7187
7/8	12	.875	51/64	0.7969
	18		53/64	0.8281
	27		27/32	0.8437
1	12	1.000	59/64	0.9219
	27		31/32	0.9687

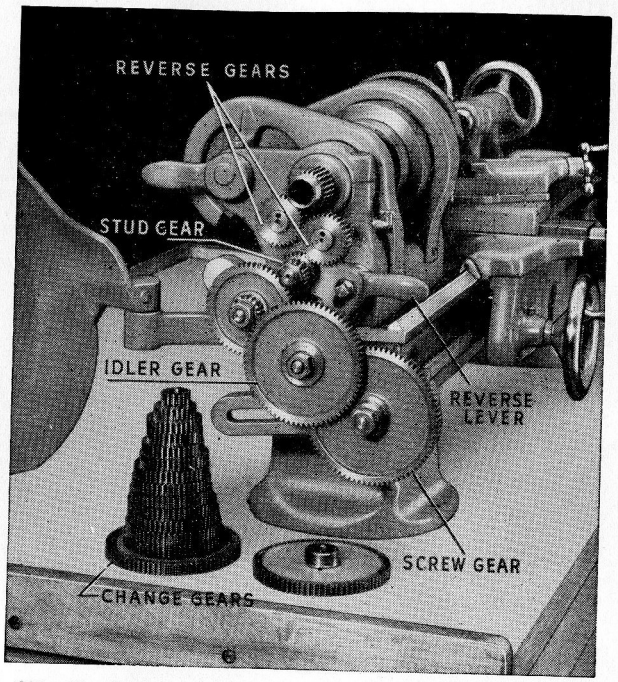


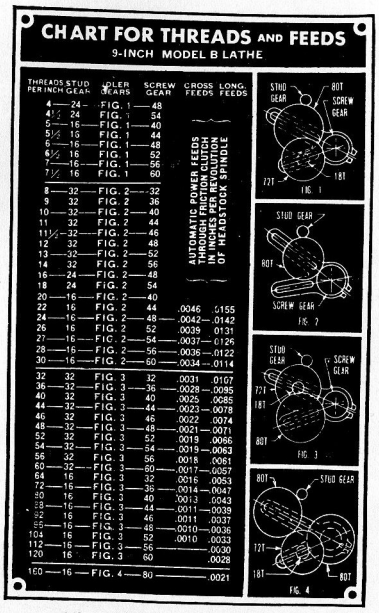
Fig. 209. Standard Change Gear Lathe Set Up for Cutting Screw Threads

Cutting Threads on Standard Change Gear Lathes

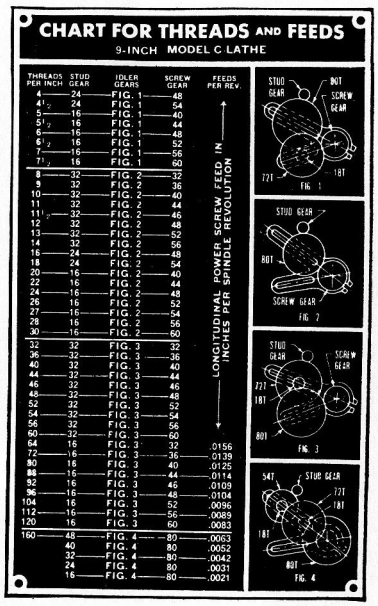
Screw threads are cut on Standard Change Gear Lathes by engaging the apron half nuts with the lead screw. The pitch of thread to be cut is determined by the number of teeth in the change gears used on the reverse stud and the lead screw, also the compound gears used.

To set up the lathe for cutting a screw thread, first determine the number of threads per inch to be cut. By referring to the change gear chart attached to the lathe (Fig. 210) the change gears required can be determined. The thread to be cut should be located in the first column under the heading "Threads Per Inch." In the second column under the heading "Stud Gear" is listed the number of teeth in the change gear which should be placed on the reverse stud of the lathe. (See Fig. 209.) In the third column under the heading "Idler Gear" is listed the figure number of the diagram on the index chart showing the arrangement of idler gear and compound gears. In the fourth column under the heading "Screw Gear" is listed the number of teeth in the gear to be placed on the lead screw "C".

After selecting the change gears necessary for cutting the desired thread, place them on the reverse stud and lead screw respectively and connect them with the idler gear and compound gears, as shown on the change gear chart.



9", Model B, Index Chart



9", Model C, Index Chart

Fig. 210. Change Gear Charts for Standard Change Gear Lathes

Position of Spacing Collar

The spacing collar on the lead screw must be placed outside the screw gear, as shown in Fig. 210A, when simple gearing (Fig. 2 on index charts) is used, and inside the screw gear, as shown in Fig. 210B, when compound gearing (Fig. 1 on index charts) is used. See page 114.

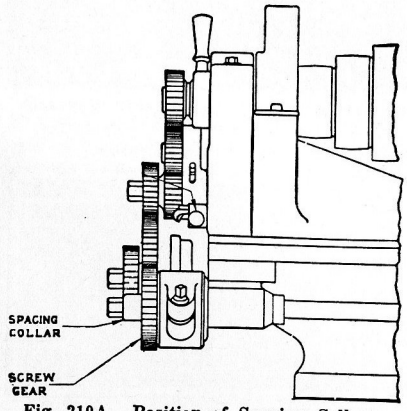


Fig. 210A. Position of Spacing Collar for Simple Gearing

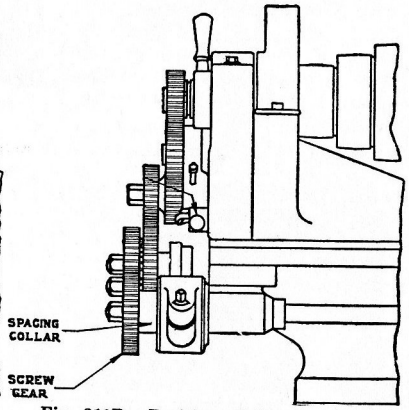


Fig. 210B. Position of Spacing Collar for Compound Gearing

Cutting Screw Threads on Quick Change Gear Lathe

The quick change gear lathe has a gear box, shown in Fig. 211, which eliminates the necessity of using loose change gears for cutting various pitches of screw threads and for obtaining various power cross-feeds and longitudinal feeds. Standard screw threads from 8 to 224* per inch are obtained by placing the two tumbler levers in the positions indicated by the direct reading index chart attached to the gear box. A close-up of this chart is shown in Fig. 211B.

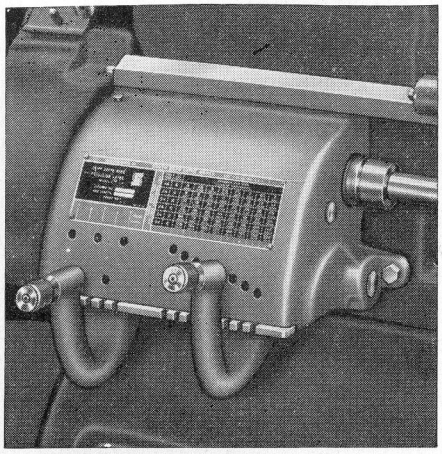


Fig. 211. Quick Change Gear Box

To cut coarse pitches of screw threads from 4 to 7 per inch, the small stud gear must be replaced by a large stud gear which, for safe keeping, is attached to the gear box drive shaft as shown in Fig. 211A. The number of teeth in the stud gear required is indicated on the index chart in the column of figures under the heading "Stud Gear."

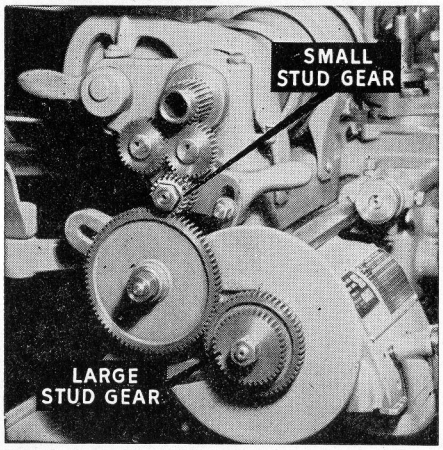
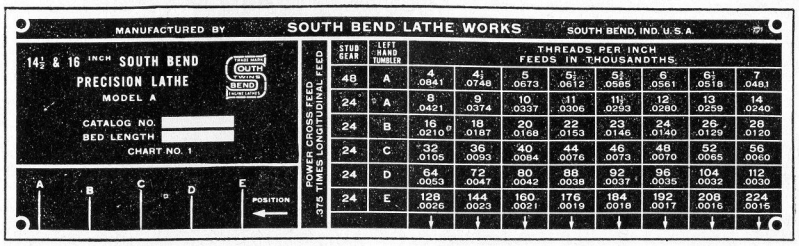


Fig. 211A. Lathe set up for cutting threads 8 to 224 per inch

Almost any special thread (one not shown on the index chart) can be obtained by using a special stud gear in place of the standard gears. Special stud gears are usually made to order for the specific thread required.

*Gearbox on 10"-1" Collet Lathes cuts 70 threads, 4 to 480 per inch, without changing stud gear



STUD GEAR	LEFT HAND TUMBLER	THREADS PER INCH / FEEDS IN THOUSANDTHS.							
48	A	4 .0841	4½ .0748	5 .0673	5½ .0612	5¾ .0585	6 .0561	6½ .0518	7 .0481
24	A	8 .0421	9 .0374	10 .0337	11 .0306	11½ .0293	12 .0280	13 .0259	14 .0240
24	B	16 .0210	18 .0187	20 .0168	22 .0153	23 .0146	24 .0140	26 .0129	28 .0120
24	C	32 .0105	36 .0093	40 .0084	44 .0076	46 .0073	48 .0070	52 .0065	56 .0060
24	D	64 .0053	72 .0047	80 .0042	88 .0038	92 .0037	96 .0035	104 .0032	112 .0030
24	E	128 .0026	144 .0023	160 .0021	176 .0019	184 .0018	192 .0017	208 .0016	224 .0015

Fig. 211B. Direct Reading Index Chart for Quick Change Gear Lathe

Tools for Cutting Screw Threads

The shape or form of a screw thread cut on the lathe is determined by the shape of the cutter bit, which must be carefully ground and set if an accurate thread form is to be obtained. The most common thread forms are shown on pages 70, 82, 83 and 84. A gauge should be used for grinding the lathe tool to the required shape for any form of screw thread.

Use of Center Gauge

The point of the cutter bit must be ground to an angle of 60° for cutting American National screw threads in the lathe, as shown in Fig. 213 at right. A center gauge having a 60° included angle is used for grinding the tool to the exact angle required. The top of the tool is usually ground flat, with no side rake or back rake. However, for cutting threads in steel, side rake is sometimes used.

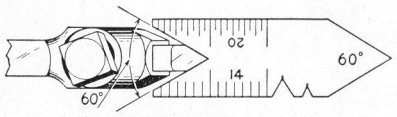


Fig. 213. Cutter Bit for Cutting Screw Threads is Ground to 60° Center Gauge

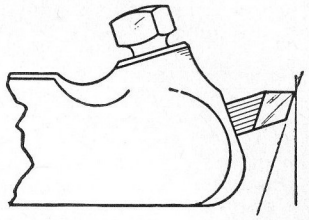


Fig. 214. Side View of Lathe Tool Cutter Bit Ground for Cutting Screw Threads

Front Clearance

There must be sufficient front clearance on the cutter bit to permit it to cut freely. Usually the front clearance is sufficient to prevent the tool from dragging in the helix angle of the thread so that except for very coarse pitches the helix angle may be ignored.

Formed Threading Tool

A formed threading tool is sometimes used if considerable threading is to be done. Fig. 215 illustrates a good type of formed threading tool. The formed threading tool requires grinding on top only to sharpen, and therefore always remains true to form and correct angle.

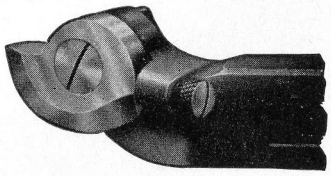


Fig. 215. Formed Thread Cutting Tool, Solid Type

Thread Tool Gauge

A gauge for grinding threading tools to the exact shape required for various pitches of American National screw threads is shown in Fig. 216.

For American National Screw Threads finer than 10 per inch, the point of the tool is usually left sharp or with a very small flat. However, for coarser pitches of threads and when maximum strength is desired, the flat on the point of the tool should be one-eighth of the pitch. (See Fig. 208, page 70.)

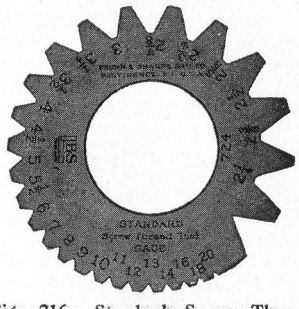


Fig. 216. Standard Screw Thread Tool Gauge for Grinding Thread Cutting Tools

Setting Cutter Bit

The top of the threading tool should be placed exactly on center, as shown in Fig. 217 at right, for cutting external screw threads. Note that the top of the tool is ground flat and is in exact alignment with the lathe center. This is necessary to obtain the correct angle of the thread.

The threading tool must be set square with the work, as shown in Fig. 218. The center gauge is used to adjust the point of the threading tool, and if the tool is carefully set a perfect thread will result. Of course, if the threading tool is not set perfectly square with the work, the angle of the thread will be incorrect.

Internal Threads

The point of the threading tool is also placed exactly on center, as shown in Fig. 219 at right for cutting internal screw threads. The point of the tool must be set perfectly square with the work. This may be accomplished by fitting the point of the tool into the center gauge, as shown in Fig. 220 at right.

When adjusting the threading tool for cutting internal threads, allow sufficient clearance between the tool and the inside diameter of the hole to permit backing out the tool when the end of the cut has been reached. However, the boring bar should be as large in diameter and as short as possible to prevent springing.

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

Engaging the Half-Nuts

After the work has been mounted in the lathe and the cutting tool properly adjusted, the half-nuts may be engaged with the lead screw for cutting the screw thread. Once the thread has been started, the half-nuts should not be disengaged from the lead screw, unless the thread dial indicator is used. See page 81.

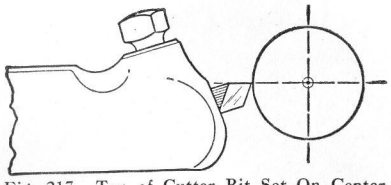


Fig. 217. Top of Cutter Bit Set On Center for Cutting Screw Threads

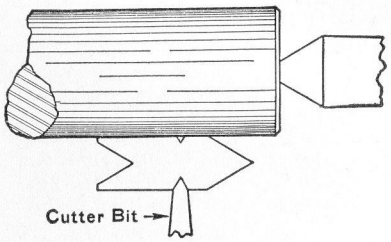


Fig. 218. Cutter Bit Set Square With Work, for Cutting External Screw Threads

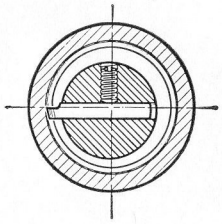


Fig. 219. For Cutting Internal Screw Threads, Top of Cutter Bit Should be Set Exactly on Center

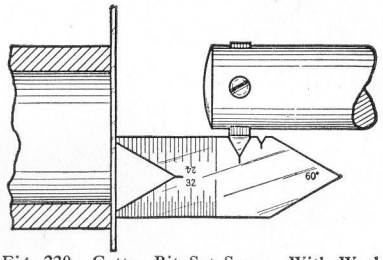


Fig. 220. Cutter Bit Set Square With Work for Cutting Internal Screw Threads

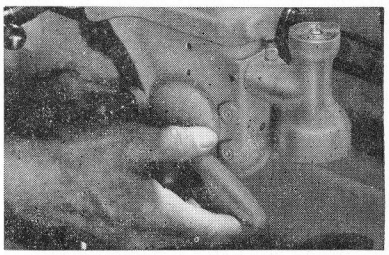


Fig. 220A. Engaging the Half-Nuts

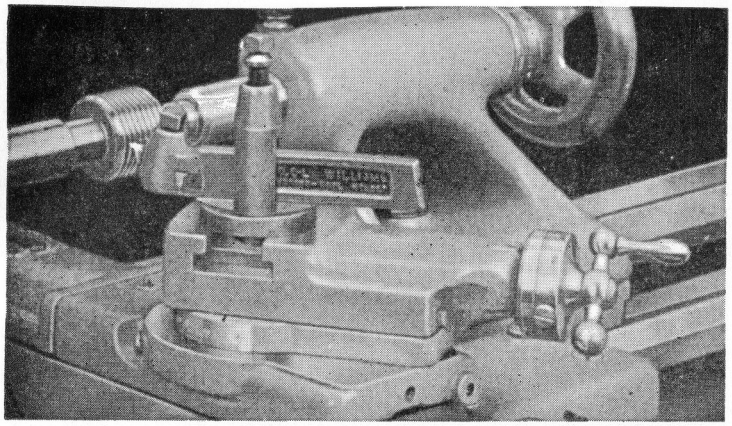


Fig. 221. Compound Rest Set at 29° Angle for Cutting 60° Screw Threads

Position of Compound Rest for Cutting 60° Screw Threads

In manufacturing plants where maximum production is desired, it is customary to place the compound rest of the lathe at an angle of 29° for cutting 60° screw threads.

The compound rest is swung around to the right, as shown in Figs. 221 and 224. The compound rest screw is used for adjusting the depth of cut and most of the metal is removed by the left side of the threading tool. (See Fig. 223.) This permits the chip to curl out of the way better than if tool is fed straight in.

The right side of the tool will shave the thread smooth and produce a good finish, although it does not remove enough metal to interfere with the main chip which is taken by the left side of the tool.

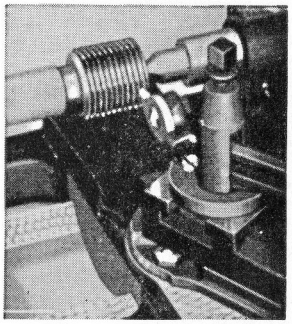


Fig. 222. Cutting a Screw Thread with Compound Rest Set at 29°

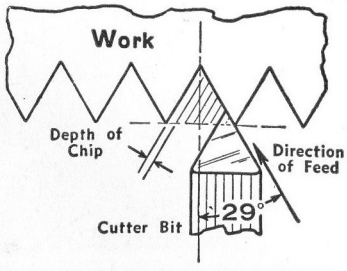


Fig. 223. Action of Thread Cutting Tool when Compound Rest is Set at 29° Angle

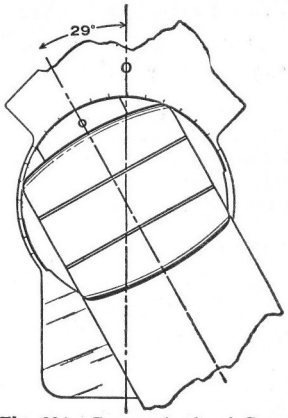


Fig. 224. Correct Angle of Compound Rest for Thread Cutting

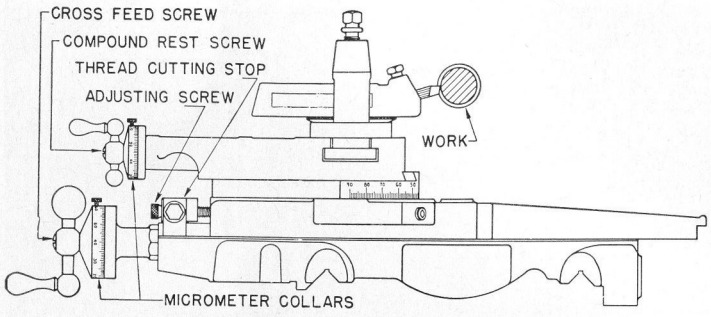


Fig. 225. Thread Cutting Stop Attached to Dovetail of Saddle

Use of Thread Cutting Stop

Because of the lost motion caused by the play necessary for smooth operation of the change gears, lead screw, half nuts, etc., the thread cutting tool must be withdrawn quickly at the end of each cut, before the lathe spindle is reversed to return the tool to the starting point. If this is not done, the point of the tool will dig into the thread and may be broken off. The thread cutting stop may be used for relocating the cutting tool for each successive chip.

The point of the tool should first be set so that it just touches the work; then lock the thread cutting stop to the saddle dovetail approximately ¼ of an inch from the compound rest base and turn the thread cutting stop screw until the shoulder is tight against the stop. When ready to take the first chip run the tool rest back by turning the cross feed screw to the left several turns and move the tool to the point where the thread is to start. Then 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, and by turning the compound rest feed screw in .002 in. or .003 in. the tool will be in a position to take the first cut.

Using Micrometer Collar

The micrometer collar on the cross feed screw of the lathe may be used in place of the thread cutting stop, if desired. To do this, first bring the point of threading tool up so that it just touches the work, then loosen small lock screw "A," adjust the micrometer collar on the cross feed screw to zero, and tighten lock screw.

All adjusting for obtaining the desired depth of cut should be done with the compound rest screw. Withdraw the tool at the end of each cut by turning the cross feed screw to the left one complete turn, return the tool to the starting point and turn the cross feed screw to the right one turn, stopping at zero. The compound rest feed screw may then be adjusted for any desired depth of chip.

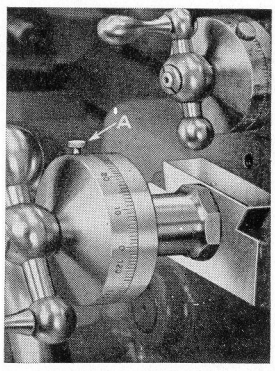


Fig. 226. Micrometer Collar on Cross Feed Screw of Lathe

Taking the First Cut

After setting up the lathe, as explained on the preceding pages, take a very light trial cut just deep enough to scribe a line on the surface of the work, as shown in Fig. 227. The purpose of this trial cut is to make sure the lathe is arranged for cutting the desired pitch of thread.

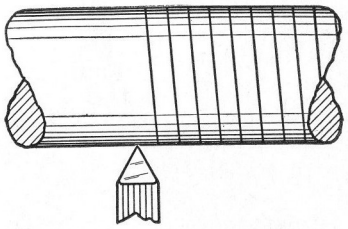


Fig. 227. Trial Cut to Check the Set-up for Thread Cutting

Measuring Screw Threads

To check the number of threads per inch, place a scale against the work, as shown in Fig. 228, so that the end of the scale rests on the point of the thread or on one of the scribed lines. Count the spaces between the end of the scale and the first inch mark, and this will give you the number of threads per inch. Fig. 228 shows eight threads per inch.

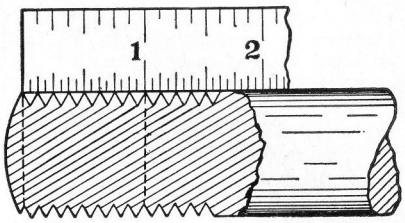


Fig. 228. Measuring Screw Threads

Screw Pitch Gauge

A screw thread gauge as illustrated in Fig. 229 is very convenient for checking the finer pitches of screw threads. This gauge consists of a number of sheet metal plates in which are cut the exact form of the various pitches of threads.

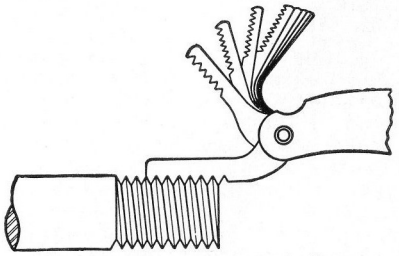


Fig. 229. Screw Thread Pitch Gauge

Fitting and Testing Threads

The final check for both the diameter and pitch of the thread may be made with the nut that is to be used or with a ring thread gauge, if one is available. Fig. 230 shows how the nut may be used for checking the thread. The nut should fit snugly without play or shake but should not bind on the thread at any point.

If the angle of the thread is correct and the thread is cut to the correct depth it will fit the nut perfectly. However, if the angle of the thread is incorrect, the thread may appear to fit the nut but will only be touching at a few points. For this reason, the thread should be checked by other methods in addition to the nut or ring gauge.

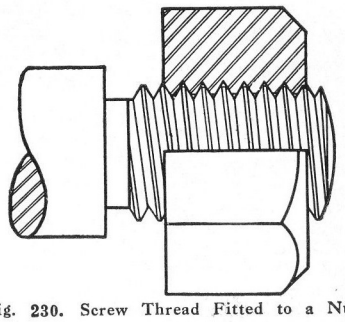


Fig. 230. Screw Thread Fitted to a Nut

Resetting Tool After Cut Has Been Started

If for any reason it is necessary to remove the thread cutting tool before the thread has been completed, the tool must be carefully readjusted so that it will follow the original groove when it is replaced in the lathe.

Before adjusting the tool, take up all the lost motion by pulling the belt forward by hand.

The compound rest top should be set at an angle, and by adjusting the cross feed screw and compound rest feed screw simultaneously the point of the tool can be made to enter exactly into the original groove.

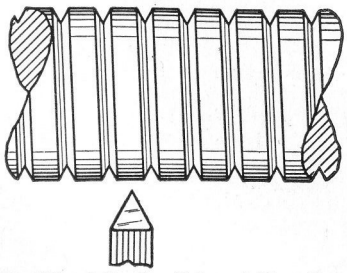


Fig. 231. Adjusting Point of Threading Tool to Conform with Thread

Finishing the End of a Thread

The end of the thread may be finished by any one of several methods. The 45° chamfer on the end of the thread, as shown in Fig. 232, is commonly used for bolts, cap screws, etc. For machine parts and special screws the end is often finished by rounding with a forming tool, as shown in Fig. 233.

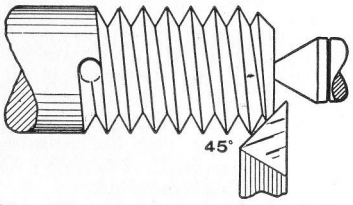


Fig. 232. Finish End of Thread with 45° Chamfer

It is difficult to stop the threading tool abruptly so some provision is usually made for clearance at the end of the cut. In Fig. 232 a hole has been drilled in the end of the shaft, and in Fig. 233 a neck or groove has been cut around the shaft. The groove is preferable as the lathe must be run very slowly in order to obtain satisfactory results with the drilled hole.

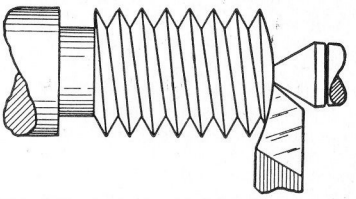


Fig. 233. Finishing End of Thread with Forming Tool

Cutting a Left Hand Screw Thread

A left hand screw is one that turns counter-clockwise when advancing (looking at head of screw) as shown in Fig. 235. This is just the opposite of a right hand screw. Left hand threads are used for the cross feed screws of lathes, the left hand end of axles for automobiles and wagons, one end of a turnbuckle, some pipe threads, etc.

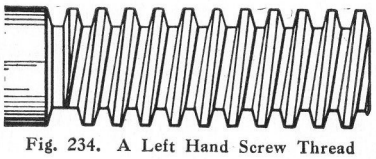


Fig. 234. A Left Hand Screw Thread

In cutting left hand screw threads the lathe is set up exactly the same as for cutting right hand screw threads, except that the lathe must be arranged to feed the tool from left to right, instead of from right to left, when the spindle is revolving forward.



Fig. 235. A Left Hand Screw Advances When Turned Counter-clockwise

Use of Thread Dial Indicator

The thread dial indicator, shown in Fig. 237, is used to save time, especially when cutting long screw threads. When the lathe is set up for cutting screw threads, the thread dial indicates the relative position of the lead screw, spindle, and carriage of the lathe. This permits dis-engaging the half-nuts from the lead screw at the end of a cut, returning the carriage quickly to the starting point by hand without reversing the lathe spindle, and re-engaging the half-nuts with the lead screw at a point which will assure the tool following exactly in the original cut.

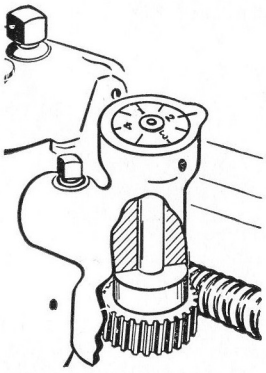


Fig. 237. Thread Dial Indicator Attached to Lathe Carriage

The 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. Provided the thread is started when a numbered graduation on the dial coincides with the witness mark on the frame, the points at which the half-nuts may be engaged for successive cuts will be indicated as follows:

For all even numbered threads, close the half-nuts at any line on the dial, or each ⅛ revolution.

For all odd numbered threads, close the half-nuts at any numbered line on the dial, or each ¼ revolution.

For all threads involving one-half of a thread in each inch, such as 11½, close the half-nuts at any odd numbered line, or each ½ revolution.

For all threads involving one-fourth of a thread in each inch, such as 4¾, return to the original starting point before closing the half-nuts.

Metric Thread Dial Indicator

Because metric threads are measured in mm pitch (the distance the thread advances each turn) instead of in threads per unit of measurement as is done in the English system, the thread dial indicator is a little more complicated. A special thread dial indicator is required for cutting metric screw threads. See page 85.

Use Oil When Cutting Threads in Steel

Lard oil (or machine oil) should be used when cutting screw threads in steel in order to produce a smooth thread. If oil is not used, a very rough finish will be caused by tearing of the steel by the cutting tool.

The oil should be applied generously preceding each cut. A small paint brush is ideal for applying the oil when cutting external screw threads, as illustrated in Fig. 238.

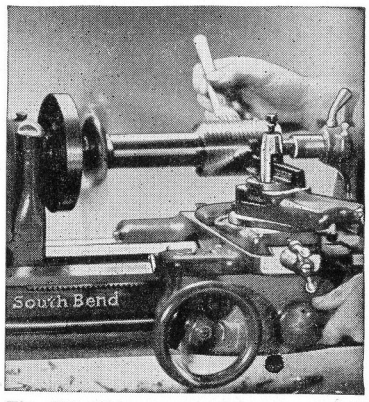


Fig. 238. Using a Small Brush to Apply Oil When Cutting Screw Threads

Tapered Screw Threads

Tapered screw threads such as pipe threads may be cut with the aid of a taper attachment, as shown in Fig. 239, or by setting over the tailstock off center, as shown in Fig. 240.

Regardless of which method is used, it is important that the thread tool is set square with the straight portion of the work, as shown in Figs. 239 and 240 and not to the tapered portion. The angle of the sides of the thread will be incorrect if the tool is not set as shown.

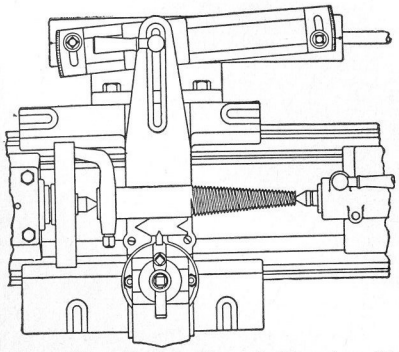


Fig. 239. Cutting Tapered Threads with Taper Attachment

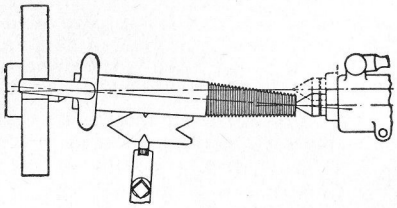


Fig. 240. Cutting Tapered Threads with Tailstock Center Set Over

Square Screw Threads

Square threads are used for vise screws, jack screws, etc. The sides of the tool for cutting square threads should be ground at an angle to conform with the helix angle of the thread, as shown in Fig. 241.

To determine the helix angle of a screw thread, draw line A-C2 equal to the circumference of the thread to be cut. Draw line C2-C equal to the lead of the thread and at right angles to line A-C2. Complete the triangle by drawing line A-C. Angle B in the triangle is the helix angle of the thread. The sides of the tool E and F should be given 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 one thousandth to three thousandths of an inch larger, to permit a free fit on the screw.

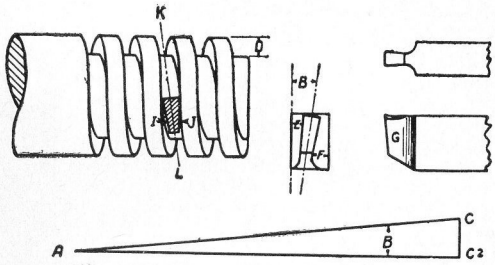
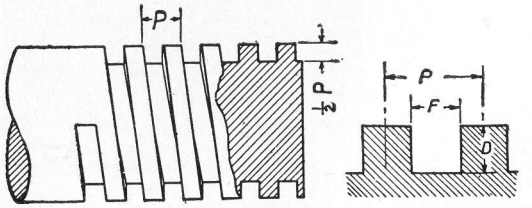


Fig. 241. Tool for Cutting Square Threads



FORMULA

$$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$$

D = Depth = P x .500

F = Space = P x .500

Fig. 242. Design and Proportions of Square Screw Threads

Acme Screw Threads

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

$D = \text{Depth} = \frac{1}{2} P + .010 \text{ In.}$

$F = \text{Flat} = .3707\ P$

$C = \text{Flat} = .3707\ P - .0052 \text{ In.}$

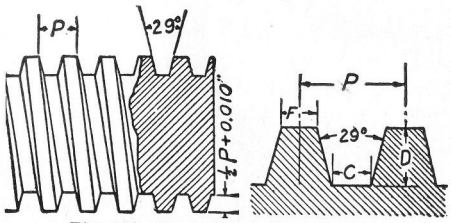


Fig. 243. Acme Screw Thread Form

Acme screw threads are used for the feed screws and adjusting screws of machine tools and machinery of all kinds. Acme threads are preferable to square threads because they are easier to cut.

While the top and the bottom of the threads are similar to a square thread in that they are flat, the sides of the thread have a 29° included angle, as shown in Fig. 243.

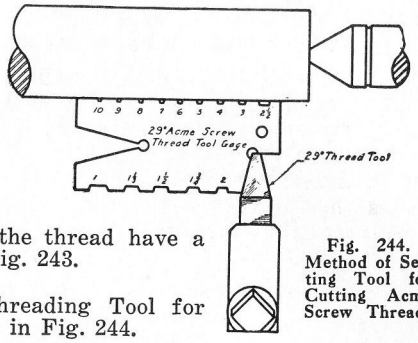


Fig. 244. Method of Setting Tool for Cutting Acme Screw Threads

The method of setting a Threading Tool for cutting an Acme Thread is shown in Fig. 244.

29° Worm Thread
(Brown & Sharpe)

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

$D = \text{Depth} = .6866\ P$

$F = \text{Flat} = .31\ P$

$C = \text{Flat} = .335\ P$

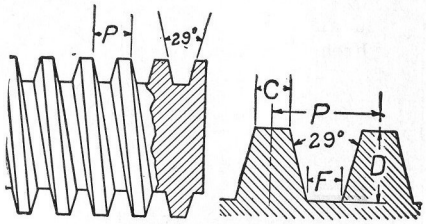


Fig. 245. 29° Worm Thread Form

A 29° Worm Thread should 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 shown above.

Whitworth Thread

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

$D = \text{Depth} = P \times .6403$

$R = \text{Radius} = .1373 \times P$

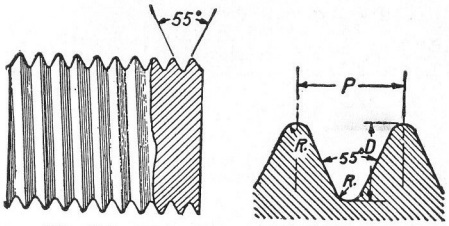


Fig. 246. Whitworth Screw Thread Form

The Whitworth thread form is used mostly in England. Great care is required in grinding the thread tool so that it will produce the radius at the top and bottom of the thread.

Metric and English Transposing Gears

When it is desired to cut both English and metric screw threads on the same lathe, transposing gears are required.

English transposing gears are used for cutting English screw threads on lathes having metric lead screws. Metric transposing gears are used for cutting metric screw threads on lathes having English lead screws.

The form of the metric thread is similar to the American National Screw Thread form, having a 60° included angle and a flat at the top of the thread, but a small radius at the root of the thread provides greater clearance. (See Fig. 250.)

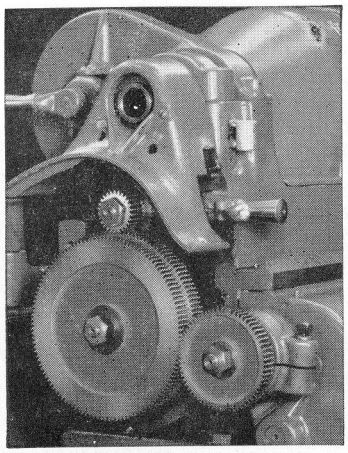
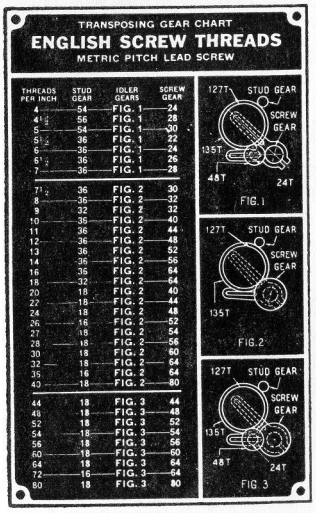
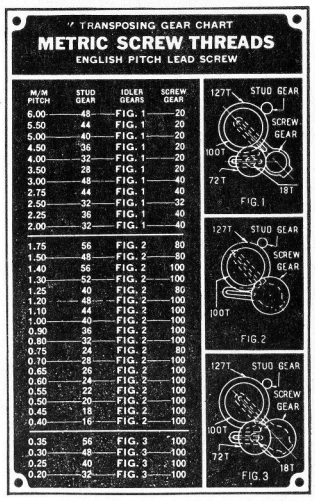


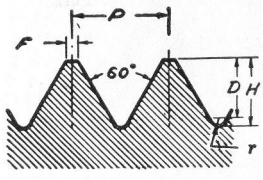
Fig. 247. Lathe Equipped with Transposing Gears



Left—Fig. 248 Index Chart Showing English Threads Cut on Metric Lathe with English Transposing Gears



Right—Fig. 249 Index Chart showing Metric Threads Cut on English Lathe with Metric Transposing Gears



FORMULA

P=Pitch in MM
D=Depth of Engagement=P x .6495
H=Depth of Thread =P x .6945
r=Maximum Radius =P x .0633
F=Flat=$\frac{P}{8}$

Fig. 250. International Standard Metric Screw Thread Form

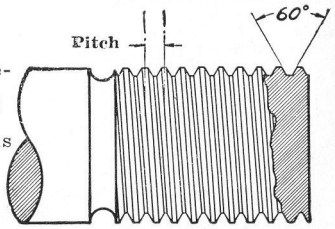


Fig. 251. 2.5 mm. Pitch Metric Screw Thread

Metric Lathe with Metric Lead Screw

Metric lathes equipped with metric lead screws are preferable in locations where metric screw threads are used exclusively. The metric lathe is identical with the English lathe, except that the lead screw, cross-feed screw and compound rest screw have metric threads, and all graduations are in the metric system.

Metric lathes are made in both the Standard Change Gear and Quick Change Gear types. Metric Quick Change Gear Lathes have a quick change gear box which permits cutting a wide range of metric screw threads and feeds, as listed on the index chart, which is illustrated below in Fig. 253. Metric Standard Change Gear Lathes have a similar range of metric screw threads and feeds.

MANUFACTURED BY SOUTH BEND LATHE WORKS SOUTH BEND, IND., U.S.A.

PITCHES IN mm—PASOS EN mm—PAS EN mm								POSITION POSICION	STUD ARBOL ARBRE
7.500	7.000	6.500	6.000	5.500	5.000	4.500	4.000	D	50
3.750	3.500	3.250	3.000	2.750	2.500	2.250	2.000	C	"
1.875	1.750	1.625	1.500	1.375	1.250	1.125	1.000	B	"
1.500	1.400	1.300	1.200	1.100	1.000	0.900	0.800	C	20
0.750	0.700	0.650	0.600	0.550	0.500	0.450	0.400	B	"
0.375	0.350	0.325	0.300	0.275	0.250	0.225	0.200	A	"
FEEDS IN mm—AVANCES EN mm									
0.512	0.478	0.444	0.410	0.375	0.341	0.307	0.273	C	20
0.256	0.239	0.222	0.205	0.188	0.171	0.154	0.137	B	"
0.128	0.119	0.111	0.102	0.094	0.085	0.077	0.068	A	"

TRADE MARK SOUTH BEND ENGINE LATHES

9-Inch—235 mm
SOUTH BEND LATHE
MODEL A

CATALOG NO.
BED LENGTH

Positions Posiciones A B C D

CROSS FEEDS 1/10 OF PITCHES
AVANCES TRANSVERSALES 1/10 DES PASOS
AVANCES TRANSVERSALES 1/10 DES PAS

Fig. 253. Index Chart Showing Metric Threads and Feeds on a 9-inch Swing Metric Quick Change Gear Lathe

Metric Thread Dial Indicator

The thread dial used for cutting metric screw threads on lathes equipped with metric lead screws is shown in Fig. 254. To provide for the various pitches of metric threads, several gears having different numbers of teeth are mounted on the lower end of the shaft. The vertical position of the thread dial indicator is changed as required so that the correct gear for the pitch of the thread to be cut will mesh with the lead screw.

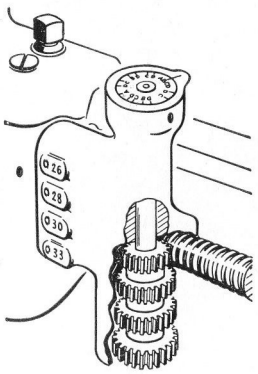


Fig. 254. Metric Thread Dial Indicator

Each graduation on the dial is marked with a letter which indicates the points at which the half-nuts may be engaged for certain threads. A chart (Fig. 254A) is supplied with the thread dial to show which gear and which graduations must be used for each pitch of metric screw thread.

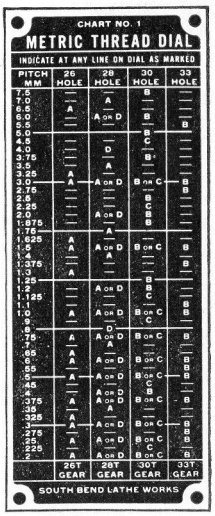


Fig. 254A. Index Chart for Metric Thread Dial

Cutting Multiple Screw Threads

A multiple thread having two grooves is known as a double thread, three threads a triple thread, etc. (See Fig. 255.) The pitch and lead of a multiple thread should not be confused. The pitch is the distance from a point on one thread to the corresponding point on the next thread, while the lead is the distance a screw thread advances in one turn.

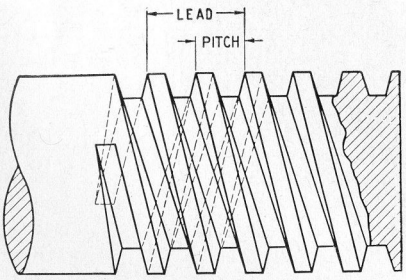


Fig. 255. A Multiple Screw Thread Having Two Grooves (Double Thread)

When cutting multiple threads 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. In order to obtain an 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. For a double thread, two slots; a triple thread, three slots, etc. If it is not convenient to cut slots in the face plate, equally spaced studs may be attached to the face plate and a straight tail lathe dog used.

Another method for indexing the work when cutting multiple threads is to disengage the change gears after the first thread has been completed and turn the spindle to the required position for starting the next cut.

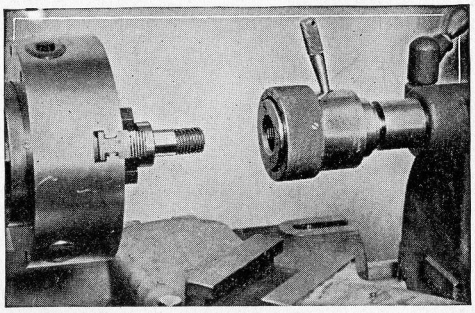


Fig. 256. Die Mounted in Tailstock of Lathe for Threading Studs

Cutting Threads with Die in Tailstock

A die may be mounted on tailstock of lathe for cutting screw threads, as shown in Fig. 256. A lathe tool may also be mounted in the tool post for turning operations or for cutting-off if desired. This method is often used for threading a large number of small pieces.

A die may be mounted on lathe carriage for cutting screw threads, as shown in Fig. 257. The lead screw and half nuts are used to feed the die so that threads with a perfect lead are obtained.

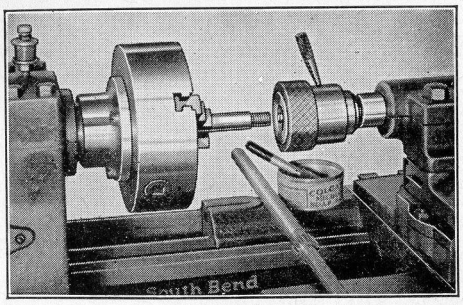


Fig. 257. Die Mounted on Lathe Carriage for Cutting Accurate Screw Threads

Chapter XI

SPECIAL CLASSES OF WORK

There are many special classes of lathe work such as knurling, filing, polishing, coil winding, etc. The most important are illustrated and described briefly on the following pages.

Knurling

Knurling is the process of embossing the surface of a piece of work in the lathe with a knurling tool (Fig. 258) in the tool post of the lathe.

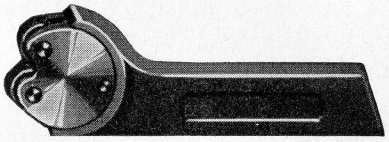


Fig. 258. Knurling Tool for Lathe

Three examples of knurling on a piece of steel are shown in Fig. 259. The pattern of the knurl is alike in all three cases but is of different grades, coarse, medium and fine.

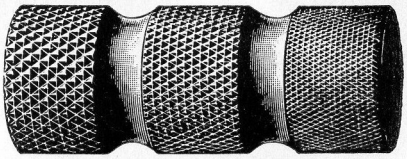


Fig. 259. Sample of Knurling

For all knurling operations the lathe should be arranged for the slowest back geared speed. After starting the lathe force the knurling tool slowly into the work at the right end until the knurl reaches a depth of about $\frac{1}{64}$ in. Then engage the longitudinal feed of the carriage and let the knurling tool feed across the face of the work. Plenty of oil should be used on the work during this operation.

When the left end of knurl roller has reached the end of work, reverse the lathe spindle and let the knurling tool feed back to the starting point. Do not remove the knurling tool from the impression but force it into the work another $\frac{1}{64}$ in., and let it feed back across the face of the work. Repeat this operation until the knurling is finished.

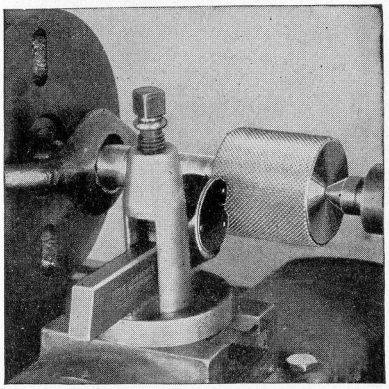


Fig. 260. Knurling a Steel Piece in the Lathe

Machining Work on the Face Plate

Before mounting a face plate on the spindle nose of the lathe, all dirt and chips should be removed from the threaded hole. Also clean the thread and the shoulder on the spindle nose because any dirt, chips or burrs will prevent the face plate from running true.

Oil the threads of the spindle so that the face plate will screw on easily and can be easily removed. If it seems difficult to screw on the face plate, unscrew the plate, remove the dirt, burrs, etc., and try again. The face plate hub should screw tight against the shoulder of the spindle but the face plate should not be spun up to the shoulder suddenly as this makes removal difficult.

The face plate is especially valuable in tool room work for machining holes in tools and jigs. In this class of work the holes must be accurately spaced, with an allowance usually not more than .001 of an inch.

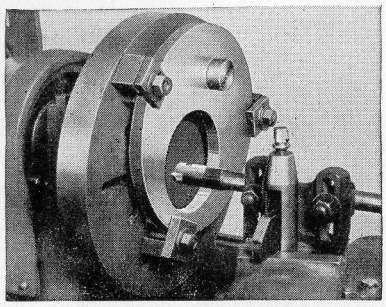


Fig. 261. Boring an Eccentric Hole on the Face Plate of the Lathe

Clamping Work on Face Plate

Some care should be exercised when clamping work on the face plate so that neither the work nor the face plate will be sprung. A piece of paper placed between the face plate and the work will reduce the danger of the work slipping. Balance weights should be used as shown in Fig. 264.

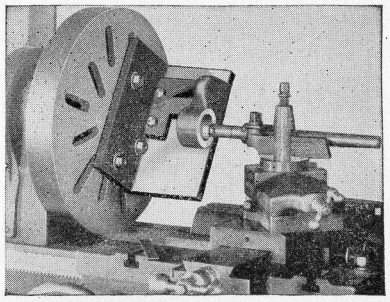


Fig. 262. Boring a Bracket with an Angle Plate Attached to the Face Plate

Centering the Work

A center indicator may be used, as shown in Fig. 263, for accurately locating work on the face plate for drilling and boring. A dial indicator may also be used, as shown in Fig. 264.

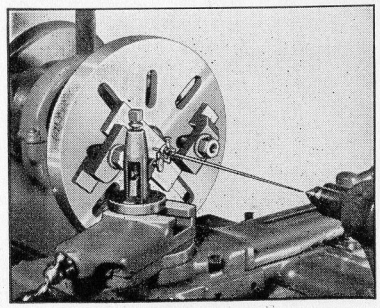


Fig. 263. Centering Work on the Face Plate with a Center Indicator

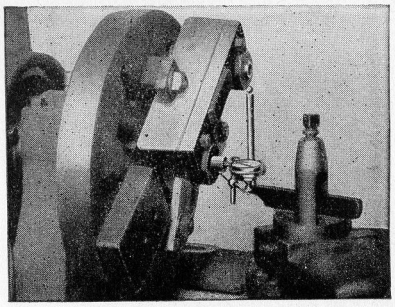


Fig. 264. Locating Work on the Face Plate with a Dial Indicator

Filing and Polishing

All tool marks can be removed and a smooth, bright finish obtained on the surface of a piece of work by filing and polishing, as shown in Figs. 265 and 266.

Use a fine mill file and file with the lathe running at a speed so that the work will make two or three revolutions for each stroke of the file. File just enough to obtain a smooth surface. If too much filing is done the work will be uneven and inaccurate. See page 119.

Keep the left elbow high and the sleeves rolled up so there will be no danger from the lathe dog.

Keep the file clean and free from chips, using a file card frequently.

A very smooth, bright finish may be obtained by polishing with several grades of emery cloth after filing. Use oil on the emery cloth and run the lathe at high speed. Be careful not to let the emery cloth wrap around the revolving work.

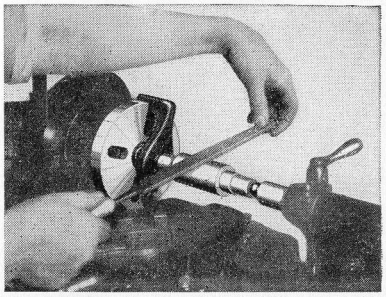


Fig. 265. Filing to Remove Tool Marks

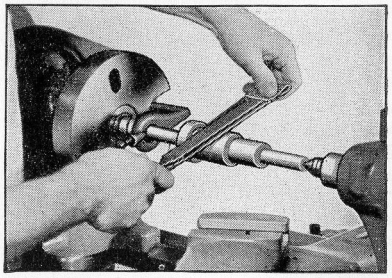


Fig. 266. Polishing with Emery Cloth and Oil

Lapping

Hardened gauges, bushings and bearings are often finished in the lathe by lapping, as shown in Fig. 267. Emery cloth, emery dust and oil, diamond dust and other abrasives are used. Usually the lathe spindle is operated at high speed. See page 119.

The lap may be very simple, consisting of a strip of emery cloth attached to a shaft, or it may be elaborately constructed of lead, copper, cast iron, etc. Some very fine and precise work may be accomplished by careful lapping.

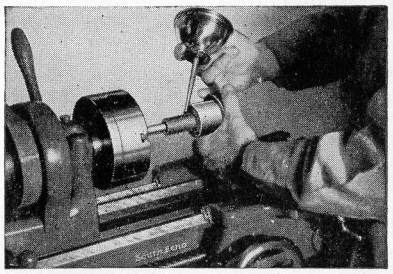


Fig. 267. Lapping the Inside of a Hardened Steel Bushing with Emery Dust and Oil

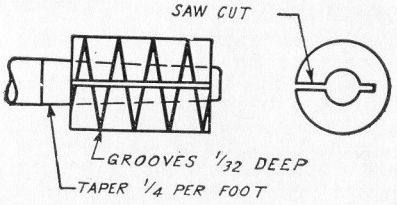


Fig. 268. A Cast Iron Lap for Emery Dust

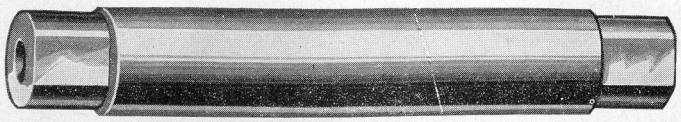


Fig. 269. A Steel Lathe Mandrel

Machining Work on a Lathe Mandrel

Cylindrical work that has been bored and reamed in a chuck is usually further machined on a mandrel between the lathe centers, as shown in Figs. 270 and 271. The mandrel is slightly tapered and must be driven into the hole tight enough so that the work will not slip on the mandrel while it is being machined.

Large diameter work such as pulleys should be driven with a pin or driver attached to the lathe face plate if it can be arranged as this will eliminate possibility of the work slipping on the mandrel.

Before driving the mandrel into the hole in the work, oil both the mandrel and the hole so that the work will be easy to remove from the mandrel. If there is no lubricant on the mandrel it may "freeze" in the work, in which case it cannot be driven out without ruining both the work and the mandrel.

In driving a mandrel out of a piece of work be sure that it is driven in the opposite direction from that which it entered the work.

Standard lathe mandrels can be purchased in the various sizes. These mandrels are hardened and tempered and the surface that receives the work is ground usually to a taper of about .006 in. per foot.

In the case of special jobs having odd diameter holes, a soft mandrel may be used, turning and filing it to the proper diameter and tapered for a driving fit in the hole in the work. See page 102.

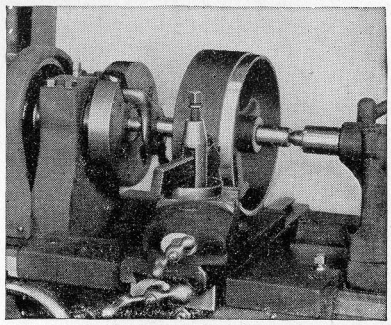


Fig. 270. Turning a Pulley on a Mandrel

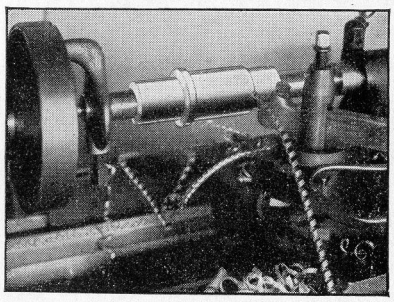


Fig. 271. Finishing a Bushing on a Mandrel

Special Mandrels

Special types of mandrels are often used for special classes of work. A nut mandrel for finishing the outside diameter of gear blanks is shown in Fig. 272. Expansion mandrels of various types are also available and are used where there is considerable variation in hole sizes.

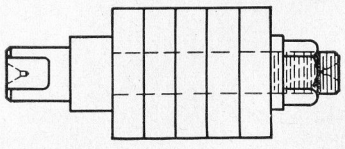


Fig. 272. Nut Mandrel for Finishing Gear Blanks

Winding Coils in the Lathe

The unusually wide range of positive power longitudinal feeds available on the lathe make it an ideal machine for winding electrical coils of all kinds. A revolution counter may be attached to register the number of turns, as shown in Fig. 273. Special gearing may be obtained for odd leads not in the usual thread cutting range of the lathe. Any type of coil form or wire guide required may be used.

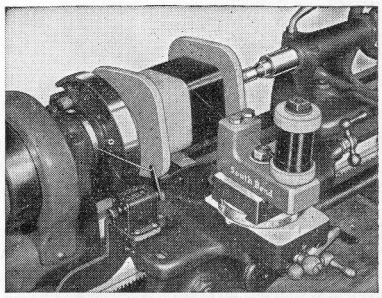


Fig. 273. Winding a Coil

Spring Winding

Coil springs of all kinds may be wound on the lathe, as shown in Fig. 274. Special mandrels are used for irregular shaped springs. The lead screw and half nuts of the lathe are usually used to obtain a uniform lead so that the coils are all equally spaced.

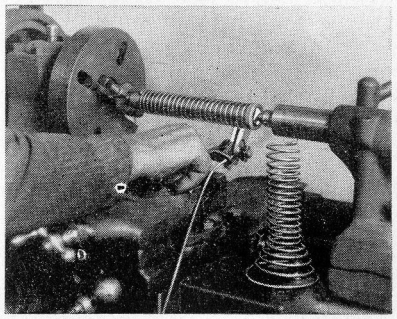


Fig. 274. Winding a Spring

Boring Work Mounted on Lathe Carriage

Large work may be mounted on the lathe carriage for boring, as shown in Fig. 277.

The boring bar is held between centers and driven by a lathe dog. The work is clamped to the top of the lathe saddle and is fed to the tool by the power longitudinal feed of the carriage.

Several good types of boring bars for this class of work are shown in Figs. 275, 276, and 278.

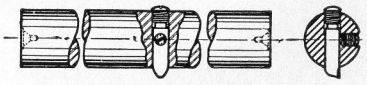


Fig. 275. Boring Bar with Fly Cutter

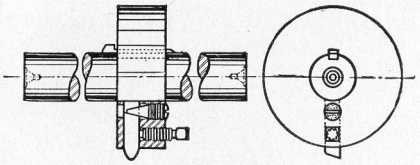


Fig. 276. Boring Bar with Boring Head

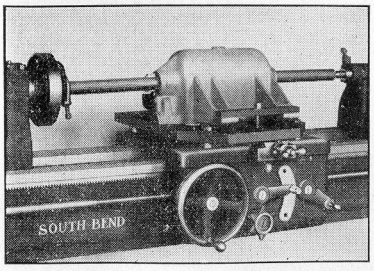


Fig. 277. Boring on the Lathe Carriage

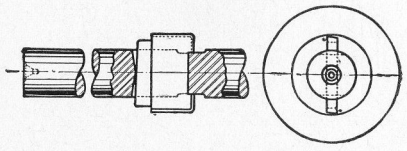


Fig. 278. Boring Bar for Sizing the Hole

Use of the Steady Rest (Center Rest)

The steady rest (or center rest) is used to support long slender shafts as shown in Fig. 280, or to support the outer end of shafts or spindles for drilling, facing, boring and similar operations as shown in Fig. 281. A steady rest with rectangular jaws is shown in Fig. 279. An improved steady rest with telescoping jaws is shown in Fig. 280.

To mount work in the steady rest, first clamp the open steady rest on the bed as shown in Fig. 279, then mount the work in the lathe. Close and lock the top portion of the steady rest. The jaws may then be adjusted to the work, which should be round and smooth at point of contact. Careful adjustment is necessary, as all three jaws should just touch the work. If one of the jaws is moved in too far, the work will be sprung and cannot be accurately machined. Make sure the work turns freely. Oil the jaws and work before starting the lathe.

For drilling, boring, and similar operations, one end of the work may be held in the chuck as shown in Figs. 281 and 282. However, for fine, accurate work most mechanics prefer to mount the left end of the work on the headstock center as shown in Fig. 283. To do this, the face plate is first unscrewed several turns. The work with lathe dog attached is then tied securely to the face plate with strong rawhide belt lacing, and the face plate is screwed onto the spindle. This tightens the lacing on the work and holds it firmly against the center point.

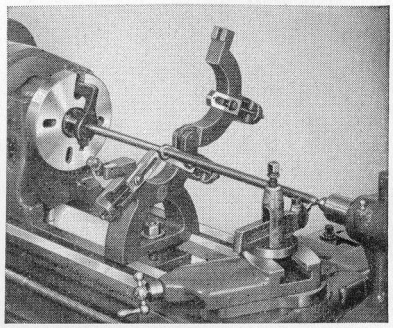


Fig. 279. Steady Rest Mounted on Lathe Bed with Top Open to Receive Work

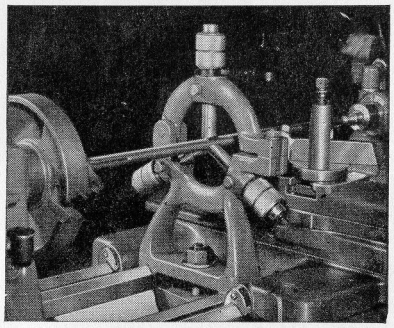


Fig. 280. Telescoping Jaw Steady Rest with Jaws Adjusted to Support Work

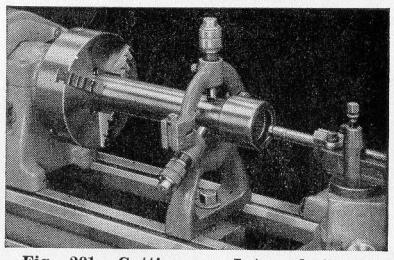


Fig. 281. Cutting an Internal Screw Thread with Work Supported in Steady Rest

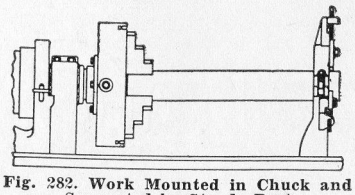


Fig. 282. Work Mounted in Chuck and Supported by Steady Rest

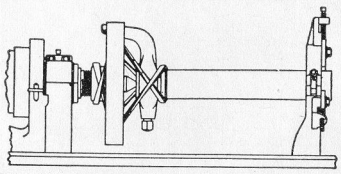


Fig. 283. Work Mounted on Center and Supported by Steady Rest

The Use of the Follower Rest

The follower rest is attached to the saddle of the lathe to support work of small diameter that is liable to spring away from the cutting tool. The adjustable jaws of the follower rest bear directly on the finished diameter of the work, as shown in Figs. 284 and 285. As the tool feeds along the work, the follower rest being attached to the saddle travels with the tool.

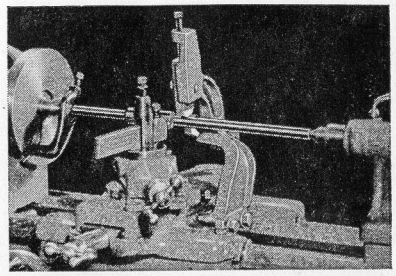


Fig. 284. Threading a Long Slender Shaft with the Aid of a Follower Rest

For the machining of small shafts in quantity, small rollers are sometimes substituted for the rigid adjustable jaws, and the device is then known as the roller bearing follower rest.

The application of both the steady rest and follower rest at the same time is shown in Fig. 286. The shaft to be machined, while very small in diameter is of considerable length, and in order to do an accurate job it is necessary to support the shaft with both the steady rest and follower rest.

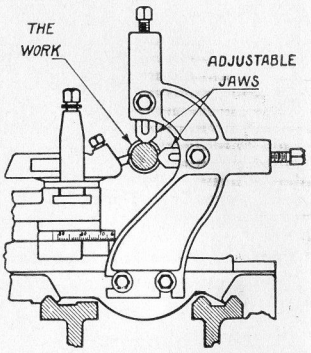


Fig. 285. Follower Rest Mounted on Lathe Saddle

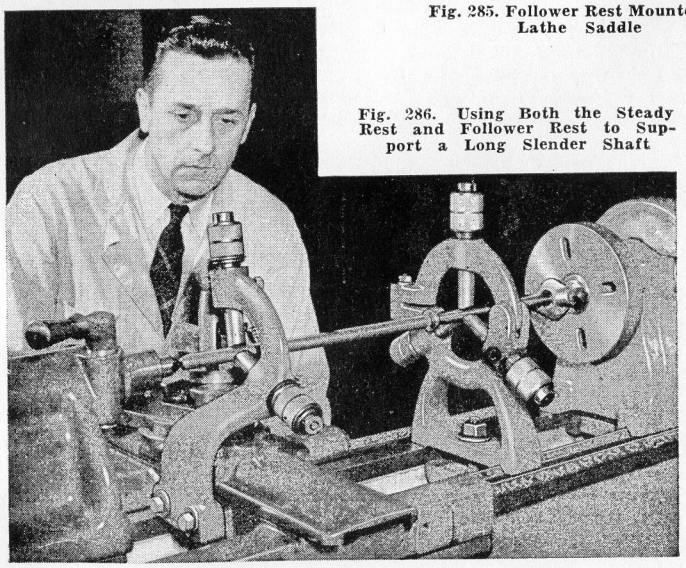


Fig. 286. Using Both the Steady Rest and Follower Rest to Support a Long Slender Shaft

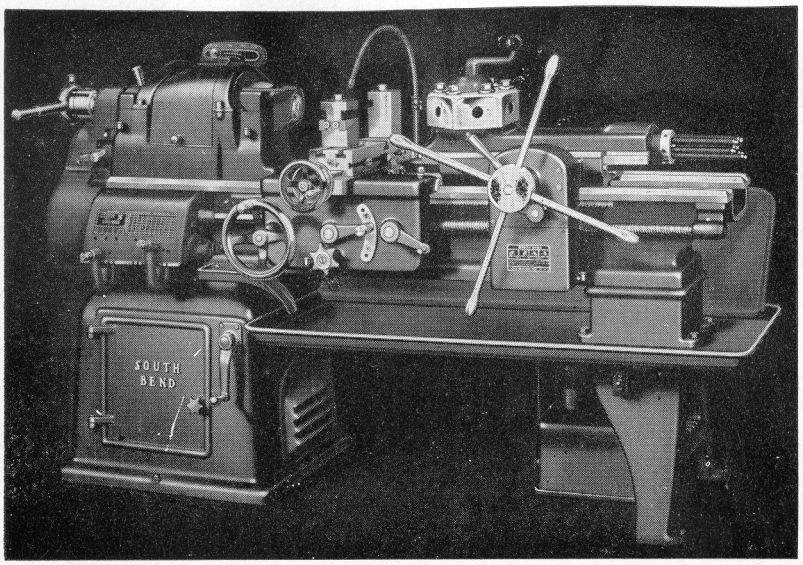


Fig. 287. No. 2-H South Bend Turret Lathe with Power Feed Bed Turret

Turret Lathes for Manufacturing

Turret Lathes are designed for the efficient production of duplicate parts. They are equipped with a power feed or hand lever feed turret having six faces, with automatic indexing and individual stop for each face. Cutting tools may be mounted in each of the six turret faces and indexed into position as required for performing various operations.

Turret lathes are usually equipped with either screw feed or hand lever operated double tool rest on the carriage cross slide. This permits using front and back tools for turning, facing, cutting off, and similar operations. A four-way turret tool post may be used on the cross slide.

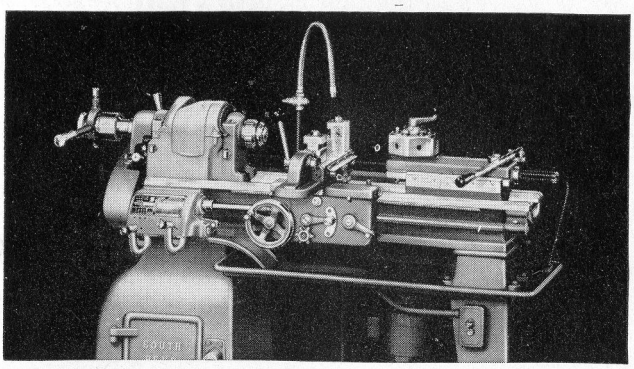


Fig. 288. 10-inch South Bend Turret Lathe with Hand-lever Operated Bed Turret

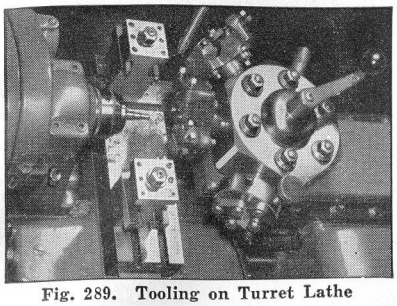


Fig. 289. Tooling on Turret Lathe

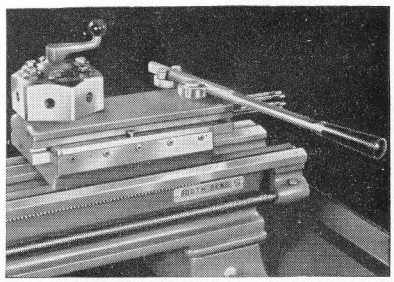


Fig. 290. Handlever Bed Turret

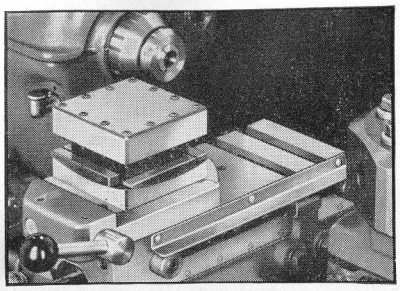


Fig. 291. Square Turret Tool Block

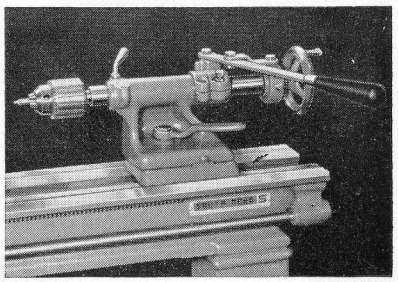


Fig. 292. Handlever Tailstock

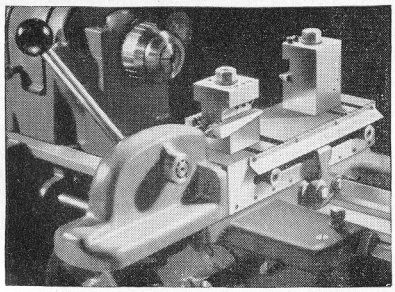


Fig. 293. Handlever Double Tool Rest

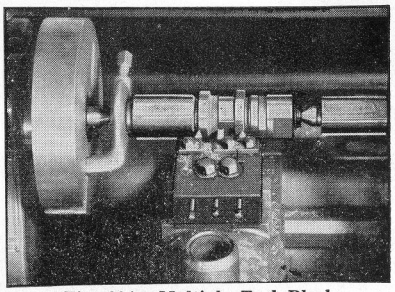


Fig. 294. Multiple Tool Block

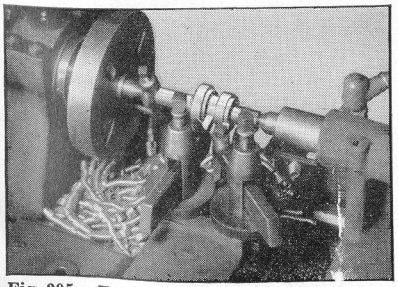


Fig. 295. Two Tools Cutting Simultaneously on a Piece of Work

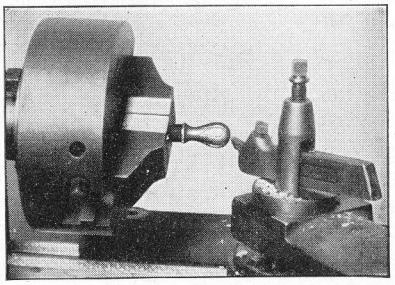


Fig. 296. An Irregular Shaped Piece Held in a Two-Jaw Chuck

Milling in the Lathe

The Milling and Keyway Cutting Attachment illustrated in Figs. 298 and 300 will take care of a great deal of milling in the small shop that does not have enough work to install an expensive milling machine.

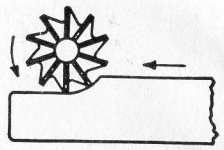


Fig. 297. Direction of Feed for Milling Operations

The cut is controlled by the hand wheel of the lathe carriage, the cross feed screw of the lathe and the vertical adjusting screw at the top of the milling attachment.

All milling cuts should be taken with the rotation of the cutter against the direction of the feed, shown in Fig. 297.

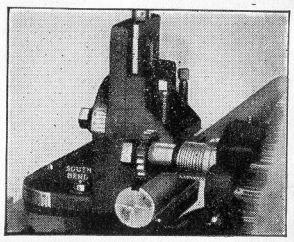


Fig. 298. Milling a Standard Keyway in a Shaft

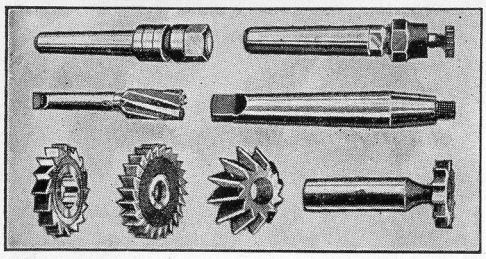


Fig. 299. An Assortment of Milling Cutters and Arbors

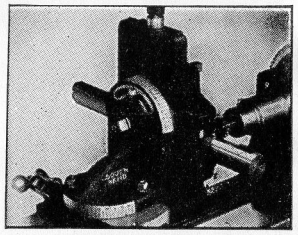


Fig. 300. Milling a Woodruff Keyway in a Shaft

Standard Keyways

The recognized standards for the depth and width of keyways in pulleys, gears, etc. are shown in Fig. 301 and the tabulation below. The same specifications are used for the depth and width of keyways in shafts.

The key should fit snugly in the keyway but must not be too tight.

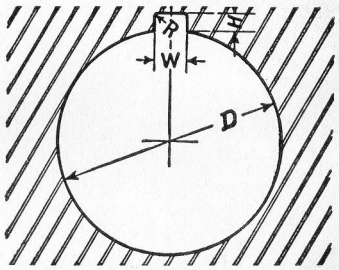


Fig. 301. Standard Keyway Dimensions

Specifications of American Standard Keyways

Diameter Hole D Inches	Width W Inches	Depth H Inches	Radius R Inches	Diameter Hole D Inches	Width W Inches	Depth H Inches	Radius R Inches
½	3/32	3/64	.020	2½	5/8	7/32	1/16
5/8 to 7/8	1/8	1/16	3/32	3	3/4	1/4	3/32
1	1/4	3/32	1/16	3½	7/8	5/16	3/32
1¼	5/16	1/8	1/16	4	1	3/8	3/32
1½	3/8	5/32	1/16	4½	1⅛	7/16	1/16
1¾	7/16	3/16	1/16	5	1¼	1/2	1/8
2	1/2	3/16	1/16				

Cutting Gears on the Lathe

The gear cutting attachment for the lathe, shown in Fig. 302, 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.

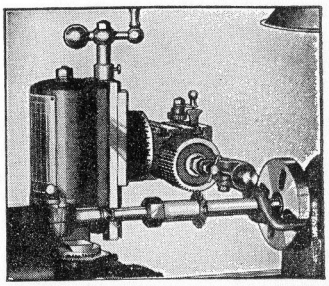


Fig. 302. Gear Cutting Attachment

This attachment is practical for cutting small gears and for milling small light work of various kinds on the screw cutting lathe.

The dividing head construction is based on the principle of interchangeable gears, the same as regularly used on gear cutting machines. The index plate shows the proper gears to use for divisions from 2 to 360.

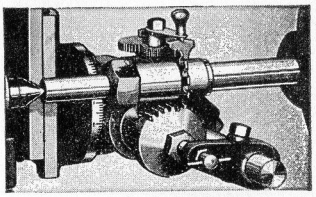


Fig. 303. Cutting a Gear on a Lathe

Turning Wood, Fiber, and Plastics

Turning wood in a metal working lathe is a very simple matter. Spur and cup centers are substituted for the 60° centers, 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.

Other materials may be machined as well. Alabaster, Catalin, Bakelite, fiber and other plastics, synthetic resins, etc., may be turned and polished with complete satisfaction.

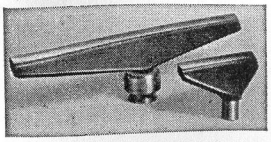


Fig. 304. Hand Rest



Fig. 305. Spur Center



Fig. 306. Cup Center



Fig. 307. Wood Turning in a Metal Working Lathe

Ten in One Tool Holder

The Ten in One Tool Holder replaces the conventional tool post and various tool holders ordinarily used for general lathe work. It provides rigid support for turning, boring, threading, and cutting-off tool bits. In addition, it is equipped with a self-aligning knurling head. Screw adjustments for tool height are easily made, and they stay put. No readjustment is required when replacing tools. The application of the Ten in One Tool Holder on various types of operations is shown in Fig. 308, and in the illustrations below.

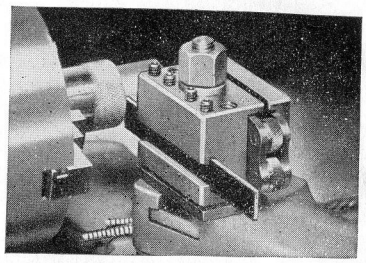
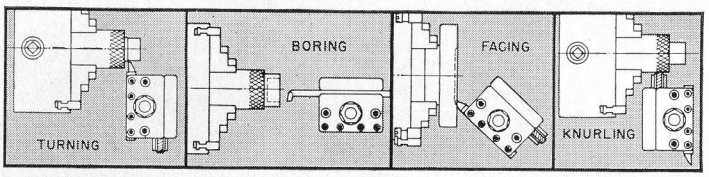


Fig. 308. Ten in One Tool Holder



Machinability of Various Grades of Steel

The machinability or free cutting quality of steel varies with the chemical analysis and the physical processes employed in its manufacture. The machinability ratings of popular grades of steel are listed in the table below. These ratings have been calculated to show the per cent of the relative cutting speeds based on a machinability rating of 100 for Bessemer Screw Steel No. B1112. The table shows the approximate cutting speeds in surface feet per minute for high speed steel cutting tools on average work. Higher or lower cutting speeds may be found more practical, depending on the cutting tool used, the rate of feed, the depth of cut, and the type of operation performed. See also pages 36 and 50.

Machinability Ratings and Cutting Speeds for Steel

AISI Number	Machinability Rating	Approximate Cutting Speed F.P.M.	AISI Number	Machinability Rating	Approximate Cutting Speed F.P.M.
B1111	94	155	3130*	72	120
B1112	100	165	3135*	70	115
B1113	136	225	3140*	66	110
C1117	91	150	4130*	72	120
C1118	91	150	4140*	66	110
C1119	100	165	4620	66	110
C1137	72	120	5120	76	125
C1141	70	115	5130	57	95
C1141*	81	135	5140*	70	115
C1144	76	125	6120	57	95
C1010†	...	120	8620	66	110
C1017	72	120	8630*	72	120
C1019	78	130	8640*	66	110
C1020	72	120	8650*	60	100
C1030	70	115	8720	66	110
C1035	70	115	8740*	66	110
C1040	64	105	E9310*	51	85
C1045	57	95	E9315*	49	80
2330*	70	115	E9317*	49	80
2340*	57	95	9763*	54	90

*Annealed †Light Feeds

Micrometer Carriage Stop

The micrometer carriage stop consists of a micrometer spindle mounted in a clamp which may be securely locked onto the front V-way of the lathe bed, as shown in Fig. 310. A lock screw is provided for locking the spindle at any point.

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

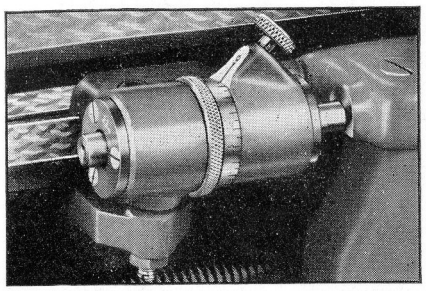


Fig. 310. Micrometer Carriage Stop

Four Position Carriage Stop

Much time can be saved in positioning the cutting tool for repetitive operations, by using the four position carriage stop shown in Fig. 311. It clamps onto the V-ways of the lathe bed, much the same as the micrometer stop. Each of the four adjustable stop screws may be set for a different tool position, and may be revolved into position to locate the carriage for each of four successive cuts. This attachment is especially desirable for spacing shoulders in shafts and similar operations.

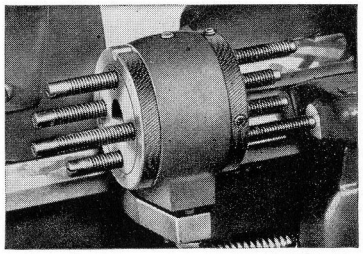


Fig. 311. Four Position Carriage Stop

Metric Graduated Collars

Metric cross-feed screw, metric compound rest screw and metric graduated collars are supplied on lathes that are to be used exclusively for working in the metric system.

The metric graduated collars read in tenths of a millimeter and are adjustable so that they may be set at zero whenever desired.

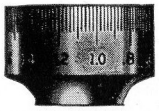


Fig. 312. Metric Graduated Collar

Metric Graduations on Taper Attachment

Taper attachments on lathes that are to be used for cutting tapers in the metric system are equipped with graduations reading in the metric system. Usually these graduations read in millimeters per centimeter and are in addition to the regular graduations.

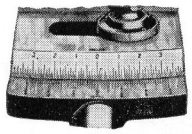


Fig. 313. Taper Attachment with Metric Graduations

Metric Graduations on Tailstock Spindle

The tailstock spindle of the lathe may be graduated in centimeters, as shown in Fig. 314. The graduations are to aid in drilling accurately to the required depth.

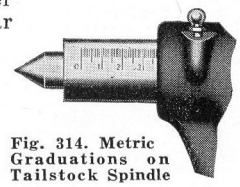


Fig. 314. Metric Graduations on Tailstock Spindle

Grinding in the Lathe

When equipped with a good electric grinding attachment the lathe can be used for sharpening reamers and milling cutters, grinding hardened bushings and shafts and many other grinding operations.

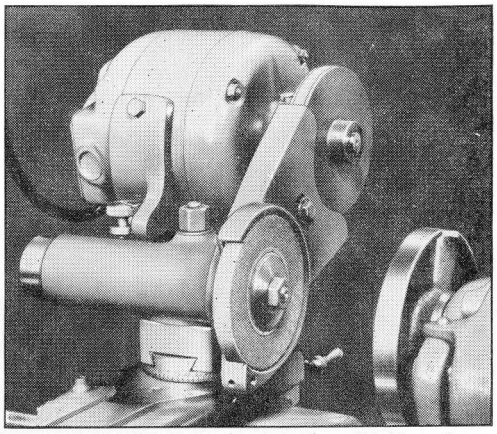


Fig. 315 External Grinding Attachment

The V-ways of the lathe bed should be covered with a heavy cloth or canvas to protect them from dust and grit from the grinding wheel, and the lathe spindle bearings should also be protected. A small pan of water or oil placed just below the grinding wheel will collect most of the grit.

A large, powerful grinder is most satisfactory for external grinding. The wheel should be at least four inches in diameter and the grinder should be mounted direct on the compound rest of the lathe, as shown in Fig. 315.

Internal Grinding

For grinding the inside of hardened drill jig bushings and other internal grinding, a high speed internal grinding attachment is used.

Fig. 315A shows an internal grinding attachment with compound V-belt drive which provides a spindle speed of 30,000 r.p.m.

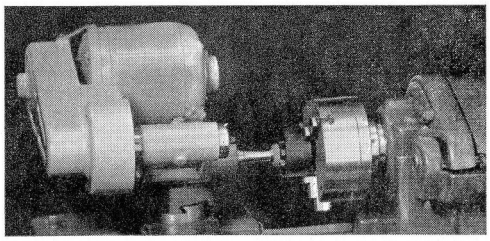


Fig. 315A. Internal Grinding Attachment

GRINDING WHEELS FOR VARIOUS KINDS OF WORK

Tabulation shows grade of Norton Grinding Wheels.

Kind of Work	Rough Grind	Finish Grind
Cast Iron	37C36-KV Crystolon	37C60-JV Crystolon
Soft Steel	57A46-M8VBE	57A60-M8VBE
Hardened Steel	57A46-L8VBE	57A60-L8VBE
High Speed Steel	57A46-K8VBE	57A60-K8VBE
Brass or Bronze	37C36-KV Crystolon	37C60-JV Crystolon
General Work	57A46-M8VBE	57A46-M8VBE
Aluminum	A30-M3E Shellac	A36-M3E Shellac
Bakelite	37C36-KV Crystolon	37C46-KV Crystolon
Soft Rubber	37C20-K5B-2 Crystolon Resinoid	37C46-K5B-2 Crystolon Resinoid
Hard Rubber	37C30-K5B-2 Crystolon Resinoid	37C60-K5B-2 Crystolon Resinoid
Automobile Valves	57A60-M8VBE	57A80-L8VBE
Tungsten Carbide	39C601-17V Crystolon	39C1002-H7V Crystolon

Diamond Dresser for Truing Grinding Wheel

The grinding wheel must be balanced and must be dressed with a diamond dresser if a smooth, accurate ground finish is to be obtained. The grinding wheel must be dressed frequently as it is used to keep it true and free from particles of metal which become embedded in the periphery of the wheel.

The diamond dresser consists of a small industrial diamond mounted in a steel shank, as shown in Fig. 318. The dresser must be rigidly supported in a fixture for truing the grinding wheel, as shown in Fig. 317.

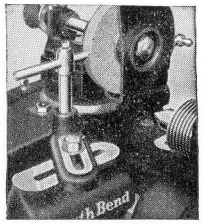


Fig. 317. Truing a Grinding Wheel with a Diamond Dresser

The diamond point of the dresser should be placed on center, or slightly below center and the revolving grinding wheel passed back and forth across the diamond. Remove about .001 in. from the wheel at each cut and dress the wheel just enough to make it run true.



Fig. 318. Diamond Dresser

Grinding Hardened Steel Parts

Hardened steel parts should be carefully ground in order to produce a smooth, accurate finish. The part should be machined to within a few thousandths of the finished size before it is hardened. After hardening, all scale should be removed before grinding. Remove only a few thousandths at each pass of the grinding wheel for if the part is ground too fast it may become overheated and warp, or the temper may be drawn.

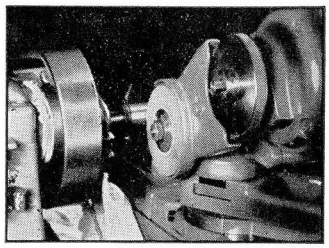


Fig. 319. Grinding a Hardened Steel Bushing

Sharpening Reamers and Cutters

Reamers and milling cutters may be sharpened by grinding in the lathe, as shown in Figs. 320, 321 and 322. Some reamers are first circular ground, then relieved by grinding with a tooth rest set slightly below center, as shown in Fig. 320, leaving a land .002 in. to .005 in. wide. Other reamers and most milling cutters are ground with about 2° relief.

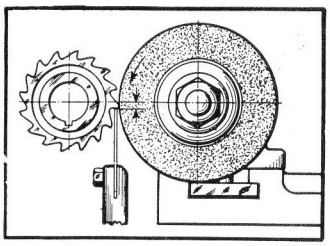


Fig. 320. Grinding Clearance on a Milling Cutter

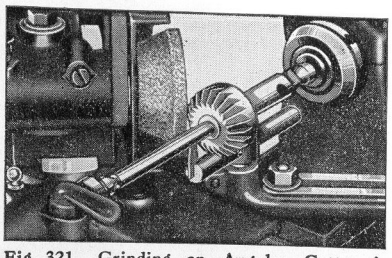


Fig. 321. Grinding an Angular Cutter in the Lathe

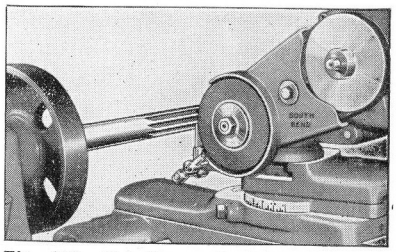


Fig. 322. Grinding a Straight Reamer in the Lathe

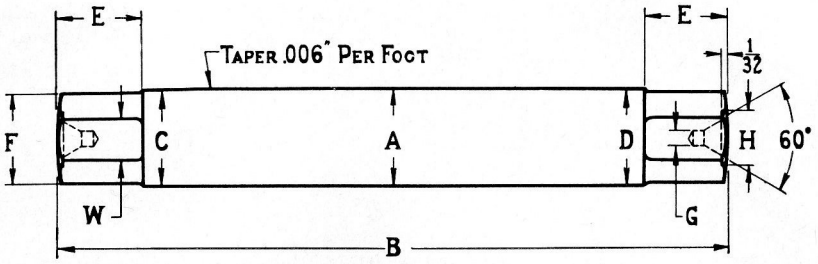


Fig. 323. Mandrel for machining work between centers in the lathe

How to Make Lathe Mandrels

Any good grade of machine steel can be used for making lathe mandrels. Old automobile axles are excellent. If only a few parts are to be made, the mandrel does not need to be hardened.

The tabulation below shows the dimensions recommended for standard lathe mandrels. A slight taper is required so that the mandrel can be pressed into the part tightly. The size of the mandrel is always stamped on the large end.

The center holes in the ends of the mandrel are very important. They should be large enough to provide a good bearing and they must be perfectly concentric with the outside diameter of the mandrel.

If a large quantity of parts are to be made, it is advisable to case-harden the center holes or harden the entire mandrel. The outside diameter of the mandrel must be finished after hardening; otherwise it will not run true because the steel will warp during the hardening process.

Before mounting a part on the mandrel, always oil both the inside of the part and the mandrel. See page 90.

All Dimensions Shown in Tabulation Below are in Inches

Nominal Diameter A	Total Length B	Small End C	Large End D	Undercut Length E	Undercut Diameter F	Center Drill G	Recess for Center H	Width of Flat W
3/16	3 3/4	.1870	.1884	7/16	11/64	3/64	1/8	5/64
1/4	3 3/4	.2495	.2509	7/16	15/64	3/64	1/8	7/64
5/16	4	.3120	.3136	7/16	9/32	3/64	3/16	1/8
3/8	4 1/4	.3745	.3760	1/2	11/32	1/16	1/4	5/32
7/16	4 1/2	.4370	.4387	9/16	13/32	1/16	9/32	1/4
1/2	5	.4995	.5014	5/8	15/32	3/32	5/16	1/4
9/16	5 1/4	.5620	.5640	5/8	17/32	3/32	5/16	1/4
5/8	5 1/2	.6245	.6265	3/4	19/32	3/32	3/8	1/4
11/16	5 3/4	.6870	.6891	3/4	5/8	3/32	3/8	5/16
3/4	6	.7495	.7517	13/16	11/16	1/8	3/8	5/16
13/16	6 1/4	.8120	.8142	7/8	3/4	1/8	3/8	7/16
7/8	6 1/2	.8740	.8764	7/8	13/16	1/8	3/8	7/16
15/16	6 3/4	.9370	.9394	15/16	7/8	1/8	3/8	1/2
1	7	.9995	1.0020	15/16	15/16	5/32	1/2	1/2
1 1/16	7 1/4	1.0615	1.0641	1	1	5/32	1/2	1/2
1 1/8	7 1/2	1.1240	1.1267	1	1 1/16	5/32	1/2	1/2
1 3/16	7 3/4	1.1865	1.1889	1	1 1/8	5/32	1/2	1/2
1 1/4	8	1.2490	1.2520	1	1 3/16	3/16	5/8	1/2
1 5/16	8 1/4	1.3115	1.3144	1 1/4	1 1/4	3/16	5/8	1/2
1 3/8	8 1/2	1.3740	1.3760	1 1/4	1 5/16	3/16	5/8	5/8
1 7/16	8 3/4	1.4365	1.4396	1 1/4	1 3/8	7/32	5/8	5/8
1 1/2	9	1.4990	1.5022	1 1/4	1 7/16	7/32	5/8	5/8

Press Fits and Running Fits

Standard tolerances for press fits and running fits, etc., are given in the tabulations below. The hole is usually made a standard size and the shaft made the required size for the desired type of fit. The figures shown in the tabulations below indicate the amount to increase or decrease the shaft diameter provided a standard hole size is maintained. The tolerance observed as standard for holes is usually +.000" −.001".

Since working conditions vary a great deal it may sometimes be advisable to increase or decrease the allowances shown in the tabulations. For example, the length of the bearing, the material used, and the speed should all be taken into consideration when calculating the tolerance for a running fit.

Standard Tolerances for Press Fits

Light Press Fit

Dia. of Hole, in inches	Shaft Dia. More Than Hole, in inches
Up to ½	+.0004 to +.0006
½ to 1	+.0005 to +.0010
1 to 2	+.00075 to +.0020
2 to 3	+.0015 to +.0030
3 to 4	+.0020 to +.0040
4 to 5	+.0020 to +.0045
5 to 6	+.0030 to +.0050

Heavy Press Fit

Dia. of Hole, in inches	Shaft Dia. More Than Hole, in inches
Up to ½	+.0005 to +.001
½ to 1	+.001 to +.003
1 to 2	+.002 to +.004
2 to 3	+.003 to +.006
3 to 4	+.005 to +.008
4 to 5	+.006 to +.010
5 to 6	+.008 to +.012

Standard Tolerances for Running Fits

Speeds up to 1000 r.p.m.

Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to ½	—.0005 to —.0010
½ to 1	—.00075 to —.0015
1 to 2	—.0015 to —.0025
2 to 3	—.0020 to —.0025
3 to 4	—.0025 to —.0030
4 to 5	—.0030 to —.0035
5 to 6	—.0035 to —.0040

Speeds over 1000 r.p.m.

Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to ½	—.0005 to —.0010
½ to 1	—.0010 to —.0020
1 to 2	—.0020 to —.0030
2 to 3	—.0025 to —.0035
3 to 4	—.0030 to —.0040
4 to 5	—.0035 to —.0045
5 to 6	—.0040 to —.0050

Standard Tolerances for Push Fits

Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to ½	—.00025 to —.00075
½ to 1	—.0005 to —.0010
1 to 2	—.0005 to —.0015
2 to 3	—.0005 to —.0015
3 to 4	—.00075 to —.0020
4 to 5	—.00075 to —.0020
5 to 6	—.00075 to —.0020

Standard Tolerances for Sliding Fits

Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to ½	—.0005 to —.001
½ to 1	—.00075 to —.0015
1 to 2	—.0015 to —.0025
2 to 3	—.0020 to —.0030
3 to 4	—.0025 to —.0030
4 to 5	—.0025 to —.0035
5 to 6	—.0025 to —.0040

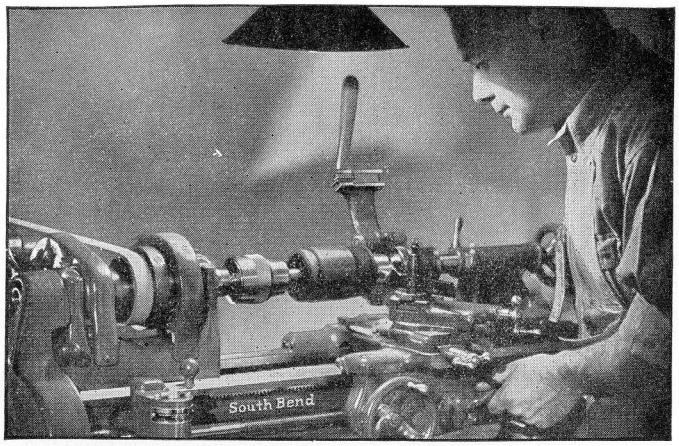


Fig. 327. Truing and Undercutting an Armature Commutator in the Lathe

The Lathe in the Auto Service Shop

The Back-Geared Screw Cutting Lathe is frequently called the "Universal Tool," and this applies in automotive service work as well as in general industry. Most of the mechanical parts of the automobile, bus, truck, tractor and airplane are originally made on lathes or in special machines which are adaptations of the lathe.

A lathe with 9-in. or 10-in. swing is very practical for handling such jobs as refacing valves; truing armature commutators and undercutting mica; finishing pistons; beveling piston skirts; reaming piston pin holes; making bushings, bearings and glands; cutting screw threads, testing and straightening bent shafts, and many other jobs. Special attachments used on the lathe greatly increase its versatility.

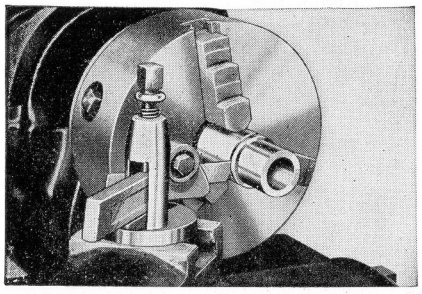


Fig. 328. Making a Replacement Bushing Complete in a 9-in. Lathe

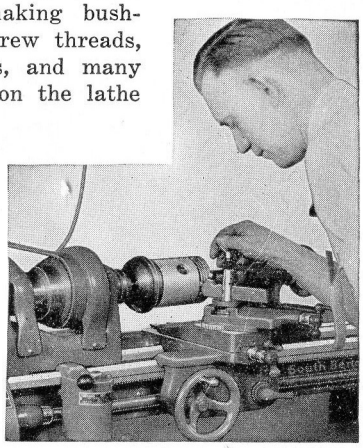


Fig. 329. Turning a Semi-Machined Piston to Size in a 9-in. Lathe

Armature Truing

Machining the commutator of an armature true and under-cutting the mica are two of the most important jobs in auto electrical work, and these jobs are most easily handled in the lathe.

A small lathe equipped for these jobs is shown in Fig. 330. The undercutting attachment is mounted on the lathe in such a way that it is ready for instant use, yet it does not interfere with turning the commutator.

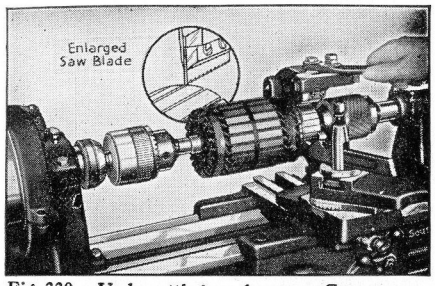


Fig. 330. Undercutting an Armature Commutator in the Lathe

Refacing Valves

A lathe equipped with a grinding attachment and a special hollow valve chuck for refacing valves is shown in Fig. 331.

Other valve jobs done in the lathe include: Truing the valve tappet face and rocker arm face; making valve guide bushings and valve seat replacement rings, etc.

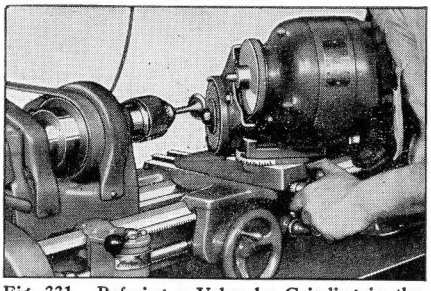


Fig. 331. Refacing a Valve by Grinding in the Lathe

Finishing Pistons

Pistons of all sizes and types can be rough and finish turned in the lathe, as shown in Fig. 332. The lathe can also be used for reaming and honing piston pin holes, cutting oil grooves in pistons, remachining piston ring grooves, beveling piston skirts, etc.

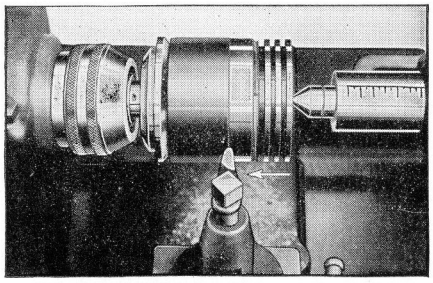


Fig. 332. Finishing a Piston in the Lathe

Straightening Bent Shafts

With the aid of a piece of chalk, the high spot on a bent shaft can easily be located and marked for straightening as it revolves between the lathe centers. The shaft should be removed from the lathe and placed in a press or on an anvil for straightening.

A dial test indicator mounted in the tool post of the lathe permits testing and straightening shafts to a high degree of accuracy. See Fig. 337.

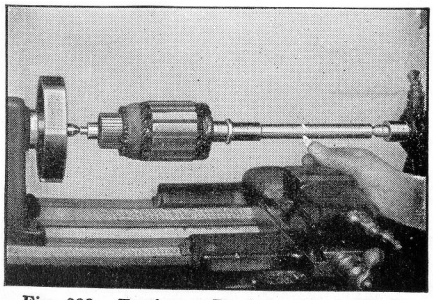


Fig. 333. Testing a Bent Armature Shaft in the Lathe

Machining Eccentrics

A simple eccentric can be machined on a straight mandrel having two sets of center holes as shown in Fig. 334. One set of centers is used for machining the concentric hub and the other set of centers is used for machining the eccentric part.

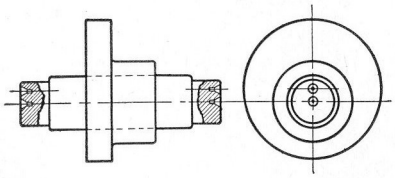


Fig. 334. A Mandrel with Two Sets of Center Holes for Machining an Eccentric

Crankshaft Turning

Crankshaft turning is an adaptation of eccentric machining. A single throw crankshaft mounted in the lathe for machining the throw bearing is shown in Fig. 335. The adapters attached to each end of the crankshaft have offset center holes corresponding to the throw of the crankshaft.

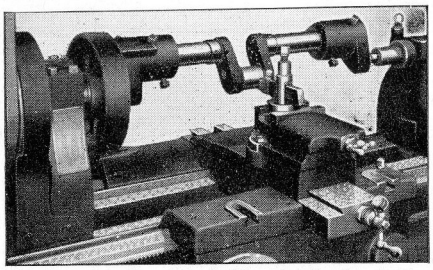


Fig. 335. A Crankshaft Mounted in the Lathe for Machining the Throw Bearing

Truing Crankshaft Bearings

Throw bearings of automobile crankshafts are often worn out of round or scored and must be reground. A special crank pin grinding attachment shown in Fig. 336, permits truing the throw bearings without the use of offset centers. The grinding wheel travels around with the throw bearing in such a way that it will grind the bearing round and straight. The lathe spindle must revolve very slowly (about 10 r.p.m.) while this tool is being used.

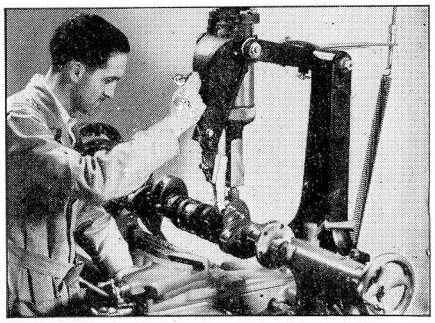


Fig. 336. Truing the Throw Bearings of a Crankshaft by Regrinding in the Lathe

Testing Crankshafts

Crankshafts may be tested between the lathe centers, as shown in Fig. 337. The dial indicator mounted in the tool post of the lathe reads in thousandths of an inch and will show exactly how much the crankshaft is sprung and will also disclose any out-of-round condition of the bearing. Straightening a crankshaft is a delicate job and should be attempted only by an experienced mechanic.

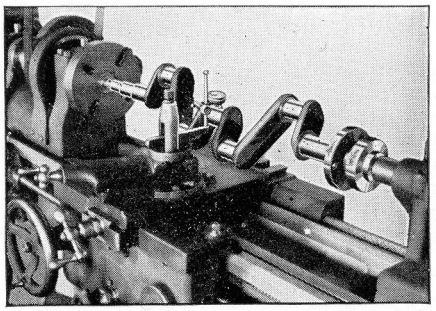


Fig. 337. Testing a Crankshaft in the Lathe

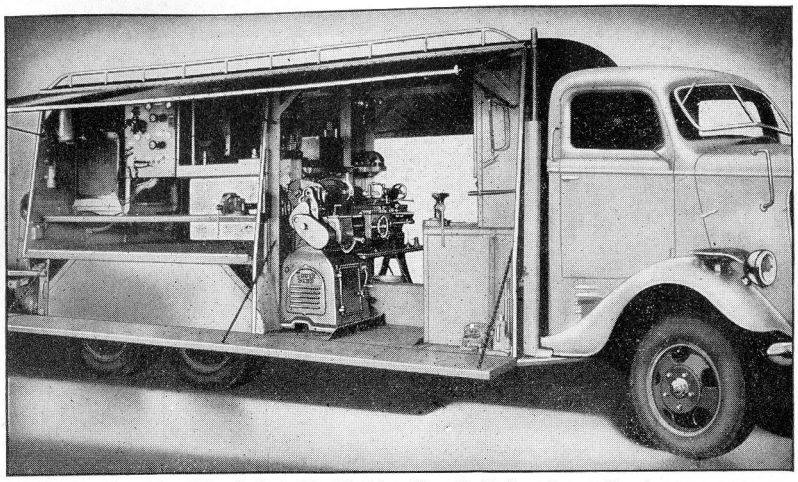


Fig. 338. A Portable Machine Shop Built in a Large Truck

Portable Machine Shop

The portable machine shop shown in Fig. 338 is rapidly gaining in popularity. This type of shop is especially valuable for service in oil fields, construction camps, air ports, army posts, etc., also for the maintenance of road building equipment and for repairing construction machinery and equipment on large engineering projects. The advantage in taking the shop to the job is obvious when the delay and difficulty involved in transporting heavy, awkward parts to and from the shop are taken into consideration.

The equipment of the portable machine shop may be quite complete, consisting of a 16 in. by 8 ft. lathe, a 20-in. drill press, a forge, anvil, grinder, welding outfit, etc., as shown above, or it may be limited to a small lathe and a good assortment of small tools. The equipment will vary with the purpose of the shop and the amount to be invested.

The lathe is the most important of all of the tools in the portable machine shop because it can be used for so many classes of work. When equipped with the necessary attachments, the lathe may be used as a milling machine, gear cutting machine, grinding machine, drill press, etc.

Regardless of the size of the lathe and other equipment, it is important that the truck or trailer in which the shop is installed be of substantial construction with a good solid floor. Provision should be made for blocking and leveling the floor while the machinery is in use. All parts must be securely fastened in place so there will be no danger of damage while the shop is being transported from one location to another.

Power for operating the lathe and other machinery is usually obtained through a generator installed in such a way that it can be operated by the truck engine. This same generator also supplies current for electric lights to illuminate the shop and also large flood lights which permit working near the unit after night.

Cutting Speeds for Turning—Drilling—Tapping With High Speed Steel Cutting Tools

Material	Turning Speeds		Drilling Speeds		Tapping Speeds	
	Ft. per Minute	Lubricant	Ft. per Minute	Lubricant	Ft. per Minute	Lubricant
Aluminum	300-400	Comp. or Kerosene	200-330	Comp. or Kerosene	90-110	Kerosene & Lard Oil
Brass, leaded	300-700	Dry or Comp.	200-500	Comp.	150-250	Comp. or Lt. Base Oil
Brass, red and yellow	150-300	Comp.	75-250	Comp.	60-150	Comp. or Lt. Base Oil
Bronze, leaded	300-700	Comp.	200-500	Comp.	150-250	Comp. or Lt. Base Oil
Bronze, phosphor	75-150	Comp.	50-125	Comp.	30- 60	Comp. or Lt. Base Oil
Cast Iron	50-110	Dry	100-165	Dry	70- 90	Dry or Comp.
Cast Steel	45- 90	Comp.	35- 45	Comp.	20- 35	Sul. Base Oil
Copper, leaded	300-700	Comp.	200-500	Comp.	150-250	Lt. Base Oil
Copper, electro.	75-150	Comp.	50-125	Comp.	30- 60	Lt. Base Oil
Chrome Steel	65-115	Comp.	50- 65	Comp.	20- 35	Sul. Base Oil
Die Castings	225-350	Com-pound	200-330	Com-pound	60- 80	Kerosene & Lard Oil
Duralumin	275-400	Com-pound	250-375	Com-pound	90-110	Comp. or Ker. and Lard Oil
Fiber	200-300	Dry	175-275	Dry	80-100	Dry
Machine Steel	115-225	Com-pound	80-120	Com-pound	40- 70	Comp., Sul. Base Oil or Kero. & Para
Malleable Iron	80-130	Dry or Comp.	80-100	Dry or Comp.	35- 70	Comp. or Sul. Base Oil
Mang. Bronze	150-300	Comp.	75-250	Comp.	60-150	Lt. Base Oil
Mang. Steel	20- 40	Comp.	15- 25	Comp.	10- 20	Comp. or Sul. Base Oil or Ker. & Para
Moly. Steel	100-120	Comp.	50- 65	Comp.	20- 35	Sul. Base Oil
Monel Metal	100-125	Comp. or Sul. Base	40- 55	Sul. Base	20- 30	Sul. Base or Kero. and Lard Oil
Nickel Silver 18%	75-150	Comp.	50-125	Comp.	30- 60	Sul. Base or Kero. and Lard Oil
Nickel Silver, leaded	150-300	Comp.	75-250	Comp.	60-150	Sul. Base or Kero and Lard Oil
Nickel Steel	85-110	Comp. or Sul. Base	40- 65	Sul. Base Oil	25- 40	Sul. Base Oil
Plastics, hot-set molded	200-600	Dry	75-300	Dry	40- 54	Dry or Water
Rubber, Hard	200-300	Dry	175-275	Dry	80-100	Dry
Stainless Steel	100-150	Sul. Base	30- 45	Sul. Base	15- 30	Sul. Base
Tool Steel	70-130	Comp.	50- 65	Comp.	25- 40	Sul. Base or Kero. and Lard Oil
Tungsten Steel	70-130	Comp.	50- 65	Comp.	20- 35	Sul. Base
Vanadium Steel	85-120	Comp.	45- 65	Sul. Base	25- 40	Sul. Base

The above speeds have been collected from several sources and are suggested as practical for average work. Special conditions may necessitate the use of higher or lower speeds for maximum efficiency.

Use of Coolant

Coolants are used extensively in the machining of steel parts in order to permit higher cutting speeds, produce a better finish, and increase tool life. The principal function of a coolant is, as its name implies, to cool the work and the cutting tool. The coolant also facilitates production by lubricating the cutting tool, flushing away chips, and preventing rust.

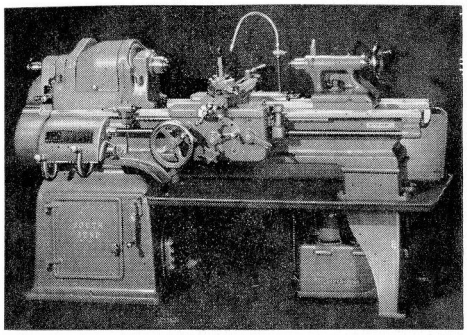


Fig. 341. Lathe Equipped with Coolant Pan, Pump, Reservoir, and Piping

Coolant Equipment

Very simple equipment can sometimes be used effectively in applying a coolant. For example, a small paint brush may be used as shown in Fig. 238, page 81. An ordinary oil can is also convenient for applying a limited amount of coolant. However, on continuous high speed production the lathe should be equipped with an oil pan, coolant pump, and reservoir as shown in Fig. 341.

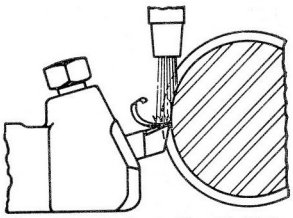


Fig. 342. Coolant Should Make Contact with Work Where Cutting Action Takes Place

Application of Coolant

For effective cooling, it is important that the coolant be properly applied to the work. A large stream at slow velocity is preferable to a small, high velocity stream. The coolant should make contact with the work at the exact spot where the cutting action takes place, not above or to one side of the cutting tool. See Fig. 342.

Types of Coolants

Coolants considered most effective for various classes of work are listed in the table on page 108. Each coolant has some qualities which make it especially desirable for certain applications. The characteristics of the most popular coolants are as follows:

Lard Oil—One of the oldest and best coolants, and most expensive. Especially good for cutting screw threads, drilling deep holes, and reaming. Provides excellent lubrication, increases tool life, and produces a smooth finish on the work. Prevents rust.

Mineral Lard Oil Mixtures—Various mixtures of lard oil and petroleum base mineral oils are used in lieu of lard oil because they are more fluid, less expensive and are almost as effective.

Mineral Oils—Petroleum base oils compounded with chemicals to improve their lubricating and anti-welding qualities. Less expensive than lard oil and mineral oil mixtures.

Soluble Oils—Mineral oils which have been treated so that they may be mixed with water to form an emulsion and provide an excellent low cost coolant. Although they carry away heat better than lard oil or mineral oil, their lubricating qualities are comparatively poor. Their use is usually limited to rough turning operations. Even though they are mixed with water, they leave a protective film on metal which resists rusting.

Soda Water Mixtures—Cheapest of all coolants, soda water mixtures are very effective for cooling but have practically no lubricating qualities and will cause steel or iron to rust. A popular mixture consists of 1 pound of sal-soda (carbonate of soda), 1 quart of lard oil, 1 quart of soft soap, and 10 gallons of water, boiled together for ½ hour.

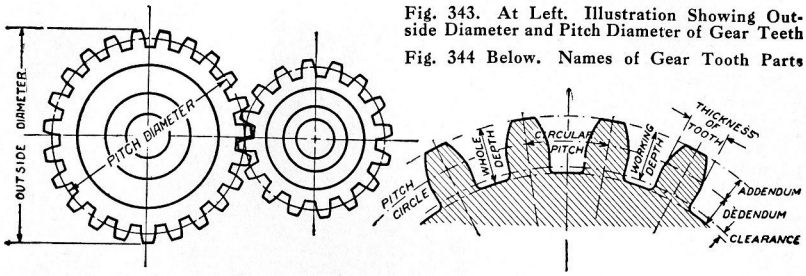


Fig. 343. At Left. Illustration Showing Outside Diameter and Pitch Diameter of Gear Teeth

Fig. 344 Below. Names of Gear Tooth Parts

Information on Gears

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

Diametral Pitch—Number of teeth divided by pitch diameter, or 3.1416 divided by circular pitch.

Example: If a gear has 40 teeth and the pitch diameter is 4 in., the diametral pitch is 40 divided by 4, and the diametral pitch is 10, or in other words there are 10 teeth to each inch of the pitch diameter, and the gear is 10 diametral pitch.

Circular Pitch—Distance from center to center of two adjacent teeth along pitch circle, or 3.1416 divided by diametral pitch.

Pitch Diameter—Number of teeth divided by diametral pitch.

Example: If the number of teeth is 40 and the diametral pitch is 4, divide 40 by 4, and the quotient, 10, is the pitch diameter.

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

Example: If the number of teeth is 40 and the diametral pitch is 4, add 2 to the 40, making 42, and divide by 4; the quotient, 10½, is the outside diameter of gear or blank.

Addendum—1 divided by diametral pitch.

Whole Depth of Tooth—2.157 divided by diametral pitch.

Thickness of Tooth—1.5708 divided by diametral pitch.

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 10 in. and the diametral pitch is 4, multiply 10 by 4 and the product, 40, will be the number of teeth in the gear.

Example: If the outside diameter is 10½ and the diametral pitch is 4, multiply 10½ by 4 and the product, 42, less 2, or 40, is the number of teeth.

Center Distance—Total number of teeth in both gears divided by two times the diametral pitch.

Example: If the two gears have 50 and 30 teeth respectively, and are 5 pitch, add 50 and 30, making 80, divide by 2, and then divide the quotient, 40, by the diametral pitch, 5, and the result, 8 in., is the center distance.

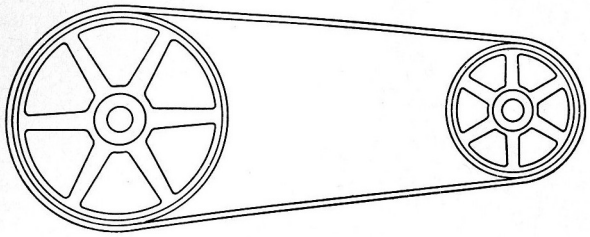


Fig. 345. A Pair of Pulleys for Flat Belt Drive

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 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 260 R.P.M. Speed of the driven pulley 390 R.P.M. Diameter of the driven pulley 8 in.

To find the diameter of the driving pulley.

$$390 \times 8 = 3120$$
$$3120 \div 260 = 12$$

The diameter of the driving pulley is 12 in.

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

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

Fitting a Chuck Plate to a Chuck

Before a chuck can be used on a lathe it must be fitted with a chuck plate that has been threaded to fit the spindle nose of the lathe. Semi-machined chuck plates that have been accurately threaded to fit the lathe spindle can be obtained from the lathe manufacturer.

Mounting Chuck Plate on Spindle

Before screwing the chuck plate on to the spindle nose of the lathe, clean the threads of the chuck plate and the spindle nose thoroughly. Make sure that there are no chips, burrs or small particles of dirt lodged in the screw threads, or on face of hub, and also make sure that the shoulder on the headstock spindle is perfectly clean and free from chips or burrs.

Oil the threads of the headstock spindle and the chuck plate and screw the chuck plate onto the spindle nose. Do not jam the threads tight or it may be difficult to remove the chuck plate after it has been finished.

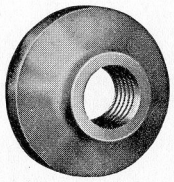


Fig. 346. Semi-Machined Chuck Plate, Threaded to Fit the Spindle Nose of the Lathe

Finishing the Flange

First, machine the face of the chuck plate, taking one roughing cut about $\frac{1}{32}$ in. deep, and then one or two finishing cuts, removing not over .001 in. in the last cut.

Measure the diameter of the recess in the back of the chuck carefully with inside calipers, and set outside calipers to correspond with the inside calipers. Machine the diameter of the chuck plate flange very carefully. Take very light finishing cuts and try the chuck on the chuck plate frequently, as the chuck plate must fit snugly into the recess in the back of the chuck.

After the chuck plate has been finished to fit the recess in the back of the chuck, remove it from the lathe spindle and chalk the face of the flange thoroughly. Place the flange in the recess in the back of the chuck and tap lightly on the chuck plate so that the edge of the bolt holes in the chuck will mark the location of bolt holes.

Drill the holes $\frac{1}{16}$ in. larger in diameter than the bolts used to secure the chuck plate onto the chuck. It is very important that the bolt holes be large enough to eliminate all possibility of the bolts binding.

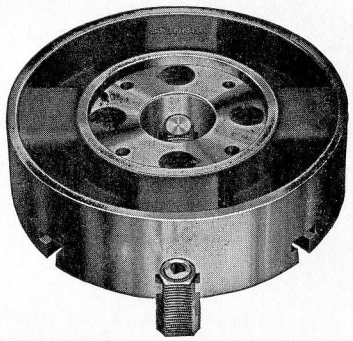


Fig. 347. Rear View of the Lathe Chuck

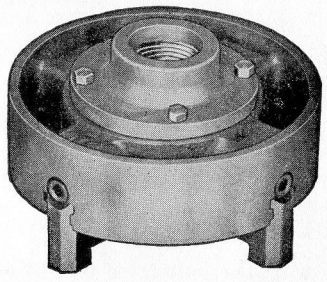


Fig. 348. Chuck with Chuck Plate Attached

Hardening and Tempering Lathe Tools

After a forged lathe tool has been used for some time, it should be re-forged, hardened and tempered. If carefully done, this will make the tool as good as new. Before attempting to harden and temper the tool, make sure of the kind of steel from which it is made.

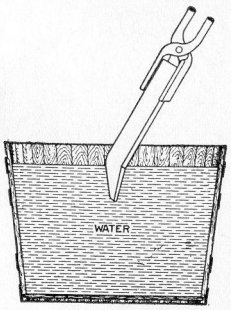


Fig. 349. Hardening a Lathe Tool

To Distinguish Carbon Tool Steel

To distinguish carbon tool steel from high speed steel, touch against emery wheel. Carbon steel gives off a shower of bright yellow sparks; high speed steel gives off a few dark red sparks.

To Harden Carbon Tool Steel

To harden a forged lathe tool made of carbon tool steel, heat the end of the tool slowly to a bright cherry red for a distance of at least an inch back of the cutting edge; then immerse the point about 1½ in. deep in cold water, but do not cool the shank. When the point is cool remove from water, polish the cutting edge with emery cloth and wipe with oily rag.

The tool is now hardened, and as the heat in the shank passes into the point it will discolor the polished surface, indicating the amount the temper is drawn. When a light straw color appears, cool the entire tool quickly in water and it will have the correct strength and toughness for metal turning.

The method outlined above may be followed for hardening and tempering any tool made of carbon tool steel. For wood cutting tools, taps and dies, draw to a dark straw color. For hatchets, screw drivers, cold chisels, etc., draw to a brown yellow; for springs dark purple.

Case Hardening

To case harden a piece of machinery steel, heat the steel to a cherry red; then remove from fire and apply cyanide of potassium to the surface you wish to case harden. The cyanide will dissolve slowly and be absorbed by the steel. After the surface has received a thorough coat of cyanide return the steel to the fire and heat slowly for about one minute so that the cyanide will be thoroughly absorbed by the steel. Remove from the fire and quench in cold water.

How to Anneal Tool Steel

Carbon tool steel may be annealed by heating slowly and evenly to a cherry red and then placing in a box of lime or ashes to cool slowly. The steel should be completely covered and when it is cooled to room temperature will be ready for machining.

How to Anneal Brass

Brass that has been hardened through cold working may be annealed by heating to a dull red when held in dark shadow and plunging into cold water. Care must be taken not to overheat the brass.

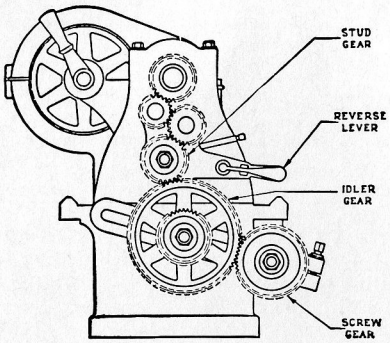


Fig. 350. Simple gearing

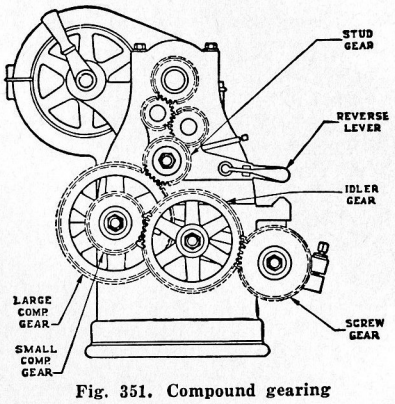


Fig. 351. Compound gearing

How to Calculate Change Gears for Thread Cutting

If it is necessary to cut a special thread that does not appear on the index chart of a lathe or if no index chart is available, the gears required can easily be calculated. All South Bend Lathes are even geared; that is, the stud gear revolves the same number of revolutions as the headstock spindle, and when gears of the same size are used on both the lead screw and stud, the lead screw and spindle revolve the same number of revolutions, so it is 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, as shown in Fig. 350, 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 6 threads per inch, the ratio of the change gears would be 6 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 number of threads per inch to be cut and the number of threads per inch in the lead screw by the same number.

Example: Problem—To cut 10 threads per inch on lathe having lead screw with 6 threads per inch.

Solution— $6 \times 8 = 48$ — No. of teeth in gear on stud.

$10 \times 8 = 80$ — 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.

When compound gearing, as shown in Fig. 351, is used, the ratio of the compound idler gears must also be taken into consideration, but otherwise the calculations are the same as for simple gearing. Usually, the compound idler gear ratio is 2 to 1, so that the threads cut are just twice the number per inch as when simple gearing is used.

DECIMAL EQUIVALENTS OF FRACTIONAL PARTS OF AN INCH

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

METRIC AND ENGLISH LINEAR MEASURE

The measuring rules shown below are graduated, in the Metric system and in the English system. This shows at a glance the comparison of the fractions of the Metric and English units, the meter and the inch. It will be noticed that one inch is equal to 25.4001 + mm.

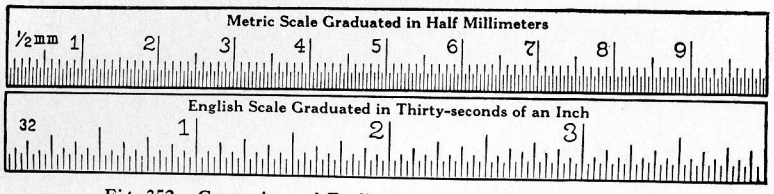


Fig. 352. Comparison of English and Metric Scales, Actual Size

TABLE OF METRIC LINEAR MEASURE

10 Millimeters	= 1 Centimeter	1 Centimeter	= .3937 inch
10 Centimeters	= 1 Decimeter	1 Decimeter	= 3.937 inches
10 Decimeters	= 1 Meter	1 Meter	= 39.37 inches

Equivalents of Millimeters in Decimals of Inches

1/10 mm = .00394 in.	8 mm = .31496 in.	18 mm = .70866 in.
1/5 mm = .00787 in.	9 mm = .35433 in.	19 mm = .74803 in.
1/2 mm = .01969 in.	10 mm = .39370 in.	20 mm = .78740 in.
1 mm = .03937 in.	11 mm = .43307 in.	21 mm = .82677 in.
2 mm = .07874 in.	12 mm = .47244 in.	22 mm = .86614 in.
3 mm = .11811 in.	13 mm = .51181 in.	23 mm = .90551 in.
4 mm = .15748 in.	14 mm = .55118 in.	24 mm = .94488 in.
5 mm = .19685 in.	15 mm = .59055 in.	25 mm = .98425 in.
6 mm = .23622 in.	16 mm = .62992 in.	26 mm = 1.02362 in.
7 mm = .27559 in.	17 mm = .66929 in.	27 mm = 1.06299 in.

SHOP KINKS

From "American Machinist"

A good lubricant for turning, boring and milling aluminum is made of equal parts of lard oil and kerosene. Kerosene alone is also good, and cheap.

When drilling or turning hard steel (not hardened steel) in the lathe, run slowly and lubricate the tool with turpentine, or turpentine and spirits of camphor.

Red lead and graphite are good lubricants for the tail center. On heavy work, make the countersinks for the center as large as possible without making the job unsightly.

Probably the handiest lathe chuck for the jobbing shop is the four-jawed independent chuck having stepped, reversible jaws. These will hold almost any shaped piece, and hold it firmly.

Before complaining that the lathe is not in line, be sure that the bed is carefully leveled up. Don't twist the lathe bed out of shape by pulling it down on to an uneven floor with lagscrews, and then expect it to turn straight.

Squared ends and a uniform depth of lathe center are desirable for accurate and economical work. Don't make the mistake of thinking that any boy can do the centering well enough without proper instruction and supervision.

A little square of mica with a tin rim makes a good chip guard when turning brass. A simple wire spring clip can be fastened to the tin rim so that the guard can be readily fastened to or detached from the tool or tool post.

If a $\frac{1}{32}$-in. slot is made on the top of the tail center running from the point back to a little beyond the large part of the conical end, the center can be oiled without slacking it back. This is better than grinding half the conical end away.

The indicator, of the dial variety, is an extremely handy tool around the engine lathe, particularly if accurate work is to be done. Don't get the idea that this is an unnecessary frill, but get accustomed to using it if you wish to become an accurate workman.

Universal lathe chucks are very convenient to have in the shop but can rarely be depended on to run true if accuracy is desired. They may be plenty good enough for a large proportion of the work to be done, but accurate work demands separate adjustment of the jaws.

The lathe centers should be well cared for, the point ground in place, if possible, and always put into place in the same position. This may sound finicky but it is the only way to produce accurate work. Don't make the mistake of using a lathe center either for a hammer or a center punch.

DON'TS FOR MACHINISTS

From "Machinery"

Don't run a lathe with the belt too loose.

Don't run the point of your lathe tool into the mandrel.

Don't rap the chips out of your file on the lathe shears.

Don't set a lathe tool below the center for external work.

Don't start up a lathe without seeing that the tail stock spindle is locked.

Don't put an arbor or shaft on lathe centers without lubricant on them.

Don't leave too much stock on a piece of work to take off with the finishing cut.

Don't try a steel gauge or an expensive caliper on a shaft while it is running.

Don't put a mandrel into a newly bored hole without a lubricant of some kind on it.

Don't put a piece of work on centers unless you know that the internal centers are clean.

Don't try to straighten a shaft on lathe centers, and expect that the centers will run true afterwards.

Don't put a piece of work on lathe centers unless you know that all your centers are at the same angles.

Don't take a lathe center out of its socket without having a witness mark on it, and put it back again according to the mark.

Don't start polishing a shaft on lathe centers without having it loose enough to allow for the expansion by heat from the polishing process.

Don't run your lathe tool into the faceplate.

Don't try to knurl a piece of work without oiling it.

Don't run a lathe an instant after the center begins to squeal.

Don't forget to oil your machine every morning; it works better.

Don't forget that a fairly good center-punch may be made from a piece of round file.

Don't forget that a surface polished with oil will keep clean much longer than one polished dry.

Don't start to turn up a job on lathe centers unless you know that the centers are both in line with the ways.

Don't cross your belt laces on the side next the pulley, for that makes them cut themselves in two.

Don't try to cut threads on steel or wrought iron dry; use lard oil or a cutting compound.

Don't run a chuck or faceplate up to the shoulder suddenly; it strains the spindle and threads and makes removal difficult.

Don't screw a tool post screw any tighter than is absolutely necessary; many mechanics have a false idea as to how tight a lathe tool should be to do its work.

To drive the center out of head spindle use a rod and drive through the hole in spindle.

When putting a lathe chuck on the head spindle, always remove the center.

When the center is removed from the head spindle of the lathe, always put a piece of rag in spindle hole to prevent any dirt from collecting.



Fig. 353. A Trade School Shop.

INDUSTRIAL APPRENTICE TRAINING

In the United States

The industrial plants of the United States are very much interested in the Vocational and Trade Schools. These schools are doing remarkable work for the individual boys of their own communities and also for the industries of the entire United States as a whole.

The metal-working industry is aware that well equipped vocational schools can teach young men the fundamentals of the machinist's trade much better than they can be taught to apprentices in factories. Under the guidance of a capable and thoroughly trained instructor, the boys not only receive practical instruction in the operation of various machine tools, but are also taught the necessary shop mathematics, mechanical drawing, business English, economics, etc.

Industry looks to vocational and trades training schools to supply young men with sufficient vocational training so that they can be further trained in the factory for positions of expert workmen, specialty mechanics, foremen, superintendents, salesmen and advertising men—not for the ordinary jobs of operating production machines.

A number of large industrial plants in the United States have established training courses in their own plants to supplement and continue the fundamental training now being given in the Vocational and Trade Schools.

Allowance for Finish

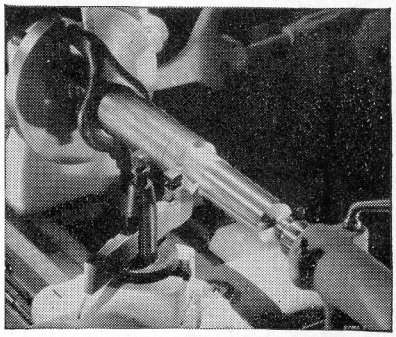


Fig. 356. Taking a Finish Turning Cut Following a Rough Turning Operation

The amount of stock which should be left when rough turning or boring a surface that is to be finish turned, reamed, ground, or otherwise finished, varies with the size of the work and the process used. Generally speaking, a minimum amount of stock should be left for the finishing cut. However, it is important that sufficient stock be left to make sure that the work will "clean up." Parts that are to be heat-treated must have ample finish to allow for warping during the heat-treating process. Allowance must also be made for warping which may result from the removal of an outer surface in which there are likely to be strains, such as castings or steel bars which have not been normalized.

Finish Turning

It has been found that an allowance of $\frac{1}{64}$" to $\frac{1}{32}$" on the diameter is usually sufficient for finish turning work of normal length up to 2" in diameter. For work over 2" in diameter, the allowance is usually between $\frac{1}{32}$" and $\frac{1}{16}$".

File Finish

When work is to be finished by filing in the lathe, the stock left for filing should be as small as possible. This not only saves time, but produces more accurate results. Very little stock need be left if the cutting tool is properly ground and set so that it will produce a smooth finish and will not scratch or tear the work. Usually about .001" to .0015" on the diameter will be ample, but if only .0005" is necessary, so much the better. See page 89.

Polishing

The amount of stock removed by polishing with emery cloth is so small that usually no allowance is made. Only a few tenths of a thousandth are required for a polishing operation, less than can be measured with ordinary micrometers. See page 89.

Grinding

The customary allowance for grinding is .010" to .012" on the diameter. Less may be required on small parts. However, if parts are to be heat-treated between the turning and grinding operations, an allowance of .015" to .020" on the diameter may be necessary to allow for warping.

Reaming

Holes that are to be finished by reaming must be drilled or bored undersize. The allowance for machine reaming in steel is usually .005" to .010", for cast iron .010" to .015". For hand reaming in steel or cast iron .0005" to .0015" is sufficient.

Lapping and Honing

Very little stock is removed by the lapping or honing processes. Usually less than .0005" on the diameter is allowed. See page 89.

MOTORS FOR OPERATING LATHES

Many different types of motors are available for operating individual motor driven lathes. In some cases any one of several different motors may be used, but usually there is one particular type of motor that will give more satisfactory service than any other. A brief study of the various types will aid in selecting the most desirable motor.

Reversing Motor Required—Several important lathe operations necessitate reversing the rotation of the lathe spindle. For this reason it is important that the motor be of the reversible type. The motor should conform exactly with the electric current that is to be used. Dual rated motors may be operated on two different voltages.

When more than one type of current is available, the motor should be selected for use with the most desirable type of current. Two or three-phase alternating current is generally considered more desirable for power service than single phase current or direct current.

The Size of Motor specified by the lathe manufacturer should always be used unless the lathe is to be operated at spindle speeds higher than standard. More power is required for high speed operation, necessitating a larger motor. The lathe manufacturer should always be consulted before equipping a lathe for high speed operation. The speed of the motor should be, as nearly as possible, the speed specified.

Polyphase Motors for 3-phase or 2-phase alternating current are instantly reversible, have a high starting torque and constant speed. These motors have no brushes or commutators to cause radio interference. They are very satisfactory for lathe operation.

Capacitor Motors for single phase alternating current are made in instant reversing, and start-stop reversing types. The instant reversing type is preferable but the start-stop reversing type is less expensive and may be used satisfactorily on certain classes of lathe work. This type of motor has a fairly high starting torque and constant speed. It has a starting switch but has no commutator to cause radio interference.

Repulsion Induction Motors for single phase alternating current have a high starting torque and fairly constant speed. They are made to order in the instant reversing type for lathe operation. Having a commutator and brushes, this motor may cause radio interference.

Split Phase Motors for single phase alternating current do not have sufficient starting torque for satisfactory use with a lathe.

Shunt Wound Motors for direct current are instantly reversible, have a good starting torque and a fairly constant speed. They are entirely satisfactory for lathe operation, although the commutator and brushes may cause radio interference.

Compound Wound Motors for direct current are instantly reversible. They have a high starting torque and a fairly constant speed. The compound wound motors are satisfactory for lathe operation but may cause radio interference due to the commutator and brush construction.

Universal Motors are designed for operating on either alternating current or direct current. This type of motor is seldom used for operating a metal working lathe.

Decimal Equivalents of Number Drills

Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent
1	.2280	14	.1820	26	.1470	39	.0995
2	.2210	15	.1800	27	.1440	40	.0980
3	.2130	16	.1770	28	.1405	41	.0960
4	.2090	17	.1730	29	.1360	42	.0935
5	.2055	18	.1695	30	.1285	43	.0890
6	.2040	19	.1660	31	.1200	44	.0860
7	.2010	20	.1610	32	.1160	45	.0820
8	.1990	21	.1590	33	.1130	46	.0810
9	.1960	22	.1570	34	.1110	47	.0785
10	.1935	23	.1540	35	.1100	48	.0760
11	.1910	24	.1520	36	.1065	49	.0730
12	.1890	25	.1495	37	.1040	50	.0700
13	.1850			38	.1015		

Decimal Equivalents of Letter Drills

Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent
A	.234	H	.266	N	.302	U	.368
B	.238	I	.272	O	.316	V	.377
C	.242	J	.277	P	.323	W	.386
D	.246	K	.281	Q	.332	X	.397
E	.250	L	.290	R	.339	Y	.404
F	.257	M	.295	S	.348	Z	.413
G	.261			T	.358		

Use of Gauge Blocks

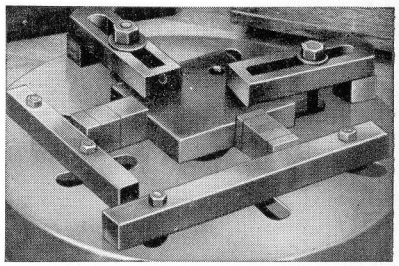


Fig. 359. Using Gauge Blocks to Position Work on Face Plate of Lathe

Precision gauge blocks are used for making extremely accurate measurements, usually for toolroom or inspection operations. Each hardened steel block has two opposite parallel faces ground and lapped to a fine finish, with the distance between them equal to the size marked on the block within a few millionths of an inch. A set of blocks includes an assortment of sizes which permits assembling various combinations to obtain any dimension within the capacity of the set, advancing in increments of one ten-thousandth of an inch. Assemblies up to 6″ in length are said to be accurate within one hundred-thousandth part of an inch at 66° F.

Gauge blocks may be used to locate jig plates and other work on the face plate of the lathe for boring accurately spaced holes to known vertical and horizontal dimensions. Accurate parallels must first be attached to the face plate in a position which will locate the hole farthest from the working edges of the work piece in line with the center of the lathe spindle. After the first hole is bored, the location of each subsequent hole is obtained by inserting gauge blocks of the required size between the work and the parallels.

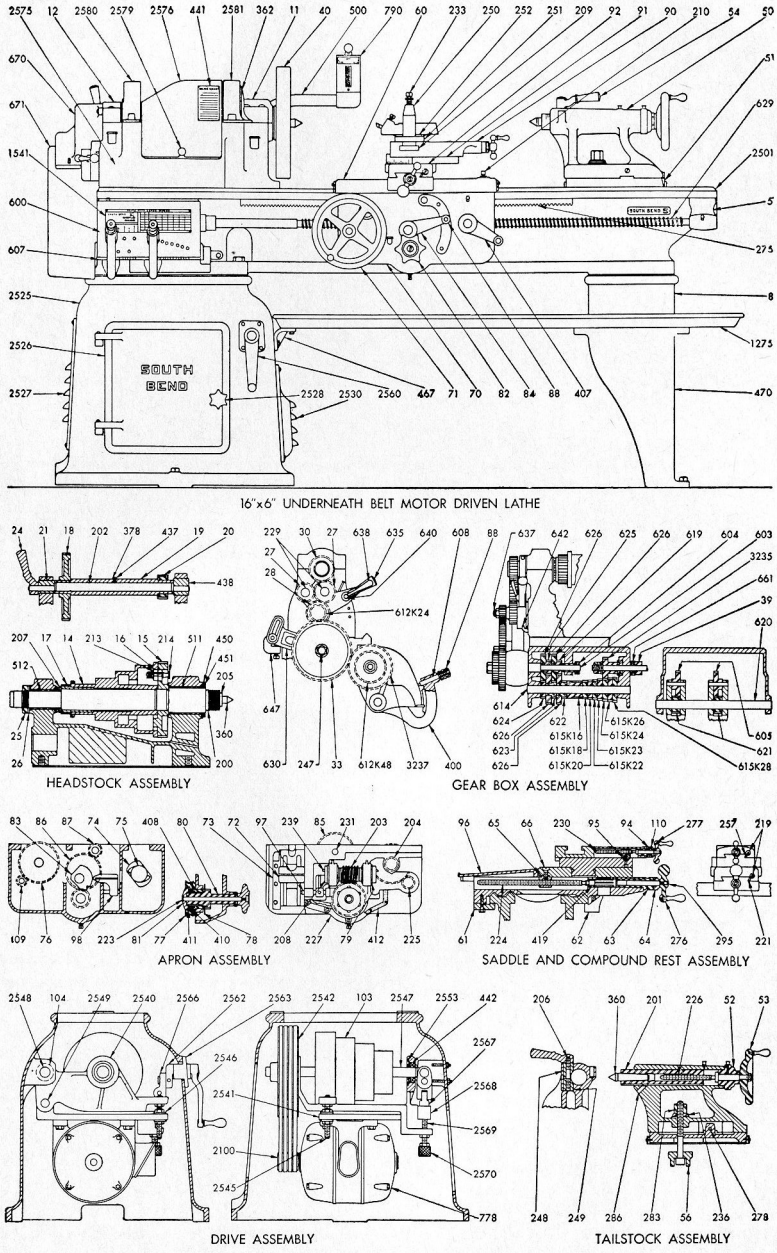


Fig. 362. Principal Parts of a South Bend Underneath Motor Drive Lathe

BASE NUMBERS AND NAMES OF LATHE PARTS

The base numbers of the parts of an Underneath Motor Drive Lathe are shown in Fig. 362. The names of the various parts are listed opposite the corresponding numbers, on this and the following page.

The base numbers identify each part in a general way, but do not designate the size or model of the lathe. A prefix or suffix is used with the base number to establish the exact part required to fit a particular size and model of lathe. However, if the serial number of the lathe is known, it is sometimes possible to identify a lathe part by the base number only.

Part No.	Name of Part
5	Rear Lead Screw Bracket
8	Tailstock Leg (Bed to Chip Pan)
11	Headstock Cap, Large
12	Headstock Cap, Small
14	Spindle Cone Pulley
15	Bull Gear
16	Bull Gear Lock
17	Cone Pulley Pinion
18	Quill Gear
19	Quill Sleeve
20	Quill Sleeve Pinion
21	Eccentric Shaft Bushing
24	Back-Gear Lever
25	Spindle Take-up Nut
26	Spindle Take-up Nut Washer
27	Twin Gears
28	Reverse Gear
30	Spindle Gear
33	Idler Gear
39	Lead Screw Thrust Collar
40	Large Face Plate
50	Tailstock Top
51	Tailstock Base
52	Tailstock Nut
53	Tailstock Handwheel
54	Tailstock Binding Lever
56	Tailstock Clamp
60	Saddle
61	Saddle Gib
62	Saddle Lock
63	Cross-Feed Bushing
64	Cross-Feed Graduated Collar
65	Cross-Feed Nut
66	Cross-Feed Nut Screw
70	Apron
71	Apron Handwheel
72	Half-Nuts
73	Half-Nut Gib
74	Half-Nut Cam
75	Half-Nut Friction Washer
76	Rack Pinion Gear
77	Worm Gear
78	Oil Distributing Washer
79	Worm Bushing
80	Clutch Sleeve
81	Apron Clutch Plate
82	Apron Clutch Knob
83	Apron Feed Change Gear Shaft
84	Apron Feed Change Lever
85	Cross-Feed Compound Gear
86	Apron Feed Change Idler Gear
87	Cross-Feed Compound Pinion
88	Knob
90	Compound Rest Top
91	Compound Rest Swivel
92	Compound Rest Base
94	Compound Rest Bushing
95	Compound Rest Nut
96	Compound Rest Chip Guard
97	Apron Feed Interlock Plunger
98	Apron Feed Interlock Arm
103	Countershaft Cone Pulley
104	Cone Cradle Shaft Collar
110	Compound Rest Graduated Collar
200	Headstock Spindle
201	Tailstock Spindle
202	Eccentric Shaft
203	Apron Feed Worm
204	Rack Pinion
205	Headstock Spindle Sleeve
206	Tailstock Binding Lever Stud
207	Spindle Thrust Bearing
208	Apron Worm Collars
209	Tool Post Block
210	Carriage Lock Screw
213	Bull Gear Lock Gib
214	Bull Gear Lock Plunger
219	Dovetail Gib Adjusting Screws
221	Compound Rest Base Gib
223	Clutch Screw
224	Cross-Feed Screw
225	Apron Handwheel Pinion
226	Tailstock Screw
227	Oil Reservoir Plate
229	Twin Gear Studs
230	Compound Rest Screw
231	Cross-Feed Compound Gear Shaft
233	Tool Post Screw
236	Tailstock Clamp Bolt Washer

Part No.	Name of Part
239	Apron Worm Key
247	Idler Gear Bolt
248	Tailstock Binding Lever Upper Plug
249	Tailstock Binding Lever Lower Plug
250	Tool Post
251	Tool Post Ring
252	Tool Post Wedge
257	Compound Rest Top Gib
275	Rack
276	Cross-Feed Ball Crank
277	Compound Rest Ball Crank
278	Tailstock Set-Over Screws
283	Tailstock Clamping Nut
286	Tailstock Spindle Key
295	Cross-Feed Ball Crank Retaining Nut
360	Center
362	Bull Gear Lock Guard
378	Quill Sleeve Oil Hole Washer
400	Tumbler
407	Half-Nut Lever
408	Worm Gear Lock Ring
409	Handwheel Pinion Thrust Spring
410	Inside Clutch Plates
411	Outside Clutch Plates
412	Oil Reservoir Gasket
419	Cross-Feed Screw Chip Collar
437	Quill Sleeve Oil Hole Plug
438	Eccentric Shaft Bushing, Large
441	Lubrication Plate
442	Countershaft Cone Cradle Bearings
450	Spindle Protecting Ring Gasket
451	Spindle Protecting Ring
467	Chip Pan Brackets
470	Tailstock Leg (Chip Pan to Floor)
500	Switch Arm
511	Headstock Spindle Bearing, Large
512	Headstock Spindle Bearing, Small
600	Gear Box
603	Main Drive Shaft
604	Main Drive Collar
605	Tumbler Idler Gear
607	Tumbler Guide Bar
608	Tumbler Plunger
612K24	Stud Gear
612K48	Stud Gear
614	Cone Gear Shaft
615K16	Gear Box Cone Gear—16 Teeth
615K18	Gear Box Cone Gear—18 Teeth
615K20	Gear Box Cone Gear—20 Teeth
615K22	Gear Box Cone Gear—22 Teeth
615K23	Gear Box Cone Gear—23 Teeth
615K24	Gear Box Cone Gear—24 Teeth
615K26	Gear Box Cone Gear—26 Teeth
615K28	Gear Box Cone Gear—28 Teeth
619	Small Compound Gear
620	Tumbler Pinion Shaft
621	Main Drive Gear
622	Small Compound Gear
623	Small Compound Gear
624	Cone Shaft Gear
625	Small Compound Gear
626	Large Compound Gear
629	Lead Screw
630	Primary Gear Bracket
635	Reverse Bracket
637	Reverse Shaft
638	Reverse Latch Lever
640	Reverse Plunger
642	Reverse Lock
647	Primary Gear Bracket Support
661	Thrust Ball Bearing
670	Upper Gear Guard
671	Lower Gear Guard
778	Motor
790	Drum Switch
1275	Chip Pan
1541	Gear Box Index Plate
2100	Motor Pulley
2501	Lathe Bed
2525	Cabinet Leg
2526	Cabinet Leg Door
2527	Left Ventilator Cover
2528	Leg Door Knob
2530	Right Ventilator Cover
2540	Countershaft Cone Cradle
2541	Motor Mounting Plate
2542	Countershaft V-Belt Pulley
2545	Motor Belt Adjusting Stud
2546	Motor Belt Adjusting Stud Washers
2547	Countershaft Cone Pulley Shaft
2548	Cone Cradle Hinge Shaft
2549	Motor Plate Hinge Shaft
2553	Countershaft Bearing Retainer Washers
2560	Belt Tension Release Crank
2562	Belt Tension Eccentric
2563	Belt Tension Eccentric Bushing
2566	Belt Tension Eccentric Clevis
2567	Belt Tension Eccentric Ball Socket
2568	Belt Tension Eccentric Ball Socket Sleeve
2569	Countershaft Cone Cradle Adjusting Screw
2570	Countershaft Cone Cradle Adjusting Screw Nut
2575	Headstock
2576	Headstock Cone Pulley Cover
2579	Cone Pulley Cover Knob
2580	Quill Gear Guard
2581	Bull Gear Guard
3235	Lead Screw Gear
3237	Screw Gear

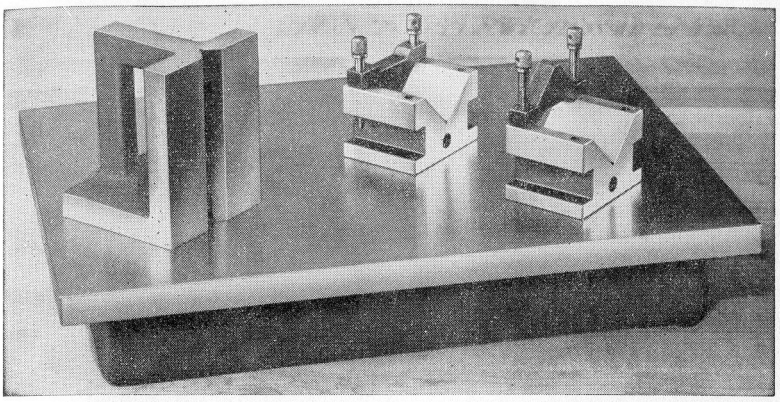


Fig. 363. Surface Plate with V-Blocks and Angle Plate for Laying Out Work

The Surface Plate

The surface plate, shown in the illustration above, is a flat cast iron plate used in the building of fine machinery to test plane surfaces. The surface plate is coated with a very thin film of pigment such as prussian blue or red lead and the surface to be tested is rubbed over it. The high spots on the surface tested will be marked with the pigment, and can be filed or scraped off. The surface is then re-tested and re-scraped until satisfactory.

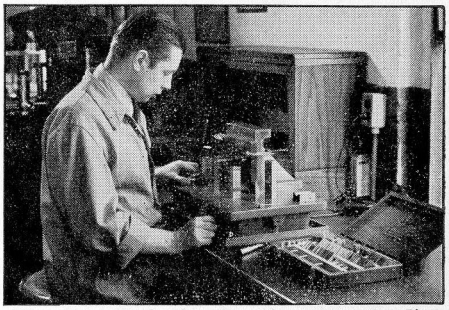


Fig. 364. Toolmaker Locating Holes in Jig Plate with Aid of Surface Plate, Surface Gauge, and Gauge Blocks

Surface plates are also used by toolmakers for locating and checking holes in jig plates, laying out work, testing, inspecting, and similar operations. With the aid of a surface gauge and a set of gauge blocks very accurate work can be done on the surface plate.

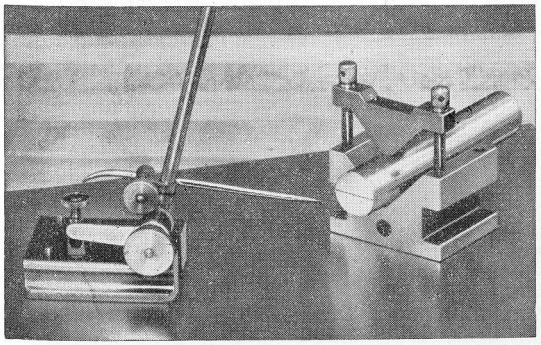
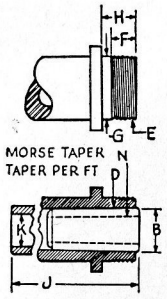
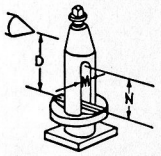
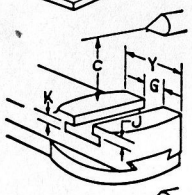
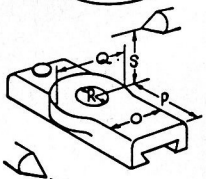


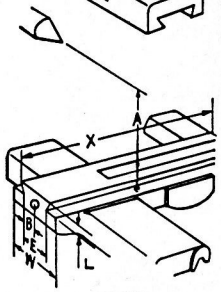
Fig. 365. Laying Out Work with Surface Plate, V-Block, and Surface Gauge

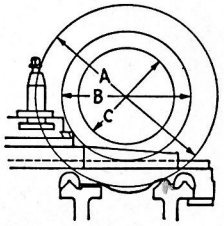












DIMENSIONS FOR TOOLING SOUTH BEND LATHES

All dimensions are in inches

SPINDLE NOSE THREAD

Size Lathe	9″ and 10K	10″ & 13″ 1″ Collet	10″-11/16″ Collet	14½″-1″ Collet	16″ & 16-24″
E	1½—8	2¼—8	1⅞—8	2⅜—6	2⅜—6
F	11/16	15/16	15/16	1⅜	1⅜
G	1.509	2.259	1.884	2.384	2.384
H	⅞	1 3/16	1 3/16	1¾	1¾

SPINDLE TAPER, SPINDLE HOLE, SPINDLE LENGTH

Size Lathe	9″	10K	10″-1″ Collet	10″-11/16″ Collet	13″-1″ Collet	14½″-1″ Collet	16″ & 16-24″
B	.938	.938	1.629	1.231	1.629	1.629	1.629
D	.602	.602	.602	.623	.602	.602	.602
K	¾	27/32	1⅜	1	1⅜	1⅜	1⅜
N	2	2	2	2	3	3	3
J	12¼	12¼	13 23/32	13 5/16	21 1/16	24 29/64	24 11/16

Note: Tailstock spindle taper is same as N above.

TOOL POST OPENING

Size Lathe	9″ and 10K	10″	13″	14½″	16″ & 16-24″
D	19/32	19/32	1 3/64	1 7/32	1 7/32
M	13/32	15/32	19/32	21/32	23/32
N	⅞	1	1¼	1¾	1¾

COMPOUND REST TOP

Size Lathe	9″ and 10K	10″	13″	14½″	16″ & 16-24″
C	1 1/32	1 1/32	1 5/8	1 27/32	2 1/16
G	⅞	1	1¼	1 7/16	1⅝
J	9/32	21/64	7/16	7/16	7/16
K	9/32	9/32	13/32	17/32	17/32
Y	2½	2⅞	3 5/8	4	4½
T	1 9/32	1 13/32	1 49/64	2 1/64	2 17/64

COMPOUND REST BASE

Size Lathe	9″ and 10K	10″	13″	14½″	16″ & 16-24″
O	2 5/8	2⅞	4	4 1/8	4 5/8
P	3 5/16	3 5/16	4½	5 1/8	5¾
Q	3½	3⅝	5¼	5 5/8	6
R	1⅜	1⅝	2¼	2½	2¾
S	2 1/16	2 51/64	3¾	4 5/16	4 9/16

60° SADDLE DOVETAIL

Size Lathe	9″ and 10K	10″	13″	14½″	16″	16-24″
A	3 19/32	3 61/64	5 7/32	5¾	6 3/16	10 15/16
B	.7655	.8328	1.1753	1.2690	1.4253	1.4253
E	1.5253	1.6032	2.1631	2.4756	2.7256	2.7256
L	23/64	7/16	9/16	9/16	9/16	9/16
W	3¼	3⅝	4 9/16	4⅞	5⅜	5⅜
X	10 1/16	11⅞	16 1/32	18 5/16	19⅞	10⅞

SWING OVER BED, SADDLE, COMPOUND REST

Size Lathe	9″	10K	10″	13″	14½″	16″	16-24″
A	9¼	10	10⅛	13⅛	14 5/8	16¼	25⅛
B	...	...	6¾	8¾	10¼	11⅛	18¾
C	5½	6¼	5⅞	7¾	8¾	9 5/8	19¼

INDEX

Subject	Page
Accuracy of a Screw Cutting Lathe	40
Acme Screw Threads	83
Alignment of Centers	48, 51
Allowance for Finish	119
Aluminum, Machining	36
American National Screw Thread	70, 71
Annealing Steel and Brass	113
Application of Lathe Tools	34
Apprentice Training	118
Apron of the Lathe	14, 24
Armature Truing and Undercutting	104, 105
Attachments for the Lathe	57, 58, 62, 81, 84, 92 to 109
Auto Service Shop Lathe Work	104
Back-Geared Headstock	12, 22
Bearings for Headstock Spindle	13
Bed of Lathe	13
Belts, Lacing, Shifting, etc.	17, 18, 19, 26, 111
Belt Tension	19
Bench Lathes	4, 10
Bits for Lathe Tool Holders	27 to 35
Boring in the Lathe	33, 42, 56, 59, 63, 91, 92
Brass, Machining	36
Bronze, Machining	36
Brown & Sharpe Worm Thread	83
Calipers, Use of	37 to 39
Capacity of the Lathe	11
Carriage of Lathe	14, 24
Carriage Stop	99
Case Hardening	113
Center Drill Holder	66
Center Gauge	60, 75
Center Holes, Locating and Drilling	43 to 46
Center Indicator	54, 88
Center Rest	92
Centers, Alignment of	48, 51
Centers, Mounting in Lathe Spindle	47
Centers, Removing from Lathe Spindle	47
Centers, Truing	60
Centering Work in Chuck	54, 55
Centering Work on Face Plate	88
Change Gear Chart	73
Change Gears, Calculating	114
Chuck Plate, Fitting to Chuck	112
Chuck Work	53
Chucks, Mounting on Lathe Spindle	54
Chucks, Practical Sizes	55
Chucks, Removing from Lathe Spindle	55
Coil Winding in the Lathe	91
Collet Chuck and Collets	57, 58
Combination Center Drill and Countersink	45
Commutator Truing and Undercutting	104, 105
Compound Gearing for Threads and Feeds	72, 114
Compound Rest of Lathe	59, 60, 77, 78
Coolants and Coolant Equipment	108, 109
Copper, Machining	36
Crankshaft Testing	106
Crankshaft Truing	106
Cross Feed of Lathe, Automatic	25, 49
Crotch Center	65
Cup Center	97
Cutter Bits	27 to 35
Cutting Screw Threads	69 to 86
Cutting Speeds for Various Metals	50, 98, 108
Decimal Equivalents	115
Dial Test Indicator	55, 88
Die Castings, Machining	36
Die for Cutting Screw Threads	86
Dimensions for Tooling Lathes	126
Direct Cone Drive for Lathe Spindle	22
Don'ts for Machinists	117
Draw-in Collet Chuck Attachment	57
Drilling in the Lathe	65
Drilling Center Holes	45, 46, 66
Drilling Cored Hole	67
Drill Chuck	56
Drill Pad	65
Drills, How to Sharpen	67
Drill Sizes	121
Eccentric Machining	106
Emery Wheel Speeds	100
Face Plate, Removing from Lathe	55
Face Plate Work	88
Facing Work on Centers	49
Filing in the Lathe	89, 119
Fine Cuts, Adjustment for	42
Fitting Chuck Plate to Chuck	112
Floating Reamer Driver	68
Follower Rest for Lathe	93
Forged Steel Lathe Tools	35, 113
Four Position Carriage Stop	99
Gauge Blocks	121
Gear Box, Operation of	25, 74
Gear Cutting in the Lathe	97
Gearing Lathe for Screw Threads	72, 114
Gears, Information on	110
Grinding Cutter Bits	29
Grinding in the Lathe	100
Grinding Wheel Speeds	100
Grinding Wheels, Truing	101
Handlever Bed Turret	95
Handlever Double Tool Rest	95
Handlever Draw-in Collet Chuck	58
Hand Rest for Wood Turning	97
Handwheel Draw-in Collet Chuck	57, 58
Hardening and Tempering	113
Headstock of Lathe	12, 22
Headstock Spindle Chuck	56
Height of Cutting edge of Cutter Bit	28, 32, 33
High Speed Steel Cutter Bits	27, to 34
History of the Lathe	3
Horizontal Motor Drive for Bench Lathe	10
Independent Chuck	54
Indicator, Center and Dial	54, 55
Industrial Apprentice Training	118
International Screw Thread Form	70
Keyways, American Standard Sizes	96
Knurling in the Lathe	87
Lacing Belts	17
Lapping in the Lathe	89
Lathe Dog, Attaching	47, 48
Lathe Tools	27
Laying Out Work	125
Left Hand Thread	80
Leveling the Lathe	15, 16
Locating Center Holes	43
Longitudinal Feed, Automatic	25, 49, 114

INDEX

Subject	Page
Machinability Ratings for Steels	98
Mandrels	90, 102
Manufacturing, Turret Lathes for	94
Measurements, How to Take Accurate	37
Magnesium Alloy, Machining	36
Measuring Screw Threads	79
Metric Graduations	99
Metric Lathes	85
Metric Measure	115
Metric Micrometer	39
Metric Screw Threads	84
Metric Thread Dial	85
Micrometer Carriage Stop	99
Micrometer Calipers	39, 42
Micrometer Collar	24, 78
Milling in the Lathe	96
Monel Metal, Machining	36
Morse Standard Tapers	64
Motor Drives for Lathes	8, 9, 10
Motors, Hints on Selecting Correct Types	120
Mounting Chuck on Lathe Spindle	54
Mounting Lathe Center in Spindle	47
Multiple Screw Threads	86
Names of Lathe Parts	21, 123
Non-ferrous Materials, Machining	36
Notes on Belts and Pulleys	17, 19, 26, 111
Notes on Gears	110
Notes on Lathe Work	26
Nut Mandrel	90
Oiling the Lathe	20
Operation of Lathe	21
Part Numbers of Lathe Parts	123
Piston Finishing	105
Pitch and Lead of Screw Thread	70
Plain Turning	43
Plastics, Machining	36
Polishing in the Lathe	89, 109
Portable Machine Shop	107
Power Carriage Feeds	25, 49
Precision Level	16
Press Fits	103
Production Attachments	95
Pulleys, Calculating Speed and Size	26, 111
Quick Change Gear Box for Threads and Feeds	74
Quick Change Gear Lathes	6, 25, 74
Reamer Sharpening	101
Reaming in the Lathe	68
Refacing Automobile Valves	105
Removing Chuck from Spindle	55
Reverse Lever on Headstock	22
Roughing Cuts, Maximum Depth	36
Rough Turning	30
Running Fits	103
School Shops	118
Screw Pitch Gauge	79
Screw Threads, calculating change gears for	114
Screw Threads, Fitting and Testing	79
Screw Thread Tables	71
Screw Thread Terms	70
Screw Thread Cutting	69
Selecting a Lathe	11
Semi-machined Chuck Plate, Fitting	112
Setting Cutter Bit for Threads	76
Setting the Lathe Tool	28, 49
Setting Over Tailstock for Taper Turning	61
Shifting Belts	19
Shop Kinks	116
Shoulders, Measuring and Machining	52
Size and Capacity of Lathe	11
Special Classes of Work	87, 125
Spindle Speeds of Lathes	23, 50
Spring Winding in the Lathe	91
Spur Center	97
Square Screw Threads	82
Square Turret Tool Block	95
Stainless Steel, Machining	36
Standard Change Gear Lathe	5, 25, 72
Standard Screw Thread Tables	71
Steady Rest	92
Step Chuck and Closer	58
Stellite Cutter Bits	35
Straightening Bent Shafts	105, 106
Surface Plate	125
Tailstock Adjustment	51
Tailstock, Handlever Type	95
Tailstock of Lathe	24
Tailstock Set-over for Tapers, Calculating	61
Tap Drill Sizes	71
Taper Attachment	62
Taper Gauges	61
Taper Turning and Boring	59 to 63
Tapered Screw Threads	82
Tapers, Standard Dimensions of	64
Tapping in the Lathe	68
Tempering and Hardening	113
Ten-in-One Tool Holder	27, 98
Test for Alignment of Lathe Centers	48, 51
Test for Taper Fit	61
Testing Bent Shafts	105
Testing Instruments for Chuck Work	54, 55
Testing the Lathe for Accuracy	16, 40
Thread Cutting	69
Thread Cutting Stop	78
Thread Cutting Tool	32, 75
Thread Dial	81, 85
Thread Tool Gauge	75
Threads, Terms Relating to	70
Tolerances for Fits	103
Tool Bits, High Speed Steel	27 to 34
Tool Holders for Lathes	27, 98
Toolmaker's Buttons	41
Toolroom Lathes	7
Transposing Gears for Metric Screw Threads	84
Tungsten Carbide Cutter Bits	35
Turning Between Centers	43
Turret for Lathe Bed	95
Turret Lathes for Manufacturing	94
Undercutting Armature Commutators	105
Underneath Motor Drive for Lathe	9
Universal Chuck	56
U. S. Standard Screw Thread	70
Valve Refacing in the Lathe	105
V-Block	125
Whitworth Screw Thread	83
Wood, Machining	36
Wood Turning on Lathe	97
Worm Thread, 29°	83

D5824-ROXM-10-58

How to Become a Machinist

1. Keep your cutting tools sharp.
2. Look at your drawing carefully before starting your job.
3. Be sure your machine is set up right before starting the work.
4. Take your measurements accurately.
5. Keep your machine well oiled, clean and neat. Personal neatness will give you personality.
6. Take an interest in your job; don't feel that you are forced to work.
7. Learn the fundamentals of mechanical drawing.
8. Keep your belts tight and free from oil.
9. Take as heavy a cut as the machine and cutting tool will stand until you are near the finished size; then finish carefully and accurately.
10. Try to understand the mechanism of the machine you are operating.
11. Hold yourself responsible for the job you are working on.
12. Keep your eyes on the man ahead of you; you may be called on to take his place some day.
13. Have a place for everything, and keep everything in its place.
14. Read one or two of the technical magazines relating to your line of work.
15. If a boy learns a trade properly he becomes a first-class mechanic, but if he has ability he need not stop at that.

    Henry Ford, George Westinghouse and others got their start because they were mechanics.
16. If you have spoiled a job, admit your carelessness to your foreman, and don't offer any excuses.
17. Before starting to work on a lathe, roll up your sleeves and remove your necktie—safety pays.

CE3450