MANUAL OF

LATHE OPERATION

and

MACHINISTS TABLES

Atlas
ATLAS PRESS CO.
KALAMAZOO, MICH.
U. S. A.

MANUAL OF

LATHE OPERATION

AND

MACHINISTS TABLES

ENGINEERING DEPARTMENT

ATLAS PRESS COMPANY

1822 North Pitcher Street
Kalamazoo, Michigan, 49001

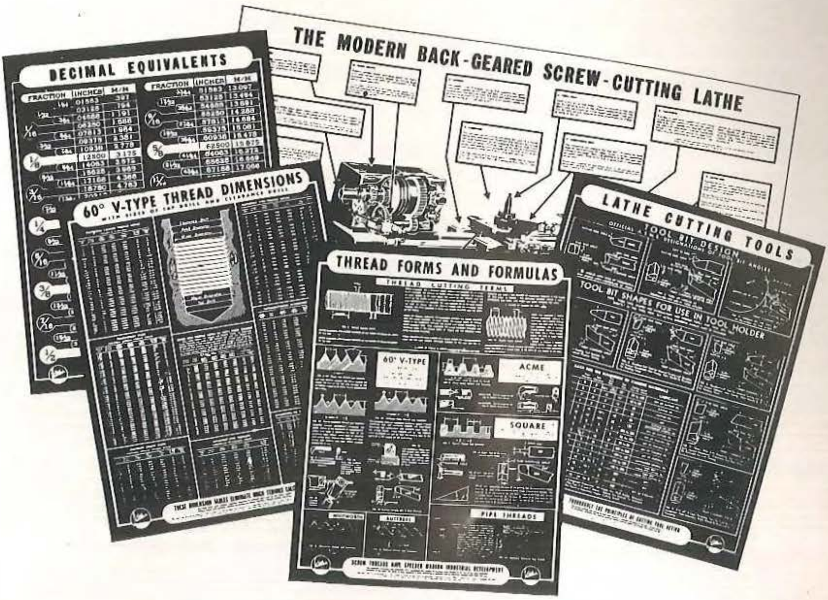
Price
$2.50

Twenty-third Edition

Copyrighted
ATLAS PRESS COMPANY
Printed in U. S. A.

Reprinted 1938, 1939, 1940, 1941, 1942, 1943, 1944, 1945,
1946, 1947, 1948, 1949, 1950, 1951, 1952, 1954,
1955, 1957, 1960, 1961, 1963, 1965, 1967.

CONTENTS



SET OF FIVE WALL CHARTS

Printed blue-print style on durable ledger paper. Four charts measure 16¼" x 21" — feature **Decimal Equivalents, Thread Forms and Formulas, Lathe Cutting Tools,** ***and*** **60° V-Type Thread Dimensions** in easy-to-read form. The fifth chart measures 35" x 23" and describes lathe parts and functions of each. **$1.00 per set.**

PREFACE

This Manual of Lathe Operation has been prepared to provide authentic, up to date, and complete operating information for owners of all types of metal cutting lathes.

Fundamental and concrete theory, as well as operating procedure, is included in order to make this book suitable for students, apprentices and vocational schools. Much of the data will prove invaluable to the machinist and the more experienced lathe operator.

It is our hope that this Manual will further the advancement of the lathe user in all walks of industry. If we have helped him, even in a small way, the research and labor involved in the preparation of this book will have been well worth while.

Atlas Press Company

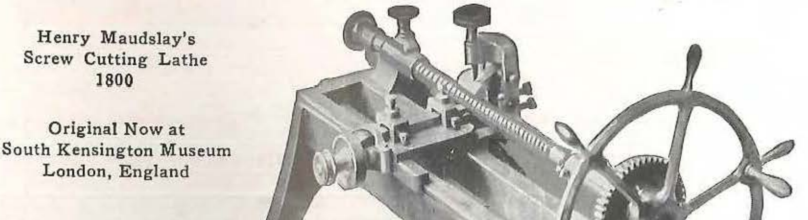
We wish to extend our sincere appreciation to the many manufacturers, engineers and machinists who have assisted in the preparation of the technical material in this material. If the reader desires further information on any of the metals or plastics mentioned, we will gladly furnish the name and address of the manufacturer.

F O R E W O R D

The history of modern machinery started in the last years of the eighteenth century when Henry Maudslay, an Englishman, built the first practical screw-cutting lathe. When compared with a modern precision lathe, this machine was slow and clumsy, but from the basic principles of Maudslay's lathe have come nearly all modern machine tools. The skill of early New England machinists in developing his theories soon put the United States in the front rank among industrial nations of the world.

Henry Maudslay's
Screw Cutting Lathe
1800

Original Now at
South Kensington Museum
London, England

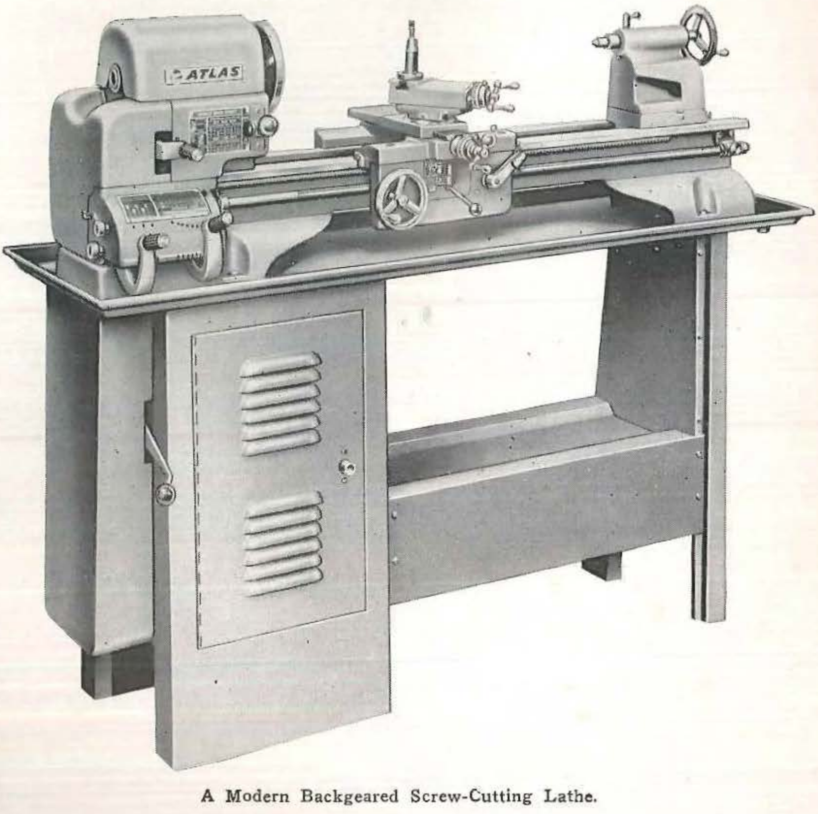


Courtesy Popular Mechanics Magazine

Today, more than 150 years after Maudslay, the screw-cutting lathe is still the heart of industrial manufacturing. It seems odd to consider the lathe so vitally important when large batteries of automatic machines are used in every modern factory. But pay a visit to the factory tool room where the machining is done which makes possible the construction of these huge automatic machines. There you will find a lathe, easily the most important tool, busy at the hands of an expert machinist turning the plans of designers and engineers into new tools and machines for modern industry.

The lathe is the "King of All Tools"—more jobs of a mechanical nature can be done on a lathe than with any other dozen tools. In the machine shop, experimental shop, or home workshop, the metal lathe is called upon for many operations. Turning, milling, grinding, drilling and boring must be performed on iron and steel; wood, plastics, alloys and soft metals must be shaped into form; threads of all sizes and shapes have to be cut; and machine parts need repairing or replacing. Manufacturers, tool and die makers, experimenters, automotive men, model builders, inventors—thousands of businesses, hobbies, and professions depend on a precision screw cutting lathe with its many attachments.

THE MODERN BACK GEARED SCREW-CUTTING LATHE



A Modern Backgeared Screw-Cutting Lathe.

Part 1

LATHE CARE AND CONSTRUCTION

PART 1

LATHE CARE AND CONSTRUCTION

SETTING UP THE LATHE

Most bench lathes are shipped completely assembled. All unpainted surfaces have been greased thoroughly and wrapped in oil paper, and the entire lathe strongly crated. Take care in removing the crate—a crow bar or hammer can slip easily and damage some part of the lathe. If the side boards are loosened at the bottom, the entire top of the crate can be lifted off.

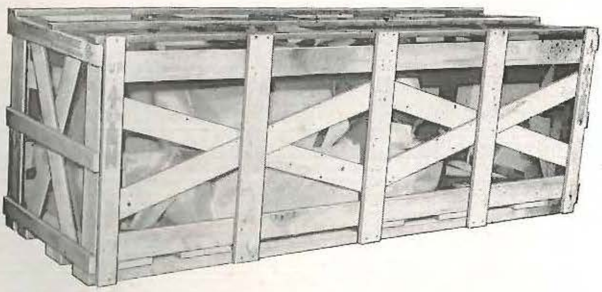


FIG. 1

Bench Lathe ready for domestic shipment. All machined surfaces are greased, and the lathe is then wrapped in paper and solidly crated.

As soon as the lathe is unpacked, oil it completely and thoroughly at all points shown on the Oiling Chart on pages 6 and 7. Choose a well lighted location that is dry and with enough room for maximum efficiency and convenience.

Floor stands and cabinets (page 2) make ideal supports for the lathe. If the lathe is to be mounted on a bench, use one that is solidly built, well braced and with a good dry lumber top at least two inches thick. The precision of any lathe, regardless of size, depends a great deal upon the rigidity of the base under the lathe bed—a flimsy, warping bench top can, in a few days, spoil a careful mounting of the lathe and in time will impair its accuracy.

A bench height of 32 to 34 inches is correct for the man of average height. Adjacent edges of the top boards should be carefully joined and planed smooth. It is suggested that the top boards either be heavily dowelled, or that four or five ⅜″ steel rods, threaded at both ends, be run edgewise through all of the top boards and pulled up tight. This latter method is preferred and calls for an accurate boring job. The top should also be planed smooth and level.

7 - THREADING
2 - METAL CUTTING
3 - CUTTING TOOLS
4 - MACHINING OF MATERIALS
5 - HOLDING WORK
6 - DRILLING

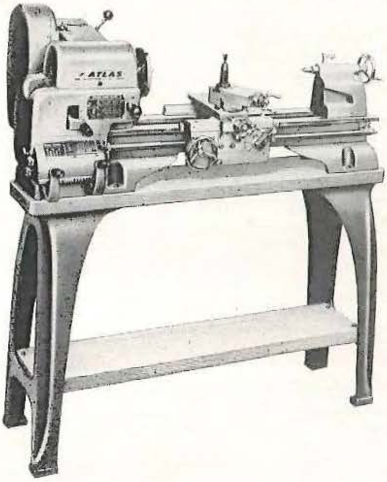


FIG. 2 (Left)
Lathe on floor stand. This type of mounting provides a rigid support and avoids imperfections of many shop benches.

FIG. 2A (Right)
Lathe on floor cabinet for tools and attachments with underneath drive.

LEVELING THE LATHE

The first step in successful lathe operation is to keep the lathe perfectly level at all times. When carelessly mounted any lathe bed will become twisted or bent, and with a slight amount of twist the centers become out of alignment and accurate work is impossible. Expert machinists agree that the better the leveling, the more accurate the lathe.

Here is the proper way to mount and level the lathe: With the lathe in position on the bench, mark and drill five ⅜" holes for machine bolts under the corresponding holes in the legs. Differences in height must then be detected with a good machinists level. Be sure the lathe bed is level ALL WAYS, including crosswise and longitudinally near both the headstock and tailstock ends (see Fig. 3). Differences in height are then taken up by the use of wide

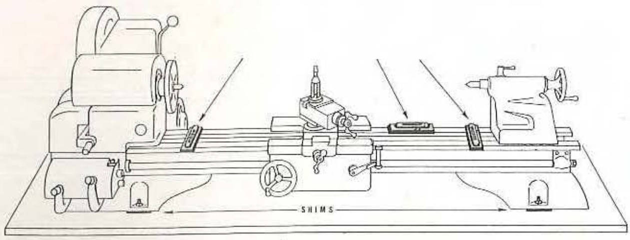
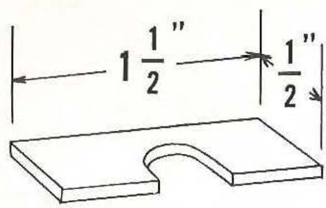


FIG. 3. Three Different Level Positions (only one level is required).

FIG. 3A
Approximate shim dimension.



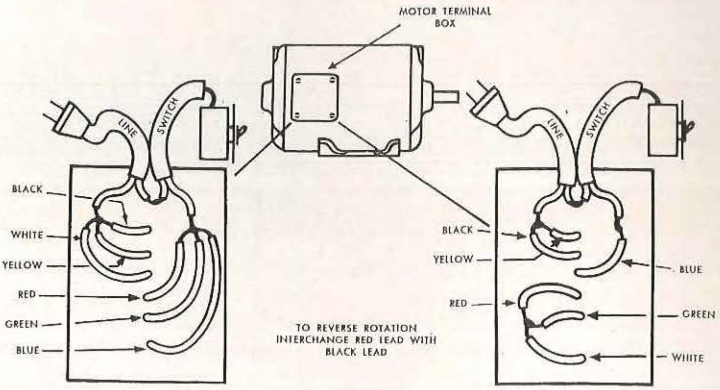
shims to insure a firm base. Shims should be thin metal strips—see Figure 3A for approximate size.

Repeat the checking operation after the legs have been bolted down tightly. It may be necessary to relax the bolts and adjust height by adding more shims. Most machinists check these leg shims regularly and whenever the lathe is expected to be in use for a long period of time. Before heavy work or whenever the lathe is moved to a new shop location, it is advisable to repeat the checking of the level position.

Do not slight the leveling of your lathe. In order to make precision cuts on long work it is absolutely necessary to have the bed perfectly aligned and horizontal. The precision built into a lathe can be made entirely useless by faulty, uneven mounting. Extra care and time spent in installation and leveling will give the lathe every chance to perform the accurate work for which it is built.

MOUNTING THE MOTOR

These lathes are designed to be run from a 1725 R.P.M. motor, either ½ or ¾ H.P., depending upon the type of work being handled. With the lathe in place, mount the motor on the motor bracket and connect the switch wires as shown in Fig. 4. Before bolting down the motor, run it for a moment to make sure that the



TO WIRE OTHER MAKE MOTORS
FOLLOW MANUFACTURERS INSTRUCTIONS

FIG. 4

direction of rotation is clockwise when facing the pulley end of the shaft. If the motor pulley does not fit readily on the motor shaft, scrape the pulley hole or dress the inside with emery cloth wrapped around a wooden dowel. Sight along the edge of the large pulley on the countershaft to obtain alignment with the motor pulley. V belt drives require only medium tension, and the motor drive belt should be adjusted with the tension lever in the back position on horizontal countershaft and lowest position on underneath drive lathes.

KEEP YOUR LATHE WELL OILED

Before using the lathe, oil it thoroughly at the points shown in the chart on pages 6 and 7. It is well to memorize the exact order of the chart. Use a good grade of machine oil — automotive oil, S.A.E. No. 20, is excellent for general lathe use.

Both top and side surfaces of the bed ways should be oiled whenever using the lathe. These ways, as well as all other unpainted surfaces, should be covered with a generous film of oil when the lathe is not in use. Keep the lathe completely covered when it is a dusty location or standing idle for a long time. Some types of gritty dust or soot are nearly as hard as emery dust and will cause wear unless lathe bearing surfaces are protected. Be sure to cover the bed ways during grinding operations.

Form the habit of oiling your lathe regularly.

FIG. 5 Modern Lathe Production Lines.

FIG. 6

CONSTRUCTION OF THE LATHE HEADSTOCK

The precision of a lathe depends to a great extent upon the care taken in the manufacture of the headstock. The headstock shown on page 8 is heavy, close-grained grey iron, ribbed and reinforced for absolute rigidity and solidly anchored to the bed. The front of the headstock casting extends up to the bearing height, providing a heavy, permanent truss between the right and left bearings and insuring perfect alignment even under the heaviest loads.

IMPORTANT — LUBRICATE LATHE BEFORE OPERATING

LUBRICATION CHART

12-INCH METALWORKING LATHES

CODE

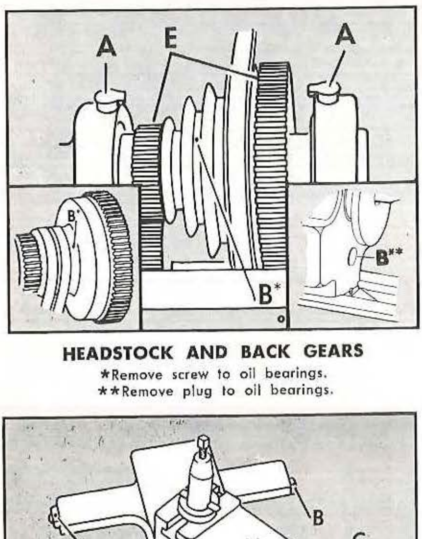
A – OIL DAILY with S.A.E. No. 20 oil

B – OIL WEEKLY with S.A.E. No. 20 oil

C – OIL MONTHLY with S.A.E. No. 20 oil

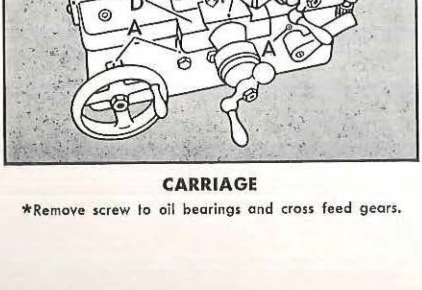
D – KEEP CLEAN and well oiled at all times

E – LUBRICATE gear teeth with Keystone No. 122 gear lubricant, or equivalent, to obtain smoother, more quiet operation. Remove oil and dirt before applying grease.



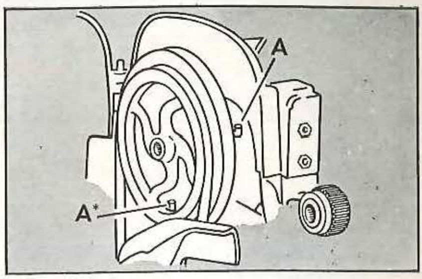
HEADSTOCK AND BACK GEARS

*Remove screw to oil bearings.
**Remove plug to oil bearings.



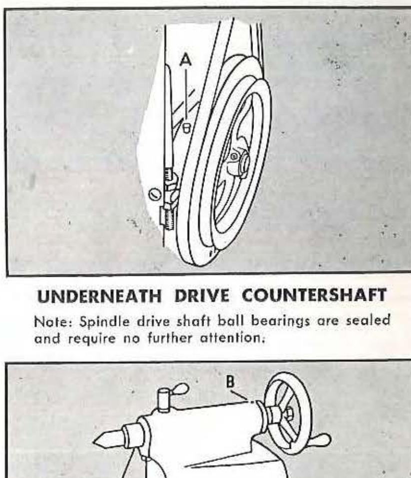
CARRIAGE

*Remove screw to oil bearings and cross feed gears.



HORIZONTAL COUNTERSHAFT

*Lubricate rocker-shaft pin at this point.



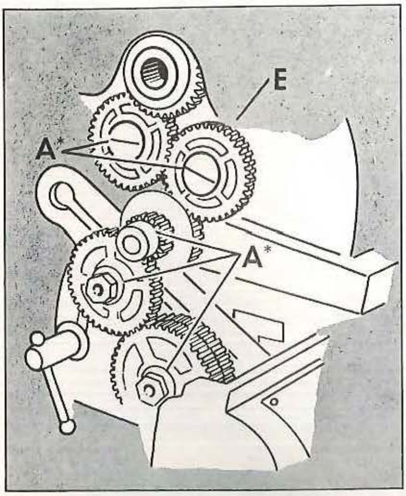
UNDERNEATH DRIVE COUNTERSHAFT

Note: Spindle drive shaft ball bearings are sealed and require no further attention.

TAILSTOCK — LEADSCREW — LEADSCREW BEARING — RACK

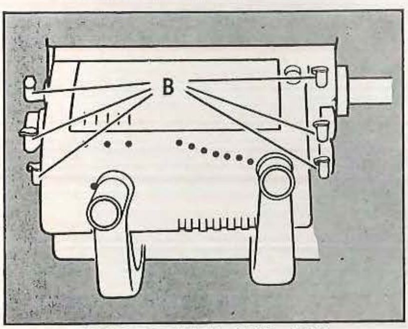
*About once a month clean with kerosene and a brush, then cover with oil.

QUICK-CHANGE LATHE



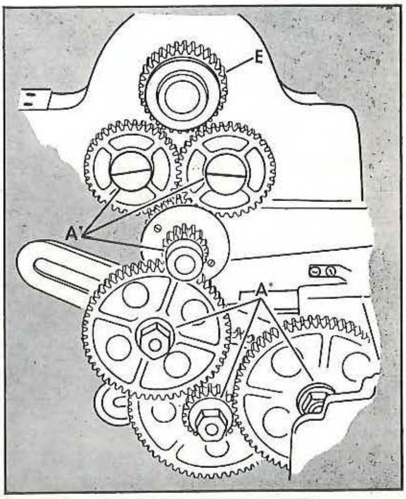
QUICK-CHANGE GEAR TRAIN

*Lubricate gear bearings at these points.



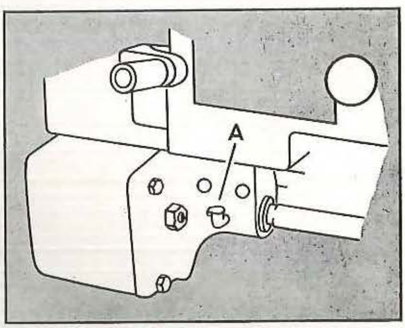
QUICK-CHANGE GEAR BOX

STANDARD CHANGE-GEAR LATHE



STANDARD GEAR TRAIN

*Lubricate gear bearings at these points.



STANDARD REVERSE GEAR BOX

KEEP YOUR LATHE CLEAN

Oil and dirt form an abrasive compound which can easily damage carefully fitted bearing surfaces. Wipe the bed and all polished parts with a clean oily rag at frequent intervals. Use a brush to clean spindle threads, gear teeth, lead screw threads, etc.

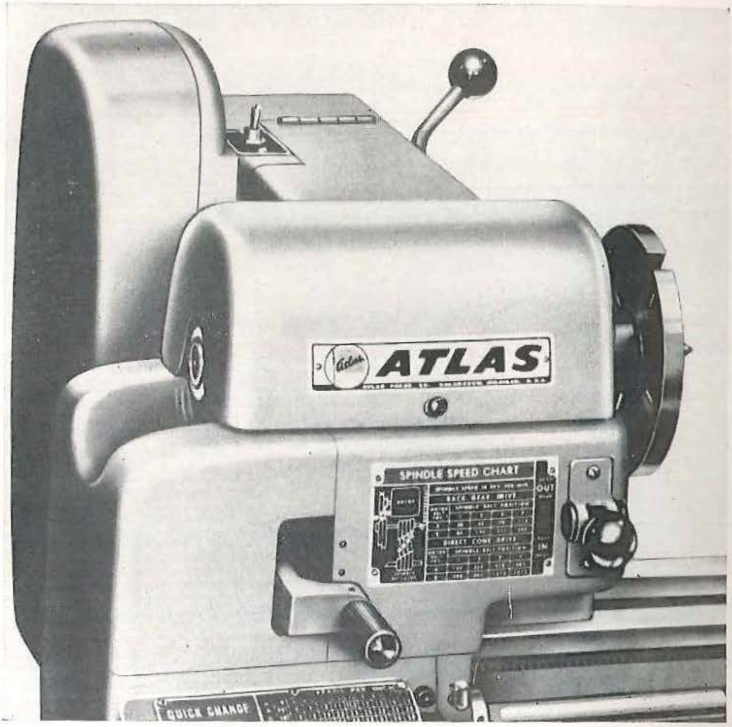


FIG. 7 Lathe Headstock

HEADSTOCK SPINDLE

The headstock spindle is special alloy steel — accurately ground and polished to extremely close tolerances to provide a perfect surface for the bearing. The spindle diameter is 1½"—the nose has 8-pitch National Form threads. A 25/32-inch hole is bored through its entire length, allowing full-sized ¾-inch stock to be fed through the spindle (see Fig. 212). The spindle nose is reamed for a No. 3 Morse Taper, and a reducing sleeve is furnished to permit the use of a standard No. 2 Morse Taper center.

TIMKEN SPINDLE BEARINGS

Lathes are equipped with Timken Tapered Roller Bearings and are recommended whenever the lathe spindle speed must be exceptionally high for long intervals. These anti-friction bearings are ideal for continuous production jobs, wood turning and metal spinning as well as the usual work at normal speeds. A Timken-

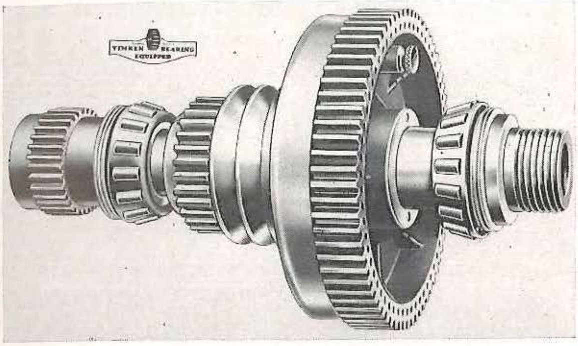


FIG. 8
Lathe Headstock Spindle is equipped with Timken Tapered Roller Bearings. The tapered design and positively aligned rolls mean that both radial and thrust loads are carried with minimum of friction.

equipped lathe also makes an excellent "combination machine" for the shop handling quantities of both wood and metal work. Timken Bearings are pre-loaded to insure a tight bearing even under the most severe use—the tapered design and positively aligned rolls carry both radial and thrust loads with minimum of friction.

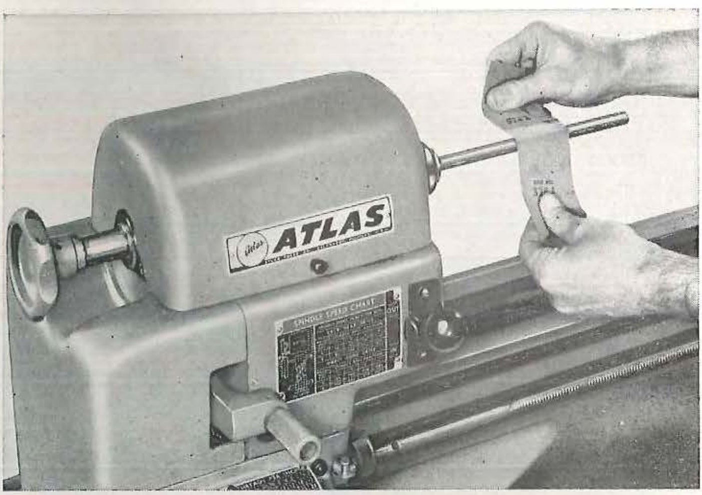


FIG. 9
Timken bearings permit high speeds required for polishing—machining small diameters, plastics, aluminum, etc.

ADJUSTMENT OF TIMKEN BEARINGS

Adjustment of the Timken Bearing is not often necessary, but if the spindle spins too freely or play is noticeable when the spindle is pushed back and forth, the following simple procedure will adjust the headstock bearings:

Run the lathe between thirty minutes and an hour to warm up the spindle (a temperature rise of 50° Fahr. increases the spindle about .002 inch between bearings). Then loosen the set screw A (in Fig. 11) on the thrust nut, B, at the extreme left end of the spindle, C, and turn it up to a point where no play can be detected in the spindle. Advance this thrust nut 1/16 turn past that point (equal to two teeth on the spindle gear) in order to provide the correct pre-load. Tighten the set screw.

CARE OF TIMKEN BEARINGS

Lathes equipped with Timken Bearings can be set to work immediately. Oil the bearings every time the lathe is in use with S.A.E. No. 10 motor oil or a good grade of machine oil.

OPERATION OF THE BACK GEARS

The back gears reduce the lathe spindle speed, providing power for heavy cuts and correct surface speeds for large-diameter work. The back gear ratio is approximately 6 to 1. The back gears are conveniently located, easily used and take up very little space. Aluminum guards provide a safety covering. The mechanism for changing from direct drive to back geared drive is quick and simple in operation. Adequate bearings and good gears are both vitally important in lathe consruction.

The lathe back gearing mechanism is pictured in Figure 11, and explains the details of operation. The "bull gear," D is keyed solidly to the spindle. The small gear, E, and the spindle pulley, F, are fastened together rigidly — they have wide, perfectly fitting bronze bearings for rotation on the spindle. This small gear and pulley assembly is free to rotate unless the pin, G, is pushed in, locking the pulley to the bull gear and spindle. In this locked position with the back gears disengaged, the spindle is driven directly from the countershaft.

FIG. 10

Top view of lathe headstock showing the back gears.

When the pin, G, is pulled out of the bull gear and the back gears are engaged by lowering back gear lever, H, to the "in" position the belt from the countershaft drives the small gear and pulley assembly, E, and F, the small gear meshes with and drives the large back gear, J, and the small back gear, K meshes with the bull gear, D turning the spindle.

The back gear spindle should be oiled as shown in the Oiling Chart on pages 6 and 7. A small amount of Keystone No. 122 gear lubricate on the gear teeth will sometimes give more quiet operation. Back gear drive is usually more noisy than direct drive, and the use of the back gears with the motor pulley belt in the high speed position is not recommended or necessary for general use. These high speeds are shown in Figure 55, page 45.

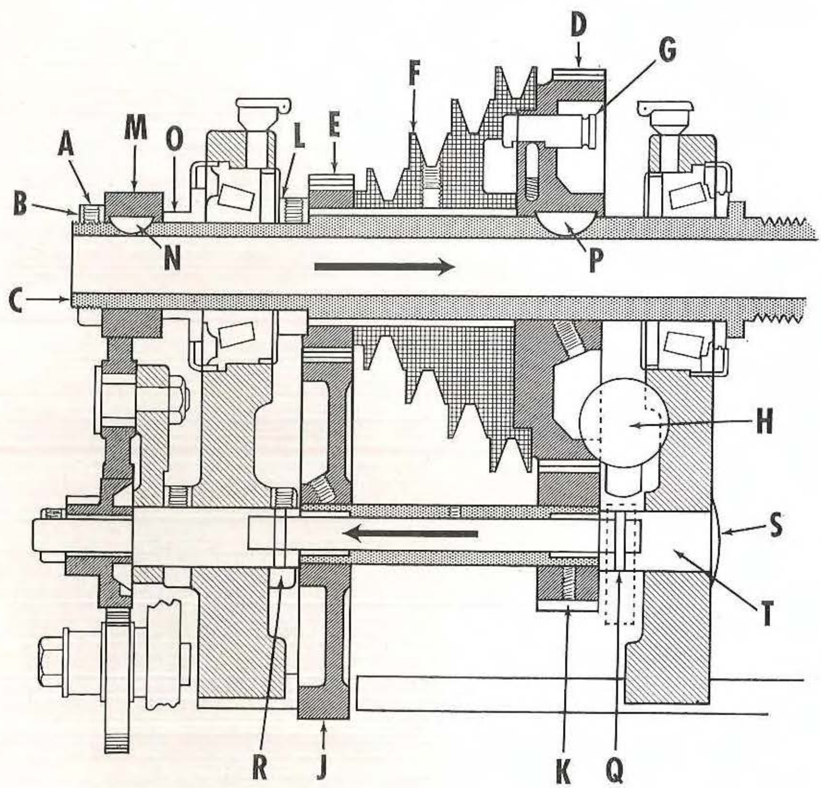


FIG. 11

Cross Section (side view) of the lathe headstock and back gearing mechanism. Note Timken bearings and adjusting nut, spindle idler pulley and bronze sleeve bearing. The operation of back gears is fully explained on pages 10 and 11.

INDEXING MECHANISM

The face of the front spindle gear has 60 evenly spaced indexing holes for such dividing operations as fluting, reeding, serrating, sprocket- and spoke-spacing.

To divide the circumference of a piece of work into a given number of equal divisions, mount the work and engage one of the indexing holes by pressing lock pin through headstock. Perform operation, release pin and, after consulting indexing table below, rotate gear the proper number of indexing holes. Engage hole and repeat operations until circle has been completed. NOTE: When using lathe dog be sure that tail of dog fits tightly in slot of face plate. In layout work it is advisable to use a pencil to mark all required divisions *before* beginning the actual operation.

INDEXING TABLE

Divisons Desired	No. of Spaces	Degrees of Arc	Divisons Desired	No. of Spaces	Degrees of Arc
1	60	360	10	6	36
2	30	180	12	5	30
3	20	120	15	4	24
4	15	90	20	3	18
5	12	72	30	2	12
6	10	60	60	1	6

THE LATHE COUNTERSHAFT

The support bracket for the horizontal countershaft is mounted on the lathe bench. This type is "quick-change" with the belt tension lever within easy reach for speed changes. Sixteen speeds are available, ranging from 28 to 2072 RPM.

This modern countershaft design does away with the irritating disadvantages of a cumbersome, space taking flat belt drive with its limited speed range and difficulties of adjustment. It provides a smooth, even flow of power to the spindle at the exact speed most efficient for the work being done.

The countershaft spindle revolves on bronze bearings, amply lubricated through the hollow housing. These fine bearings transmit maximum power to the spindle and give years of trouble-free performance.

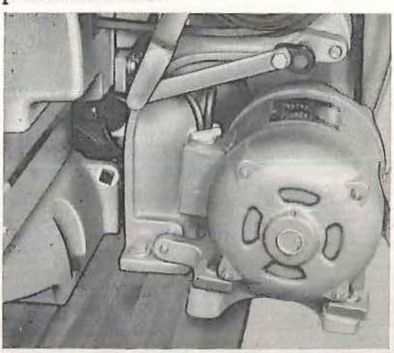


FIG. 12

Support bracket for horizontal countershaft is mounted on lathe bench or stand.

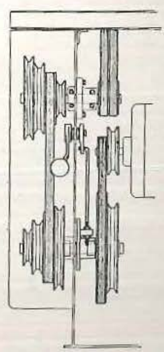


FIG. 12A

Lathes with underneath-drive have two V-belts that turn lathe spindle. Upper countershaft turns on ball bearings.

REMOVING HEADSTOCK SPINDLE

(Necessary for replacing spindle belt or bearings)
(Refer to Figure 11)

1. Disengage back gears.

2. Loosen set screw in collar B and remove collar by turning anti-clockwise. Beneath set screw is a small thread protector. Do not lose it when collar is removed.

3. Loosen set screws in collar L, and large spindle gear D.

4. Slide spindle gear M from spindle. If gear sticks, tap lightly with a piece of wood and a hammer.

5. Remove key N from spindle with a pair of pliers. Slide flanged collar O off spindle.

6. Wedge two pieces of wood between head and large spindle gear D on both sides of spindle.

7. Place a piece of wood against left end of spindle and drive in direction indicated by arrow until key P emerges from large spindle gear D.

8. Remove the wood wedges and remove key P with pair of pliers.

9. Drive spindle entirely out of headstock in direction of arrow. Be Sure To Catch The Spindle As It Is Released From The Headstock. It is not necessary to remove spindle entirely if replacing belt only on horizontal countershaft lathes.

REMOVING THE BACK GEARS

(Necessary for replacing spindle belt on underneath drive lathes)

1. Remove headstock spindle.

2. Lower quadrant assembly.

3. Loosen and remove both screws in back gear lever clamps (see Figure 13).

4. Drive out groove pin Q that holds back gear shaft to eccentric sleeve.

5. Loosen set screw in collar R and slide complete tumbler assembly and back gear shaft out of headstock in direction indicated by arrow.

6. Remove back gears and collar from headstock.

REASSEMBLING THE BACK GEARS

1. Lightly file all burrs from the back-gear shaft and eccentric.

2. Slide back-gear shaft, with tumbler assembly, through left side

of headstock, replacing collar R and back gears — be sure belt is in position.

3. Remove cap S from headstock.

4. Slide back gear shaft in right hand eccentric sleeve T — turning sleeve with a screw driver to line up groove pin hole in sleeve and shaft.

5. Replace groove pin Q and slide collar R against headstock and tighten set screw.

6. Replace back gear lever clamps — with screw driver slot in original position of about 30° from the vertical and back gear lever in the out position, tighten lever clamps.

7. After reassembling the headstock spindle, run the lathe in back gear drive to check the mesh of the back gears. If lathe runs noisy, or if there's too much play between the gears, stop the motor and shift right eccentric sleeve — see Back Gear Adjustment.

REASSEMBLING THE HEADSTOCK SPINDLE

1. Lightly file all burrs caused by set screws from spindle and keys N and P.

2. Clean all parts thoroughly, including Timken Bearings.

3. To reassemble, reverse the procedure in "Removing Headstock Spindle", page 13. Be sure belt is in position. When driving spindle into headstock, use the palm of your hand as much as possible to avoid damaging the precision surfaces of the spindle nose. Use Figure 11 as a guide, making sure all parts are installed in their proper positions. Read the following instructions for spindle and back gear adjustments.

SPINDLE ADJUSTMENTS

1. Slide collar L against shoulder of spindle and lock in place. Move pulley-gear assembly against collar and slide large spindle gear D against pulley and tighten set screw in gear. There should be a slight amount of clearance between the pulley and the large spindle gear to permit pulley to turn freely.

2. Tighten collar B until all lateral (end) and radial (side) play has been removed from spindle. Check by tapping spindle back and forth with the hand. Do not tighten spindle too tightly — spindle should rotate freely.

3. Preload spindle bearings by continuing to tighten the threaded collar B approximately 1/16″ turn (equivalent to two spindle

gear teeth). There should be a slight drag felt as the spindle is rotated by hand. This is important because as spindle and bearings warm up, the spindle expands laterally approximately .002″. Tighten set screw in collar B.

BACK GEAR ADJUSTMENT

1. Shift back gear lever upward to the Out position.
2. Loosen back-gear lever clamp screws — see Figure 13.
3. Remove cap S in right end of headstock, and with a screw driver turn the eccentric sleeve slightly. Tighten clamp screws, and check gear mesh by placing lever in back-gear position.
4. Repeat adjustment if necessary until gears are in proper mesh.

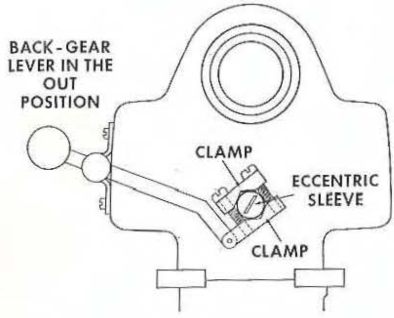


FIG. 13
When replacing the back-gear shaft, make sure shaft is positioned as shown.

ADJUSTING BELTS

The driving belt on the horizontal countershaft lathe is adjusted easily by means of the square head set screw in the countershaft arm. Make the adjustment with the belt tension lever in the back position so that the center of the belt can be pushed in about one-half inch with a moderate amount of pressure. V-belts do not have to be tight in order to drive normal loads, and belt life will be lengthened by running them fairly loose.

The motor drive belt is adjusted by moving the motor bracket up or down. This adjustment should also be made with the belt tension lever in the back position — a moderate amount of pressure should depress the center of the belt about one-half inch.

The spindle belts on the underneath drive lathe are adjusted by loosening the bracket mounting bolts (see Fig. 12A) and raising or lowering the drive shaft bracket. Adjust the tension on the belt between the four step pulleys on the spindle drive shaft and countershaft spindle by turning the tension adjusting nuts. Make this adjustment with the belt tension lever in the lowest position.

The motor drive belt is adjusted by moving the motor up or down on the mounting bracket. Adjust the belt with the tension lever in the lowest position.

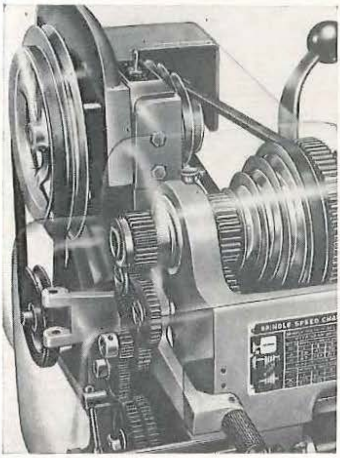


FIG. 14

THE V-BELT DRIVE

V-belt drives, due to their many advantages, are used in practically all modern machinery installations. This type of power transmission is ideal for metal lathes because it assures smoother operation, less slippage and maximum power.

V-belts of the type shown at the left have been scientifically designed to give long, efficient service, and if properly used and cared for will serve for hundreds of hours of operation.

THE LATHE BED

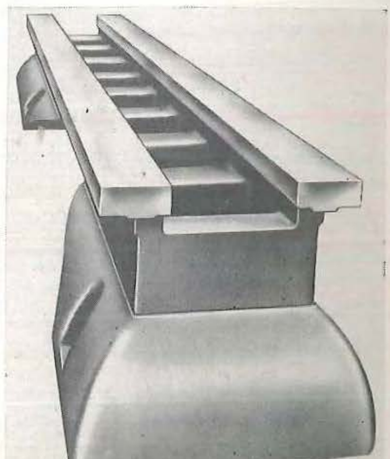


FIG. 15
The finished lathe bed—seasoned, milled, precision-ground and ready for the assembly line.

The accuracy of the lathe bed is most important from the standpoint of precision and good lathe work. The use of specially designed milling and grinding machinery and extreme care in manufacture and inspection, has succeeded in producing lathe beds today with a degree of precision previously unknown in popular-priced lathes.

The accuracy of the bed, regardless of design, is almost entirely dependent upon the finish it receives in the process of manufacture. The milling or planing operations used to reduce lathe beds to approximately final shape, do not give accuracy of more than two or three thousandths of an inch. The bed-way surfaces must then be either hand scraped or machine ground.

Modern industry has proved conclusively that surfaces can be precision-finished by grinding to unbelievably close limits—a production accuracy undreamed of ten years ago. Old fashioned, expensive methods of hand scraping have nearly disappeared—better and more adequate equipment is now being produced by machine grinding at a more moderate cost than ever before.

A precision grinding machine (Fig. 17) made especially to produce the accurate finish on the ways of the lathe shown in Fig. 15 requires a huge expenditure of money, but a precision lathe demands a precision bed—there can be no compromise.

Lathe beds are made from selected close-grained semi-steel iron castings. The entire bed, comprising the cross ribs, ways and base, is made in one piece. The heavy box-type cross ribs, spaced every four inches, rigidly brace the bed ways against heavy turning forces. The heavy ways on top and the inner bead at the bottom resist longitudinal stress. The heavy streamlined legs with cross-

TWO STEPS IN THE MACHINING OF A LATHE BED

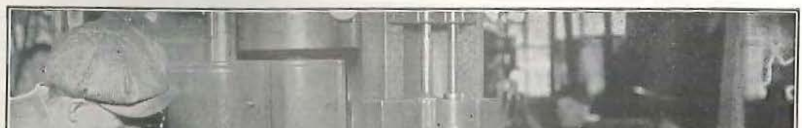


FIG. 16
A huge Kearney-Trecker Milling Machine designed and built especially for milling Lathe Beds.

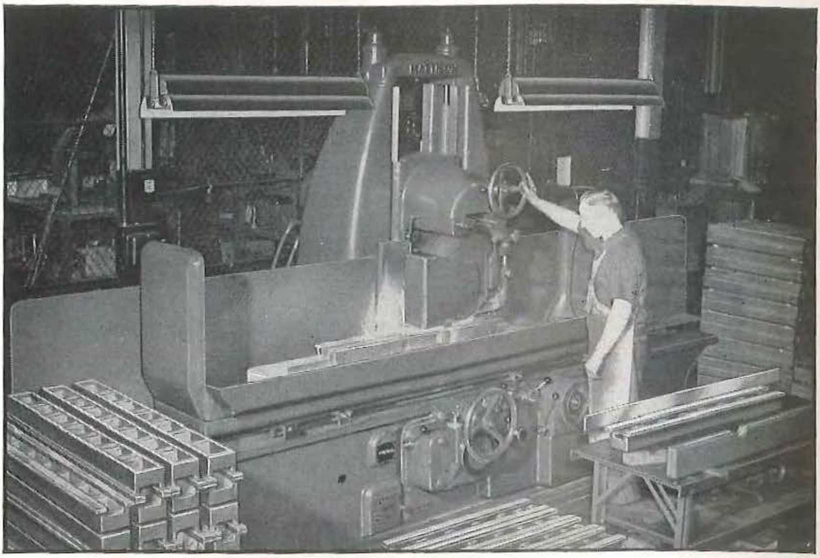


FIG. 17

Special-built modern grinding equipment of the type shown above gives the final precision finish to the lathe bed ways.

braces have a total bearing surface of 48 inches on the bed. This unusually large bearing surface provides a sturdy base for the entire lathe and keeps vibration at a minimum.

After the bed is cast, it is first rough-milled and allowed to season, or age, for a number of months. This permits internal strains in the metal to become normalized, so that warping and twisting will not occur in the finished bed. After seasoning, a finish milling cut is taken, and the ways are then finish-ground on especially designed surface grinders. The completed bed is checked thoroughly, inspected innumerable times during the assembly of the lathe, and carefully checked again by the final inspectors.

BE CONSIDERATE OF THE LATHE BED

With normal use no appreciable bed wear will occur even over a period of years, but any finely finished metal surface can be damaged by abuse, and your lathe bed is no exception.

Tools or other objects should not be dropped on the ways.

Do not use the lathe bed as an anvil.

Do not drop chucks on the bed when removing them from the spindle.

Do not allow chips to accumulate on the bed. When filing or grinding on the lathe, remove the fine dust and oil the ways liberally as soon as the operation is finished. Better still, keep the lathe bed covered during such operations.

Keep the bed well oiled when not in use—when ready to use the lathe, wipe the ways and cover them with clean oil.

FIG 18

CHECK LEVEL POSITION OF LATHE AT REGULAR INTERVALS

CARRIAGE AND COMPOUND REST

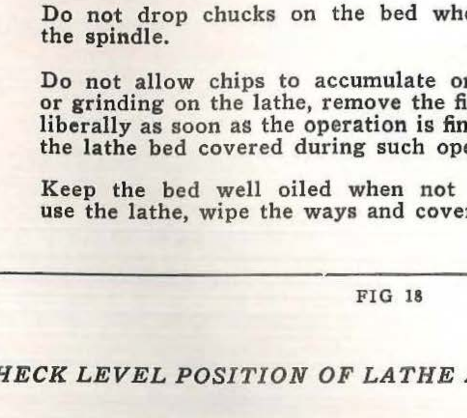


FIG. 19
Carriage and Compound Rest Assembly.

While the accuracy of cuts parallel to the bed depends upon the accuracy of the lathe bed, the accuracy of cross and compound feeds depends upon the accuracy of the carriage and compound ways. Consequently, a great amount of care is taken in the machining of these ways.

The carriage is a heavy, well proportioned grey iron casting, with six wide bearing surfaces, each 11¼ inches long, in contact with the bed. This large amount of bearing surface minimizes wear and results in more permanent accuracy. The front of the carriage, called the "apron," contains the power feed mechanisms.

Pulling the knob control on the carriage apron engages the power cross feed. The lever at the right engages longitudinal feed. Both may be reversed by shifting the feed reverse lever located on the left side of the lathe headstock. The compound can be turned in a complete circle, and it is graduated in degrees from 0 to 180, so that any angle can be cut with the compound rest.

ADJUSTMENTS OF THE CARRIAGE

Four gib screws are located on the back of the carriage for adjusting horizontal play between the carriage and the bed—these screws should be tightened just enough to give a firm sliding fit between carriage and bed. Bearing plates on the carriage, which bear on the under side of both the front and the back bed ways, anchor the carriage firmly to the bed in a vertical direction. These bearing plates have shims for adjustment of possible wear.

The large carriage hand wheel on the front of the apron operates a set of gears, the last of which meshes with the rack on the bed. These gears can be adjusted for play by loosening the screws on the front of the apron, moving the gear case toward the rack, and tightening the screws.

REMOVING THE CARRIAGE FOR CLEANING AND ADJUSTING

In order to clean or make adjustments on the inside of the apron, it is preferable to take the carriage completely off the bed. First remove the tailstock, then unbolt the bearing on the right end of the lead screw and remove the lead screw (half-nut lever must be up). With the lead screw out, it is a simple matter to loosen the gibs on the back of the carriage, and slide the carriage off the bed.

When reassembling, turn the lead screw until the keyway slips into the key of driver inside housing.

ADJUSTING CROSS FEED AND COMPOUND FEED GIBS

The gibs on the cross feed slide and the compound feed slide should be adjusted at regular intervals. The cross slide gibs should always fit snugly, because the cross slide is in almost continual use. The compound slide gibs should be kept tight unless using the compound feed.

> ***For best results, do not take heavy cuts or use the cut-off tool with the compound rest overhanging the compound rest slide.***

Be sure to loosen the two square-head set screws whenever changing the position of the compound rest.

The ball and crank handles on the cross feed screw and compound feed screw can be adjusted for play as follows. Tighten the knurled collar against the dial while holding the dial to keep it from turning. Now turn the dial and collar to remove end play in the feed screw assembly and then hold the crank and securely tighten the nut on the hub. An extremely tight fit is likely to result in a jerky feed — the turning force keeps these slides firm against the screw, and play in the handles does not affect the accuracy of the work. A nice working, snug fit is ideal.

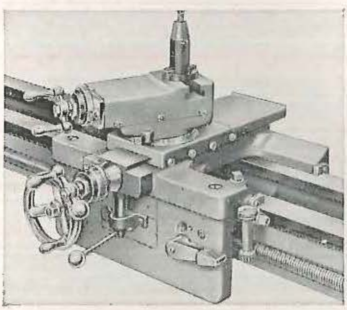


FIG. 21
Cross Slide and Compound rest.

POWER FEED MECHANISM FOR STANDARD CHANGE AND QUICK CHANGE GEAR LATHES

The automatic longitudinal power feed consists of the tumbler assembly, gear train, lead screw and half-nut mechanism. For detailed operation of the feed gears, together with the set-ups for cutting various threads, refer to the Threading Section No. 7.

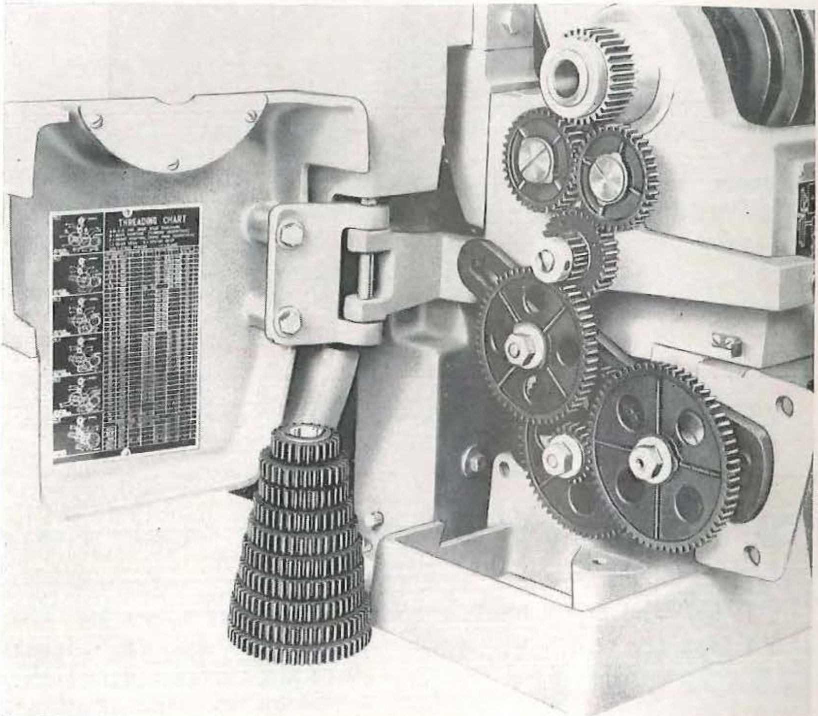


FIG. 22
Arrangement of the reverse gears, gear train and change gears for standard change lathe.

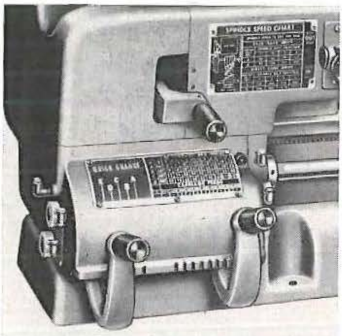


FIG. 23
Quick change mechanism speeds up thread and feed selections.

The tumbler lever with the small knob, located at the headstock end of the lathe, is the feed reverse lever. This lever is used to reverse or stop the rotation of the lead screw. Three holes are drilled in the headstock providing three positions for the lever. The center hole is neutral and the upper and lower holes are either forward or reverse positions, depending upon the gear set-up. This lever should not be removed while lathe is operating at high speeds — it may strip the gears or result in serious damage to the lathe.

LEAD SCREW

The lead screw is accurately cut with a pitch of ⅛ inch, (eight threads per inch). Its accuracy is maintained by keeping it clean and free from chips. Once a month or oftener clean the threads with a stiff bristle brush and kerosene, and oil freely along its entire length.

The lead screw bearing on the tail end of the lathe serves as a "safety valve" protecting the lead screw. One of the most common accidents on the lathe is letting the power feed drive the carriage into the headstock or tailstock. Serious and expensive results from such an accident are prevented by the light construction of this bearing. The lead screw simply forces itself out and breaks the bearing casting. In this way the light bearing prevents what would otherwise be an expensive break-down.

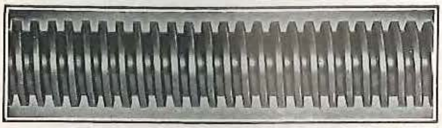


FIG. 24

Closeup of a lead screw (8 Acme threads per inch). A high degree of accuracy is essential for precision thread cutting.

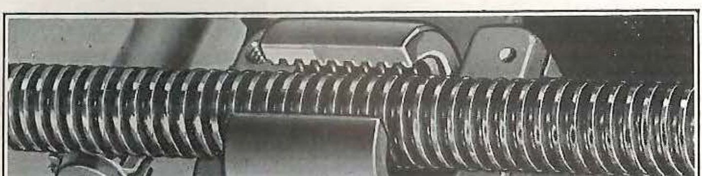


FIG. 25

A view of the half-nuts and their closing mechanism. Positive closing action, combined with the use of two half-nuts, insures smooth and accurate threads.

Figure 25 shows the construction of the half-nuts and their closing mechanism. Two half-nuts, closing on both sides of the lead screw, prevent strain on the lead screw and insure a smooth feed. In order to minimize wear on the lead screw, the half-nuts are made of a metal softer than steel. The carriage should be removed at regular intervals and the half-nuts, closing mechanism and rack cleaned thoroughly and greased. Dirt or chips will damage the half-nuts and the lead screw. Oil regularly.

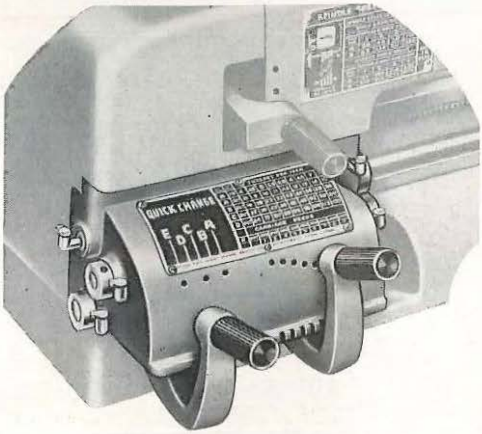


FIG. 26

QUICK CHANGE GEAR BOX

Quick change speeds up threading and every lathe operation requiring power feed by providing instant finger-tip selection of fifty-four threads or feeds — forty-five are obtained by merely shifting two levers on the gear box, and an additional nine by changing the position of a sliding gear.

Work range includes practically every feed from .0042″ to .250″ per revolution of spindle, and all threads right or left hand from 4 to 240 in the following standards: National Coarse (USS), National Fine (SAE), Acme, Square and Whitworth. Does not in any way limit the odd threads and feed range capacity of the Atlas lathe. By varying the gear train with Atlas change gears, hundreds of additional threads and feeds are available for such jobs as metric threading, coil and wire winding and special tooling.

Gear train arrangements and instructions for cutting metric threads and frequently used odd threads are contained in Section 7.

THE TAILSTOCK

The tailstock of a lathe must line up perfectly with the headstock at any point on the lathe bed. The precision of the ground ways and the extra care taken in the fitting of the tailstock assure accurate alignment at any position.

The ram is made of special steel, finish ground, and has an accurately reamed No. 2 Morse Taper hole for the tailstock center. Turning the tailstock hand wheel in a counter-clockwise direction to the end of its travel automatically ejects the center. Accurate graduations on the tailstock ram (Figure 28) simplify accurate boring and drilling. The inside tailstock bearing on the rear bed way is gibbed for take-up adjustment. Two gib screws, one on

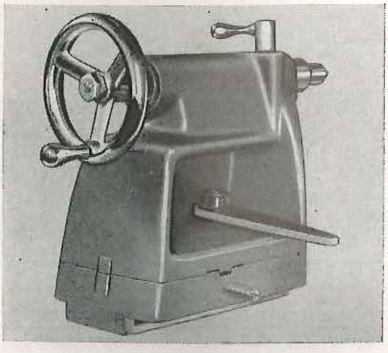
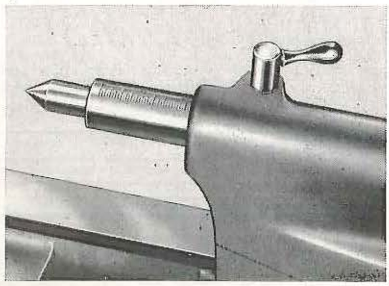


FIG. 27
Rear View of Tailstock Showing Gib and Bed Lock-Lever.

FIG. 28
View of lathe tailstock showing graduated ram and self-ejecting center. The heavy, well braced, grey iron casting assures plenty of strength for accurate turning between centers.

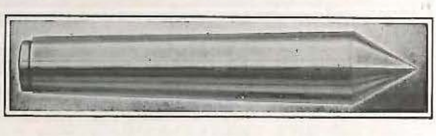


each end of the gib, regulate the tightness of the tailstock between the bed ways. These two screws should be adjusted evenly so that both ends of the gib will bear against the way with the same amount of pressure.

The tailstock can be set over ¾ inch for turning tapers. This is done by simply adjusting the two headless screws after loosening the tailstock clamp nut. Taper turning and the proper realigning of the tailstock are explained in Part 8 of this Manual. Keep the ram well oiled on the outside only. Before inserting the center in the tailstock ram, clean both tapers thoroughly with a dry cloth.

LATHE CENTERS

FIG. 29
Both the headstock and the tailstock centers are hardened and ground carbon tool steel—No. 2 Morse Taper.



A sleeve is furnished to adapt the standard No. 2 Morse Taper Center to the No. 3 Morse Taper headstock spindle nose. Before placing centers in the lathe, clean both external and internal tapers thoroughly with a dry cloth. Any dirt or chips between these tapers will score both and destroy their accuracy. Do not oil the tapers. Even a slight film of oil will prevent a firm fit and cause trouble in turning.

It is vitally important to keep all tapers very clean.

THREADING GEARS

All threads, either right or left hand, from 4 to 96 per inch in the following standards can be cut with the change gears and threading dial furnished as standard equipment: all National Form threads including National Coarse (U.S.S.), National Fine (S.A.E.), Acme, Square and Whitworth. All standard metric threads from .5 to 7 mm. can be cut with the change gears furnished.

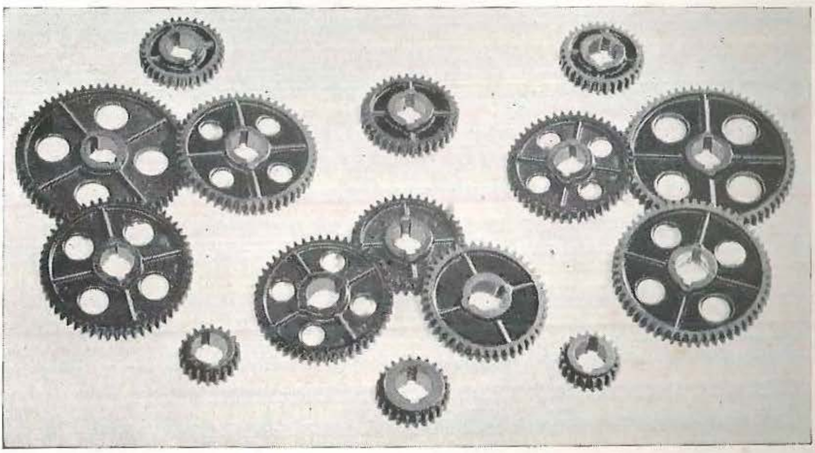


FIG. 30
Threading Gears.

Feeds are available for spring making, wire winding and electrical coil winding with all sizes of wire between No. 12 and 40 B. & S. and all types of magnet wire insulation. Multiple threads, machine screws, pipe-type threads and special screws can also be cut with the standard gears furnished. Complete set-ups and directions for the most common threads and feeds are given in the Threading Supplement.

Part 2

THEORY OF METAL CUTTING

PART 2

THEORY OF METAL CUTTING

Every lathe owner should have a basic knowledge of the cutting action of the tool bit. With this knowledge the lathe tool can be properly ground and applied to the work. Extreme care is taken in the design and manufacture of the modern lathe to provide maximum accuracy and rigidity. Clean, accurate lathe work results only when equal care is taken in the grinding and use of the cutting tool. In the next three sections of this Manual are given actual "reasons why" tool bits are ground to certain angles, how tools are set into the work and what tools are used for different types of work. One important point must always be remembered — ***Use Sharp Tools!*** Nothing is more essential for clean, accurate lathe work or does more to lengthen lathe life.

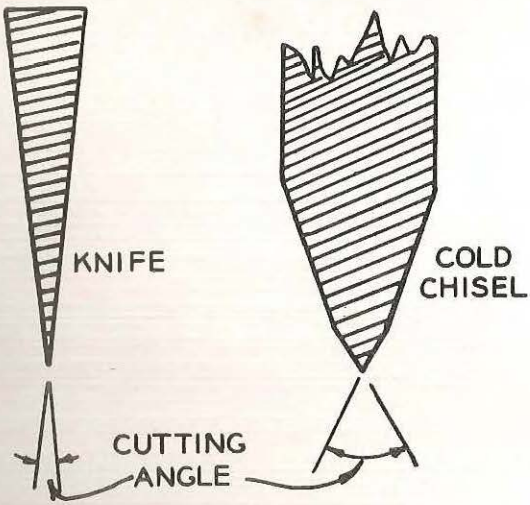


FIG. 33
Cross sections of a knife and a cold chisel, showing the great difference in cutting angles.

THE WEDGING ACTION OF CUTTING TOOLS

All cutting tools employ a wedging action. The differences are in the angle of the two sides of the tool which form the cutting edge and the manner in which the tool is applied to the work. The edge of a pocket knife would be ruined in trying to cut a nail, even though the metal in the knife is much harder than that in the nail. A cold chisel, however, shows no signs of damage in cutting the same nail, although the chisel is usually a poorer grade of steel than the knife. Obviously, the difference lies in the angle of the tool. Figure 33 shows these two tools in profile.

COMPARISON WITH WOOD TOOLS

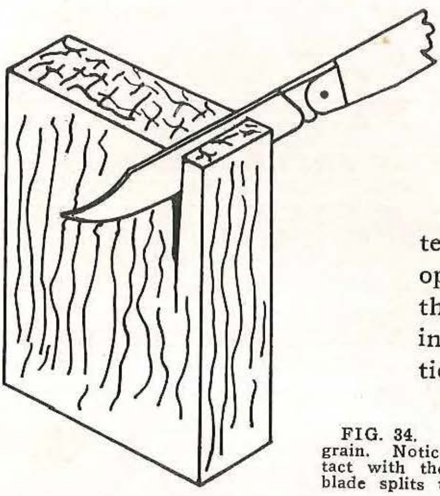


Figure 34 shows the action of a knife in cutting a piece of wood. Notice that the cutting edge of the tool is used only in the first entering cut. All wood cutting tools operate on this principle, although when cutting across grain in a piece of hardwood, the action is more complex.

FIG. 34. A knife slicing a block of wood with the grain. Notice that the edge of the knife is not in contact with the wood—the wedging action of the knife blade splits the wood ahead of the edge.

In Figure 35 a wood chisel is cutting a slice across the end of a block of wood. Here the wedge of the tool acts to shear off small sections, each one a separate cutting and wedging action. If the wood fibers are strong enough, these small sections will cling together and result in a curled chip.

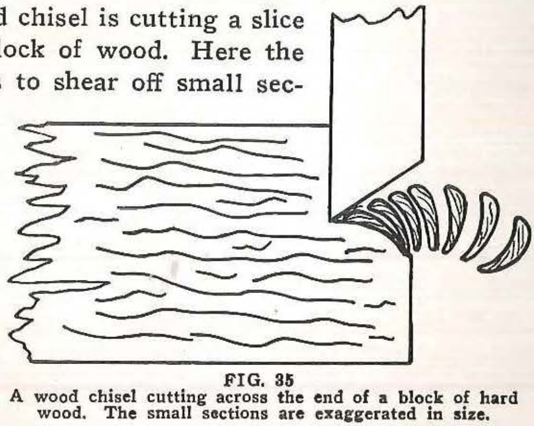


FIG. 35
A wood chisel cutting across the end of a block of hard wood. The small sections are exaggerated in size.

DIFFERENCES BETWEEN WOOD and METAL-CUTTING

Wood cutting tools are usually not clamped in a fixed position, but guided by the operator. Metal cutting, on the other hand, requires holding both the work and the tool as firmly as possible. Absolute rigidity is impossible to attain, but every effort should be made to approach it.

The cutting edge of a lathe tool for metal turning is ground to an angle of between sixty and ninety degrees. This wedge angle must be large because the tool edge must stand up under enormous pressure — an actual downward pressure as high as 250,000 pounds per square inch has been measured on a lathe tool in turning steel.

METAL CUTTING ACTION

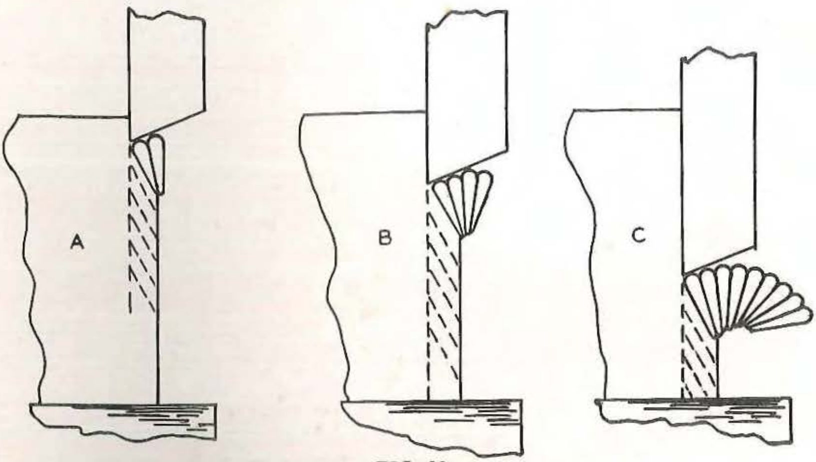


FIG. 36

Progressive steps in metal cutting. At "A" the tool is just entering the metal; at "B" the cut has progressed to a point where the triangular shape of the small sections can be seen. "C" shows the start of the curled chip. For clearness, a straight shear cut is illustrated and the size of the small sections greatly exaggerated.

Figure 36 shows the action of a metal cutting tool. It is assumed that both the tool and work are held rigidly. A shearing cut is pictured — lathe cuts are similar but made on a rounded surface.

The first action, Figure 36A, is that of the tool edge forcing into the metal — an entering cut. Figure 36B shows the wedging action more clearly, the angle of the tool forcing the metal apart and the compression squeezing the small sections into triangular shapes. Figure 36C illustrates the tool further advanced, with the sheared sections forming the start of a curled chip.

There is sufficient force from the wedge of the tool to shear off small sections of the work at short intervals. The cut is not continuous but has a finite fluctuation period measured in small fractions of a second. The wedging force rises to the shearing limit of the small section, drops and gradually rises again until the next section shears, and so on. If this fluctuation time happens to synchronize with the natural vibration period of any part of the tool, holder, or work, a vibration called "chatter" occurs.

It must be realized that the chip and the small sections in Figure 36 are greatly exaggerated in size. Actually, the chip is only several thousandths of an inch in thickness. The deformation of the metal on the inside of a reasonably thick chip can be seen clearly.

FALSE CUTTING EDGE ON THE TOOL

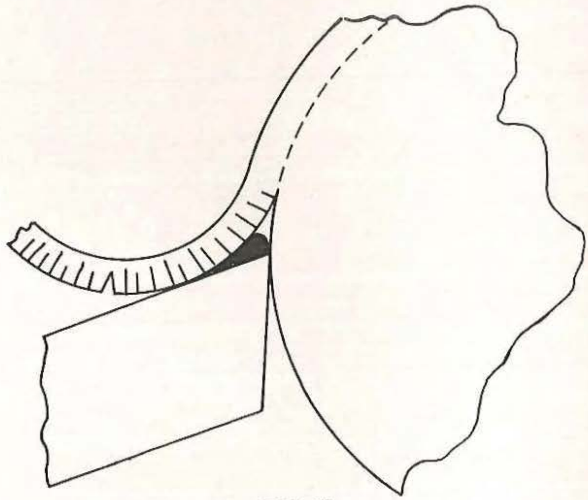


FIG. 37
The false cutting edge formed at the tip of a tool bit. The dark portion is deposited on the tool when taking heavy cuts. The wedging or cutting is done with this bit of metal, not the edge of the tool bit.

A tool bit that has been used on rather heavy cuts has a small ridge of metal directly over the cutting edge. This bit of metal is much harder than the metal being cut and is almost welded to the edge of the tool, indicating that an immense amount of heat and pressure was developed at this point.

This "false cutting edge" acts as the actual cutting edge in turning. It is a decided advantage in heavy turning because it relieves the edge of the tool bit from most of the work of cutting and lengthens tool life. For continuous heavy cuts, the speed should be kept low enough, and the rake of the tool small enough in order to build up this false edge. However, in taking fine finishing cuts, this built-up edge should be avoided by taking finer cuts at higher speeds and with larger rake angles.

There are several theories as to the forming of this false cutting edge. It is generally agreed, however, that the cutting action, aided by the heat and pressure at the end of the tool bit, causes the metal particles to deform or flow which produces what is called "work hardening" of the metal. Whether it is due to the compression of a small strip of metal ahead of the edge of the tool itself, or is simply a work-hardened portion of the main chip is a debated question. The important point to remember is that the false cutting edge is desirable for heavy cuts—on fine finish cuts it should be avoided.

THE SHARPNESS OF THE TOOL EDGE

How fine the edge of a tool bit should be depends upon the class of work (roughing or finishing) and upon the metal being cut. For heavy roughing cuts in steel, there is no point in honing the edge of the tool. A fine edge lasts for only a few feet of cutting, then it rounds off to a more solid edge and remains in approximately this same condition until the tool breaks down. A 60 grit wheel is satisfactory for grinding tools for heavy roughing cuts.

For fine finishing cuts, the tool should be ground to shape and then honed with a reasonably fine stone. In most instances, the finish is directly dependent upon the keenness of the edge of the tool. Tools for soft metals should be honed carefully to as fine an edge as possible—both the cutting action and the finished surface depend upon the edge of the tool.

For threading tools, grinding on a 60 grit wheel is sufficient for roughing cuts, but the edge of the tool should be honed before taking the finish cuts.

HEAT DEVELOPED IN CUTTING METAL

All of the power used in cutting metal is ultimately expended in heat. The shearing of the chip by the wedging action of the tool, the small sections of metal sliding over each other, the back of the chip rubbing on the face of the tool bit, the compression at the point of the tool—all of these actions generate heat which must be dissipated. The tool should have a large cutting angle to help carry this heat away from the cutting edge as rapidly as possible.

In production work, where high speed is important, coolants composed of various chemical mixtures help absorb this heat from the edge of the tool—a steady stream of cutting compound is directed at the point of the tool so that it spreads and covers both the tool and the work. A large pan under the lathe bed collects this compound, carries it to a settling tank and then to a pump.

Coolants are seldom used in small lathe work. Ordinary cutting is done dry, or sometimes with the aid of a cutting oil for lubrication only. It must be remembered when cutting dry, that the work will heat considerably higher than the surrounding temperature, often as much as 100° Fahr. This increase in temperature causes the work to expand, and the tightness of the lathe centers should be watched carefully. In taking measurements with a caliper or micrometer, be sure to cool the work before measuring to a final dimension.

Part 3

CUTTING TOOLS

PART 3

CUTTING TOOLS

LATHE TOOL BIT DESIGN

The angles of the top and sides of lathe tool bits, together with their official A.S.M.E. designations, are shown in Figures 38A, 38B and 38C.

TOP RAKE ANGLES

In the preceding section of this Manual it has been shown that the wedge or cutting angle should be as large as possible for maximum strength at the edge and to carry heat away from the cutting edge. On the other hand, the larger the wedge angle the greater the power required to force it into the work. Thus, there are two opposing factors and a compromise between them is necessary in arriving at the best rake angles. There has been a great amount of experimental work in this connection, notably by F. W. Taylor and O. W. Boston. Recommended values of both back and side rake for the various kinds of metal have been determined. Rake angles for general use with many types of metals and plastics are given in Part 4 of this Manual.

CLEARANCE ANGLES

Clearance angles allow the part of the tool bit directly under the cutting edge to clear the work while taking a chip. Too much clearance weakens the cutting edge, and the high pressure exerted downward on the tool bit demands that clearance be as small as possible and still allow the tool to cut properly.

A tool with excessive clearance also has a tendency to chatter. Taylor's experiments showed that for hand ground tools a side clearance of 12° and a front clearance of 8° is satisfactory for general turning of steel. The larger side clearance is necessary because the lathe tool feeds and cuts at the same time, making the actual path of the tool helical, or spiral, instead of straight. Recommended angles of clearance for metals other than steel are given in Part 4 of this Manual.

Whenever the tool digs into the work or refuses to cut unless forced, check the clearance of the tool bit. Digging-in occurs most often during facing and threading operations. For light turning it is usually better to allow just a little more than enough clearance rather than to risk having too little.

LATHE TOOL BIT DESIGN

OFFICIAL A.S.M.E. DESIGNATIONS OF TOOL BIT ANGLES

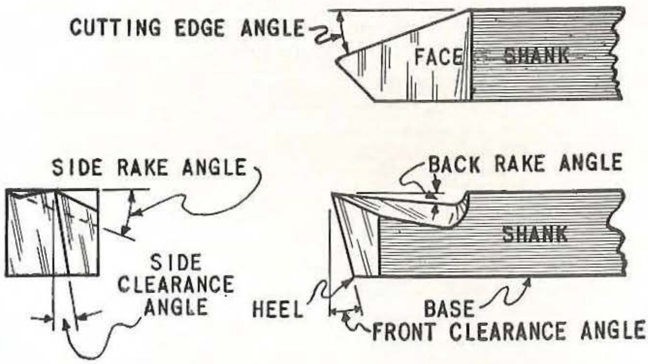


FIG. 38A

Tool bit angles with the tool bit horizontal and at a right angle with the centerline of the work

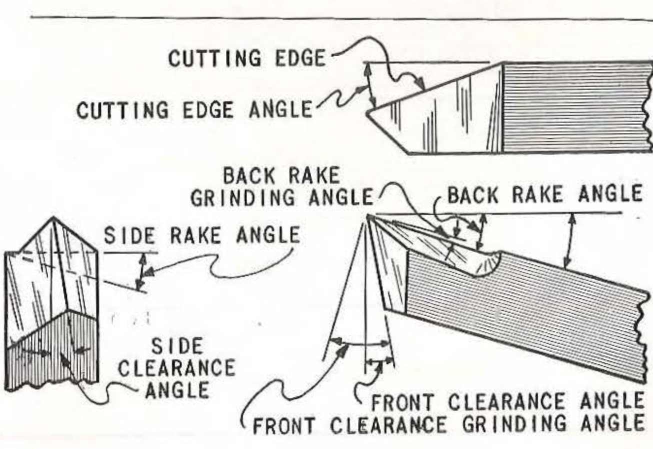


FIG. 38B

Tool bit angles as designated for use in the tool holder.

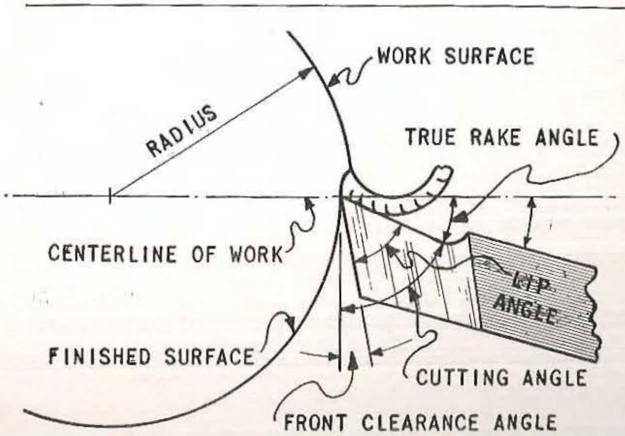


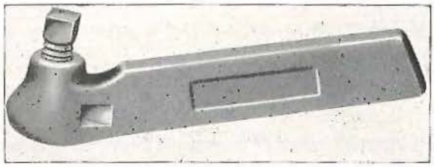
FIG. 38C

Angles of the tool bit in relation to the work.

THE TOOL HOLDER

FIG. 39

A tool holder for holding ¼" x ¼" tool bits. This holder makes unnecessary the use of larger forged tools of expensive high speed steel and provides a 7½° front rake angle without spoiling the entire end of the tool.



Tool holders of the type shown in Figure 39 are used universally on engine lathes, permitting the use of small, inexpensive and replaceable tool bits. The tool bit is set at an angle of 7½°. This angle serves two purposes: it provides a front rake angle without spoiling the entire end of the tool, and it directs a large portion of the cutting pressure directly toward the base of the tool post. Allowance for this 7½° angle must be made when grinding tool bits for use in the tool holder. All of the angles and diagrams in this and the following section take this angle into account.

In order to avoid undesirable overhang, tool bits should be clamped so that the cutting end of the tool bit is as close to the tool holder as the work will permit — also the end of the tool holder which holds the tool bit should be as close to the tool post as possible.

GRINDING TOOL BITS

Figures 40 through 44 show five forms of tool bits for use in the tool holder. These shapes are suitable for practically all lathe turning and the cutting of 60° V-type threads. Part 4 includes correct clearance and rake angles for using these tool shapes in the machining of many different metals, alloys and plastics. Threading and boring tools are described in detail in later sections of this Manual.

A good tool grinder is essential, preferably motor driven such as the one shown in Figure 45. The grinder should have one medium grit wheel (about 60 grit) on which high speed tool bits can be ground. Some practice is necessary before tools can be properly ground but by following carefully the directions given in this section, the beginner will soon become adept at this important part of lathe operation.

The tool can be sharpened on either the side or the face of the wheel, although the regular cutting face is used by most machinists and generally considered better grinding practice. Grind the shapes and angles as directed to within reasonable limits. Be careful not to burn the edges—a cup of water should be kept handy to cool the tool and avoid spoiling the temper of the steel.

Always keep tools sharp.

TOOL BIT SHAPES FOR USE IN THE TOOL HOLDER

The five standard tool forms on these two pages will be found suitable for most lathe turning. When grinding tools for special work, simply keep in mind the shapes and angles recommended for general turning and apply these principles to the special tool being ground. See the examples on page 39.

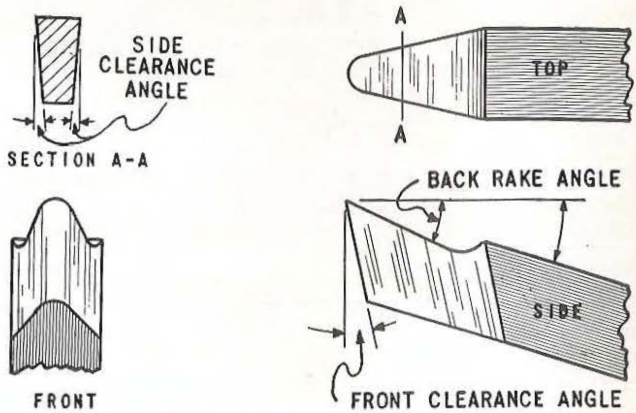


FIG. 40. Round Nose Cutting Tool suitable for roughing and general purpose turning.

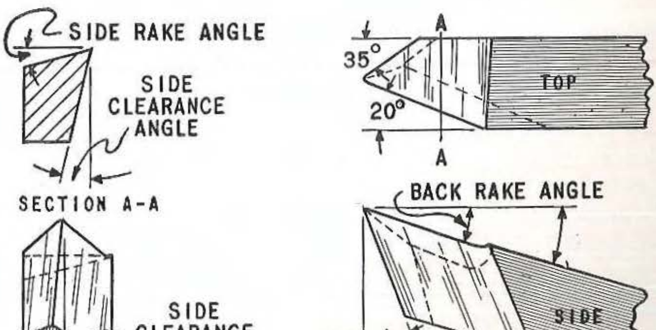


FIG. 41. Excellent R. H. Tool for general turning and shouldering toward headstock; also facing. Point should be rounded for finishing work.

FIG. 42

Excellent L. H. Tool for general turning and shouldering toward tailstock; also facing. Point should be rounded for finishing work.

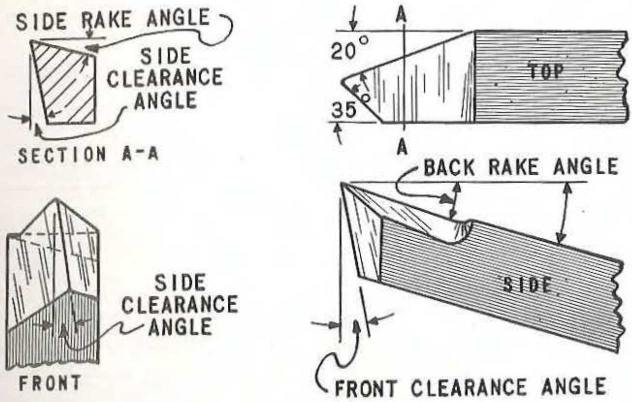


FIG. 43

Heavy Duty R. H. Roughing Tool for taking deep cuts toward headstock. Clearance and rake angles should be reversed for L. H. turning.

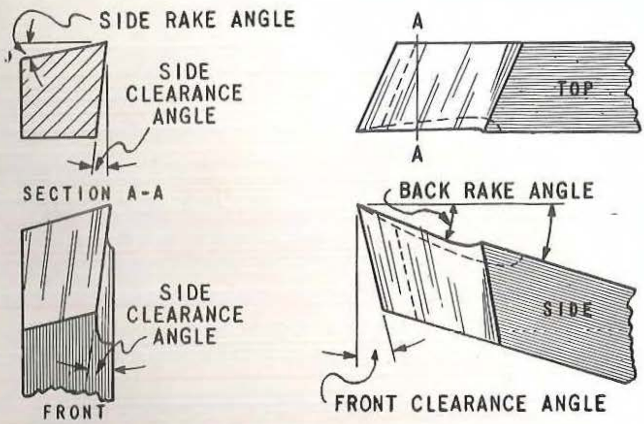
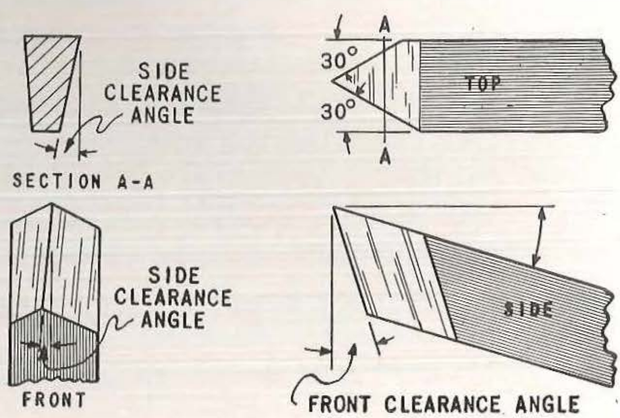


FIG. 44

R. H. 60° V-Type Threading Tool for cutting toward headstock. Side clearance angle should be reversed for L. H. threading.



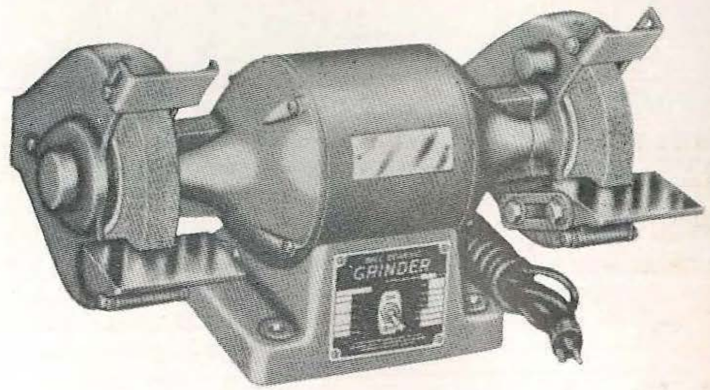


FIG. 45

Using the type of grinder shown in Figure 45, the tool is roughed to shape on the coarse wheel and finish ground on the fine wheel. A properly ground tool will have continuous wheel marks on each face — that is, each face is one clean cut all the way across. The beginner can grind tools quite accurately by comparing each side of the tool with the angles given in the drawings on pages 34, 36 and 37, while grinding the tool.

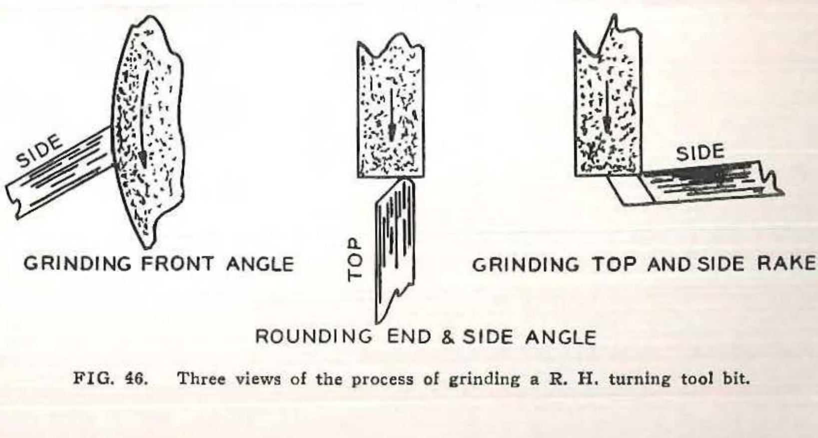


FIG. 46. Three views of the process of grinding a R. H. turning tool bit.

SPECIAL FORM-CUTTING TOOLS

In using form tools with side faces such as shown in Figures 47A and 47D, side rake is out of the question. Front rake, however, should be used except when turning brass. It is recommended that tools wider than 1/8″ never be used on steel. Form cutting tools as wide as 1/2″ can be used on brass, aluminum and similar metals.

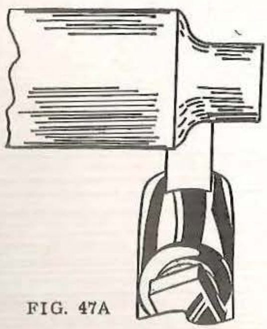


FIG. 47A

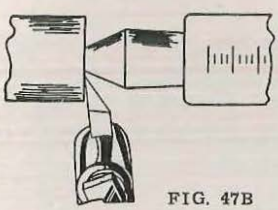


FIG. 47B

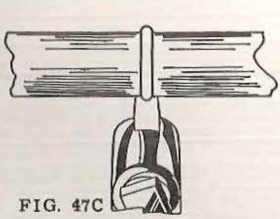


FIG. 47C

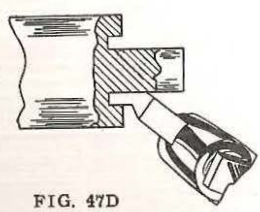


FIG. 47D

SETTING THE TOOL TO THE WORK

The tool shapes and angles appearing on the following pages show the tool being set approximately at right angles to the center line of the work (the line between the lathe centers).

The true rake angle of a tool bit is a combination of the front and side rake (see Fig. 38C) and can be changed slightly by swinging the tool at an angle with the work. For some cuts it is necessary to set the tool at an angle, and occasionally it will result in cleaner cuts and less chatter. Generally, however, the tool should be set directly into the work or at a slight angle as shown in Figure 49.

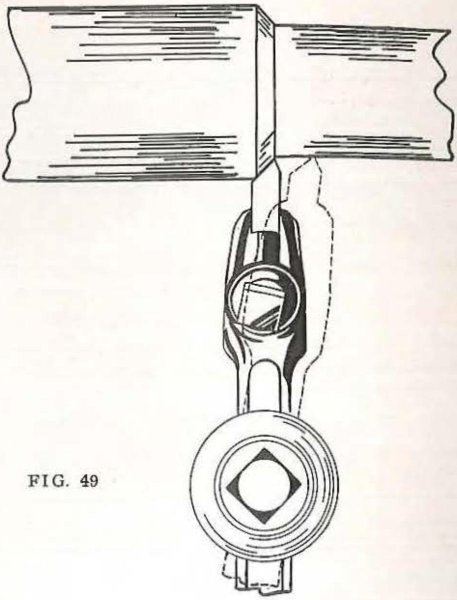


FIG. 49

TYPES OF TOOL HOLDERS

FIG. 50A

Top view of a R. H. Tool Holder. Used for cutting up to chucks, face plate, dogs, etc. at the headstock end of the work.

FIG. 50B

A Straight Tool Holder. Used for general cutting where no clearance is needed.

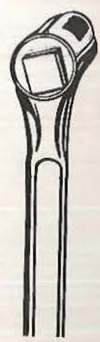


FIG. 50C

A L. H. Tool Holder. Used for cutting up to shoulders, projections, etc., at the tailstock end of the work.

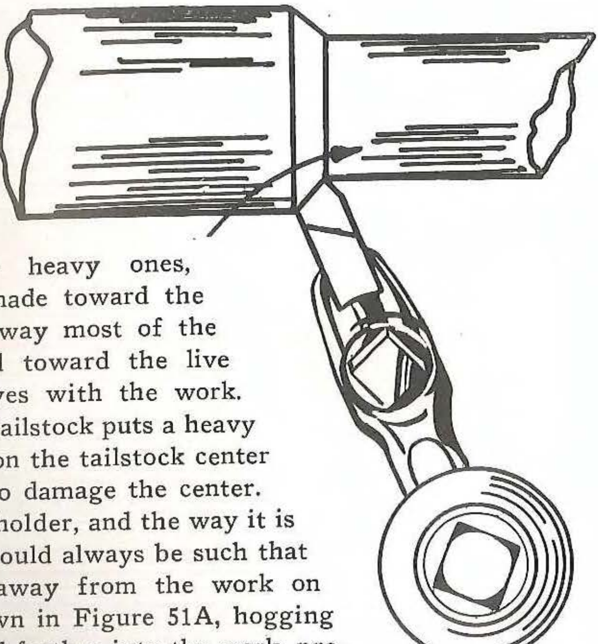


FIG. 51A

When the tool is set like this, it tends to swing into the work on heavy cuts, producing rough work.

Cuts, especially heavy ones, should always be made toward the headstock. In this way most of the pressure is directed toward the live center which revolves with the work. Cutting toward the tailstock puts a heavy additional pressure on the tailstock center and is quite likely to damage the center.

The type of tool holder, and the way it is set into the work, should always be such that it tends to swing away from the work on heavy cuts. As shown in Figure 51A, hogging tends to pull the tool farther into the work, producing a rough, inaccurate cut. If the tool is set as shown in Figure 51B, it swings out away from the work. When cutting at an

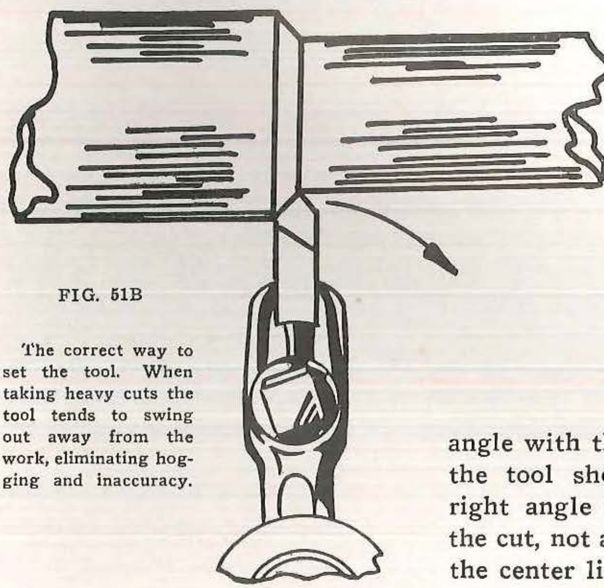


FIG. 51B

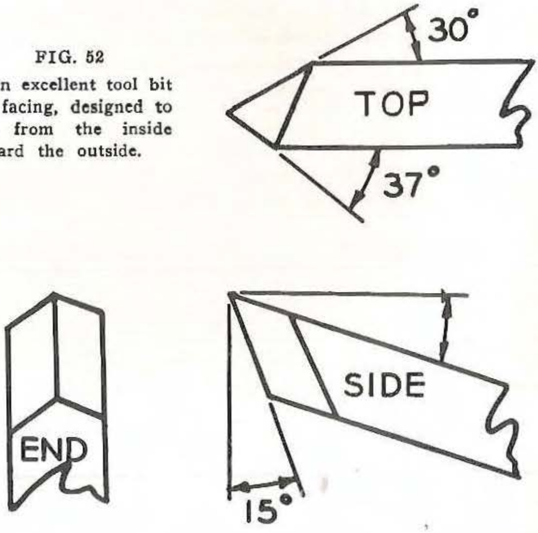
The correct way to set the tool. When taking heavy cuts the tool tends to swing out away from the work, eliminating hogging and inaccuracy.

angle with the compound rest, the tool should be set at a right angle to the surface of the cut, not at a right angle to the center line of the lathe.

FACING CUTS ON THE LATHE

Facing cuts represent different cutting relations and tool angles, and tools should preferably be special ground for that purpose. Smoother cutting and a finer finish can be obtained generally by cutting toward the outside—that is, feeding from the center of the work out. Inasmuch as the tool must cut to the center, larger clearances should be used than when turning cylindrical work.

FIG. 52
An excellent tool bit for facing, designed to cut from the inside toward the outside.



Although ordinary R. H. cutting tools can be used for small amounts of facing work, any large amount of facing should be done with a tool ground especially for the purpose. Figure 52 shows a tool which will be found excellent for this type of cutting. It is designed to cut from the center of the work toward the outside. Notice that the shape differs somewhat from that of a standard turning tool, the effective rake of the cutting edge being dependent upon the rake angle shown. This rake angle should be the same as, or slightly greater than, the angles given for front rake for the various metals and plastics in the following section of this Manual.

The clearance angles of 15° are suitable for facing practically any material. Figure 53 shows how the tool should be set into the work.

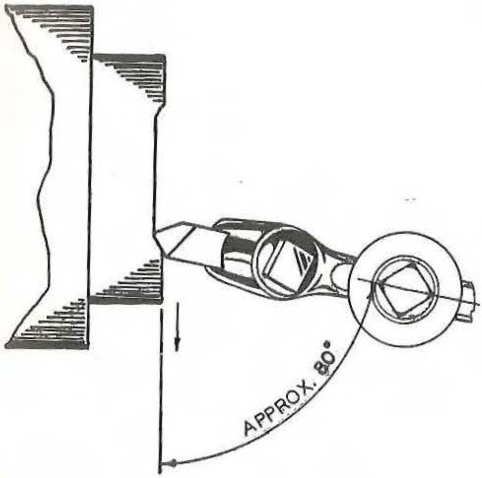


FIG. 53

When facing with a tool of the type shown in Figure 52, the tool should be set to the work in this manner The angle of 80° is approximate and can be changed for the different types of facing tools.

SET THE POINT OF THE TOOL ON THE CENTER LINE

If the tool is ground properly, the point of the tool will not have to be set above or below the center line of the work. Figure 54A shows a tool bit with an 8° front clearance set into a piece of work exactly on the center line. The clearance angle is measured between the tool and the line AB, which is tangential to the work at the point of contact of the tool. The front rake is measured

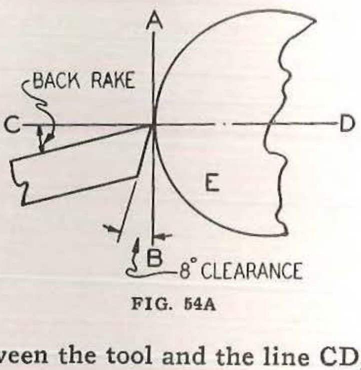


FIG. 54A

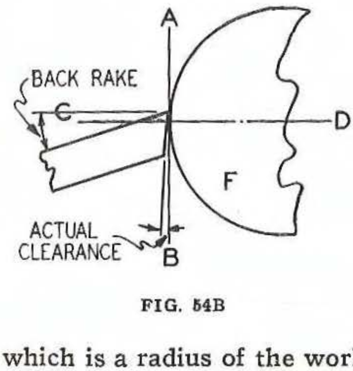


FIG. 54B

between the tool and the line CD, which is a radius of the work to the point of contact and is at right angles to the line AB.

Now, if the same tool is set above center as shown in Figure 54B, an entirely different condition exists. The front clearance is still measured between the tool and the tangential line AB, but AB is no longer vertical and the angle of clearance has been greatly reduced. The radius line CD has also been moved so that the back rake angle is now larger. A tool set to the work in this position would have to be ground entirely different in order to cut correctly.

A tool with an 8° front clearance, working on a piece of stock one inch in diameter, would have to be set only a trifle more than 1/16 inch above center in order to have the line AB coincide with the front line of the tool, producing a zero clearance. With the same setting, an original back rake of 20° would become 28°.

Some machinists make a practice of setting tools above the center line, but they must grind their tools especially for that type of setting. For the average operator, student or beginner, it is recommended that the tool bit be ground to the given angles and set exactly on the center line.

Several methods can be used to set the tool on the center line. The point can be lined up with either of the lathe centers, or the distance from the bed way to the headstock center can be measured and transferred. Another excellent method is to scribe a line along the tailstock ram: set a sharp pointed tool sidewise in the tool holder and align it with the headstock center. Then use this pointed tool to scribe a light line along the side of the tailstock ram (remove burr). This line will serve as a guide to set the tool, even when the work is in position between centers.

Part 4

THE MACHINING OF VARIOUS MATERIALS

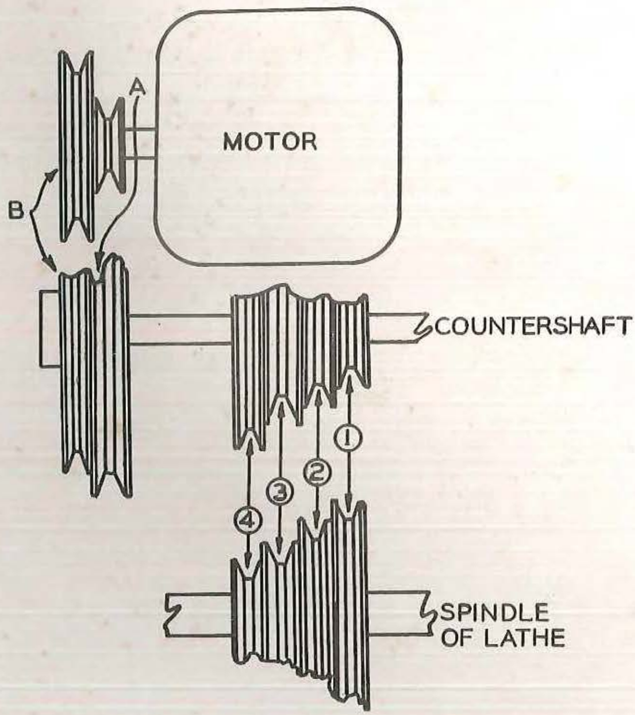
PART 4

THE MACHINING OF VARIOUS MATERIALS

PROPER CUTTING SPEEDS

Much of the success in metal cutting depends upon the choice of the cutting speed. Too slow a speed not only wastes time, but leaves a rough finish—too high a speed burns the tool. Of the sixteen speeds listed below, eight are on direct drive and eight on backgear drive. Figure 55 lists these speeds and shows how they are obtained.

FIG. 55. SPINDLE SPEEDS IN REVOLUTIONS PER MINUTE
12-INCH BACK GEARED LATHE



BACK GEAR DRIVE					DIRECT CONE DRIVE				
Motor Belt Position	Spindle Belt Position				Motor Belt Position	Spindle Belt Position			
	1	2	3	4		1	2	3	4
A	28	45	70	112	A	164	266	418	685
B	83	134	211	345	B	500	805	1270	2072

Cutting speeds for metal turning are usually expressed in feet per minute, measured on the circumference of the work. Spindle revolutions per minute are then determined by using this formula:

$$\frac{12 \times \text{SFM}}{3.1416 \times \text{D}} = \text{RPM}$$

which is simplified to

$$\frac{3.82 \times \text{SFM}}{\text{D}} = \text{RPM}$$

where SFM is the rated surface feet per minute
RPM is the spindle speed in revolutions per minute
D is the diameter of the work in inches.

In order to simplify the selection of the proper speed, Figure 56 gives the exact speeds obtainable, which correspond approximately to surface speeds in feet per minute for the various work diameters. Thus, knowing the surface speed recommended for various metals and plastics, first use Figure 56 to find the proper lathe speed for the diameter being turned, then refer to Figure 55 for the correct belt set-up to obtain that speed.

DEPTH OF CUT AND FEED PER REVOLUTION

The speeds recommended for the various metals and plastics are for cuts of $\frac{1}{8}$ of an inch or less in depth. The harder the metal, the less the depth of cut should be. Ordinary turning does not demand unusually deep cuts—more metal per minute can usually be removed by turning at recommended speed with a roughing cut of between .100 and .125 inch in depth.

Finish cuts are taken after the roughing cuts and should be approximately .015 inch or less in depth, taken at the recommended speed. The work can be roughed down to within approximately .015 inch of the final diameter, then finished with a sharp tool, using light cuts. Before taking finish cuts to size be sure that the work has cooled to approximately room temperature—the shrinkage of a hot piece of work can easily spoil the intended fit.

The six most common carriage feeds are shown on the lathe threading chart: .0087, .0070, .0060, .0050, .0035 and .001877

FIGURE 56

TABLE OF CUTTING SPEEDS

Correct Spindle Speeds to Give Approximately the Surface Speeds Shown

Diameter of Work Inches	*Surface Speed in Feet Per Minute*											
	30	*40*	*50*	*60*	*70*	*80*	*100*	*120*	*150*	*200*	*300*	*500*
1/16..........	2072	—	—	—	—	—	—	—	—	—	—	—
1/8...........	805	1270	1270	2072	2072	2072	—	—	—	—	—	—
3/16..........	685	805	805	1270	1270	1270	2072	2072	—	—	—	—
1/4...........	418	685	805	805	805	1270	1270	2072	2072	—	—	—
3/8...........	266	418	500	685	685	805	805	1270	1270	2072	—	—
1/2...........	266	266	418	418	500	685	685	805	1270	1270	2072	—
5/8...........	164	266	266	418	418	500	685	685	805	1270	2072	—
3/4...........	164	164	266	266	418	418	500	685	805	1270	1270	2072
1.............	112	164	164	266	266	266	418	418	500	805	1270	2072
1¼...........	83	112	164	164	164	266	266	418	418	685	805	1270
1½...........	70	112	112	164	164	164	266	266	418	500	805	1270
1¾...........	70	83	112	112	164	164	164	266	266	418	685	1270
2.............	45	70	83	112	112	164	164	266	266	418	500	805
2½...........	45	70	70	83	112	112	164	164	266	266	418	805
3.............	45	45	70	70	83	112	112	164	164	266	418	685
3½...........	28	45	45	70	70	83	112	112	164	164	266	500
4.............	28	45	45	45	70	70	83	112	164	164	266	500
5.............	28	28	45	45	45	70	70	83	112	164	266	418
6.............	28	28	28	45	45	45	70	70	83	112	164	266
7.............	28	28	28	28	45	45	45	70	83	112	164	266
8.............	28	28	28	28	28	45	45	45	70	83	164	266
9.............	28	28	28	28	28	28	45	45	70	83	112	164
10...........	28	28	28	28	28	28	45	45	45	70	112	164

inches per spindle revolution. Ordinary cutting, where the final finish can be touched up by filing and with emery cloth, should be done with the .0087 inch feed. The .0050, .0035 and .001877 inch feeds are used for a fine finish and for working on tough, hard-to-machine metals. The .0035 inch feed is ideal for taking trueing cuts on commutators. The .0060 inch feed is an intermediate feed often useful for work not falling into these other classes. Gear set-ups for the various carriage feeds are given in Part 7.

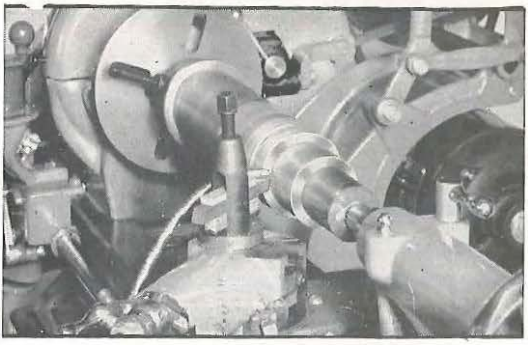


FIG. 57

Reducing the diameter of a steel shaft ½" in one cut. Except for the experienced machinist cuts like this should never be taken — use the speed recommended for the metal being turned and take cuts of about ⅛ of an inch for roughing.

CUTTING COMPOUNDS

Ordinary turning on the lathe is done dry. During threading operations the use of a cutting compound, oil or fluid results in a better class of work. Lard oil or any one of the general purpose cutting fluids should be kept handy for this purpose. Continuous production work usually requires the use of liberal quantities of a cutting compound to carry the heat away from the tool bit.

MACHINING STEEL

Steel is manufactured in hundreds of grades, each with a different carbon and alloy content. The grades of steel listed and described in this Manual are carried in stock by most steel suppliers. They are purchased from the warehouse by their S.A.E. (Society of Automotive Engineers) numbers, listed in detail in Part 10. Some of the harder grades should be purchased annealed for machining purposes.

The tool angles and cutting speeds given in Figure 59 are approximate and will be found suitable for average work. They

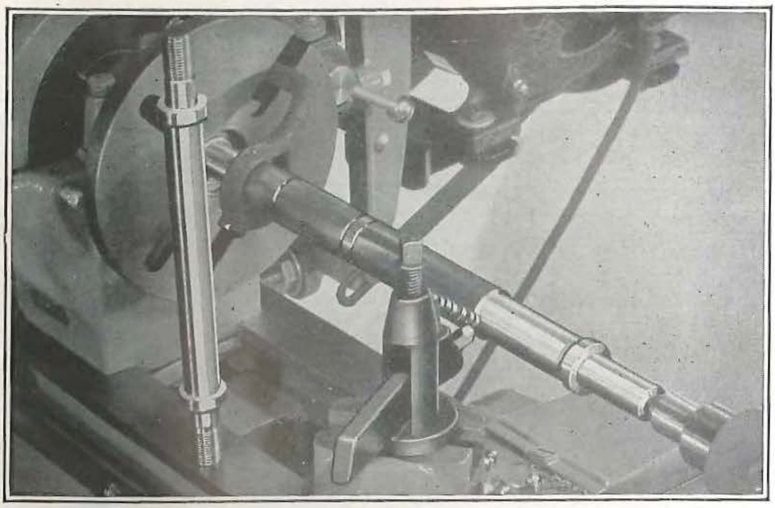


FIG. 58
A small alloy steel grinder shaft turned in the lathe. The finished shaft is also shown. Note the grooves made by the cut-off tool for blocking out the work for roughing (see page 160).

represent the consensus of opinion of a large number of factories, steel companies and machinists. It is impossible to give precisely the tool angles and speeds most satisfactory for each grade of steel, since feeds, depths of cuts, temper of work and other conditions vary for each job. Some experimenting may be necessary for production work, form tools and special shapes.

The machinability rating of each steel is an arbitrary figure determined by averaging machining time over hundreds of jobs and operations. Using S.A.E. 1112 steel as a basis and rating it 100%, the percentages given for the other steels indicate the ease of machining, or machinability—the lower the rating the more difficult is the machining.

In general, for steel, the clearance angles of the tool should be as small as can be used without hogging or having the tool edge break down too soon. The front clearance of 8° and side clearance of 10° to 12° are fairly standard for hand ground tools. Smaller clearance angles can be used in some cases, but are not recommended except for production work.

On screw machine work where a long, curled chip is undesirable, the rake angles of the tools should usually be reduced in order to break up the chips. More power will be required when using these smaller rake angles.

7 - THREADING
8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
5 - HOLDING WORK
6 - DRILLING

FIGURE 59

TOOL ANGLES AND SPEEDS FOR MACHINING STEEL

These Angles Refer to Tool Shapes on Pages 36, 37.

Description of Steel	S.A.E. No.	Machin-ability	Speed feet per minute	Side Clear-ance Angle	Front Clear-ance Angle	Back Rake Angle	Side Rake Angle
Bessemer Screw Stock	1112	100%	120	12°	8°	16½°	22°
Special Screw Stock	X1112	120	150	12°	8°	16½°	22°
High Manganese Screw Stock	X1314	95	100	12°	8°	16½°	22°
High Manganese Screw Stock	X1315	95	100	12°	8°	16½°	22°
High Manganese Screw Stock	X1335	75	100	12°	8°	16½°	18°
Open Hearth Screw Stock	1120	80	100	12°	8°	16½°	18°
Carbon Steel	1020	60	80	12°	8°	16½°	14°
Carbon Steel	X1020	70	80	12°	8°	16½°	14°
Carbon Steel	1035	62	80	12°	8°	16½°	14°
Carbon Steel	1040	61	80	12°	8°	16½°	14°
Nickel Molybdenum	4615	60	80	12°	8°	16½°	14°
Carbon Steel	1045	55	70	10°	8°	12°	14°
3½% Nickel Alloy..	2315	50	80	10°	8°	12°	14°
3½% Nickel Alloy..	2320	50	80	10°	8°	12°	14°
3½% Nickel Alloy..	2330	50	80	10°	8°	12°	14°
3½% Nickel Alloy Annealed	2335	50	70	10°	8°	12°	14°
Nickel Chromium Alloy	3115	50	70	10°	8°	12°	14°
Nickel Chromium Alloy	3120	50	70	10°	8°	12°	14°
Chrome Molybdenum	4140	50	70	10°	8°	12°	14°
Manganese Alloy ...	T1335	50	60	10°	8°	12°	14°
3½% Nickel, Annealed	2340	45	70	10°	8°	10°	12°
3½% Nickel, Annealed	2345	45	60	10°	8°	10°	12°
3½% Nickel, Annealed	2350	40	50	10°	8°	10°	12°
Nickel Chromium ..	3130	45	70	10°	8°	10°	12°
Nickel Chromium, Annealed	3135	45	60	10°	8°	10°	12°
Nickel Chromium, Annealed	3140	45	60	10°	8°	10°	12°
Chrome Vanadium, Annealed	6140	40	60	10°	8°	10°	12°
High Carbon Steel..	1095	35	50	10°	8°	8°	12°
Nickel Chromium, Annealed	3250	35	50	10°	8°	8°	12°
Chrome Vanadium, Annealed	6145	35	50	10°	8°	8°	12°

High speed tool bits are perfectly satisfactory for turning any of the steels listed in Figure 59. As mentioned before, tools for roughing cuts can be ground satisfactorily on a 60 grit wheel without honing. For finish cuts, the tool should be honed to as fine an edge as possible.

The cutting speeds are given in surface feet per minute; for the correct lathe spindle speed see Figure 56. Speeds shown are for machining dry. For best results and easier machinability, lard oil, or equivalent should be used, especially with the harder-to-machine steels. A lubricant also permits approximately 25% higher cutting speeds. For production and automatic screw machine work, commercial types of sulphurized mineral oils, or base compounds mixed with mineral oil or water are used both as coolants and for their lubricating properties.

In machining the softer grades of steel, roughing cuts can be taken with the .0087 inch feed, with depths of cuts of about 1/16 to 1/8 inch when turning at the rated speed. Deeper cuts can be taken easily at slower speeds, but it is recommended that the machinist never take roughing cuts of more than 1/8 inch—deeper cuts at rated speeds require a larger driving motor than the size recommended for the lathe. Finish cuts can be taken with any of the four feeds available, the finer feeds producing the smoothest finish. The depth of the finish cut should be .015 inch or less.

A little experimenting soon tells the operator the proper feed and depth of cut for a given steel. The figures in Figure 59 are suggestions only, and the machinist can usually tell from experience and "feel" just how much cut and feed to use.

MACHINING TOOL STEELS

It is impossible to group the many hundreds of tool steels or give definite tool angles or any description of their properties. Only annealed tool steels should be machined on the lathe. Some experimenting is necessary to determine the proper rake and clearance angles for tool bits. The harder grades, such as high speed tool steel or high carbon steel, will machine best with tool angles similar to those given for S.A.E. 1095 steel in Figure 59. Some of the die steels, while exceptionally hard when tempered

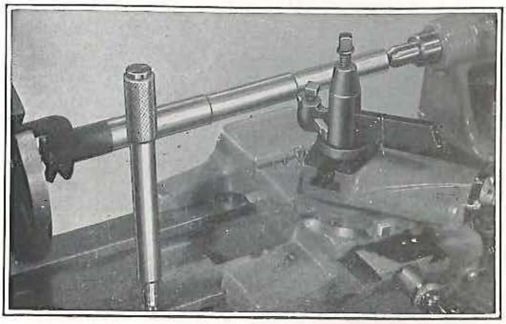


FIG. 60
Machining a bushing driver from tool steel. The finished tool is also shown. Tool angles for tool steel will require some experimenting on the part of the operator —there are hundreds of grades, each having different characteristics.

are furnished annealed. These steels can be machined best with rake angles as large as those recommended for S.A.E. 1112 steel in Figure 59.

MACHINING FORGED STEEL PARTS

Forgings are made from practically any type of steel and are usually annealed after forging. They are machined in the same manner as the bar stock from which they are made (see Fig. 59). Cuts should be deep enough to cut through the scale.

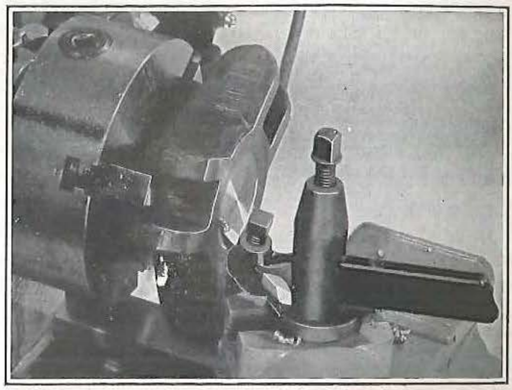


FIG. 61
Machining a forged machine part on the lathe.

MACHINING CAST IRON

Common cast iron, sometimes called "grey iron," is not so easy to machine as soft steel, nor can it be turned at so high a speed. The structure of this metal causes the chips to break out in small sections, not in a continuous chip. Rake angles must be smaller than for the softer steels. The tool nose should be sharper than for steel.

Approximate Tool Angles for Cast Iron	
Front Clearance	8°
Side Clearance	10°
Back Rake	5°
Side Rake	12°

Tool Bit Shapes, Pages 36-37

A turning speed of 50 feet per minute is generally satisfactory for cast iron, although higher speeds are sometimes used in production or with special tool bits.

CUTTING BENEATH THE SCALE

A hard scale containing sand particles forms on the outside of iron castings. Unless the first cut is taken deep enough to cut through this scale and into the softer metal, the cutting edge of the tool will be dulled quickly. First cuts on castings should be at least 1/16 or .0625 inch in depth. A speed slower than 50 feet per minute may be necessary for this depth of cut, but the speed should be reduced, not the depth of cut.

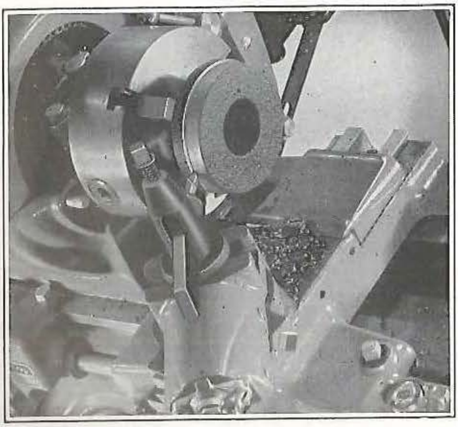


FIG. 62
Machining a cast iron collar on the lathe. Note that the cut is deep enough to get beneath the scale.

The .0087 inch feed can be used for turning most cast iron, with a depth of cut of between 1/16 and ⅛ inch running at rated speed after the scale is removed. In this connection it is interesting that much superior finishes can be obtained on cast iron by the use of high speeds and shallow cuts. Exceptionally fine finishes have been produced with speeds as high as 150 feet per minute, a depth of cut of .015 inch and the .0087 inch feed.

Cast iron is machined dry, with no lubricant or cutting oil. The structure of the metal contains a great deal of free carbon which provides the needed lubrication.

Cast iron parts that are to be machined to very accurate limits should be rough turned to within .015 to .030 inch of their finished size, and then allowed to age for three months or longer. Internal strains are set up in the metal while it is being cast, and if time is not allowed for these strains to normalize, the casting will warp after it is machined.

MACHINING STAINLESS STEEL

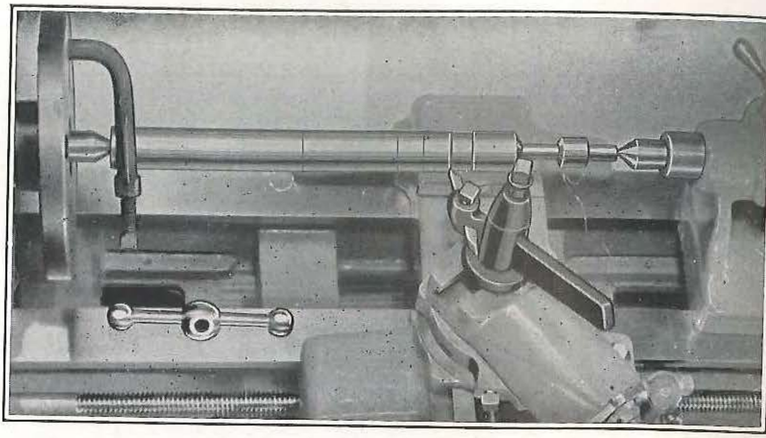


FIG. 63
Machining a stainless steel handle for use on a creamery plant machine. These steels are often used where corrosion must be avoided.

Stainless steels are either high chromium (12 to 14% chromium) or chrome-nickel (18% chrome, 8% nickel). The addition of this alloy makes stainless steels highly resistant to corrosion as well as unusually tough and strong—they are being used more and more wherever these qualities are desired. Some stainless steels are hard to machine, but if proper grades are selected they will machine fairly well. Two grades, No. 303 and No. 416, are furnished as "Free Machining Quality" in rod and bar form and can be obtained from your steel supply house.

Stainless steel is a tough, draggy metal and requires more rake than would be expected. Small rake angles will invariably cause hogging, and the material will "work-harden" badly, that is, the action of the tool in cutting causes the surface of the finished work to harden.

To prevent rubbing, clearance angles are slightly more than standard. Threading tools should have a pronounced side rake of 5° to 10°. Chips produced when turning stainless steel are stringy and hard to manage and should be pulled away from the work. They will be hot and sharp and should be handled with heavy cotton gloves or a thick cloth.

Tool angles suitable for most grades of stainless steel:

Front Clearance	10°	Back Rake	16½°
Side Clearance	12°	Side Rake	10°

Tool Bit Shapes, Pages 36-37

The tool angles on page 56 and the following speeds and feeds apply to both Nos. 303 and 416. Some experimenting may be necessary for other grades. These figures can be changed somewhat if conditions are unusual.

Slow speeds and heavy cuts are best for turning stainless steel. Speeds of 40 feet per minute will be satisfactory in most cases—much higher speeds cause the tool to break down after a short time. For roughing, the .0087 inch feed should be used with depths of cuts of between 1/16 and 1/8 inch. Finish cuts should be taken with a well rounded tool, preferably using the .0050 inch feed, and with a depth of cut of .010 to .015 inch or more. If possible, only one finish cut should be taken—the work-hardening of the metal makes a second shallow cut difficult.

Both No. 303 and No. 416 can be cut dry, but a standard lubricant results in a better finish and easier cutting. High-sulphur cutting compounds, lard oil or equivalent will be found satisfactory. A lubricant should always be used when threading.

CAUTION: When machining stainless steel, check the tightness of the work between centers after each cut. When heated, stainless steel expands approximately twice as much as ordinary steel, especially when cut dry. The tailstock center can be ruined quickly if extreme care is not taken in keeping just the right amount of pressure between the centers.

This tendency to heat up must be remembered when turning pieces to an exact size, and measurements should not be taken while the work is hot. A good method is to rough down to within about .015 inch of the finished diameter, remove the work and cool it with water or oil, then mount it and proceed to take the finishing cuts.

MACHINING COPPER

Copper, due to its combination of toughness and softness, requires different tools than brass or other copper alloys. These tool angles will generally prove satisfactory:

Front Clearance	12°	Back Rake	16½°
Side Clearance	14°	Side Rake	20°

Tool Bit Shapes, Pages 36-37

MACHINING COPPER (Continued)

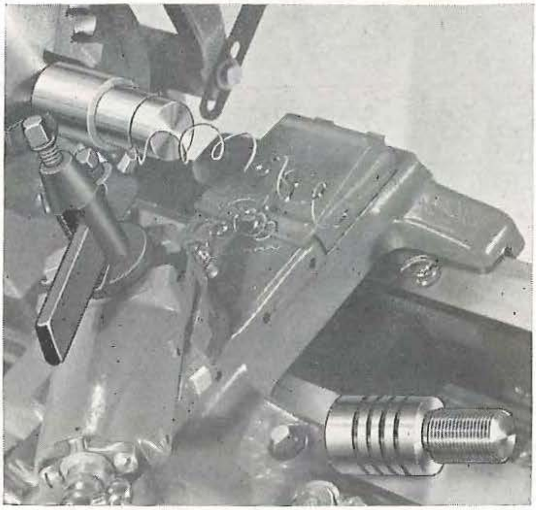


FIG. 64
Machining a solid copper electrode on the lathe. A finished electrode is also shown.

A turning speed of 120 surface feet per minute is recommended for copper, although slightly lower speeds may sometimes be necessary with wide faced tools. The .0087 inch feed should be used except for fine finish cuts, where the .0035 inch feed is best. The depth of cut for roughing can be .030 to .050 inch, and for finishing about .010 inch. Rather deep cuts at rated speed will generally be most satisfactory. On finish cuts, use a round nose tool with about 1/16 inch radius. To produce a smooth finish on copper, tools should be honed to as keen an edge as possible. Chips are tough and stringy and should be pulled away from the work—wear gloves or use a thick cloth to prevent burning your hands.

No lubricant is necessary, but it is suggested that lard oil or paraffin oil be used for threading.

Using the cut-off tool on soft copper is unusually difficult, due to the tendency of the chip to spread and jam in the groove. A method recommended by many machinists is to start a groove wider than the cut-off blade and move the cut-off tool back and forth continually as it is fed into the work, allowing the chip to clear the work without jamming. Allowance for the extra width of the groove should be made when laying out the work.

MACHINING BRASS AND COPPER ALLOYS

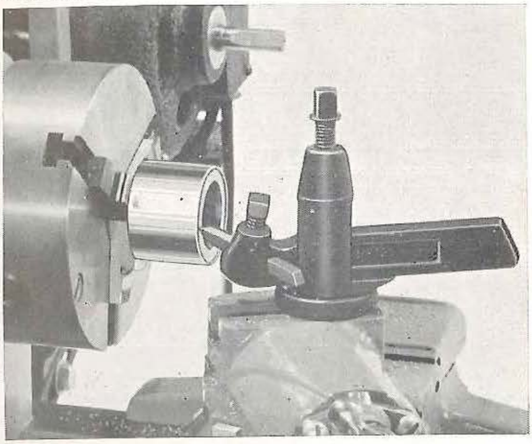


FIG. 65
A brass bushing being turned on the lathe. On many production jobs brass will be found more economical than steel, due to increased production and higher scrap value.

Free cutting brass, commercial bronze, commercial yellow brass, red brass, cast bronze, and other of the softer copper alloys are machined quite differently than steel. Because the tool has a tendency to hog into the soft metal, tool angles are required as follows:

Front Clearance	8°	Back Rake	0°
Side Clearance	10°	Side Rake	0°

Tool Bit Shapes, Pages 36-37

A very slight side rake of not more than 5° can often be used on the free-machining grades of brass and bronze. On some of the tougher alloys, a negative side rake of 2° to 4° is sometimes used to prevent hogging. If hogging and a rough finish occurs, check the clearance angles and try a slight amount of negative rake.

On production work free-machining brass is turned at speeds as high as 600 feet per minute. For small lathe work when production is not important, these speeds are recommended:

Free Cutting Brass	300 feet per minute
Yellow Brass	200 feet per minute
Commercial Bronze	80 feet per minute
Cast Bronze	50 feet per minute

Use light cuts at rated speeds rather than deep cuts with slower speeds. For roughing, depths of cuts should be from 1/16 to 1/8 inch. The .0087 inch feed can be used for roughing, and the .0035 inch feed for finishing. Lubricants are not generally used, although paraffin oil or equivalent will assist in threading.

THE HARDER COPPER ALLOYS

Special bronzes and nickel silvers, which contain elements giving high strength, hardness and toughness, are more difficult to machine than the freer cutting brasses. Phosphor bronze and silicon bronze are in this class. Experiment with tool angles for these metals if large amounts of work are to be done. Suggested angles:

Front Clearance ..... 12°	Back Rake .... 10°
Side Clearance .... 10°	Side Rake ...... 0° to -2°

Tool Bit Shapes, Pages 36-37

If hogging occurs, indicated by a rough irregular finish, reduce the rake angles.

Speeds may vary from 80 to 120 feet per minute. A feed of .0087 inch is recommended for roughing, .0035 for finishing. Take cuts of .015 inch to .125 inch in depth at rated speeds. No lubricant is necessary, although paraffin oil or equivalent will be found helpful for both turning and threading.

Some of these alloys have a small percentage of lead in their composition which improves their machinability. Rake angles of 0° should be used in turning the leaded copper alloys.

MACHINING HARD BRONZE

The alloy known commercially as hard bronze is sold under various trade names. It is used for such purposes as non-sparking wrenches and tools and has many applications where inflammable materials are handled. Its strength is comparable to tempered mild steel. Tool bits should be ground to these angles:

Front Clearance ..... 8°	Back Rake .... 0°
Side Clearance ...... 10°	Side Rake ...... 0° to -2°

Tool Bit Shapes, Pages 36-37

Cutting speeds for the various grades of hard bronze range between 40 and 100 feet per minute — the manufacturer's recommendations should be followed. Use the .0087 inch feed with moderately deep cuts about .030 inch in depth. No lubricant is necessary, although, in turning some of the harder grades, kerosene will be helpful.

MACHINING ALUMINUM

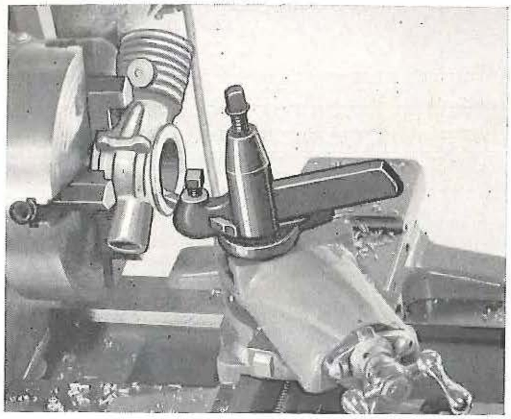


FIG. 66
Machining an alloy aluminum cylinder for a model gas engine. Engines for model airplanes and motor boats are generally made of aluminum to reduce weight.

Aluminum especially with high scrap or silicon content is difficult to machine, due to its tendency to hog and pile up in front of the tool. However, free-machining aluminum alloys have been developed and are available at most warehouses.

Two of these alloys, Alcoa 17S-T and Alcoa 11S-T3, are particularly interesting to the machinist. No. 17S-T has been on the market for some years, while No. 11S-T3 is comparatively new. Both have a tensile strength approximately equal to that of mild steel—the highest strength of any of the aluminum alloys available in rod form. Similar alloys have been marketed under various trade names, such as "Duraluminum" and are used wherever strength and lightness are desired, such as in automobiles, airplanes and dirigibles. Both Alcoa 17S-T and 11S-T3 are easily machined on the lathe with only a few special precautions.

Cast aluminum alloys have lower tensile strengths than the wrought aluminum alloys mentioned above and are ordinarily more difficult to machine. Most of the reputable foundries casting aluminum use pure alloys and turn out castings that can be machined without trouble. However, if large percentages of scrap are used or if the silicon content of the metal is allowed to become too high, there is considerable difficulty in machining.

Alcoa alloys No. 12 and No. 112 are quite easy to machine, and if castings are made from these alloys, they can be worked satisfactorily. High silicon aluminum alloys can be turned better with special tool bits—higher speeds can be used and the cutting edge of the tool stands up longer.

TOOL ANGLES: Tool bits for turning aluminum usually have more rake on both side and front than for steel. The following angles are satisfactory for turning practically all types of aluminum alloys, both cast and wrought:

Front Clearance ..... 8°	Back Rake .... 35°
Side Clearance .... 12°	Side Rake .... 15°

Tool Bit Shapes, Pages 36-37

The edge of the tool in contact with the work should be rounded but not too bluntly. If chatter occurs, decrease the radius of the tool point. Cut-off tools for aluminum should have about 15° back rake and only 4° to 5° front clearance.

The proper cutting speed is very important in turning aluminum, and in many cases trouble can be traced to the use of too low a cutting speed. While surface speeds for turning steel vary between 75 and 150 feet per minute, aluminum is turned best at speeds from 200 feet to as high as 800 feet per minute. For general work, it is recommended that wrought aluminum alloys such as 17S-T and 11S-T3 be cut at surface speeds of 300 to 500 feet per minute, while cast aluminum should be turned between 200 and 300 feet per minute, depending upon the composition of the casting. To determine actual spindle speeds for various diameters of work refer to Figure 56.

Both Alcoa 17S-T and 11S-T3 can often be turned dry, but for best results on all aluminum some form of cutting oil should be used. Equal parts of kerosene and lard oil or equivalent make a very satisfactory cutting compound. Pure lard oil is quite satisfactory for heavy cuts and slow feeds. Alcohol and some commercial cutting compounds produce excellent finishes.

GENERAL PRECAUTIONS: Light cuts and feeds and higher speeds give best results with aluminum. The .0050 inch feed, with a depth of cut of about .020 inch, will produce excellent work, but finishing cuts should be shallower. On finishing cuts, the edge of the tool bit should be honed very sharp and smooth. Even slightly rough tool edges will leave marks on the work.

Roughing cuts will often leave a built-up "false cutting edge" of work-hardened material on the edge of the tool bit. This edge should be removed and the top of the tool bit honed before it is used for finishing cuts.

When heated, aluminum expands more than steel or brass. Care should be taken when turning between centers—the lathe should be stopped frequently to check the tightness of the work against the centers. The work should be allowed to cool before taking measurements with a caliper or micrometer. ***This is important when turning aluminum.***

MACHINING MONEL METAL AND NICKEL

Due to the toughness of monel metal and nickel, the proper tool angles, speeds and feeds are especially important. A special quality of monel metal, Type R, is available and will prove fairly easy to machine. A round-nose tool with a radius of about 1/16 inch is best with the following rake and clearance angles:

Front Clearance	13°
Side Clearance	15°
Back Rake	8°
Side Rake	14°

Tool Bit Shapes, Pages 36-37

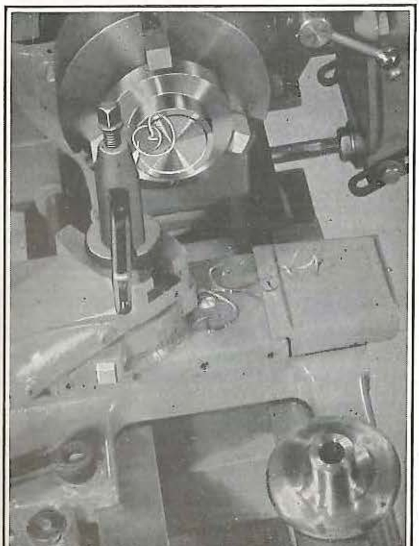


FIG. 67

Machining a monel metal hand wheel for use on a dyeing machine. A finished hand wheel is also shown. After being machined to approximate size, the hand wheel is drilled and reamed and pressed on a mandrel for finishing (see page 79).

Tool bits should be honed after grinding. A good cutting lubricant should be used for turning and drilling as well as for threading. Cutting speed should be about 100 feet per minute for cast monel metal and nickel and 120 feet per minute for rolled monel metal. Take cuts of not more than .020 to .030 inch, using the .0087 inch feed. For smooth finishing cuts use the .0050 inch feed. Deeper cuts can be taken at lower speeds but are not recommended. Tough, stringy chips are produced when machining these metals and should be kept clear of the work—use gloves or a heavy cloth in handling.

MACHINING PLASTICS

The term "plastic" applies to many types of artificially produced solids. One of the earliest plastics was celluloid — it has been followed by various other plastics, moulded and cast from such materials as phenol, urea, casein and cellulose acetate.

For machining purposes plastics can be divided into two groups: Group I includes molded Bakelite, Formica and Durez, all of which are phenol plastics moulded under heat and pressure. Group II includes all of the cast and formed plastics of various bases, sold under such trade names as Catalin, Plaskon, cast Bakelite (called Bakelite Transparent), Marblette, Joanite, Beetle, Ameroid, Pyralin, Celluloid, Tenite and Trafford.

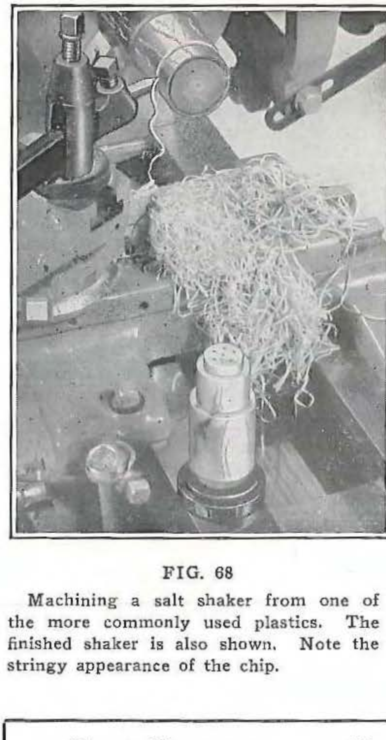


FIG. 68

Machining a salt shaker from one of the more commonly used plastics. The finished shaker is also shown. Note the stringy appearance of the chip.

MACHINING PLASTICS IN GROUP I

The machining of plastics in Group I is done best with special tool bits, and if any quantity of plastic turning is necessary, such tools will save both time and money. For a small amount of machining, high speed tool bits may be used, although it may be necessary to resharpen them several times before the job is finished. The tool should be ground to these angles:

Front Clearance ..... 8°	Back Rake .... 0°
Side Clearance ...... 12°	Side Rake .... 0°

Tool Bit Shapes, Pages 36-37

Cutting speeds of 100 to 120 feet per minute should be used. No lubricant is necessary or advisable. Take rather heavy cuts, using the .0087 inch feed.

Because of the heat generated when drilling plastics, the finished hole becomes smaller than the drill. For an exact sized hole, use an oversized drill or a drill ground slightly off center.

Apply plenty of oil when drilling and back out the drill frequently to remove chips. Special drills for Bakelite are available if any quantity of drilling is done.

MACHINING PLASTICS IN GROUP II

Regulation high speed tool bits are perfectly satisfactory for the general turning of plastics in Group II. Tool bit angles:

Front Clearance	10°	Back Rake	0° to −5°
Side Clearance	14°	Side Rake	0°

Tool Bit Shapes, Pages 36-37

For most turning the 0° angle of back rake will be satisfactory, but where there is evidence of hogging, grind a negative rake of about −5°.

The cutting speed should be around 200 feet per minute. No lubricant is necessary or advisable for turning. Light cuts of about .010 inch or less should be taken, using the .0087 inch feed, or, for a finer finish, the .0050 inch feed. If the work is being turned between centers, watch the tightness of the work against the tailstock center, as these plastics expand considerably when heated. For threading, use plenty of good cutting lubricant and reasonably high speeds.

When drilling these plastics, refer to the information listed for Group I plastics.

MACHINING FORMICA GEAR MATERIAL

Formica is a laminated plastic made of cotton duck impregnated with a phenolic resin. Tools with the following angles will be satisfactory:

Front Clearance	10°	Back Rake	16½°
Side Clearance	15°	Side Rake	10°

Tool Bit Shapes, Pages 36-37

The best cutting speeds are between 200 and 300 feet per minute with the .0050 inch feed. Depths of cuts of about .020 inch or less should be used. No lubricant is necessary. Special tool bits are advisable if any quantity is to be turned. Grind drills to an included angle of 55°.

7 - THREADING
8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
5 - HOLDING WORK
6 - DRILLING

MACHINING MICARTA

Grind tools as for Formica gear material. High speeds around 200 to 300 feet per minute are recommended, using the .0050 inch feed, and light cuts of .010 to .020 inch. Machine dry.

MACHINING TEXTOLITE

Use tools ground as for Group II plastics. A very keen edge must be maintained and special tool bits should be used if any quantity of this material is to be machined. Cutting speed should be around 200 feet per minute when using high speed tool bits and 300 feet per minute with special tool bits. The .0050 inch feed is recommended with depths of cuts of .015 to .025 inch. All machining is done dry.

MACHINING FIBER

Fiber is an extremely hard, tough material, made in the form of sheets, rods and tubes and is used extensively due to its relatively low cost. It is not commonly termed a plastic. Tools should be ground with these angles:

Front Clearance	12°	Back Rake	0°
Side Clearance	15°	Side Rake	0°

Tool Bit Shapes, Pages 36-37

Cutting speed should be about 80 feet per minute, using the .0087 inch feed and cuts of .010 to .025 inch. Keep the tool edge honed sharp with a rather broad nose at the point. Machine dry.

MACHINING HARD RUBBER

Tools should be ground to the following angles:

Front Clearance	15°	Back Rake	0°
Side Clearance	20°	Side Rake	0°

Tool Bit Shapes, Pages 36-37

If the type of hard rubber used causes hogging or tearing, make the back rake negative, about –5°.

High speed tool bits are perfectly satisfactory. Speeds of about 150 feet per minute should be used when cutting dry, but care must be taken that the work does not become too warm. The .0087 inch feed is satisfactory with depths of cuts of about .010 to .020 inch.

FINISHING AND POLISHING

Figures 69 and 70 show two steps in obtaining a finely finished surface. First, the work is filed until the tool marks disappear. Never hold the file stationary while the work is revolving. Take full-cutting strokes across the work with a slow spindle speed so that the "bite" of the file can be felt. Always file dry and keep the file perfectly clean and free from oil. Filing is also a favorite method for such jobs as rounding work corners, smoothing concave cuts, finishing off handwheels and similar jobs.

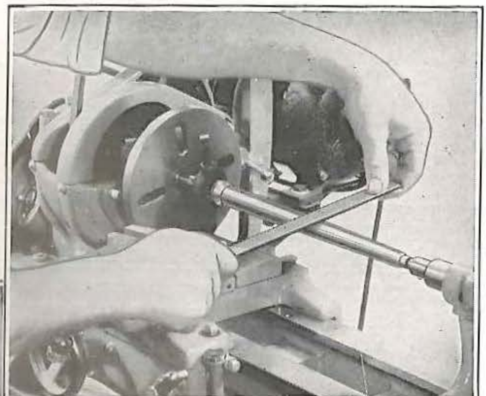


FIG. 69
Filing a taper before polishing with emery cloth.

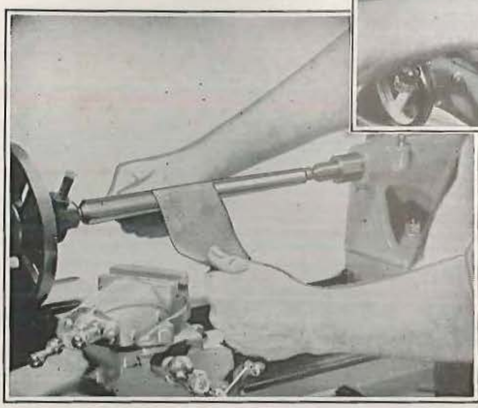


FIG. 70
Polishing steel with abrasive cloth—the emery is not held in one place but moved back and forth continually.

After filing, the work can be further polished with emery or some other abrasive cloth. See that the work is turning at a rather rapid speed. Do not hold the emery in one place — keep moving it back and forth. A few drops of oil placed on the work tends to give a better finish and eliminates scratches. Crocus cloth is also recommended for a highly polished finish.

Part 5

HOLDING THE WORK

7 - THREADING
8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
5 - HOLDING WORK
6 - DRILLING

PART 5

HOLDING THE WORK

This section describes the most common methods of holding the work in the lathe: between centers, in a chuck, on the face plate, in a collet, and on a mandrel.

BETWEEN CENTERS

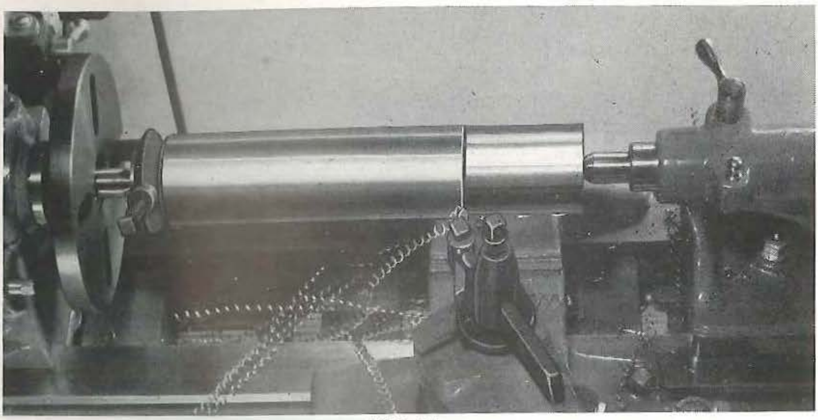


FIG. 71
Turning a piece of bar steel between centers, showing positions of the lathe dog, work and centers.

Whenever practicable, the work is held between centers. This method is usually more accurate and has the advantage of permitting removal and replacement of the work without affecting accuracy. There are two steps in mounting work between centers: locating the center points at each end of the work, and countersinking and drilling the ends to accommodate the lathe centers.

LOCATING THE CENTERS

On round work, centers are usually located with either the hermaphrodite caliper or the center head attachment for a steel scale. In the centering of square, hexagon and other regular-sided stock, lines are scribed across the ends from corner to corner. The work is then center punched at the point of intersection.

In using the hermaphrodite caliper, set the caliper to a little more than half the diameter of the work and scribe four lines as shown in Figure 72. Hold the work in a vise and center punch as accurately as possible in the center of these marks. A little chalk rubbed over the end of the work before scribing makes the marks easily seen.

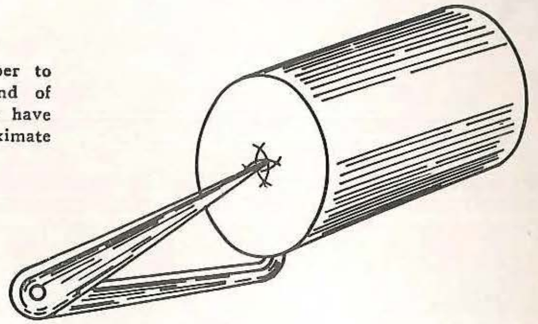


FIG. 72

Using the hermaphrodite caliper to locate center points on the end of round shafting. The four lines have been scribed to mark the approximate center position.

When the center head is used, set the center head as shown in Figure 73 and scribe two lines approximately at right angles. Use a sharp scriber and keep the lines as close to the edge of the scale as possible. Then hold the work in a vise and center punch at the intersection of the two lines.

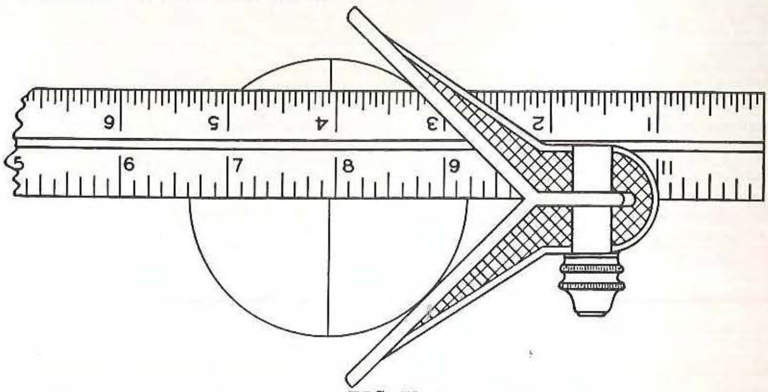


FIG. 73
Using the center head to locate work center.

If the rough stock is large enough to permit a trueing cut, the ends may be countersunk after punching. However, when the finished diameter is smaller than the stock by only a few thousandths of an inch, it is necessary to check for trueness before countersinking.

Figure 74 shows the most common method for checking trueness: Mount the work on lathe centers. Hold a piece of chalk so that it just touches the high spots of the work as it is rotated by hand. A tool bit mounted in the tool post can be used in place of chalk. Make marks close to each end, then remove the work. Hold the work in a vise and drive the two center-punched marks toward the chalk marks by striking at an angle with the center punch and then slowly bringing it back to a straight position.

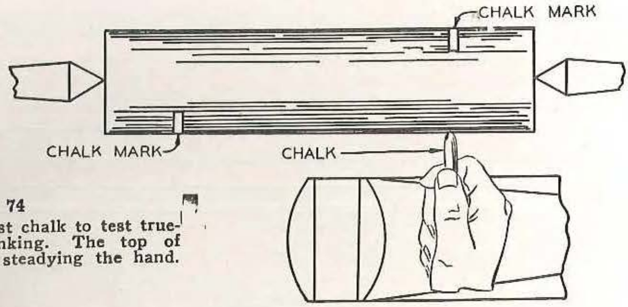


FIG. 74
Rotating work against chalk to test trueness before countersinking. The top of the compound rest is steadying the hand.

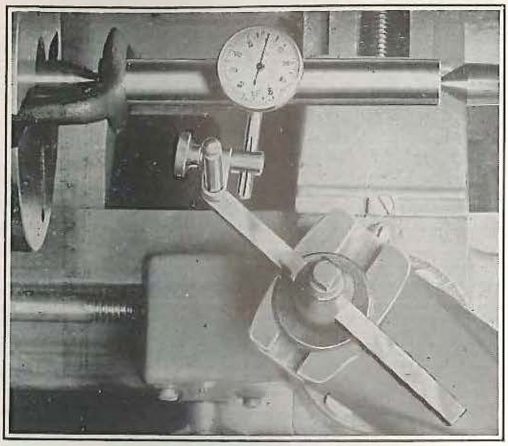


FIG. 75
Using a dial gauge to check trueness of work before countersinking.

When the center must be accurate to within one or two thousandths of an inch or when the diameter of the work is too small to permit a trueing cut, check trueness with the dial gauge before countersinking. The dial gauge is mounted in the tool post as shown in Figure 75.

COUNTERSINKING

There are three methods of countersinking the ends of the work after center punching. If a drill press is available, the work is held firmly on the table during the countersinking operation. The other two methods are illustrated in Figures 77 and 78. The size and shape of the work usually determine which method is better.

FIG. 76. 60° countersink drill for accurate centering of work to be mounted between lathe centers. The sides of the drill form an angle of 60° which exactly matches the angle of the lathe centers and provides the proper bearing surface.

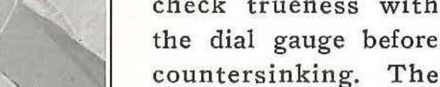


Figure 77 shows the quickest and probably the most common way to countersink centers for stock up to three inches in diameter. The left end of the work is mounted in a three-jaw universal chuck. If the work is more than ten or twelve inches long, the right end is held in position with the steady rest, relieving strain from the chuck jaws—otherwise there is no need for supporting the right end. The center punching is tested for trueness with chalk, tool bit or dial gauge. The right end can be tapped lightly with a hammer until the work runs true. Then with the spindle turning at the proper speed, the countersinking hole is bored with the 60° countersink drill held in the tailstock with a drill chuck. Do not make the centers too large.

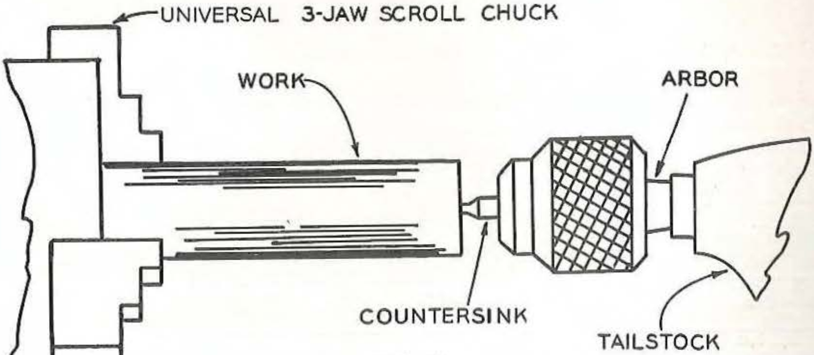


FIG. 77
The quickest way to countersink.

Another method of countersinking is illustrated in Figure 78. The countersink drill is chucked in the headstock and supports the left end of the work. The right end is supported by the tailstock. With the spindle turning at 685 or 805 R.P.M., the work is fed to the countersink drill from the tailstock and kept from turning with the left hand. Do not force the drilling or feed too fast—the advance can be felt when turning the tailstock hand wheel. If the countersink is forced and breaks off, the simplest way to remove the broken piece is to cut about one-half inch from the end of the stock. If the work cannot be shortened, heat the piece of countersink, cool slowly by covering with ashes or annealing compound, and drill out.

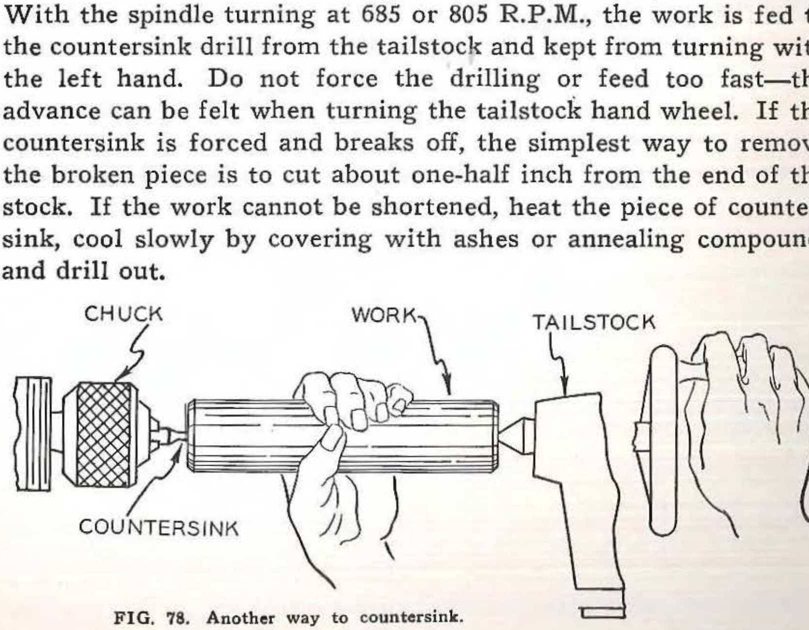


FIG. 78. Another way to countersink.

MOUNTING WORK BETWEEN CENTERS

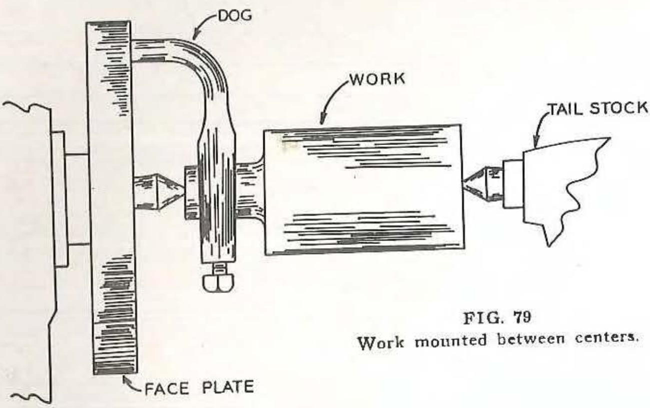


FIG. 79
Work mounted between centers.

Figure 79 shows how work is mounted between centers after the ends have been countersunk. The set of four dogs in Figure 80 handles diameters up to 1½ inches. Care must be taken in the selection of the size of the dog. The "tail" or bent portion must fit into the face plate slot without resting on the bottom of the slot. Figure 81 shows the result of making this mistake. The dog tail rests on the face plate at A and the headstock center does not "seat" properly in the countersunk hole at B.

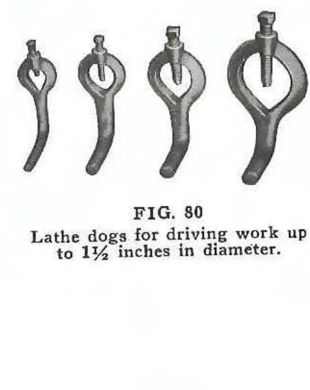


FIG. 80
Lathe dogs for driving work up to 1½ inches in diameter.

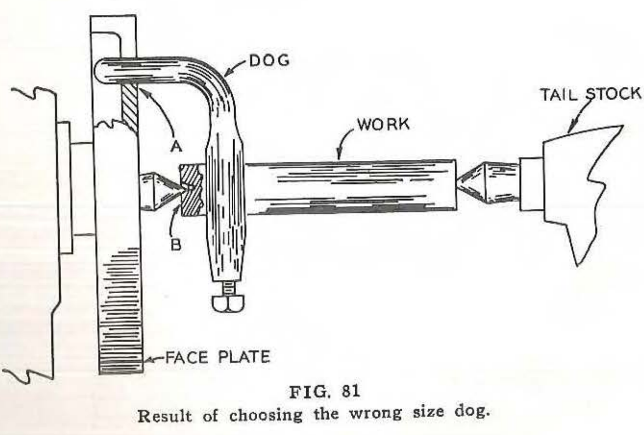


FIG. 81
Result of choosing the wrong size dog.

Work over 1½ inches in diameter can be held in the clamp type dog (Fig. 83A) or adapted to the 1½ inch dog as shown in Figure 82. The latter method requires light cuts, a rather loose tailstock center and is not recommended as standard practice. The two sizes of clamp type dogs hold stock up to 3½ inches in size and have several other advantages. They drive work of many different shapes (Fig. 83B) and can be applied if necessary without removing work already mounted between centers.

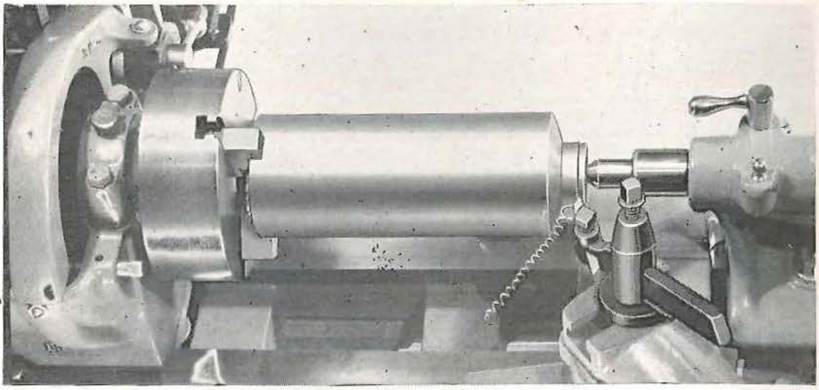


FIG. 82
Turning down a shoulder to fit the 1½" dog. This method of adapting large work to a dog is not advisable for general turning.

THE CLAMP TYPE DOG

FIG. 83A
(Left) Clamp Type Dog.

FIG. 83B
Holding rectangular work in the clamp-type dog.

THE FOUR JAW INDEPENDENT CHUCK

Much of the work to be turned or threaded on the lathe is not of a size or shape which permits mounting between centers. In such cases it is customary to mount the work on a face plate or hold it in a chuck, a device with jaws which grip the work rigidly while it is being machined.

If only one chuck is to be purchased, it should be the four-jaw independent chuck shown in Figure 84. It is easily the most versatile type of chuck. The four jaws are adjusted separately and are reversible so that work of any shape can be clamped from the inside or the outside. Some independent chucks are threaded to fit directly on the spindle nose, others are bolted to an adapter plate which fits the spindle.

Mounting work in the four-jaw chuck is largely a matter of centering. Determine the portion of the rough work that is to run true, then clamp the work as closely centered as possible, using as a guide the concentric rings on the face of the chuck. Test for trueness, marking the high spots with chalk rested against the tool post or a tool bit mounted in the tool post (see Fig. 85). The chuck jaws should be adjusted until the chalk or tool bit contacts the entire circumference of the work.

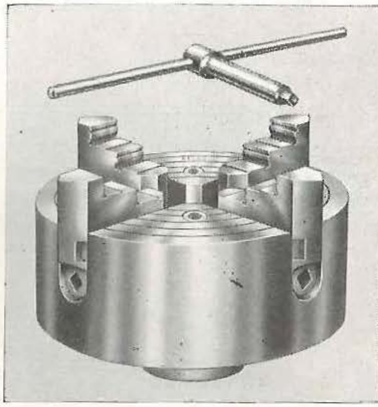
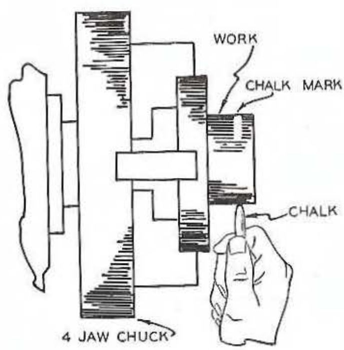


FIG. 84

The four-jaw independent chuck. The concentric rings on the face aid in adjusting the position of the work.

If especially accurate centering is desired, the trueness of the work should be checked with the tailstock center by means of an instrument called a center tester (see Fig. 86).

FIG. 85
(Right) Testing for trueness.



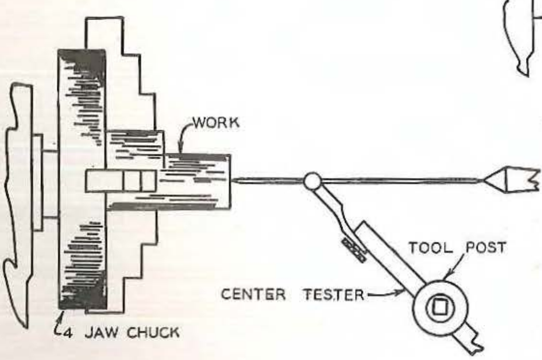


FIG. 86
Using the center tester.

THE THREE-JAW UNIVERSAL SCROLL CHUCK

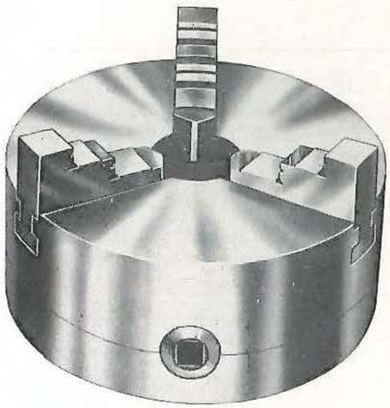


FIG. 87
The three-jaw universal scroll chuck.

The three jaws of the universal scroll chuck are self centering and adjusted by turning one screw. This construction saves time in the centering of round or hexagon work, but means that the universal chuck cannot be used for square or irregular shapes. ¾ inch stock can be fed through the headstock spindle and held in the universal chuck for turning or drilling.

Careful machining of the scroll controlling the jaws makes most universal chucks accurate to within .003 inch. For extremely accurate work, check for trueness with chalk and place shims over one of the jaws until the work runs true. To insure accuracy, the piece being machined should never be removed or reversed until all operations have been completed.

The teeth of the jaws are cut in a circular shape to mesh with the scroll threads. Consequently, the universal chuck jaws cannot be reversed. An extra set of jaws, carefully fitted to the chuck, is furnished so that large diameters can be held from the inside or outside.

To change universal chuck jaws, first remove jaws from slots by turning wrench. If jaws stick tap lightly with a piece of wood or a brass hammer. Note that each jaw and jaw slot is marked "1," "2," or "3." Place new jaws opposite slots with the same number. See that jaws, jaw slots, and scroll are free from dirt. Turn scroll until the outside start of the scroll thread is pust ready to pass the No. 1 jaw slot. Slide No. 1 jaw as far as possible into No. 1 slot. Turn scroll until jaw is engaged. Advance scroll and repeat for Nos. 2 and 3 jaws. Scroll thread must engage the first tooth in the No. 1, No. 2 and No. 3 jaws in order, and each jaw must be in its own slot.

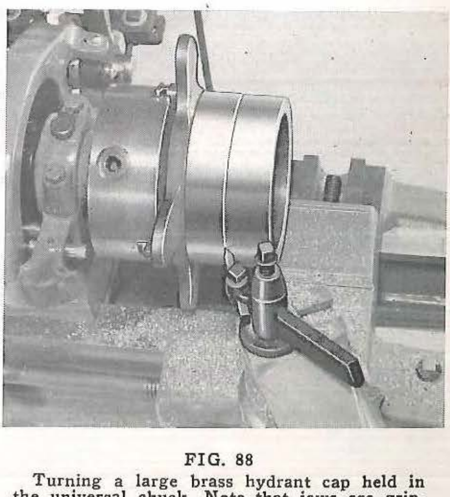


FIG. 88
Turning a large brass hydrant cap held in the universal chuck. Note that jaws are gripping the inside of the work.

THE JACOBS HEADSTOCK CHUCK

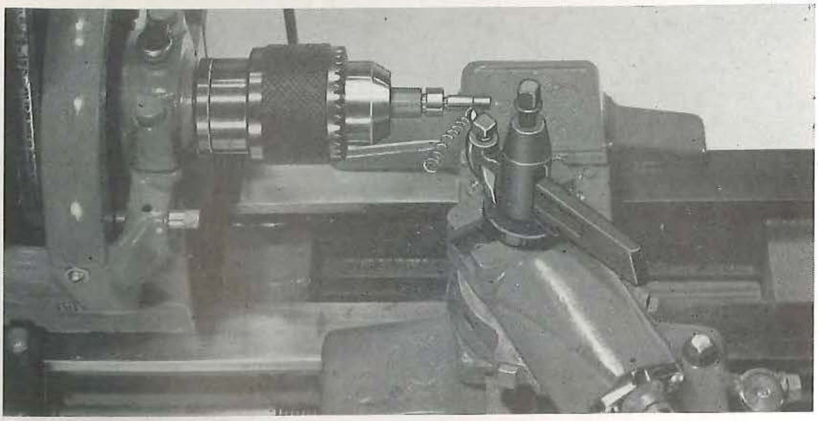


FIG. 89
Turning a small screw held in the headstock chuck.

The Jacobs headstock chuck is a most versatile chuck for holding small work in the lathe. Its accuracy is surpassed only by precision-made collets. The machinists handling any quantity of small work usually considers the headstock chuck an essential part of his equipment.

The headstock chuck is furnished in two sizes: capacities, 1/8 to 5/8 inch and 3/16 to 3/4 inch. Both are key-type chucks with a hollow construction so that work can be fed through the headstock spindle. They are threaded to fit the spindle nose of the lathe. The smaller size can also be used as a drill chuck, the inner section being tapered to fit an arbor adapter for mounting in the tailstock.

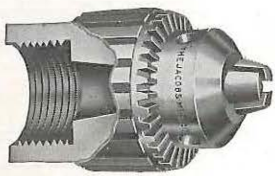


FIG. 90
The Jacobs headstock chuck showing internal taper for tailstock mounting.

When mounting work in the headstock chuck, take special care to clean between the jaws as well as the jaw surfaces. Always remove lathe center and sleeve. Never tighten the jaws until the work has been centered—keep twisting the work as the jaws are tightened.

REMOVING CHUCKS FROM THE LATHE SPINDLE

Almost every machinist has a favorite way to remove lathe chucks. The following method, illustrated in Figure 91, is simple and does not harm the chuck:

Turn the chuck until wrench hole is at the top. Lock the spindle in position by engaging the back gears without pulling out

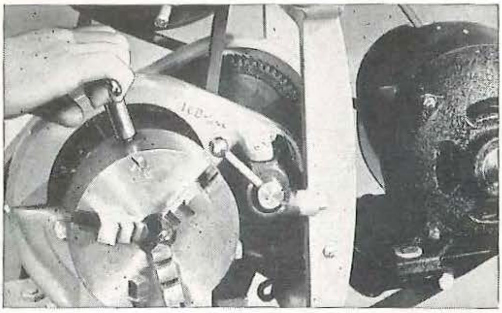


FIG. 91
A proper method for removing a chuck.

the lock pin on the face of the front spindle back gear. Put the chuck wrench in its hole and pull as shown in Figure 91. If necessary, tap the jaws with a piece of wood or a brass hammer. Do not remove the chuck carelessly. You may damage the spindle or chuck threads or drop the chuck on the bed ways.

GENERAL RULES FOR USING CHUCKS

Keep the chuck clean and do not oil excessively—a light film on all working parts is ample. Before mounting work, clean the threads in both the chuck and the lathe spindle with a piece of bent wire. Clean the face of the shoulder on the spindle nose and the back face of the chuck. Put a few drops of oil on spindle nose.

Mount the chuck carefully and not too tight, first removing the center and sleeve from the spindle. When the chuck is about 1/32 inch from the shoulder, finish with one more turning motion. The soft thud indicates a good firm seating against the shoulder. Running a chuck suddenly against the shoulder strains the spindle and makes removal difficult.

Be careful when tightening work in the chuck jaws. Too much pressure on the jaws will affect the accuracy of the chuck and may spring the work if a light piece is being turned. Try to have the jaws tighten around the more solid parts of the work. Always use the wrench which comes with the chuck. When chucking work in the universal or headstock chuck, turn the work as the jaws are tightened—an accurate "form fit" will result.

Small diameter work should not project from the chuck jaws more than four or five times its diameter—cuts should be short and light. Heavy cutting pressures will often cause small work to spring out and "ride the tool." In some instances, extra long work can be supported in the tailstock center.

Do not force a chuck to carry work larger than the diameter of the chuck body. Repeated overloading may damage the chuck.

If the jaws stick, tap lightly with a piece of wood or a brass hammer. "Sticky" jaws indicate that the chuck should be taken apart for a thorough cleaning. An old toothbrush makes an excellent chuck cleaner. Wash and brush chuck parts in a pan of kerosene. When reassembling, do not apply too much oil. Oil collects dust and chips which sooner or later clog the chuck mechanism.

Chuck jaws are carefully fitted to the chuck at the factory and are not interchangeable. When new jaws are necessary, return the complete chuck to the manufacturer. Inspect the chuck regularly to see that all parts are in good working order.

Keep the chuck protected when not in use. Dirt, dust, chips and falling tools can cause much damage.

THE FACE PLATE

Many types of lathe work which cannot be machined on centers or in a chuck are fastened to a face plate with bolts, studs or clamps. Some of the most accurate tool and die operations are handled in this way. Face plate work also includes the turning of large, flat or irregular shaped pieces such as jigs. The $8\frac{1}{2}$ inch face plate shown in Figure 92 is recommended for all types of face plate turning or boring.

FIG. 92
$8\frac{1}{2}$ inch Face Plate

The face plate should be mounted carefully in the same manner as a chuck (see page 76). For ordinary turning the work is simply bolted or clamped directly to the face plate. When maximum accuracy is desired, a light trueing cut is first taken across the face of the face plate. The face plate can be removed by tapping the slot at the outside edge with a piece of wood or a brass hammer.

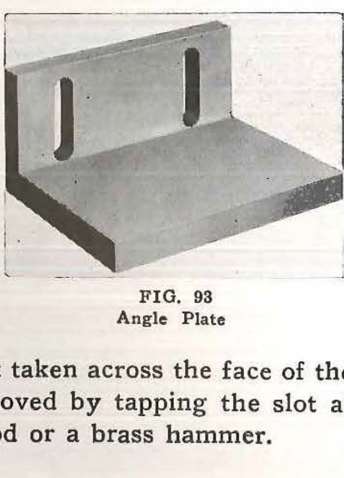


FIG. 93
Angle Plate

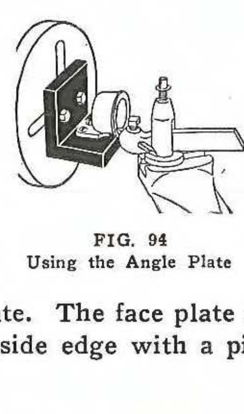


FIG. 94
Using the Angle Plate

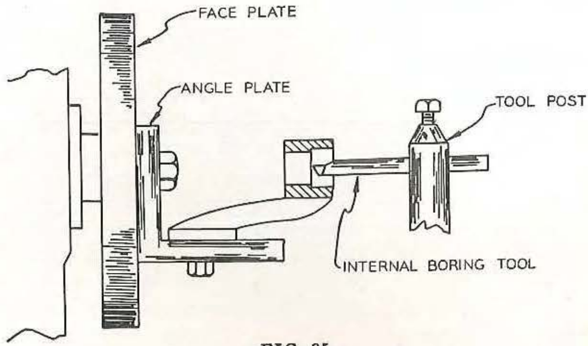


FIG. 95
How the angle plate centers a portion of an irregular piece of work.

The angle plate shown in Figure 93 is bolted to any point on the face plate for machining irregular shapes and for off-center drilling and boring. Figures 94 and 95 show two typical jobs.

Note: When heavy pieces are mounted off center, bolt a counter-balance of equal weight on the opposite edge of the face plate. The counter-balance protects lathe accuracy by equalizing pressure on the bearings and reduces excessive vibration caused by out-of-balance turning.

DRAW-IN COLLET CHUCK ATTACHMENT

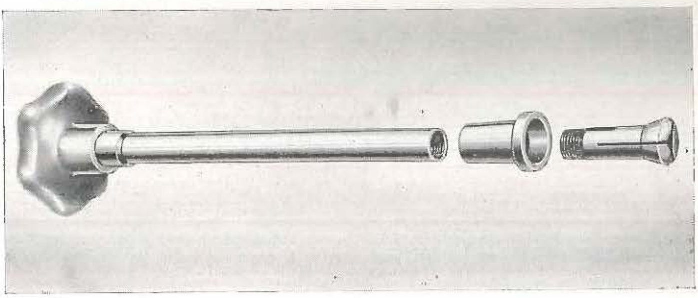


FIG. 96
Draw-in collet chuck attachment showing units in order of their assembly into the lathe headstock: draw-in spindle, tapered closing sleeve, and split holding collet.

Whenever extreme accuracy is required on small diameters, the draw-in collet chuck attachment is the logical method of chucking. When equipped with the collet assembly and the various size collets, the lathe handles the most exacting work in tool rooms and tool and die shops. Some typical collet work: precision tools, instruments, gauges and small production parts.

The collet attachment, as shown in Figure 96, includes a hollow draw-in spindle which extends through the lathe headstock spindle, a tapered holding sleeve and the split holding collets. The collets are released or tightened on the work by turning the hand wheel (see Fig. 97). Work can be fed through the lathe headstock spindle. The individual collets are furnished in all 32nds between 1/32 and ½ inch. Special sizes and shapes including metric diameters are also availabie.

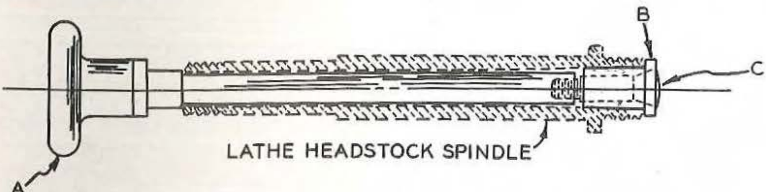


FIG. 97
Cross section showing draw-in collet assembly in lathe headstock. Turning handle A pulls collet C into sleeve B, tightening collet on work.

There are two important rules for the use of the draw-in collet chuck attachment—first, absolute cleanliness and, second, selection of the proper size collet. The collets, tapered sleeve, and the inside of the spindle nose *must* be wiped clean and dry. A collet must never be used to hold work which is more than .005 inch larger or smaller than the rated diameter of the collet. A collet attachment is the most accurate type of precision chucking and must be treated with greatest care.

MOUNTING WORK ON THE MANDREL OR ARBOR

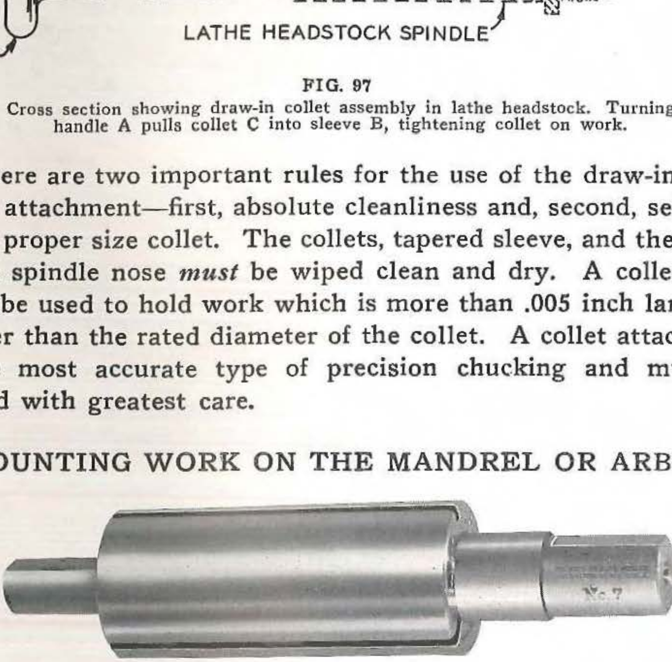


FIG. 98
Expanding mandrel.

Figure 98 shows a commercial type of expanding mandrel or arbor designed to provide work centers for facing or turning the outside diameter of work that is nearly finished or difficult to mount in a chuck. The machining of pulleys and gears is a typical mandrel job.

The mandrel consists of the ground and hardened body, tapered through its entire length, and a cast iron expansion sleeve with an internal taper to fit the body. Forcing the sleeve on the mandrel causes it to expand and hold the work firmly in position.

A mandrel, such as the one used in Figure 99, is often made on the lathe for any special piece of work. These mandrels are

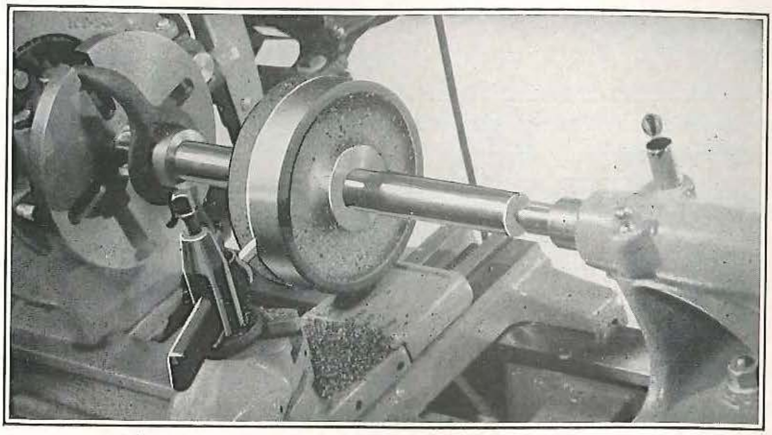


FIG. 99
Turning a cast iron pulley on a mandrel

turned from round bar machine steel stock and the ends case-hardened if possible. Cast iron, with hardened tool steel plugs for the ends, is often used in making a mandrel for large work. The mandrel should be tapered about .006 or .008 inch per foot and polished or ground. When finished, the mandrel diameter should be a force fit for the hole in the work and the tailstock end should be .003 or .004 inch smaller. It is recommended that the mandrel be turned undersized at both ends for about 3/4 inch to prevent damage.

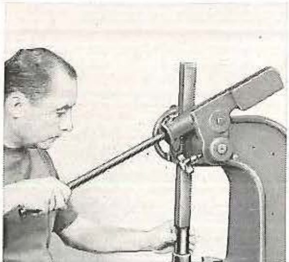


FIG. 100
Pressing mandrel on bushing before machining.

A mandrel is a precision tool for accurate work and must be handled with care. The ends are centered and countersunk exactly like other work. To make removal easier, put a drop or two of oil on the portion of the mandrel which will grip the work. Never drive a mandrel with a steel hammer without protecting the end. The best tool for forcing a mandrel in or out of the work is an arbor press, or mandrel press (Fig. 100). Be sure the work is started perfectly straight and on the entering end of the mandrel. Do not allow the tailstock center to become too hot during the machining operation.

Part 6

DRILLING AND BORING

PART 6

DRILLING AND BORING

DRILLING

Lathe drilling can be handled in two ways. Figure 101 shows the work revolving while the drill is held stationary in the tailstock. This method results in a straighter hole and insures greater accuracy than any other method. The second method of drilling is shown in Figures 114 and 115—the work is held rigid while the drill turns in the headstock. The shop with considerable drilling, reaming and tapping will find a drill press a profitable investment, because the lathe requires special attachments for production drilling.

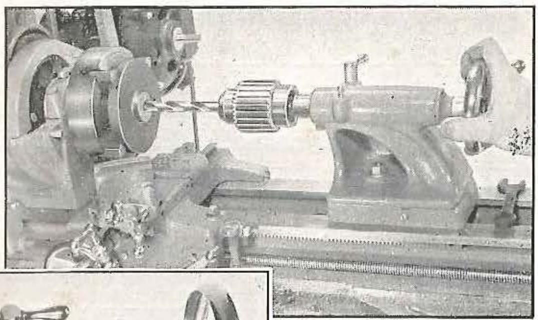


FIG. 101

Drilling with the work revolving in the headstock. This type of set-up insures maximum accuracy. Note use of graduated tailstock ram to indicate depth.

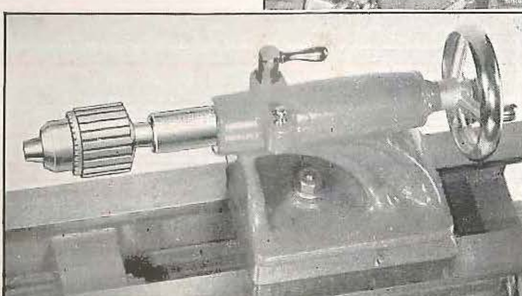


FIG. 102

Jacobs drill chuck held in the tailstock on an arbor. These chucks hold work up to ½ inch in diameter and can be used in headstock or tailstock. Follow general rules for using chucks—Part 5.

TWIST DRILLS

After the drill point is dulled for the first time, its effectiveness depends entirely upon how it is reground. For clean, accurate drilling, the operator must know how to resharpen the drill properly. Figures 103 and 104 give the usual shop terms used in drill grinding. The cone-shaped surface at the end of the drill is called the "point," and the edge at the extreme tip end is the "dead center."

THE TWIST DRILL

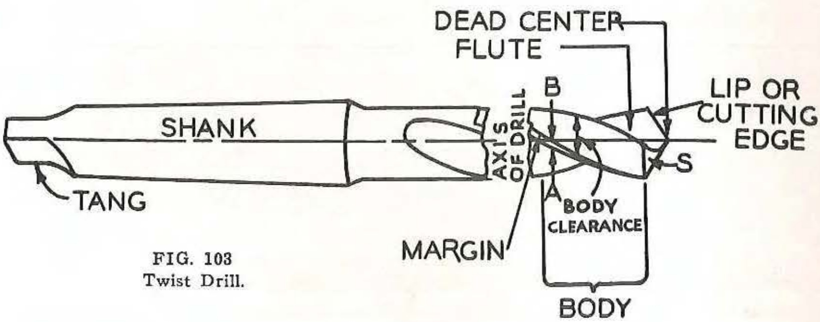


FIG. 103
Twist Drill.

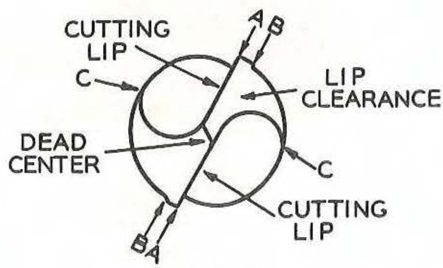


FIG. 104
Point of Twist Drill—End View.

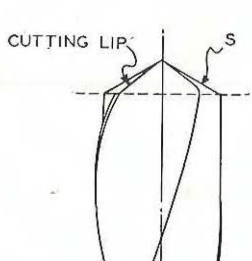


FIG. 105A
Drill without lip clearance. The cutting lip and heel, S, are in the same plane.

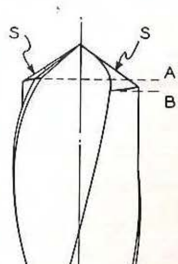


FIG. 105B
Drill with proper lip clearance. Heel line, B, is lower than cutting lip line, A. Distance between A and B measures amount of lip clearance.

Basically, a drill cuts metal exactly like a lathe tool. In order to penetrate the work, the cutting edge must have the correct cutting angle and "lip clearance" at the center of the drill (Fig. 104). Figure 105B shows how the "heel," the part directly back of the cutting edge, must be ground away. The word "heel," when used in this sense, includes the entire surface back of the cutting edge, not the circumference only.

FIG. 106. THE PROPERLY GROUND DRILL

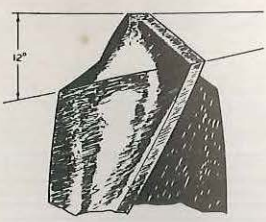


FIG. 106A
Drill point showing proper lip clearance angles at the circumference of the drill.

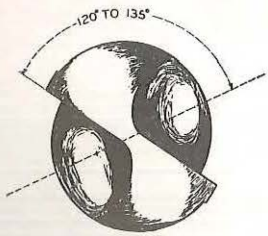


FIG. 106B
End view of drill point showing proper angle between point and lip.

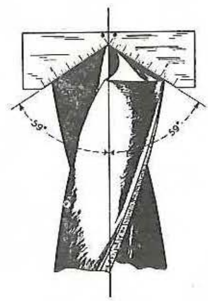
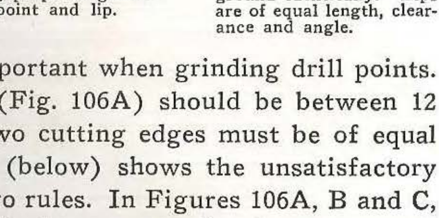


FIG. 106C
Drill point with lips ground identically. Lips are of equal length, clearance and angle.

Two rules are especially important when grinding drill points. First, the lip clearance angle (Fig. 106A) should be between 12 and 15 degrees. Second, the two cutting edges must be of equal length and angle. Figure 107 (below) shows the unsatisfactory results of disregarding these two rules. In Figures 106A, B and C, the properly ground drill point is shown—note lip clearance, angle between point and lip, and the identical lips. Refer to these drawings while the drill is being ground—they will aid in grinding drills which will cut true-sized holes with a minimum of drill wear. The angle of 59° given in Figure 106C is satisfactory for the general drilling of steel, iron and brass—larger angles are used frequently in production work and on softer metals. Both lip angle and lip length should be checked with a drill gauge (Fig. 108).

FIG. 107

Common mistakes of drill grinding. Note that in each case the resulting hole must be oversize. (Left) Lips of unequal angle and unequal length. Drill point actually travels AROUND the center of the hole. (Center) Lips of unequal angle. The right lip is doing all the work. (Right) Lips of equal angle, but unequal length, causing excessive wear on right lip.



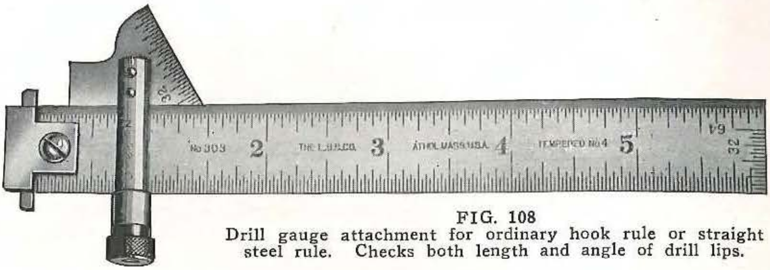
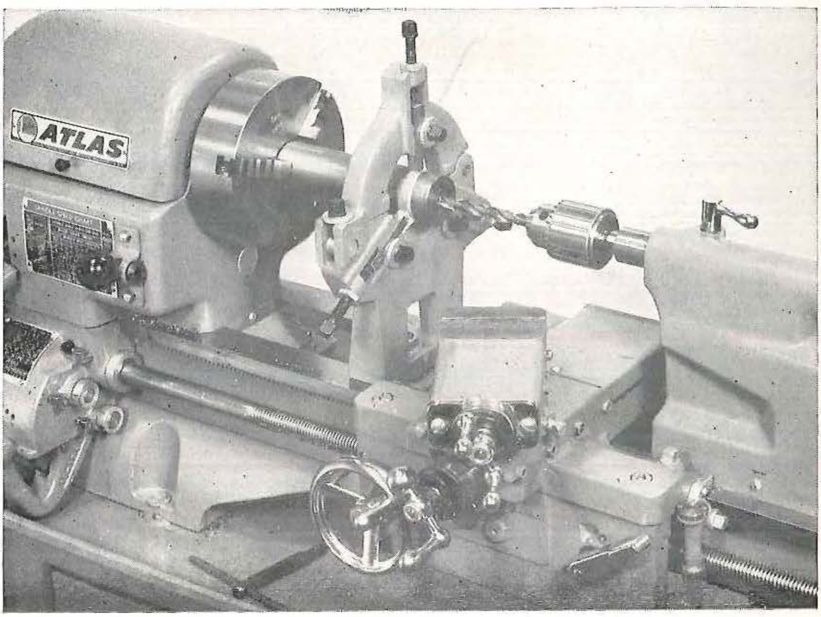


FIG. 108
Drill gauge attachment for ordinary hook rule or straight steel rule. Checks both length and angle of drill lips.



DRILLING SPEEDS

When high speed drills are used, drilling speeds in surface feet per minute for the various metals are the same as the speeds for general turning given in Part 4. The upper portion of the

Table of Cutting Speeds, page 47, will assist in the selection of the proper drilling speed. The figures in the column below "Diameter of Work" can be considered as drill sizes. Belt positions are determined by locating the proper spindle speed in Figure 56 and then referring to Figure 55, page 45. The speed should be reduced one-half with carbon drills.

Make sure that the drill runs true when starting—it may be necessary to countersink the work (see page 69). **Small drills should be fed into the work carefully since they are designed to be run at very high speeds. Avoid too high a speed, especially with the larger drills—Figure 110 shows how an excessive speed wears off drill corners. Too high a speed also draws the temper of the drill and may even burn or break the drill tip.**

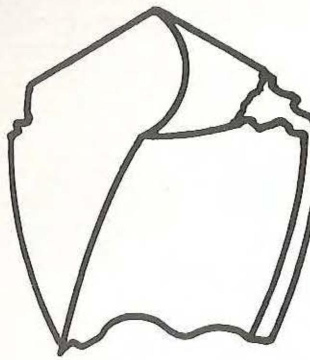


FIG. 110
Drill with edges burned by excessive heat from high speeds or drilling hard material.

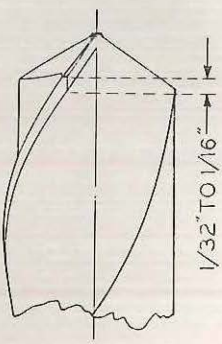


FIG. 111
Drill point for drilling brass.

NOTE: When drilling brass, aluminum, lead and other soft materials which cause the tool to "hog in," reduce the rake angle of the cutting edge by grinding as shown at the left. This reduced rake angle is also desirable when drilling very hard materials because it lessens the strain on the drill. This change makes drilling easier and smoother and results in a more accurate drilled hole.

LUBRICATION

A cutting compound is essential when drilling practically any metal. The following compounds will give best results:

Hard, tough steels	Turpentine or kerosene
Softer steels	Lard oil or equivalent
Aluminum and other soft alloys	Kerosene
Brass	Drill dry or use paraffin oil
Die castings	Drill dry or use kerosene
Cast iron	Drill dry

7 - THREADING
8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

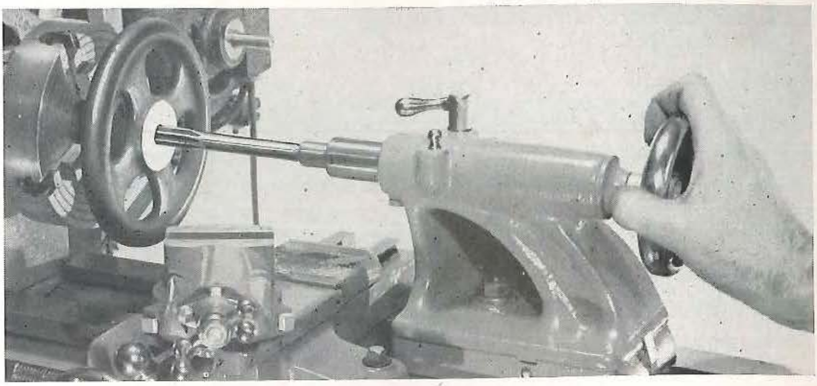


FIG. 112. Reaming a cast iron handwheel.

REAMING

When a hole must be accurate to within .002 inch or less, it is first drilled a few thousandths of an inch undersize and then hand-reamed or reamed on the lathe to the finish-diameter. Figure 112 shows a typical reaming job on the lathe. For best results, follow the same rules in reaming as in drilling and general turning. Use slow speeds, feed in evenly and be sure there are no burrs on the reamer teeth. The type of reamer shown in Figure 113 is generally used in the lathe.

FIG. 113 Reamer with Morse Taper shank for tailstock.

A reaming allowance between .010 and 1/64 inch is usually sufficient for machine-reaming holes with diameters of 1 inch or less—an allowance of 1/64 to 1/32 inch is recommended for machine-reaming holes between 1 and 2 inches in diameter. .003 to .005 inch is usually allowed for hand reaming operations.

CROTCH CENTER AND DRILL PAD

The crotch center and drill pad are two important attachments recommended for drilling work that cannot be chucked in the lathe. Both are mounted in the tailstock ram as shown in Figures 114 and 115.

The drill pad serves as a table for flat or square work and is especially valuable for drilling large holes when a drill press is not available. The crotch center automatically centers round work for cross drilling. The work is held in the left hand and advanced against the drill by turning the tailstock handwheel. The left

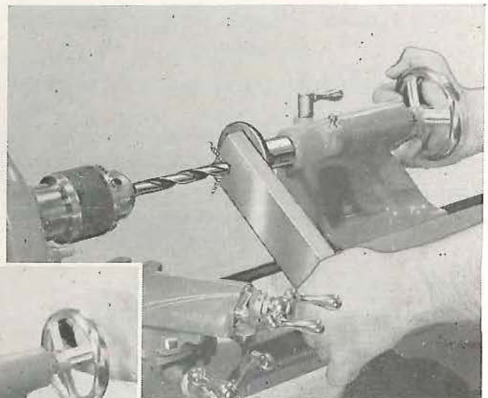


FIG. 114

Using the drill pad to support flat work while drilling a large hole.

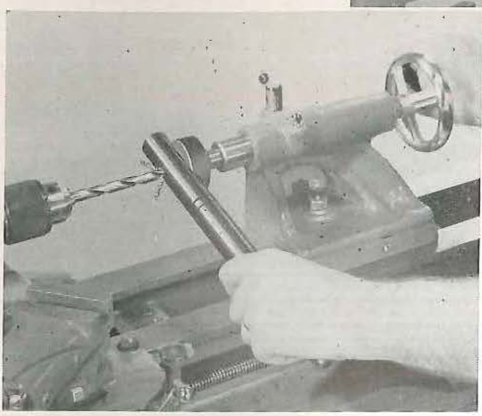


FIG. 115

Cross drilling a round shaft centered in the v-slot of the crotch center.

hand and work can be rested on a piece of wood to steady the work and protect the bed way as shown in Figure 114 above.

DRILL SETS

Every shop requires an assortment of the more commonly used drills. The sizes necessary depend upon the amount and character of the operations ordinarily performed. There is a marked trend toward the high speed drill in preference to the carbon drill. The drill set in Figure 116 includes high speed or carbon drills between 1/16 and 1/2 inch by 64ths and is adequate

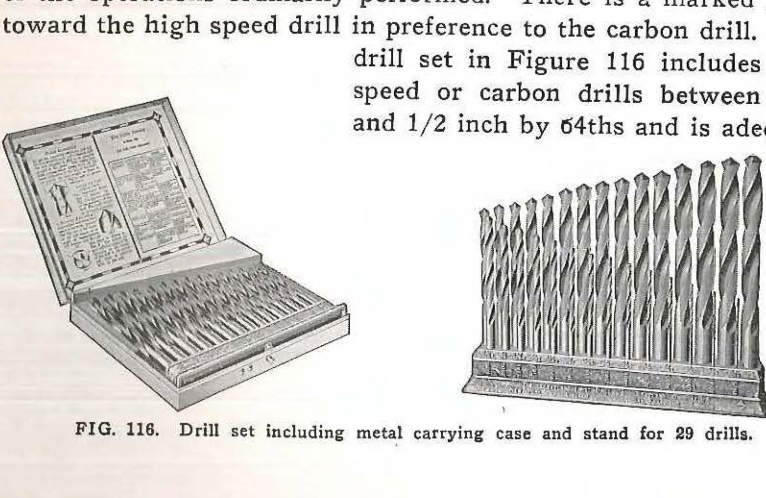


FIG. 116. Drill set including metal carrying case and stand for 29 drills.

for most small shops. The metal stand has a hole for each drill with the drill size and its decimal equivalent clearly marked. The drills can also be purchased separately.

The tables in Part 10 of this Manual give the decimal equivalents of the numbered and lettered drills and the proper drills for use with various sizes of taps. Drills in metric sizes are also available.

BORING OPERATIONS

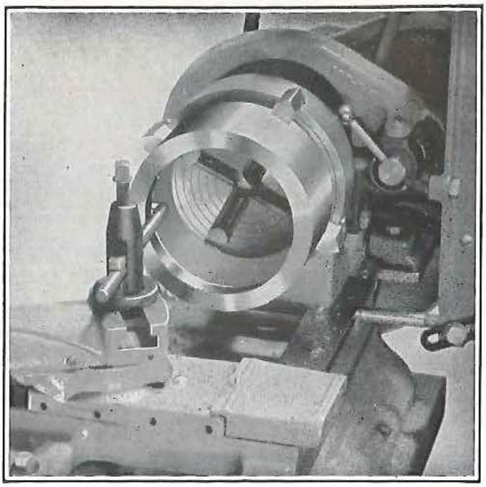


FIG. 117
Boring the inside of a large steel bushing. Note high-speed boring tool mounted directly in tool post for maximum rigidity.

Boring operations require only slightly different tools and methods than those for external turning. The big problem is that of tool rigidity, because most internal tools project considerably from their support. Figure 117 shows a typical boring operation.

There are several types of boring tools and mounting methods. The tools shown in Figure 119 are mounted directly in the tool post. The solid one-piece construction adds to rigidity by eliminating the extra joint which would result if the tool were held in a separate holder. In addition to five internal tools, this set includes a small v-block, two blocks for height spacing, and two ⅜-inch heavy-duty external tools for use directly in the tool post.

TOOL SHAPES FOR BORING

Although boring tool angles in relation to the work are somewhat different than those of an external tool, the terms in Figures 118A and 118B are fairly standard and will aid in proper tool grinding.

BORING TOOLS

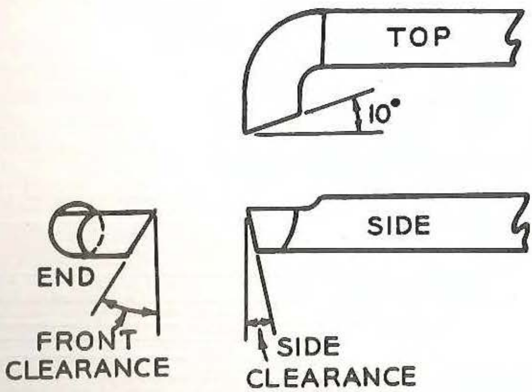


FIG. 118A

This drawing shows construction and angles of the boring tools shown in Figure 119 below. These angles make this type of tool extremely practical for all-around boring.

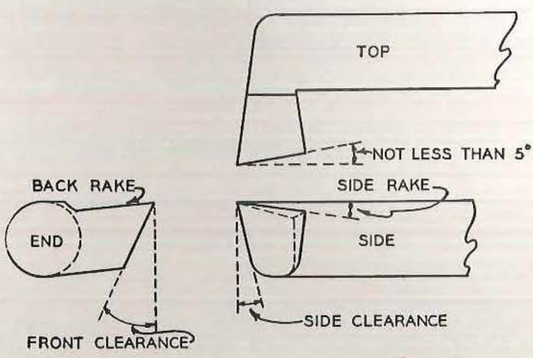


FIG. 118B

The boring tool angles shown in Figure 118A often resemble angles like these after continued grinding. New angles, such as the rake angles shown here, may be ground as desired for special jobs.

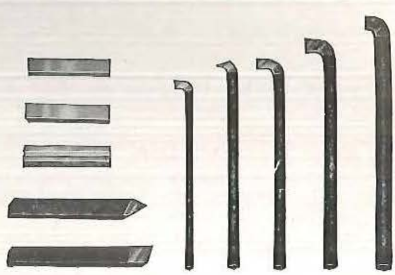


FIG. 119

Set of tools for use directly in the tool post. This set includes four boring tools, one inside threading tool, two spacers, v-block, ⅜ inch high-speed threading tool, and ⅜ inch high-speed turning tool.

TOOL SHAPES FOR BORING (Continued)

Front clearance must be increased in order to prevent the heel from rubbing on the surface of the cut. The exact amount of front clearance depends upon the size of the hole being bored. Figure 120 shows how a front clearance angle can be too small for one hole but satisfactory for a larger hole.

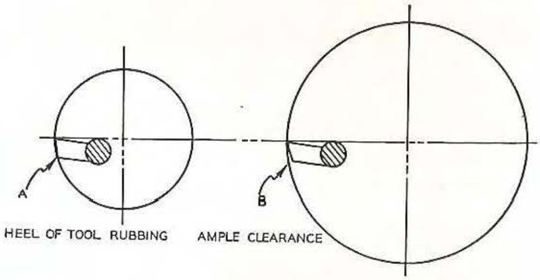


FIG. 120
This drawing shows how a certain angle of front clearance may be too small for one hole but satisfactory for a larger hole. At "A" the heel of the tool is rubbing. At "B" in the larger hole there is ample clearance.

Boring also requires smaller rake angles, and finer cuts and feeds, due to two reasons: (1) the strength of the tool edge has been reduced by the larger clearance and (2) the boring tool has a tendency to twist and "spring." The tools shown in Figures 118A and 118B are excellent for most boring operations.

BORING TOOL ANGLES

Front Clearance: *Depends upon size of hole. Never less than 10°; up to 20° for very small holes.*
Side Clearance: *Same as for external tools.*
Back and Side Rake: *About half of external angles—in some cases, less than half.*

SETTING THE BORING TOOL

With the round tool shank parallel to the lathe center line, set the boring tool into the work with the shank below the center line. Then by putting the cutting edge on exact center, the correct amount of back rake is provided. The general rules for the use of external tools apply to boring tools, except that rake angles depend a great deal on how the boring tool is set in the holder. For maximum rigidity, choose the largest possible boring tool.

BORING HINTS

When enlarging an out-of-round hole, take several small cuts rather than one heavy cut. This gradual process avoids spring in the tool—the final finish cut should be continuous.

After the last finish cut it is common practice to shift the reversing lever at the end of the forward cut and take a last fine shiving cut with the tool coming out of the work. This last cut is taken without resetting or disturbing the tool and avoids a slightly undersized hole which might otherwise result from tool spring.

Use the .0035 or .0050 inch feed and take shallow cuts.

BORING WITH THE WORK HELD STATIONARY

Figure 121 shows a method of taking long or heavy boring cuts. The work is clamped rigidly in a boring table and vise on the carriage, and a boring tool bit is set into an arbor mounted between centers. The tool bit is reset after each cut. Larger rake angles and heavier feeds and cuts may be used, since the tool has less spring.

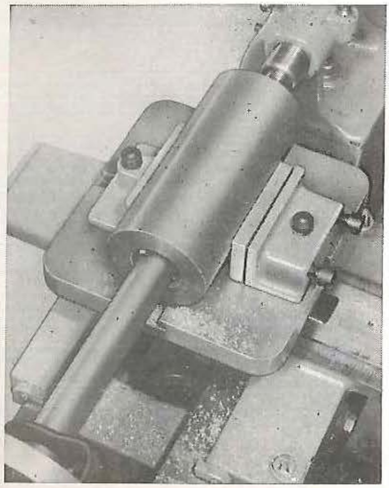


FIG. 121
Boring a small grinder spindle bearing housing held in the boring table and vise.

Part 7

THREAD CUTTING

THREAD CUTTING INFORMATION

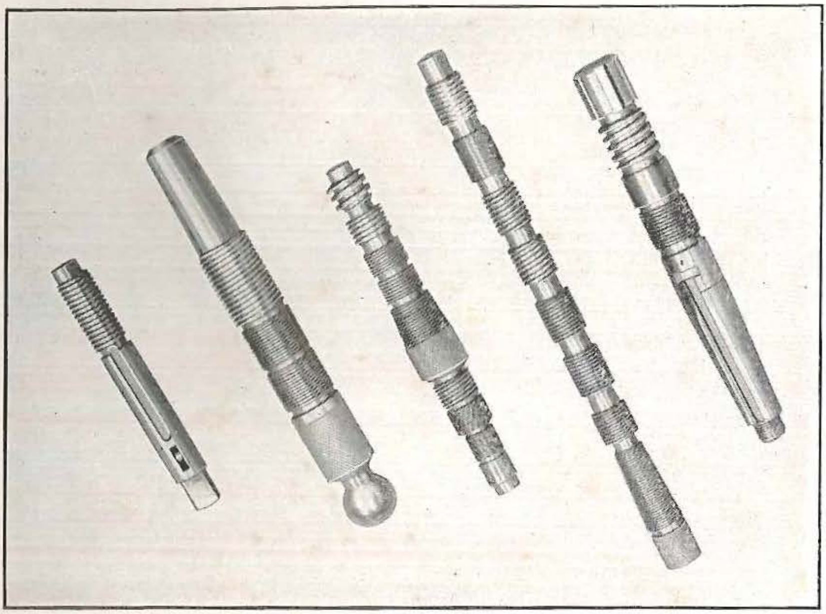


FIG. 123. A few of the threads that can be cut on the lathe

No phase of lathe operation is more interesting or profitable than the cutting of screws and threads; and no operation requires more care and study. The thread cutting range of the modern lathe is practically unlimited —a few sample threads are shown in Fig. 123.

Every lathe comes equipped for cutting threads in the following standards: National Coarse (U.S.S.), National Fine (S.A.E.), Acme, Square, and Whitworth.

THREAD CUTTING TERMS

MAJOR DIAMETER—The largest diameter of the thread of either the screw or the nut.

MINOR DIAMETER—The smallest diameter of the thread of either the screw or the 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. In Figure 124 the lines representing the diameter "PD," are located so as to make spaces "aa" and "bb" equal. On a 60° Vee-type thread and on National Form threads, the pitch diameter is simply the major diameter less the depth of the thread.

DEPTH OF THREAD—One-half the difference between the major diameter and the minor diameter. In lathe work, the

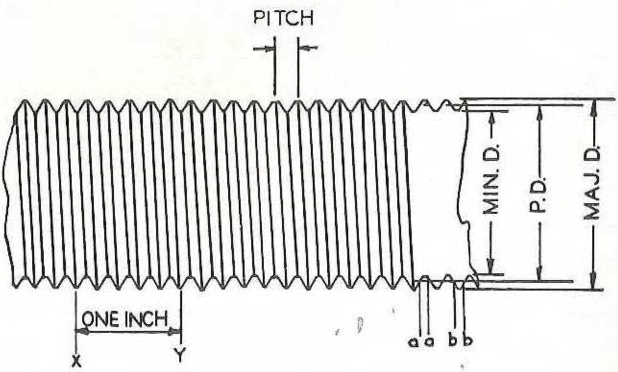


FIG. 124 Thread Cutting Terms.

DOUBLE DEPTH OF THREAD, which is the difference between the major and minor diameters, is a quite common term. Thus, knowing the major diameter required, subtracting from it the double depth of thread for the required pitch, gives the minor diameter. Information on double depths of National Form threads for different pitches will be found on page 110.

PITCH—The distance from a point on a screw thread to a corresponding point on the next thread, measured parellel to the axis (see Fig. 124).

$$p = \text{Pitch of thread in inches} = \frac{1}{\text{Number of threads per inch}}$$

THREADS PER INCH—The number of complete threads in the space of one inch. In Figure 124, the distance between points X and Y represents one inch, and there are five threads per inch.

$$n = \text{Number of threads per inch} = \frac{1}{\text{pitch}}$$

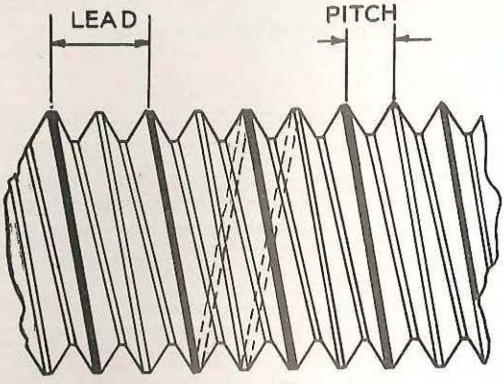


FIG. 125
Double Thread Screw. The lead is double the pitch.

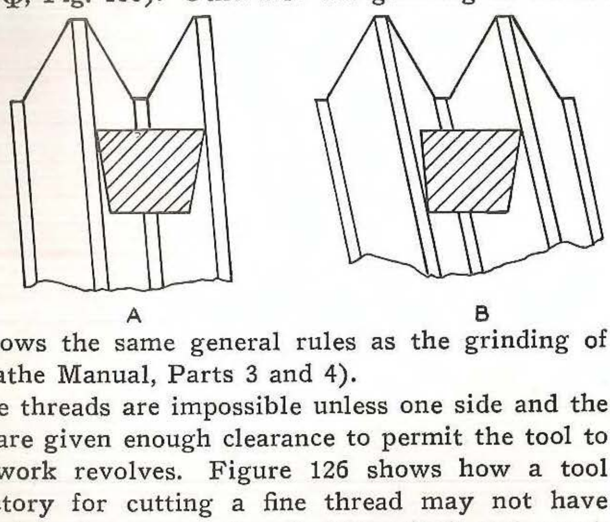
LEAD — The distance a screw thread advances axially in one turn. On a single thread screw, the lead and the pitch are identical; on a double thread screw, the lead is twice the pitch; on a triple thread screw the lead is three times the pitch, etc.

Figure 125 shows a double thread screw. There are two separate grooves or helices around the screw, each of which advances twice the pitch in a single turn. If the pitch of this screw is 1/8 inch, the lead is 1/4 inch.

THREAD CUTTING TOOLS

Thread cutting tools must be ground to the form of thread desired. Clearance must be increased because of the rapid advance of the tool, (See Φ, Fig. 150). Otherwise the grinding of thread

FIG. 126
"A" shows tool with sufficient clearance. When thread pitch is increased, as at "B," same tool has inadequate clearance.



cutting tools follows the same general rules as the grinding of external tools (Lathe Manual, Parts 3 and 4).

Clean, accurate threads are impossible unless one side and the front of the tool are given enough clearance to permit the tool to advance as the work revolves. Figure 126 shows how a tool which is satisfactory for cutting a fine thread may not have enough clearance to cut a coarse thread. "Hogging" and rough threads are usually the result of insufficient clearance.

Thread tools are ground nearly flat across the top. When the tool is fed into the work at an angle, as with National Form threads, the tool should have a few degrees of side rake. When the tool is fed into the work at right angles, as with square threads, it should have a small amount of back rake.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

CUTTING 60° TYPE THREADS

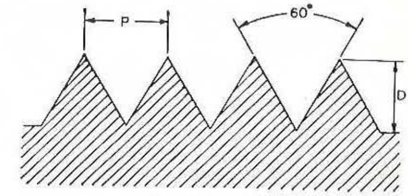


FIG. 127.

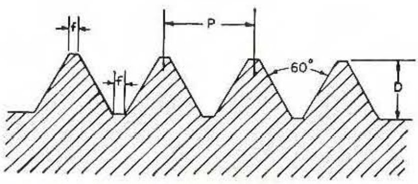


FIG. 128. American National or National Form Thread and Formulas.

60 degree type threads include the 60° Vee thread (Fig. 127) and the American National Screw Thread, (Fig. 128). The 60° Vee thread is cut very seldom, usually for small screws on which the flat

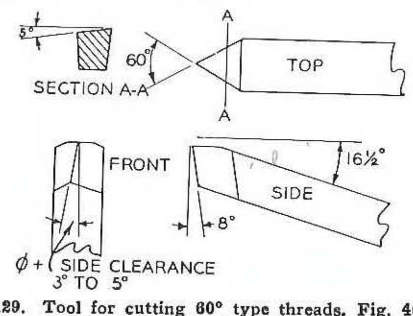


FIG. 129. Tool for cutting 60° type threads. Fig. 40 explains how angle Φ must be determined.

at the top and bottom of the National Form thread would be so small that it approaches the Vee form. Small taps usually produce Vee-type threads, and the resulting holes accommodate the standard National Form Screws.

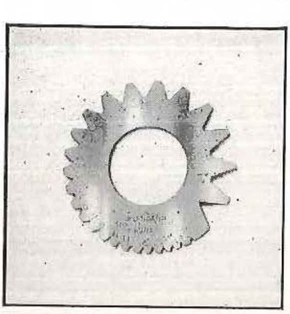


FIG. 130. N. F. Thread Gauge.

The American National Screw threads, (National Fine and National Coarse) are practically standard for automotive and machine shop work in the United States. These threads are 60° Vee threads with the points cut off so

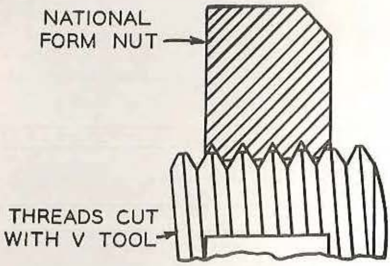


FIG. 131
The National Form nut fits the screw cut with a 60° Vee tool.

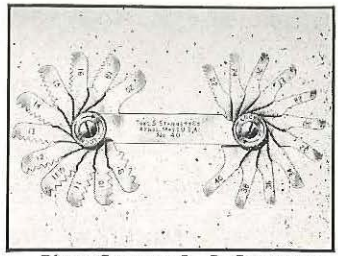


Photo Courtesy L. S. Starrett Co.
FIG. 132
National Screw Pitch Gauge

that the depth is 75% of the depth of a Vee thread of the same pitch.

Figure 129 shows a tool bit ground for cutting sharp pointed Vee threads. This tool will also cut an exact National Form Screw thread when the point is ground flat to fit the proper slot in the National Form thread gauge (Fig. 130). Generally, however, the tool is left sharp pointed and the thread is cut with the regulation Vee bottom, but the top is left with the proper amount of flat. Figure 131 shows how a screw cut in this manner fits a National Form nut. Only when desiring absolute maximum strength is the tool ground to the exact National Form.

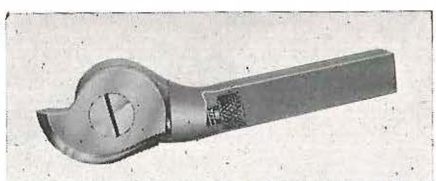


FIG. 133. Threading Tool.

The screw pitch gauge shown in Figure 132 is used to determine the exact pitch of a V-thread screw or nut. This gauge has thirty separate leaves with pitches between 4 and 42 per inch.

THREADING TOOL

The threading tool shown in Figure 133 has become extremely popular because it can be used to cut all pitches of National Form threads with the slight difference in form mentioned above.

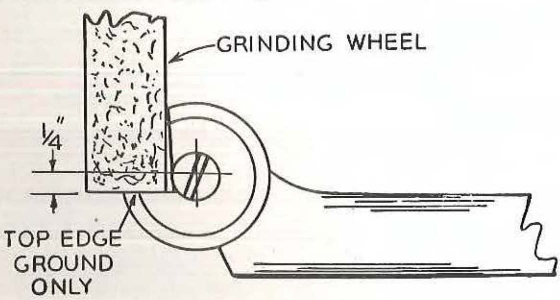


FIG. 134
Proper method of grinding the threading tool shown in Figure 133. The side faces are never ground.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

The sides of this tool are ready ground to an included angle of approximately 65 degrees. The extra 5° compensates for rake angle and the grinding of the tool—a perfect 60° thread is produced when the tool is set into the work properly (see page 99). The form of this tool also provides ample clearance for even the coarsest threads. The tool is resharpened by simply grinding the top edge, adjusting the tool as it wears.

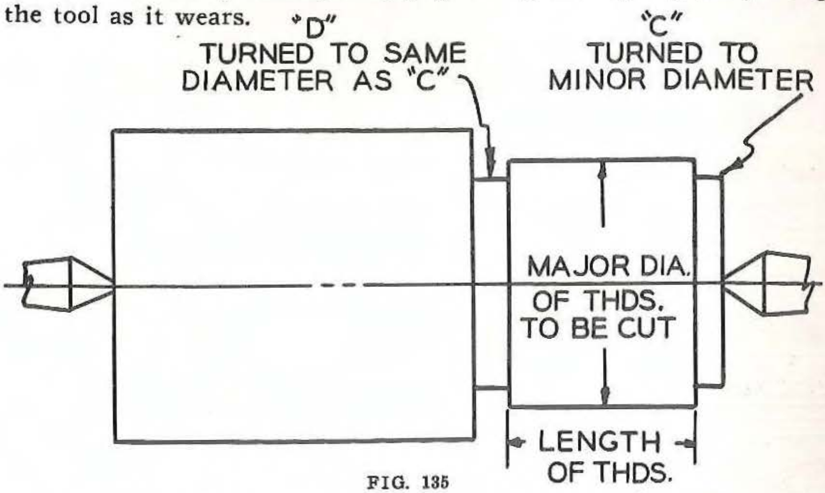


FIG. 135

PREPARING THE WORK FOR AN EXTERNAL 60° NATIONAL FORM THREAD

The work to be threaded is first turned to the exact major diameter of the desired thread. The beginner often finds it helpful to turn the grooves C and D (Fig. 135) to the exact minor diameter. The size of the minor diameter depends upon the form of the threading tool. Theoretically, if the thread were to be cut with a sharp pointed 60° tool, the minor diameter would be equal to the major diameter less the Vee-Form Double Depth of Thread (Table I, page 110) or the major diameter less 1.732 $\times$ pitch. In common practice, however, a tool bit is formed especially for a National Form thread, and the correct minor

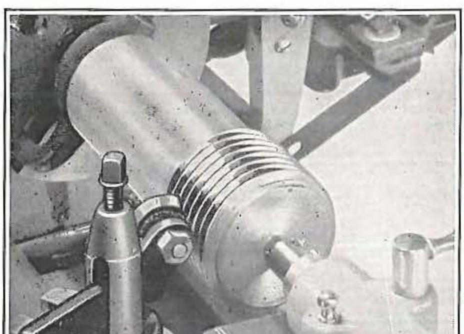


FIG. 136

Correct setting of tool and compound rest when cutting a 60° right hand thread.

diameter is listed in Table II or Table III, pages 111 and 112 (major diameter less 1.299 $\times$ pitch).

Groove C permits accurate measurement with a micrometer of the bottom of the thread. When the tool point has cut to the depth of the groove C, the thread has been finished. Groove D permits the work to revolve freely at the end of each cut. ***As soon as the beginner has become a little more familiar with threading practice, these grooves can be omitted.***

FIG. 137. Center Gauge

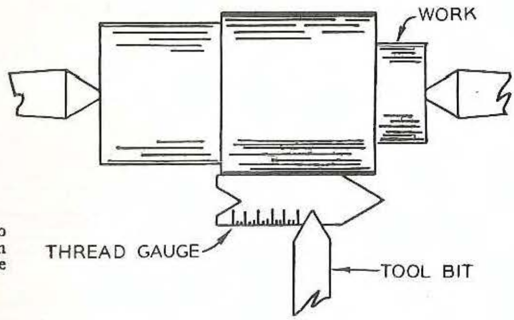


FIG. 138 (Right)
Using the center gauge to set the threading tool at an exact right angle to the work.

SETTING THE 60° THREADING TOOL

After the work has been properly prepared for threading, set the compound rest at the 29° angle shown in Figure 136. Mount the tool holder in the tool post so that the point of the tool is exactly on the lathe center line—tighten tool post screw just enough to hold the tool holder. Then use a center or thread gauge **(Fig 137)** to set the tool point at an exact right angle to the work as shown in Figure 138. Tap lightly on the back of the tool holder when bringing it into position. A piece of white paper placed under the center gauge will aid in checking the fit of the tool in the Vee of the gauge. With the tool point at an exact right angle to the work, recheck the center line position and tighten tool post screw.

THE CUTTING OPERATION

Before starting the actual cutting of a right hand thread, be sure that the change gear train is assembled properly and that the reverse lever is in the correct position to feed the carriage toward the headstock. Adjust lathe for lowest possible speed.

Set the compound rest approximately in the center of its ways and advance the cross feed so that it is set at 0 with the tool close to the work. With the point of the tool about an inch to the right of the start of the thread, advance the tool with the compound rest so that the first cut will be about .003 inch.

Start the lathe and engage the half-nut lever on the carriage. The 29° angle of the compound rest should allow the back of the tool to take a fine chasing cut on the finished side of the thread while the cutting edge does the work of forming the thread. Appply plenty of lubricant to the work. When the point of the tool reaches the groove at the end of the thread (groove D in Figure 135), raise the half-nut lever on the carriage, back out the cross feed a turn or two, and return the carriage by hand to the starting point. Advance the cross feed to its original position at 0, advance the compound rest for the desired depth of cut, and engage the half-nut lever for the second cut. All feeding is done with the compound rest. Follow the same routine on all succeeding cuts.

DEPTH OF CUT: The first two or three cuts should be approximately .005 inch advance of the compound feed and the following cuts gradually reduced until the last few cuts taken are only .001 inch or even .0005 inch. A final pass through the thread with no advance whatever will often clean up any remaining high spots. Take the last cuts with extreme care. Heavier cuts can be taken on soft metals such as brass or aluminum, but if a fine finish is desired, the last cuts should be very light.

LUBRICANTS: When cutting steel use liberal quantities of a commercial cutting compound, lard oil or equivalent. With other metals use the type of lubricant recommended for general turning operations.

CUTTING INTERNAL 60° NATIONAL SCREW THREADS

The tool shown in Figure 139 is designed for cutting internal 60° form threads and is mounted directly in the tool post exactly like a boring tool. Such a tool is included in the set of boring tools described in Manual page 89. The angles shown are typical and satisfactory for threads as coarse as 12 per inch and holes as small as ⅝ inch. The point is ground to 60° and has a slight side rake as shown in the front view.

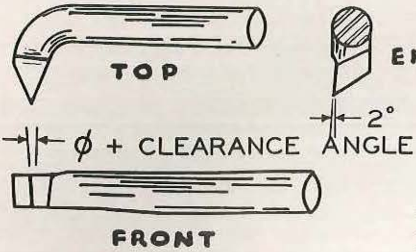


FIG. 139

Tool for cutting internal 60° threads. (When threading brass and plastics, omit side rake.) Fig. 40 explains how the angle Φ must be determined.

It is very important to have plenty of front and side clearance —much more important than with the plain boring tool. The point of the tool is set exactly on the center line of the work.

PREPARING THE WORK FOR INTERNAL NATIONAL FORM THREADS

Work to be threaded internally is prepared much in the same manner as for cutting an external thread (see page 98). The work is first bored to the exact minor diameter. Beginners often turn grooves C and D to the exact major diameter as shown in Figure

PRECAUTIONS IN CUTTING THREADS

Never disengage the half-nut lever in the middle of the thread without first backing out the tool with the cross feed.

Do not shift the reverse feed lever until the thread is completed.

If the work must be removed for checking the fit of a cut or for any other reason, be sure to replace the work with the tail of the lathe dog in the same slot of the face plate as before. Never remove work held in a chuck until the thread is completed.

When a long, heavy thread is being turned, considerable heat may be generated, causing the work to expand. If the work is mounted between centers, stop the lathe at regular intervals and check the tightness of the work against the centers. Take a light cut after checking in this way, because the work may have shifted a trifle in relation to the position of the tool bit. If the tool has a tendency to "hog in," check tool clearance.

141. If the thread is to be cut with a sharp pointed 60° tool, the major diameter is equal to the minor diameter plus the Vee-form Double Depth of Thread (Table I, page 110). If the tool bit is formed especially for a certain National Form thread, the correct major diameter is listed in Table II or Table III, pages 111 and 112.

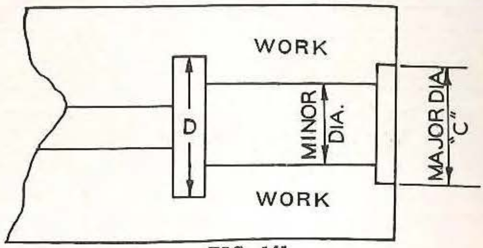


FIG. 141

Grooves C and D help the beginner when threading internally.

Groove C permits the beginner to measure accurately the bottom of the thread with a micrometer or caliper and serves as a guide for depth. When the tool point has cut to the depth of groove C, the thread has been finished. This outer groove

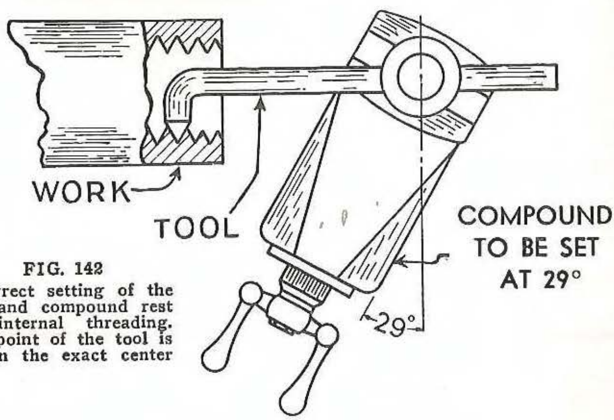


FIG. 142

Correct setting of the tool and compound rest for internal threading. The point of the tool is set on the exact center line.

is not necessary if the thread is being cut to fit a certain screw—the proper depth is then reached when the screw fits the thread correctly.

Groove D should be about twice as wide as the thread pitch and a few thousandths larger than the major diameter. This groove provides a brief interval at the end of each cut during which the work can revolve freely while the half-nut lever is disengaged.

The grooves C and D can be omitted after the operator has learned internal thread cutting operations.

CUTTING INTERNAL THREADS

The internal cutting operation is the same as the cutting of an external thread (page 99), with the following exceptions: First, the 29° angle of the compound rest is measured from the opposite side of the graduated base (Fig 142).

Second, the compound rest feed is *toward* the operator for cutting and the cross feed is *advanced* to clear the work.

Due to the spring of an internal tool, cuts should be much lighter than when cutting external threads. The last finish cuts should be taken without changing the setting of the compound rest.

CUTTING LEFT HAND THREADS

Figure 143 shows the cutting of a left hand thread. The direction of carriage feed is toward the tailstock. Gear set-ups and general cutting procedure are exactly the same as for right hand threads with the changes in tool angles made necessary by the different direction of carriage travel. Clearance angles and side

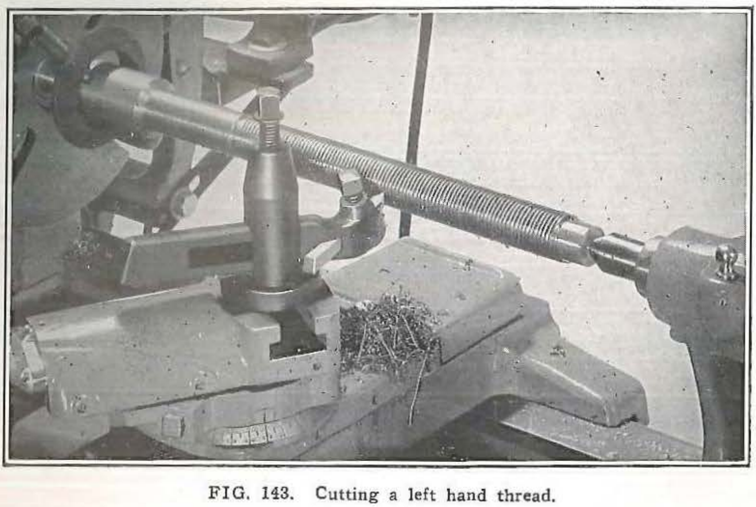
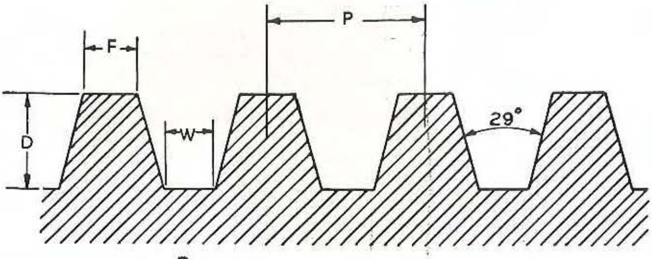


FIG. 143. Cutting a left hand thread.

rake should be the opposite of those shown in Figure 129. In cutting left hand 60° type threads, the compound rest should be set at 29° in the direction shown in Figure 143 which is opposite that for right hand threads.

CUTTING ACME THREADS



$D = \frac{P}{2} + .010''$ $F = .3707 \times P$

$W = .3707 \times P - .0052''$ MINOR DIA. = MAJOR DIA. $-(P + .020'')$

FIG. 144. Acme Screw Thread and Formulas.

The Acme screw thread (Fig 144) is often found in power transmissions, where heavy loads necessitate close-fitting threads. Another common application is in the lead screws and feed screws of precision machine tools. The lead screw, cross feed and compound rest feed screw of most lathes have Acme threads.

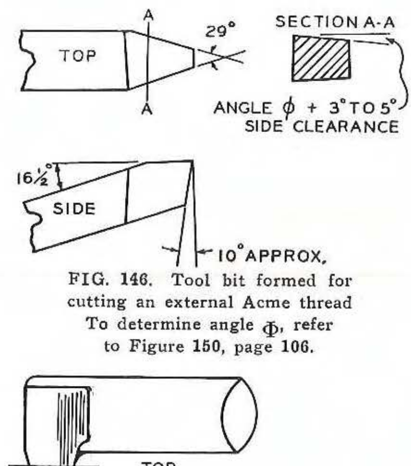


FIG. 146. Tool bit formed for cutting an external Acme thread To determine angle Φ, refer to Figure 150, page 106.

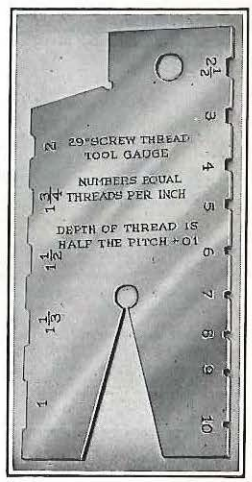


FIG. 145
Acme Thread Gauge.

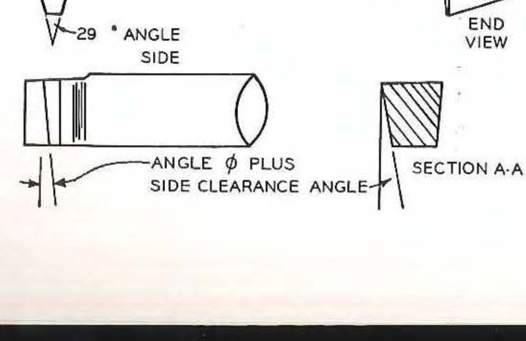


FIG. 147 (Left)
Tool bit formed for cutting an internal Acme thread. To determine an angle Φ, refer to Figure 150, page 106.

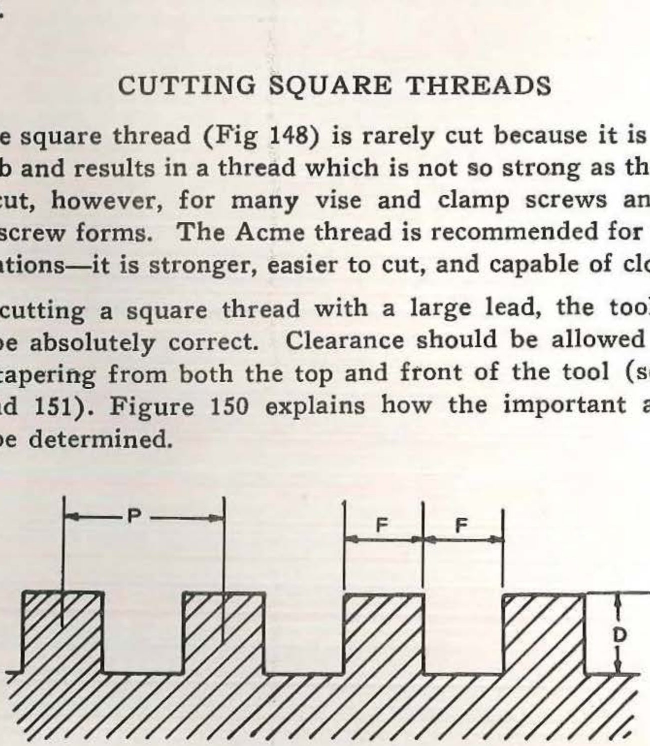
Figures 146 and 147 show the proper tool forms for cutting external and internal Acme threads. The forms must be checked with the Acme thread gauge (Fig. 145) during the cutting process.

The various steps in the cutting of an Acme thread are similar to those for 60° type threads (96 to 101). Set the compound rest at 14½° and advance compound feed after cut, returning cross feed each time to the same setting. Take lighter cuts than with 60° type threads because the total cutting face of the tool is longer.

CUTTING SQUARE THREADS

The square thread (Fig 148) is rarely cut because it is a difficult job and results in a thread which is not so strong as the Acme. It is cut, however, for many vise and clamp screws and other worm-screw forms. The Acme thread is recommended for all such applications—it is stronger, easier to cut, and capable of closer fits.

In cutting a square thread with a large lead, the tool angles must be absolutely correct. Clearance should be allowed on two sides, tapering from both the top and front of the tool (see Figs. 149 and 151). Figure 150 explains how the important angle Φ must be determined.



$F = \frac{P}{2}$ $D = \frac{P}{2}$

FIG. 148. Square Thread and Formulas.

External square threads should be cut to the minor diameter plus about .005 inch, internal square threads to the major diameter plus about .005 inch. The additional .005 inch allows a small clearance at the bottom of the thread, which helps to compensate for any small inaccuracies in the tool or cutting.

The tool must be fed directly into the work with the cross feed (or compound rest feed), and care must be taken to avoid chatter and "hogging-in." The simplest method is to set the compound rest at 0°, feed in with the compound, and back out and return the tool with the cross feed. The very light cuts when turning or boring a square thread.

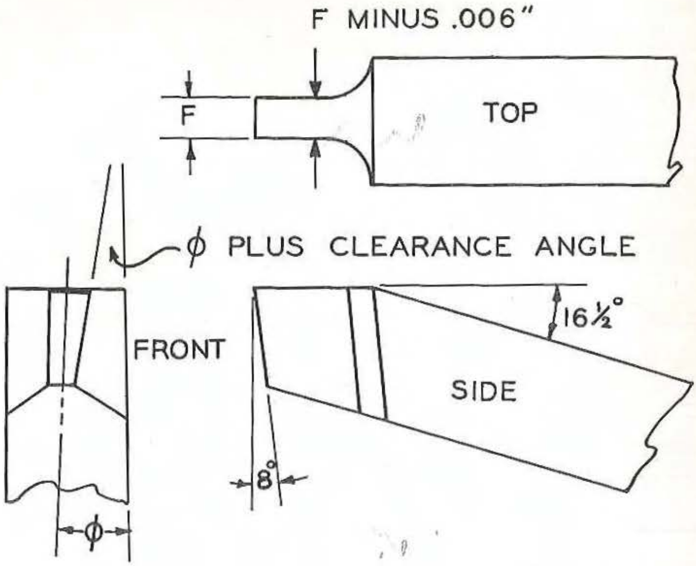


FIG. 149. Tool bit for cutting external square threads

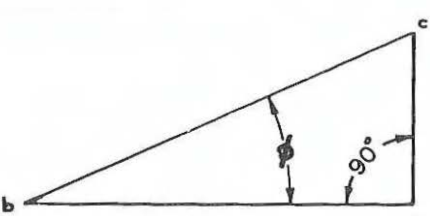


FIG. 150

Determining the angle ϕ. Draw line "ab" equal to the circumference of the thread (3.1416 × major diameter). Then draw line "ac" at right angles to "ab" and equal in length to the thread pitch (or lead, if a multiple thread). Draw line "bc." The angle ϕ is equal to the angle made by lines "ba" and "bc."

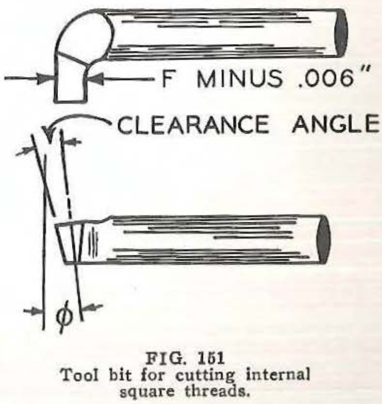


FIG. 151
Tool bit for cutting internal square threads.

WHITWORTH FORM THREAD

Figure 152 shows the Whitworth thread, a form which is standard in the British Isles for nearly all types of threads. The smaller sizes of the Whitworth form are called British Standard Fine.

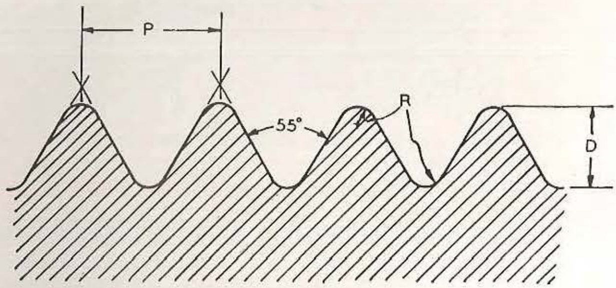


FIG. 152. Whitworth Thread and Formulas.

A Whitworth thread is cut in much the same manner as an Acme thread. There are two major differences: The thread angle is smaller, and the radius at the top and bottom of the thread must be shaped properly with a formed tool.

CUTTING PIPE-THREADS

Figure 153 shows the exact form of the American Standard Pipe thread when cut correctly in a pre-formed die. When turned into the receiving nut, the tapered lines cause the tight "jamming" for which the pipe thread is so well known. In a straight form this thread is used in oil cups and several types of electrical fittings.

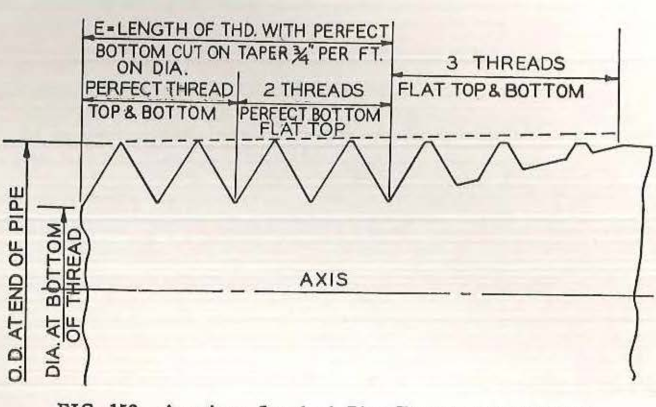


FIG. 153. American Standard Pipe Thread and Formulas.

In order to cut the American Standard Pipe thread on the lathe without special dies or equipment, some variation in form is necessary. Excellent pipe-type threads, satisfactory for commercial use and having the same jamming effect when forced into the nut or coupling, can be cut with a 60° Vee type tool and a set-over of the tailstock to obtain a taper of approximately 3/4 inch per foot. If the stock cannot be mounted between lathe centers, the taper attachment (Part 8) is required for the cutting operation. The threading operation is similar to that for a standard Vee thread and produces a thread resembling the threaded portion shown in Figure 154. Figure 155 shows a type of pipe center recommended for supporting the stock while cutting pipe type threads.

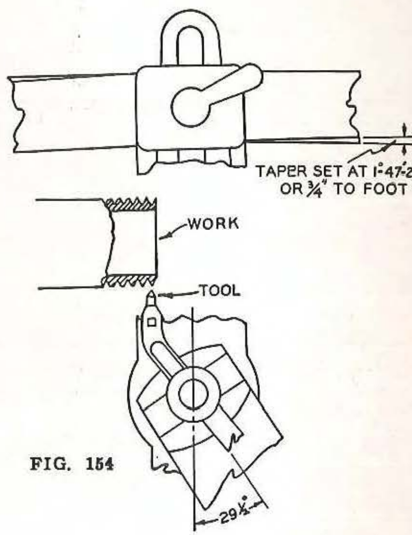


FIG. 154

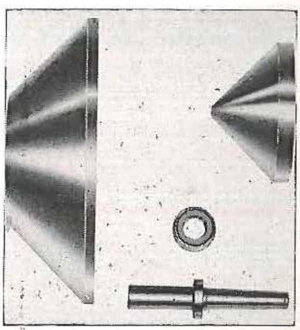
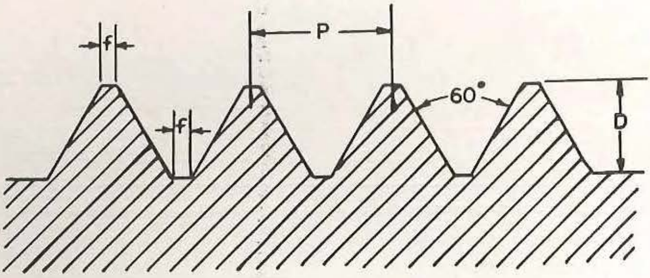


FIG. 155. Pipe Centers.

CUTTING METRIC THREADS

(Also Special Fractional Threads)

The Metric Standard screw thread form shown in Figure 156 (page 109) is accepted almost universally wherever the metric system is the standard of measurement. The metric thread angle and form is identical to that of the National Form thread, and the cutting operation is exactly the same, with one important exception: the motor must be reversed after each cut. This procedure is necessary because metric threads have no definite relation to the threading dial.



$$D = .64952 \times P \qquad f = \frac{P}{8}$$

FIG. 156. Metric Standard Screw Thread Form and Formulas.

The following cutting method applies to metric threads and also to special fractional threads, wire feeds, and the threads in Table I of 10″ Standard Change-Gear Lathe section, not marked " Exact": After the half-nut lever on the carriage is engaged for the first cut, it should not be moved until the thread has been completed. As the tool reaches the end of each cut, back out the cross feed, stop the lathe, and reverse the motor until the tool has been returned to the starting position. Then advance the cross feed to its original 0 position, turn in the compound rest feed for the next cut, start the motor and repeat the cutting operation.

MULTIPLE THREADS

Multiple threads of almost any pitch and number of starts can be cut by two methods. The threading dial is quick, simple and accurate for some double threads and some quadruple or "multiple-four" threads.

Multiple threading requires larger tool clearance angles. Figure 125 shows a double screw thread and Figure 157 shows a

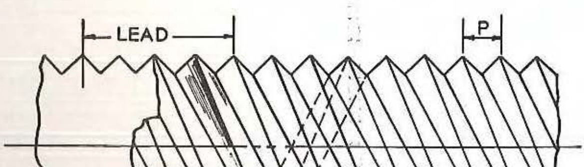


FIG. 157
Quadruple screw thread. The lead is four times the pitch.

quadruple or multiple four thread. These drawings illustrate how the angle of advance has been increased—*the tool clearance must be sufficient for the lead, not merely the pitch.*

TABLE I

DEPTH AND DOUBLE DEPTH OF NATIONAL FORM THREADS

This table shows (I) Depth and Double Depth for National Form Threads cut with a NF formed tool, and (II) Depth and Double Depth of NF threads cut with a 60° V-type tool, making a V bottom but leaving top of thread with proper amount of flat (see text, page 97). Two columns at extreme right give proper depth of compound feed to obtain correct depth of thread with compound rest at 29° (page 99).

Threads per Inch	Pitch Inches	(I) When Cut with NATIONAL FORM TOOL		(II) When Cut with VEE FORM TOOL		Depth of Compound Feed Single Depth	
		Single Depth of Thread	Double Depth of Thread	Single Depth of Thread	Double Depth of Thread	N. F. Tool	Vee Form Tool
4	.2500	.1624	.3248	.1894	.3789	.186	.216
4½	.2222	.1443	.2887	.1684	.3368	.165	.193
5	.2000	.1299	.2598	.1516	.3031	.148	.173
5½	.1818	.1181	.2362	.1378	.2755	.135	.157
6	.1667	.1083	.2165	.1263	.2525	.124	.144
7	.1429	.0928	.1856	.1082	.2165	.106	.123
8	.1250	.0812	.1624	.0947	.1894	.093	.108
9	.1111	.0722	.1443	.0842	.1684	.083	.095
10	.1000	.0650	.1299	.0758	.1515	.074	.087
11	.0909	.0590	.1181	.0689	.1377	.067	.078
12	.0833	.0541	.1083	.0631	.1263	.062	.072
13	.0769	.0500	.0999	.0583	.1166	.057	.067
14	.0714	.0464	.0928	.0541	.1082	.053	.062
16	.0625	.0406	.0812	.0473	.0947	.046	.054
18	.0556	.0361	.0722	.0421	.0842	.041	.047
20	.0500	.0325	.0650	.0379	.0758	.037	.043
22	.0454	.0295	.0590	.0345	.0690	.034	.038
24	.0417	.0271	.0541	.0316	.0632	.031	.036
27	.0370	.0241	.0481	.0281	.0562	.028	.032
28	.0357	.0232	.0464	.0270	.0541	.027	.031
30	.0333	.0217	.0433	.0253	.0506	.025	.029
32	.0313	.0203	.0406	.0237	.0474	.023	.027
36	.0278	.0180	.0361	.0211	.0421	.021	.024
40	.0250	.0162	.0325	.0189	.0379	.019	.021
44	.0227	.0148	.0295	.0172	.0345	.017	.020
48	.0208	.0135	.0271	.0157	.0315	.015	.018
50	.0200	.0130	.0260	.0151	.0303	.015	.017
56	.0179	.0116	.0232	.0135	.0271	.013	.016
64	.0156	.0101	.0203	.0118	.0237	.012	.014
72	.0139	.0090	.0180	.0105	.0210	.010	.012
80	.0125	.0081	.0162	.00945	.0189	.009	.011
96	.0104	.0068	.0136	.00901	.01802	.008	.010

Note: Using Formed Tool—Minor Diameter = Major Diameter minus Double Depth of Thread in National Form Tool column.
Using Vee Tool—Minor Diameter = Major Diameter minus Double Depth of Thread in Vee Form Tool column.

TABLE II

NATIONAL COARSE THREAD SERIES

(Formerly U. S. Standard)

THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for 75% Thread	Clearance Drill Size*
1	64	.0730	.0527	.0629	53	47
2	56	.0860	.0628	.0744	50	42
3	48	.0990	.0719	.0855	47	36
4	40	.1120	.0795	.0958	43	31
5(1/8)	40	.1250	.0925	.1088	38	29
6	32	.1380	.0974	.1177	36	25
8	32	.1640	.1234	.1437	29	16
10	24	.1900	.1359	.1629	25	13/64″
12	24	.2160	.1619	.1889	16	7/32″
1/4″	20	.2500	.1850	.2175	7	17/64″
5/16″	18	.3125	.2403	.2764	F	21/64″
3/8″	16	.3750	.2938	.3344	5/16″	25/64″
7/16″	14	.4375	.3447	.3911	U	29/64″
1/2″	13	.5000	.4001	.4500	27/64″	33/64″
9/16″	12	.5625	.4542	.5084	31/64″	37/64″
5/8″	11	.6250	.5069	.5660	17/32″	41/64″
3/4″	10	.7500	.6201	.6850	21/32″	49/64″
7/8″	9	.8750	.7301	.8028	49/64″	57/64″
1″	8	1.0000	.8376	.9188	7/8″	1- 1/64″
1 1/8″	7	1.1250	.9394	1.0322	63/64″	1- 9/64″
1 1/4″	7	1.2500	1.0644	1.1572	1- 7/64″	1-17/64″
1 3/8″	6	1.3750	1.1585	1.2667	1- 7/32″	1-25/64″
1 1/2″	6	1.5000	1.2835	1.3917	1-11/32″	1-33/64″
1 3/4″	5	1.7500	1.4902	1.6201	1- 9/16″	1-49/64″
2″	4 1/2	2.0000	1.7113	1.8557	1-25/32″	2- 1/32″
2 1/4″	4 1/2	2.2500	1.9613	2.1057	2- 1/32″	2- 9/32″
2 1/2″	4	2.5000	2.1752	2.3376	2 1/4″	2-17/32″
2 3/4″	4	2.7500	2.4252	2.5876	2 1/2″	2-25/32″
3″	4	3.0000	2.6752	2.8376	2 3/4″	3- 1/32″
3 1/4″	4	3.2500	2.9252	3.0876	3″	3- 9/32″
3 1/2″	4	3.5000	3.1752	3.3376	3 1/4″	3-17/32″
3 3/4″	4	3.7500	3.4252	3.5876	3 1/2″	3-25/32″
4″	4	4.0000	3.6752	3.8376	3 3/4″	4- 1/32″

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE III

NATIONAL FINE THREAD SERIES

(Formerly S. A. E.)

THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for 75% Thread	Clearance Drill Size*
0	80	.0600	.0438	.0519	3/64"	51
1	72	.0730	.0550	.0640	53	47
2	64	.0860	.0657	.0759	50	42
3	56	.0990	.0758	.0874	45	36
4	48	.1120	.0849	.0985	42	31
5(1/8)	44	.1250	.0955	.1102	37	29
6	40	.1380	.1055	.1218	33	25
8	36	.1640	.1279	.1460	29	16
10	32	.1900	.1494	.1697	21	13/64"
12	28	.2160	.1696	.1928	14	7/32"
1/4"	28	.2500	.2036	.2268	3	17/64"
5/16"	24	.3125	.2584	.2854	I	21/64"
3/8"	24	.3750	.3209	.3479	Q	25/64"
7/16"	20	.4375	.3726	.4050	25/64"	29/64"
1/2"	20	.5000	.4351	.4675	29/64"	33/64"
9/16"	18	.5625	.4903	.5264	33/64"	37/64"
5/8"	18	.6250	.5528	.5889	37/64"	41/64"
3/4"	16	.7500	.6688	.7094	11/16"	49/64"
7/8"	14	.8750	.7822	.8286	13/16"	57/64"
1"	14	1.0000	.9072	.9536	15/16"	1- 1/64"
1 1/8"	12	1.1250	1.0168	1.0709	1- 3/64"	1- 9/64"
1 1/4"	12	1.2500	1.1418	1.1959	1-11/64"	1-17/64"
1 3/8"	12	1.3750	1.2668	1.3209	1-19/64"	1-25/64"
1 1/2"	12	1.5000	1.3918	1.4459	1-27/64"	1-33/64"

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE IV
FRACTIONAL SIZES
NATIONAL SPECIAL THREAD SERIES
THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for 75% Thread	Clearance Drill Size*
1/16"	64	.0625	.0422	.0524	3/64"	51
5/64"	60	.0781	.0563	.0673	1/16"	45
3/32"	48	.0938	.0667	.0803	49	40
7/64"	48	.1094	.0823	.0959	43	32
1/8"	32	.1250	.0844	.1047	3/32"	29
9/64"	40	.1406	.1081	.1244	32	24
5/32"	32	.1563	.1157	.1360	1/8"	19
5/32"	36	.1563	.1202	.1382	30	19
11/64"	32	.1719	.1313	.1516	9/64"	14
3/16"	24	.1875	.1334	.1604	26	8
3/16"	32	.1875	.1469	.1672	22	8
13/64"	24	.2031	.1490	.1760	20	3
7/32"	24	.2188	.1646	.1917	16	1
7/32"	32	.2188	.1782	.1985	12	1
15/64"	24	.2344	.1806	.2073	10	1/4"
1/4"	24	.2500	.1959	.2229	4	17/64"
1/4"	27	.2500	.2019	.2260	3	17/64"
1/4"	32	.2500	.2094	.2297	7/32"	17/64"
5/16"	20	.3125	.2476	.2800	17/64"	21/64"
5/16"	27	.3125	.2644	.2884	J	21/64"
5/16"	32	.3125	.2719	.2922	9/32"	21/64"
3/8"	20	.3750	.3100	.3425	21/64"	25/64"
3/8"	27	.3750	.3269	.3509	R	25/64"
7/16"	24	.4375	.3834	.4104	X	29/64"
7/16"	27	.4375	.3894	.4134	Y	29/64"
1/2"	12	.5000	.3918	.4459	27/64"	33/64"
1/2"	24	.5000	.4459	.4729	29/64"	33/64"
1/2"	27	.5000	.4519	.4759	15/32"	33/64"
9/16"	27	.5625	.5144	.5384	17/32"	37/64"
5/8"	12	.6250	.5168	.5709	35/64"	41/64"
5/8"	27	.6250	.5769	.6009	19/32"	41/64"
11/16"	11	.6875	.5694	.6285	19/32"	45/64"
11/16"	16	.6875	.6063	.6469	5/8"	45/64"
3/4"	12	.7500	.6418	.6959	43/64"	49/64"
3/4"	27	.7500	.7019	.7259	23/32"	49/64"
13/16"	10	.8125	.6826	.7476	23/32"	53/64"
7/8"	12	.8750	.7668	.8209	51/64"	57/64"
7/8"	18**	.8750	.8028	.8389	53/64"	57/64"
7/8"	27	.8750	.8269	.8509	27/32"	57/64"
15/16"	9	.9375	.7932	.8654	53/64"	61/64"
1"	12	1.0000	.8918	.9459	59/64"	1- 1/64"
1"	27	1.0000	.9519	.9759	31/32"	1- 1/64"
1⅝"	5½	1.6250	1.3888	1.5069	1-29/64"	1-41/64"
1⅞"	5	1.8750	1.6152	1.7451	1-11/16"	1-57/64"
2⅛"	4½	2.1250	1.8363	1.9807	1-29/32"	2- 5/32"
2⅜"	4	2.3750	2.0502	2.2126	2- 1/8"	2-13/32"

** *Standard Spark Plug Size*

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE V

MACHINE SCREW SIZES

THREAD DIMENSIONS AND TAP DRILL SIZES
NATIONAL SPECIAL THREAD SERIES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for 75% Thread	Clearance Drill Size*
1	56	.0730	.0498	.0614	54	47
4	32	.1120	.0714	.0917	45	31
4	36	.1120	.0759	.0940	44	31
5(1/8)	36	.1250	.0889	.1070	40	29
6	36	.1380	.1019	.1200	34	25
7	30	.1510	.1077	.1294	31	21
7	36	.1510	.1149	.1330	1/8"	21
8	30	.1640	.1207	.1423	30	16
8	40	.1640	.1315	.1478	28	16
9	24	.1770	.1229	.1499	29	13
9	30	.1770	.1337	.1553	27	13
9	32	.1770	.1364	.1567	26	13
10	28	.1900	.1436	.1668	23	13/64"
10	30	.1900	.1467	.1684	22	13/64"
12	32	.2160	.1754	.1957	13	7/32"
14	20	.2420	.1770	.2095	10	17/64"
14	24	.2420	.1879	.2149	7	17/64"

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE VI
BRITISH STANDARD — WHITWORTH FORM
THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for Full Thread	Clearance Drill Size*
1/16″	60	.0625	.0412	.0518	57	51
3/32″	48	.0938	.0671	.0804	50	40
1/8″	40	.1250	.0930	.1090	40	29
5/32″	32	.1563	.1162	.1362	31	19
3/16″	24	.1875	.1341	.1608	28	8
7/32″	24	.2188	.1654	.1921	17	1
1/4″	20	.2500	.1860	.2180	9	17/64″
9/32″	20	.2813	.2321	.2566	C	19/64″
5/16″	18	.3125	.2414	.2769	1/4″	21/64″
3/8″	16	.3750	.2950	.3350	5/16″	25/64″
7/16″	14	.4375	.3460	.3918	T	29/64″
1/2″	12	.5000	.3933	.4466	Z	33/64″
9/16″	12	.5625	.4558	.5091	15/32″	37/64″
5/8″	11	.6250	.5086	.5668	17/32″	41/64″
11/16″	11	.6875	.5711	.6293	19/32″	45/64″
3/4″	10	.7500	.6219	.6860	41/64″	49/64″
13/16″	10	.8125	.6844	.7485	45/64″	53/64″
7/8″	9	.8750	.7327	.8039	3/4″	57/64″
1″	8	1.0000	.8399	.9200	55/64″	1- 1/64″
1⅛″	7	1.1250	.9420	1.0335	31/32″	1- 9/64″
1¼″	7	1.2500	1.0670	1.1585	1- 3/32″	1-17/64″
1⅜″	6	1.3750	1.1616	1.2683	1- 3/16″	1-25/64″
1½″	6	1.5000	1.2866	1.3933	1- 5/16″	1-33/64″
1⅝″	5	1.6250	1.3689	1.4969	1-13/32″	1-41/64″
1¾″	5	1.7500	1.4939	1.6219	1-17/32″	1-49/64″
2″	4½	2.0000	1.7154	1.8577	1- 3/4 ″	2- 1/32″
2¼″	4	2.2500	1.9298	2.0899	1-31/32″	2- 9/32″
2½″	4	2.5000	2.1798	2.3399	2- 7/32″	2-17/32″

TABLE VII
BRITISH ASSOCIATION STANDARD
THREAD DIMENSIONS AND TAP DRILL SIZES

Number Size	Pitch m/m	Major Diameter m/m	Minor Diameter m/m	Pitch Diameter m/m	Tap Drill for Full Thread	Clearance Drill Size*
0	1.00	6.0	4.80	5.400	10	F
1	.90	5.3	4.22	4.760	17	1
2	.81	4.7	3.73	4.215	24	7
3	.73	4.1	3.22	3.660	29	15
4	.66	3.6	2.81	3.205	32	21
5	.59	3.2	2.49	2.845	37	27
6	.53	2.8	2.16	2.480	43	30
7	.48	2.5	1.92	2.210	46	32
8	.43	2.2	1.68	1.940	50	37
9	.39	1.9	1.43	1.665	53	42
10	.35	1.7	1.28	1.490	55	44
11	.31	1.5	1.13	1.315	56	48
12	.28	1.3	.96	1.130	60	50

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE VIII

INTERNATIONAL STANDARD—METRIC

THREAD DIMENSIONS AND TAP DRILL SIZES

Major Diameter m/m	Pitch m/m	Minor Diameter m/m	Pitch Diameter m/m	Tap Drill for 75% Thread m/m	Tap Drill for 75% Thread No. of Inches	Clearance Drill Size†
2.0	.40	1.48	1.740	1.6	1/16	41
2.3	.40	1.78	2.040	1.9	48	36
2.6	.45	2.02	2.308	2.1	45	31
3.0	.50	2.35	2.675	2.5	40	29
3.5	.60	2.72	3.110	2.9	33	23
4.0	.70	3.09	3.545	3.3	30	16
4.5	.75	3.53	4.013	3.75	26	10
5.0	.80	3.96	4.480	4.2	19	3
5.5	.90	4.33	4.915	4.6	14	15/64"
6.0	1.00	4.70	5.350	5.0	9	1/4"
7.0	1.00	5.70	6.350	6.0	15/64"	19/64"
8.0	1.25	6.38	7.188	6.8	H	11/32"
9.0	1.25	7.38	8.188	7.8	5/16"	3/8"
10.0	1.50	8.05	9.026	8.6	R	27/64"
11.0	1.50	9.05	10.026	9.6	V	29/64"
12.0	1.75	9.73	10.863	10.5	Z	1/2"
14.0*	1.25	12.38	13.188	13.0	33/64"	9/16"
14.0	2.00	11.40	12.701	12.0	15/32"	9/16"
16.0	2.00	13.40	14.701	14.0	35/64"	21/32"
18.0*	1.50	16.05	17.026	16.5	41/64"	47/64"
18.0	2.50	14.75	16.376	15.5	39/64"	47/64"
20.0	2.50	16.75	18.376	17.5	11/16"	13/16"
22.0	2.50	18.75	20.376	19.5	49/64"	57/64"
24.0	3.00	20.10	22.051	21.0	53/64"	31/32"
27.0	3.00	23.10	25.051	24.0	15/16"	1- 3/32"
30.0	3.50	25.45	27.727	26.5	1- 3/64"	1-13/64"
33.0	3.50	28.45	30.727	29.5	1-11/64"	1-21/64"
36.0	4.00	30.80	33.402	32.0	1-17/64"	1- 7/16"
39.0	4.0	33.80	36.402	35.0	1- 3/8 "	1- 9/16"
42.0	4.50	36.15	39.077	37.0	1-29/64"	1-43/64"
45.0	4.50	39.15	42.077	40.0	1-37/64"	1-13/16"
48.0	5.00	41.50	44.752	43.0	1-11/16"	1-29/32"

* *Special Spark Plug Sizes*

†Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE IX

FRENCH STANDARD THREADS — METRIC

THREAD DIMENSIONS AND TAP DRILL SIZES

Major Diameter m/m	Pitch m/m	Minor Diameter m/m	Pitch Diameter m/m	Tap Drill for 75% Thread m/m	Tap Drill for 75% Thread No. or Inches	Clearance Drill Size*
1.5	.35	1.05	1.273	1.1	57	48
2.0	.45	1.42	1.708	1.5	53	41
2.5	.45	1.92	2.208	2.0	47	32
3.0	.60	2.22	2.610	2.4	3/32″	29
3.5	.60	2.72	3.110	2.9	33	23
4.0	.75	3.03	3.513	3.25	30	16
4.5	.75	3.53	4.013	3.75	26	10
5.0	.90	3.83	4.415	4.1	20	3
5.5	.90	4.33	4.915	4.6	14	15/64″
6.0	1.00	4.70	5.350	5.0	9	1/4″
7.0	1.00	5.70	6.350	6.0	15/64″	19/64″
8.0	1.00	6.70	7.350	7.0	I	11/32″
9.0	1.00	7.70	8.350	8.0	5/16″	3/8″
10.0	1.50	8.05	9.026	8.6	R	27/64″
12.0	1.50	10.05	11.026	10.5	Z	1/2″
14.0	2.00	11.40	12.701	12.0	15/32″	9/16″
16.0	2.00	13.40	14.701	14.0	35/64″	21/32″
18.0	2.50	14.75	16.376	15.5	39/64″	47/64″
20.0	2.50	16.75	18.376	17.5	11/16″	13/16″
22.0	2.50	18.75	20.376	19.5	49/64″	57/64″
24.0	3.00	20.10	22.051	21.0	53/64″	31/32″
26.0	3.00	22.10	24.051	23.0	57/64″	1- 3/64″
28.0	3.00	24.10	26.051	25.0	63/64″	1- 3/64″
30.0	3.50	25.45	27.727	26.5	1- 3/64″	1-13/64″
32.0	3.50	27.45	29.727	28.5	1- 1/8 ″	1- 9/32″
34.0	3.50	29.45	31.727	30.5	1-13/64″	1-23/64″
36.0	4.00	30.80	33.402	32.0	1-17/64″	1- 7/16″
38.0	4.00	32.80	35.402	34.0	1-21/64″	1-33/64″
40.0	4.00	34.80	37.402	36.0	1-27/64″	1-19/32″
42.0	4.50	36.15	39.077	37.0	1-29/64″	1-43/64″
44.0	4.50	38.15	41.077	39.0	1-17/32″	1-3/4″
46.0	4.50	40.15	43.077	41.0	1-39/64″	1-53/64″
48.0	5.00	41.50	44.752	43.0	1-11/16″	1-13/16″
50.0	5.00	43.50	46.752	45.0	1-49/64″	2″

*Clearance drill makes hole with standard clearance for diameter of nominal size.

TABLE X

ACME STANDARD THREAD DIMENSIONS

Threads per Inch	Pitch Inches P	Depth of Thread	Double Depth of Thread	Width of Top of Thread	Width of Space at Bottom of Thread
1	1	.5100	1.0200	.3707	.3655
1⅓	3/4	.3850	.7700	.2780	.2728
2	1/2	.2600	.5200	.1853	.1801
3	1/3	.1767	.3534	.1235	.1183
4	1/4	.1350	.2700	.0927	.0875
5	1/5	.1100	.2200	.0741	.0689
6	1/6	.0933	.1867	.0618	.0566
7	1/7	.0814	.1628	.0530	.0478
8	1/8	.0725	.1450	.0463	.0411
9	1/9	.0655	.1311	.0413	.0361
10	1/10	.0600	.1200	.0371	.0319

Note: Minor Diameter equals Major Diameter minus Double Depth of Thread.

TABLE XI

SQUARE THREAD DIMENSIONS

Threads per Inch	Pitch Inches P	Depth of Thread	Double Depth of Thread	Width of Top of Thread	Width of Space at Bottom of Thread
1	1.0000	.5000	1.0000	.5000	.5000
1⅓	.7500	.3750	.7500	.3750	.3750
1½	.6667	.3333	.6667	.3333	.3333
1¾	.5714	.2857	.5714	.2857	.2857
2	.5000	.2500	.5000	.2500	.2500
2½	.4000	.2000	.4000	.2000	.2000
3	.3333	.1667	.3333	.1667	.1667
3½	.2857	.1429	.2857	.1429	.1429
4	.2500	.1250	.2500	.1250	.1250
4½	.2222	.1111	.2222	.1111	.1111
5	.2000	.1000	.2000	.1000	.1000
5½	.1818	.0909	.1818	.0909	.0909
6	.1667	.0833	.1667	.0833	.0833
7	.1429	.0714	.1429	.0714	.0714
8	.1250	.0625	.1250	.0625	.0625
9	.1111	.0556	.1111	.0556	.0556
10	.1000	.0500	.1000	.0500	.0500
11	.0909	.0455	.0909	.0455	.0455
12	.0833	.0417	.0833	.0417	.0417
13	.0769	.0385	.0769	.0385	.0385
14	.0714	.0357	.0714	.0357	.0357
15	.0667	.0333	.0667	.0333	.0333
16	.0625	.0312	.0625	.0312	.0312
18	.0556	.0278	.0556	.0278	.0278
20	.0500	.0250	.0500	.0250	.0250
22	.0455	.0227	.0455	.0227	.0227
24	.0417	.0208	.0417	.0208	.0208

TABLE XII

STRAIGHT PIPE THREADS
AMERICAN STANDARD FORM

THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Pipe Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill for Full Thread
1/8"	27	.4044	.3451	.3748	11/32"
1/4"	18	.5343	.4455	.4899	7/16"
3/8"	18	.6714	.5826	.6270	37/64"
1/2"	14	.8356	.7213	.7784	23/32"
3/4"	14	1.0460	.9318	.9889	59/64"
1"	11½	1.3082	1.1690	1.2386	1- 5/32"
1¼"	11½	1.6530	1.5138	1.5834	1- 1/2 "
1½"	11½	1.8919	1.7527	1.8223	1-47/64"
2"	11½	2.3658	2.2267	2.2963	2- 7/32"
2½"	8	2.8622	2.6622	2.7622	2- 5/8 "
3"	8	3.4885	3.2885	3.3885	3- 1/4 "
3½"	8	3.9888	3.7888	3.8888	3- 3/4 "
4"	8	4.4871	4.2871	4.3871	4- 1/4 "

TABLE XIII

STOVE BOLTS

MANUFACTURERS STANDARD FORM—60° THREAD

THREAD DIMENSIONS AND TAP DRILL SIZES

Nominal Size	Threads per Inch	Major Diameter Inches	Minor Diameter Inches	Pitch Diameter Inches	Tap Drill	Clearance Drill Size*
1/8"	32	.1250	.0910	.1080	42	29
5/32"	28	.1630	.1250	.1440	1/8"	19
3/16"	24	.1950	.1510	.1730	24	8
7/32"	22	.2220	.1740	.1980	16	1
1/4"	18	.2500	.1980	.2240	8	17/64"
5/16"	18	.3125	.2403	.2764	C	21/64"
3/8"	16	.3750	.2938	.3344	M	25/64"
7/16"	14	.4375	.3447	.3911	S	29/64"
1/2"	13	.5000	.4000	.4500	Y	33/64"

*Clearance drill makes hole with standard clearance for diameter of nominal size.

THREAD CUTTING

on the

12-INCH QUICK-CHANGE LATHE

THE following pages contain complete instructions for operating and maintaining the Quick-Change mechanism on the 12″ metalworking lathes. The instant selection of fifty-four threads and feeds available on the Quick-Change, speeds up threading and every lathe operation requiring power feed.

Thousands of additional threads and feeds can be obtained by varying the gear train with standard change gears, for such jobs as metric threading, coil and wire winding and special tooling.

Gear train arrangements and instructions for cutting metric threads and frequently used odd threads are contained in this book—see instructions, page 124 thru 127—gear train set-ups are shown in Tables I and II, pages 130 and 131.

Gear train arrangements for wire winding, or a thread or feed not listed can be obtained on request. When writing specify thread or feed required—for coil winding feeds, give name, type and size of wire.

OPERATING INSTRUCTIONS

Before operating the Quick-Change, lubricate it thoroughly with SAE. No. 20 machine oil - see Lubrication Instructions, page 128.

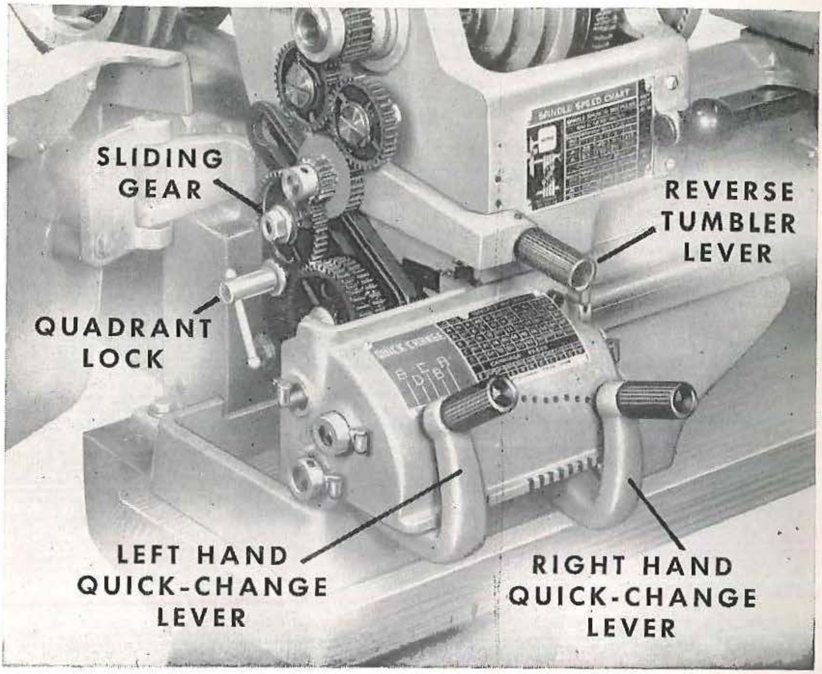


Fig. 159 Quick-Change Controls

CONTROLS

A. Left Hand Quick-Change Lever - shifts to five positions - A, B, C, D and E - shown on left end of chart. The position of the lever is indicated at the left end of the row of numbers in which you find the thread or feed desired.

B. Right Hand Quick-Change Lever - shifts to nine positions. They are numbered on bottom row of chart beneath carriage feeds. The indexing position of the lever is always directly below the thread or feed desired.

C. Reverse Tumbler Lever - has three positions - forward, neutral and reverse - top position feeds carriage to the left and cross-slide toward the operator, bottom position feeds carriage to the right and cross-slide away from operator.

D. Sliding Gear - has two positions - IN and OUT as shown on the chart. IN position is toward headstock and meshes with the 32 tooth compound gear, see Figure 160. OUT position, Figure 161, is away from headstock and meshes with the 16 tooth compound gear. Loosen T-clamp to mesh the sliding gear with the compound tumbler gear. The position of the sliding gear is shown on the chart in the same row as the thread or feed desired.

E. T-Clamp - controls the gear quadrant. Loosen clamp to mesh sliding gear with the compound tumbler gear.

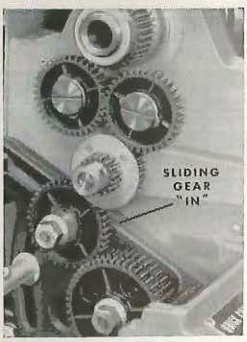


Fig. 160. Sliding Gear in IN Position—meshes with the 32 tooth compound gear.

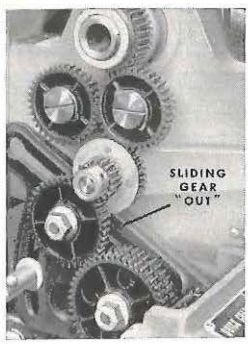


Fig. 161. Sliding Gear in OUT Position—meshes with the 16 tooth compound gear.

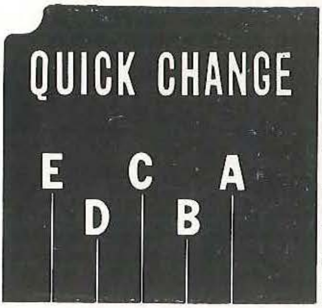
READING THE QUICK-CHANGE CHART

(See Figure 162)

Threads per Inch—forty-five standard threads are shown on Quick-Change chart. Nine additional threads, corresponding to the bottom row of carriage feeds, are also available, they are; 128, 144, 160, 176, 184, 192, 208, 224 and 240.

Carriage Feeds—shown on bottom of chart. Nine feeds are listed—.0078" to .0042" per revolution of spindle.

Sliding Gear—has two positions—IN and OUT. See Controls, Step "D", for location of gear positions. The position of the sliding gear is indicated on the chart in the same row as the thread or feed desired.



L. H. LEVER	SLIDE GEAR	THREADS PER INCH								
A	IN	4	4.5	5	5.5	5.75	6	6.5	7	7.5
A	OUT	8	9	10	11	11.5	12	13	14	15
B	OUT	16	18	20	22	23	24	26	28	30
C	OUT	32	36	40	44	46	48	52	56	60
D	OUT	64	72	80	88	92	96	104	112	120
	CARRIAGE FEEDS									
E	OUT	.0078	.0069	.0063	.0057	.0055	.0052	.0048	.0044	.0042
		1	2	3	4	5	6	7	8	9

DETACH THIS CHART BEFORE REMOVING ATTACHMENT FROM LATHE

Fig. 162. Quick-Change Chart

Left Hand Lever Position—shown in first vertical column of chart. Lever has five indexing positions—A, B, C, D and E. These positions are marked on chart directly above left hand group of indexing holes in gear box. The lever position for a thread or feed is shown in same row as thread or feed desired.

Right Hand Lever Position—figures 1 through 9 across bottom of chart are the indexing positions of the right hand lever. The indexing position of the lever is always directly below the thread or feed desired.

INSTRUCTIONS FOR CUTTING ODD AND METRIC THREADS, AND WIRE WINDING

Thousands of odd threads, metric threads, feeds for coil and wire winding, and special tooling can be obtained using standard change gears on the Quick-Change quadrant in place of the sliding gear and the two 48 tooth gears. Gear set-ups for metric threads, and odd threads from 1 to 70, are listed in this book — Tables I and II, pages 130 and 131.

Information for setting up the gear train for wire winding and threads and feeds not listed in Table II can be obtained on request. When writing specify thread or feed required - for coil winding feeds, give name, type and size of wire.

Gear train set-ups using standard change gears have been simplified by assigning gear positions A, B, C and D to the quadrant as shown in Figure 163. These positions are indicated in the odd thread and metric tables on pages 130 and 131.

The outer portion of the longest slot is position A — the inner portion of the same slot is position B. The short middle slot is position C — the lower slot, position D. These positions are approximate — they will vary with the size of the gears composing the train.

Before setting up a train of change gears, examine one of the change gear stud assemblies which hold gears to gear quadrant (Fig. 165). Each stud assembly has an outer gear bushing long enough to accommodate two gears. This bushing has a double key which fits into the keyways in the gears. Gear bushing and gears fit over a stud bushing, and this assembly is bolted to the gear quadrant. The washer is a bearing for the outer end of the gear bushing.

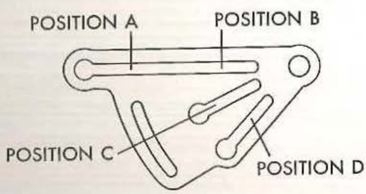


Fig. 163. Gear Positions on Quadrant.

Notice that in order to make this assembly complete, two gears must be mounted on the gear bushing at one time. When both of the gears on a gear bushing mesh with other gears in the train, they form a "Compound" gear assembly (Fig 164).

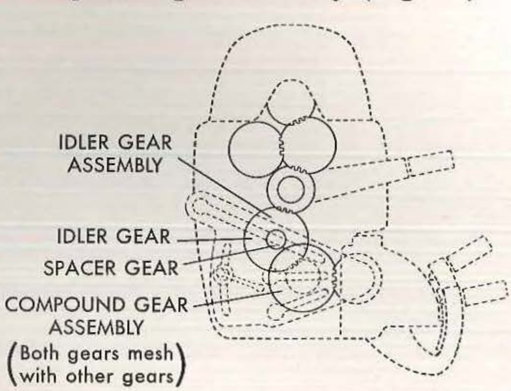


Fig. 164. Diagram showing compound and idler gears in a gear train.

When only one of the two gears meshes with another gear in the train, this gear is called an "Idler." The other gear, or spacer, is called a "Spacer" gear and does not mesh with any gear in the train.

The positions of the gears on the stud assemblies are denoted as "N"and "F" in the gear set-up tables. "N" position means the gear or spacer is positioned on stud NEXT TO Quadrant— "F" position is gear or spacer AWAY from quadrant. "SS" denotes that a double-keyway steel spacer must be used on gear stud.

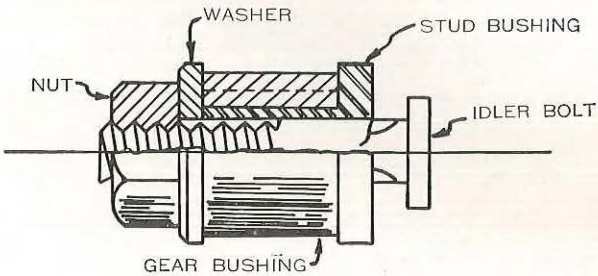


Fig. 165. Cross section of change gear stud assembly.

GEAR CLEARANCE

When setting up the gear train, be sure to allow sufficient clearance between two meshing gears (Fig. 166). Gear clearance does not reduce the accuracy of a thread cutting operation, because all play in the gears is taken up in one direction. A method often used to obtain proper gear clearance is : (1) Place a sheet of thick writing paper between the teeth of the two meshing gears, (2) tighten gears in position, and (3) remove paper. A small amount of grease, preferably graphite grease, applied to gear teeth will often aid in obtaining smoother, more quiet operation.

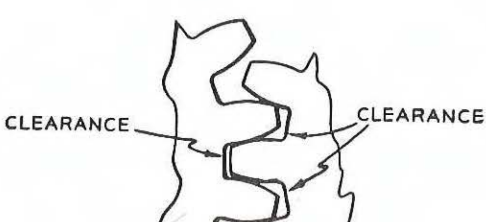


FIG. 166
Proper gear clearance.

REPLACING THE QUICK-CHANGE GEAR TRAIN

After making special set-ups for wire winding, odd or metric threads, be sure to assemble the sliding gear and the double 48 tooth gears in their original positions on the quadrant as shown in Figure 167.

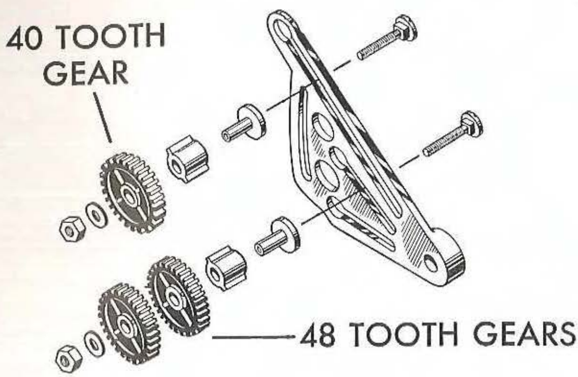


Fig. 167. Exploded view of Gears and Studs which make up the standard Quick-Change Gear Train

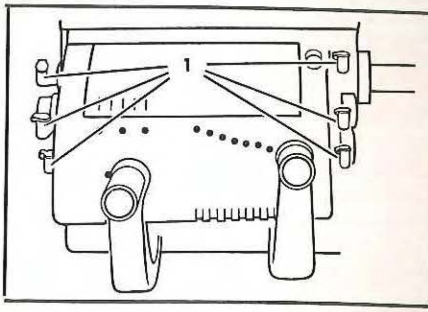
IMPORTANT

Table II, Page 131, shows gear train set-ups for threads as course as 1¼ per inch. However, extreme care must be taken when cutting threads less than 4 per inch. USE SLOW SPEEDS AND TAKE EXTREMELY LIGHT CUTS — the lead screw is revolving very rapidly, making it difficult for the operator to engage the carriage half-nuts at the right moment. Also excessive pressure is being exerted on the lead screw, resulting in rapid wear and overloading of the gear train.

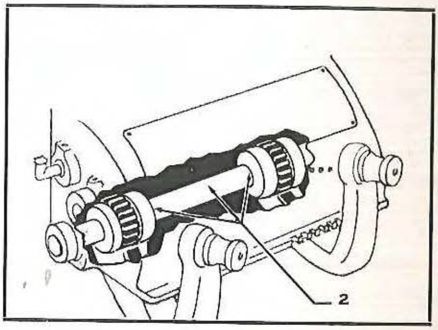
LUBRICATION

Lubricate Regularly — use S.A.E. No. 20 Machine Oil.

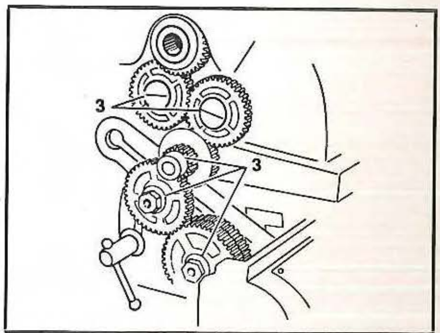
1. All the lubrication cups on the gear housing are shown in illustration at right. Put a few drops of oil in each oil cup once a week if lathe is used constantly.



2. Quick-Change lever bearings and shaft — oil once a week.



3. Feed Gears and Tumbler Gear Bearings — once a week place a few drops of oil on the gear train bearings.



Occasionally apply a small amount of heavy outer gear lubricant to the feed gears and tumbler gears—it will aid in obtaining smoother, more quiet operation. IMPORTANT—Make sure all oil has been removed from the gear teeth before applying lubricant or it will not adhere.

SUGGESTIONS FOR OPERATION AND MAINTENANCE

1. CAUTION — do not shift the reverse tumbler lever or Quick-Change gear levers when lathe is turning at high speeds or under heavy loads.

2. Do not shift sliding gear when gear train is turning.

3. If Quick-Change levers do not index, do not force, merely rotate spindle by hand until levers slide easily into position.

4. Keep metal chips from piling up underneath gear box — they may cause serious damage to the gear train.

TABLE I

GEAR SET-UPS FOR METRIC THREADS

SS in table below denotes steel spacer. Extra gears, stub assemblies, and spacers necessary to make up the gear train are available from factory at nominal cost.

Pitch in Millimeters	English Equivalent	Position A		Position B		Position C		Position D		Compound Gear	Left Hand Lever Position	Right Hand Lever Position
		N	F	N	F	N	F	N	F			
.25	.00984	SS	48	32*	SS	—	—	52	60	16	D	4
.3	.01181	48	SS	46	50	—	—	SS	24*	32	E	5
.35	.01378	48	SS	40	44	—	—	SS	24*	32	E	3
.4	.01575	SS	48	40*	SS	—	—	40	52	16	C	4
.45	.01772	56	SS	SS	24*	—	—	44	50	32	E	1
.5	.01968	SS	48	32*	SS	—	—	52	60	16	C	4
.55	.02165	48	SS	—	—	40	52*	—	—	32	D	9
.6	.02362	48	SS	46	50	—	—	SS	24*	32	D	5
.65	.02559	SS	48	36	44	—	—	32*	SS	16	C	1
.7	.02756	48	SS	40	44	—	—	SS	24*	32	D	3
.75	.02952	48	SS	—	—	40	52*	—	—	32	D	4
.8	.03150	SS	48	40*	SS	—	—	36	52	16	B	4
.85	.03346	SS	48	24*	SS	—	—	60	64	16	B	8
.9	.03543	SS	64	32*	SS	—	—	44	54	16	B	5
.95	.03740	SS	64	32*	SS	—	—	46	56	16	B	4
1.00	.03937	SS	48	32*	SS	—	—	52	60	16	B	4
1.25	.04921	SS	48	32	50	—	—	40*	SS	16	A	7
1.50	.05906	48	SS	—	—	40	52*	—	—	32	C	4
1.75	.06889	SS	60	24*	SS	—	—	54	56	16	A	8
2.00	.07874	48	SS	—	—	44	52*	—	—	32	B	9
2.50	.09842	50	32	SS	48	—	—	—	—	32	A	4
3.	.11811	48	SS	—	—	40	52*	—	—	32	B	4
3.50	.13780	56	54	—	—	SS	48*	—	—	32	A	8
4.	.15750	48	SS	—	—	44	52*	—	—	32	A	9
4.50	.17720	48	SS	36	46	—	—	SS	24*	32	A	8
5.	.19685	48	SS	—	—	50	64*	—	—	32	A	7
5.5	.21650	48	SS	—	—	40	52*	—	—	32	A	6
6.	.23620	48	SS	—	—	40	52*	—	—	32	A	4
7.	.27560	48	SS	40	44	—	—	SS	24*	32	A	1

* *This gear to mesh with gear in gear box.*

TABLE II

GEAR SET-UPS FOR FREQUENTLY USED ODD THREADS NOT SHOWN ON QUICK-CHANGE CHART

SS in table below denotes steel spacer. Extra gears, stub assemblies, and spacers necessary to make up the gear train are available from factory at nominal cost.

Threads per Inch	Feed in Inches	Position A		Position B		Position C		Position D		Compound Gear	Left Hand Lever Position	Right Hand Lever Position
		N	F	N	F	N	F	N	F			
1¼	.80000	—	—	64	SS	—	—	20	64*	32	A	1
1⅓	.75000	—	—	64	SS	—	—	20	60*	32	A	1
1½	.66666	—	—	64	SS	—	—	24	64*	32	A	1
2	.50000	48	SS	—	—	24	48*	—	—	32	A	1
2½	.40000	48	SS	—	—	24	48*	—	—	32	A	3
2⅝	.38094	—	—	64	SS	—	—	24	64*	32	A	8
2¾	.36362	48	SS	—	—	24	48*	—	—	32	A	4
2⅞	.34782	48	SS	—	—	24	48*	—	—	32	A	5
3	.33332	48	SS	—	—	24	48*	—	—	32	A	6
3¼	.30770	48	SS	—	—	24	48*	—	—	32	A	7
3½	.28570	48	SS	—	—	24	48*	—	—	32	A	8
21	.04761	24	56	—	—	—	—	64*	SS	16	A	2
25	.04000	20	50	—	—	—	—	64*	SS	16	A	3
27	.03703	20	54	—	—	64*	SS	—	—	16	A	3
33	.03030	SS	48	32	44	—	—	40*	SS	16	B	6
35	.02857	24	56	—	—	—	—	64*	SS	16	A	9
39	.02564	24	52	—	—	—	—	64*	SS	16	B	2
42	.02380	32	56	48	SS	—	—	—	—	16	B	6
45	.02222	24	54	—	—	—	—	64*	SS	16	B	3
49	.02040	32	56	48*	SS	—	—	—	—	16	B	8
50	.02000	36	60	48*	SS	—	—	—	—	16	B	9
54	.01851	32	54	48*	SS	—	—	—	—	16	C	1
55	.01818	SS	58	24	44	—	—	48*	SS	16	B	9
62	.01613	SS	64	32*	SS	—	—	44	54	16	C	7
63	.01587	24	54	—	—	—	—	48*	SS	16	B	8
65	.01538	SS	48	32	52	—	—	64*	SS	16	C	3
66	.01515	SS	48	24	44	—	—	48*	SS	16	C	2
69	.01449	24	46	48*	SS	—	—	—	—	16	C	2
70	.01428	24	56	—	—	—	—	64*	SS	16	B	9

** This gear to mesh with gear in gear box.*

THREAD CUTTING

on the

12-INCH STANDARD CHANGE GEAR LATHE

This section deals with the change gear train and its proper set-up for cutting various sizes of threads.

The 12" Standard change gear lathe comes equipped with change gears and threading dial for cutting threads in the following standards: National Coarse (U.S.S.), National Fine (S.A.E.), Acme , Square, and Whitworth. Gear set-ups for standard threads are shown on the pictorial threading chart on the inside of the change-gear guard. (Fig 168). Figure 170 is an actual-size reproduction of this threading chart. Gear data for odd-size threads are given in Table I, page 150. Metric threads may also be cut with the standard change gears furnished.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

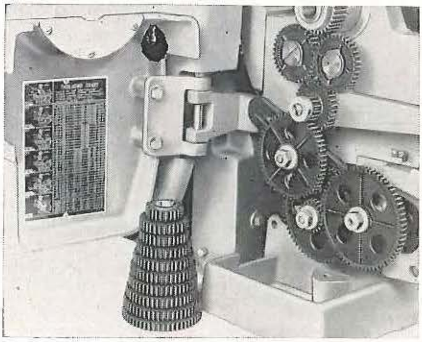


FIG. 168
Left end of lathe with gear guard open, showing change gears, gear train, and location of threading chart.

READING THE GEAR CHARTS

To simplify gear set-ups, the three different gear bracket positions have been assigned letters as shown in Figure 169. These designations will be found on the lathe threading chart as well as in all of the following gear data.

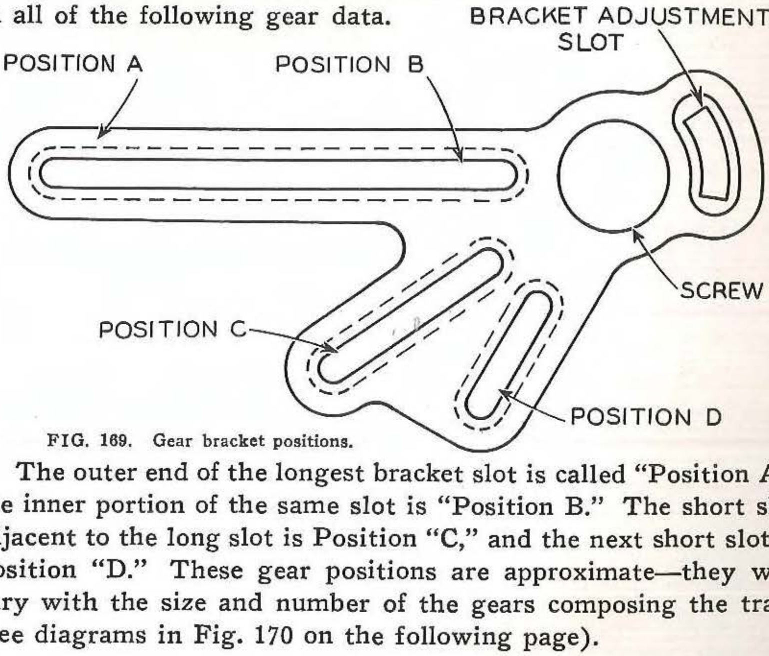


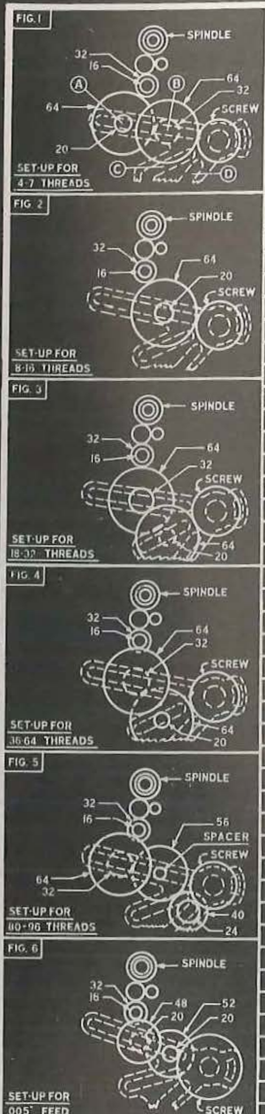
FIG. 169. Gear bracket positions.

The outer end of the longest bracket slot is called "Position A," the inner portion of the same slot is "Position B." The short slot adjacent to the long slot is Position "C," and the next short slot is Position "D." These gear positions are approximate—they will vary with the size and number of the gears composing the train (see diagrams in Fig. 170 on the following page).

CHANGE GEAR STUD ASSEMBLY

Before setting up a train of change gears, examine one of the change gear stud assemblies which hold the change gears to the gear bracket (Fig. 171). Each stud assembly has an outer gear bushing long enough to accommodate two gears. The gear bushing has a double key which fits into the keyways in the gears. The gear bushing and two gears fit over a stud bushing, and the assembly is bolted to the gear bracket. The washer is a bearing for the outer end of the gear bushing.

THREADING CHART FOR STANDARD CHANGE LATHES



THREADING CHART

A-B-C-D ARE GEAR STUD POSITIONS
B = BACK POSITION (TOWARD HEADSTOCK)
F = FRONT POSITION (AWAY FROM HEADSTOCK)
I = IDLER GEAR S = SPACER GEAR
XXS = STEEL SPACER — = BLANK

THREADS PER INCH	GEAR ON SCREW	POSITION D		POSITION C		POSITION B		POSITION A		COMPOUND TUMBLER GEAR	FIG.
		B	F	B	F	B	F	B	F		
4	32F	—	—	—	—	24	48	64I	20S	32	1
4.5	36F	—	—	—	—	24	48	64I	20S	32	1
5	40F	—	—	—	—	32	64	64I	20S	32	1
5.5	44F	—	—	—	—	32	64	64I	20S	32	1
6	48F	—	—	—	—	32	64	64I	20S	32	1
6.5	52F	—	—	—	—	32	64	64I	20S	32	1
7	56F	—	—	32	64	—	—	64I	20S	32	1
8	32B	—	—	—	—	64I	20S	—	—	32	2
9	36B	—	—	—	—	64I	20S	—	—	32	2
10	40B	—	—	—	—	64I	20S	—	—	32	2
11	44B	—	—	—	—	64I	20S	—	—	32	2
11.5	46B	—	—	—	—	64I	20S	—	—	32	2
12	48B	—	—	—	—	64I	20S	—	—	32	2
13	52B	—	—	—	—	64I	20S	—	—	32	2
14	56B	—	—	—	—	64I	20S	—	—	32	2
16	64B	—	—	—	—	64I	20S	—	—	32	2
18	36F	—	—	20S	64I	—	—	64	32	32	3
20	40F	—	—	20S	64I	—	—	64	32	32	3
22	44F	—	—	20S	64I	—	—	64	32	32	3
23	46F	—	—	20S	64I	—	—	64	32	32	3
24	48F	—	—	20S	64I	—	—	64	32	32	3
26	52F	—	—	20S	64I	—	—	64	32	32	3
27	54F	—	—	20S	64I	—	—	64	32	32	3
28	56F	—	—	20S	64I	—	—	64	32	32	3
32	64F	—	—	20S	56I	—	—	64	32	32	3
36	36B	—	—	64I	20S	24	48	—	—	16	4
40	40B	—	—	64I	20S	32	64	—	—	16	4
44	44B	—	—	64I	20S	32	64	—	—	16	4
48	48B	—	—	64I	20S	32	64	—	—	16	4
56	56B	—	—	64I	20S	32	64	—	—	16	4
64	64B	—	—	64I	20S	—	—	24	48	16	4
72	64B	—	—	—	—	24	54	20	56	16	6
80	48F	40	24	—	—	52I	XXS	32	64	16	5
96	48F	40	20	56I	XXS	—	—	32	64	16	5
FEED PER REVOLUTION OF SPINDLE											
.0024	64B	20	56	52	20	—	—	36	64	16	—
.0035	64F	—	—	—	—	64	20	20	56	16	—
.005	64F	—	—	—	—	52	20	20	48	16	6
.0071	64F	40	20	—	—	48I	XXS	20	44	16	5
.0087	64F	—	—	—	—	56	20	24	48	16	—

Fig. 170. Threading chart for cutting standard threads between 4 and 96 per inch. For additional gear train data, refer to Table I, 150.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
1 - INDEX
12 - NOTES

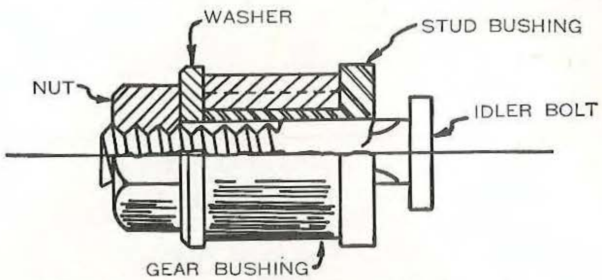


FIG. 171. Cross section of change gear stud assembly.

Notice that in order to make this assembly complete, two gears must be mounted on the gear bushing at one time. When both of the gears on a gear bushing mesh with other gears in the train, they form a "compound" gear assembly. When only one of two gears on a gear bushing meshes with the other gears in the train, it is called an "idler." The smaller gear, which is mounted on the gear bushing with an idler, is called a "spacer" gear and does not mesh with any gear in the train (see Fig. 173).

GEAR CLEARANCE

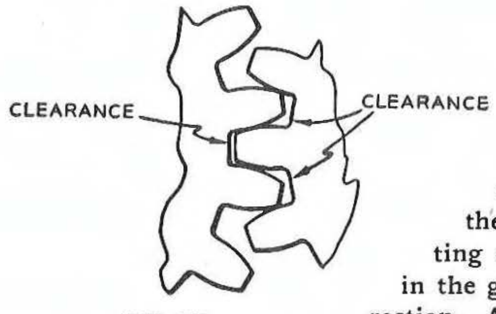


FIG. 172
Proper gear clearance.

When setting up the gear train, be sure to allow sufficient clearance between two meshing gears (Fig. 172). Gear clearance does not reduce the accuracy of a thread cutting operation, because all play in the gears is taken up in one direction. A method often used to obtain proper gear clearance is: (1) Place a sheet of thick writing paper between the teeth of the two meshing gears, (2) tighten gears in position, and (3) remove paper. A small amount of grease, preferably graphite grease, applied to gear teeth will often aid in obtaining smoother, more quiet operation.

THE REVERSING MECHANISM

Right hand threads are cut with the carriage traveling toward the headstock. Left hand threads are cut with the carriage traveling toward the tailstock.

Whenever a new gear train has been set up, shift the reverse feed lever to test the direction of the carriage travel. Because some set-ups are simple-geared and some are compounded, the carriage travel will not necessarily be to the right when the reverse lever is shifted to the right. ***Always test the direction of carriage travel before starting to cut thread.***

After the reversing lever has been lifted to the proper position, it should not be moved until the thread has been completed. *This is especially important because a shift in the lever position destroys the relation between the threading dial and the lathe spindle and causes splitting of the thread.*

GEAR TRAINS FOR STANDARD THREADS

The following pages give detailed instructions for mounting gears for the more common thread sizes. Refer to these pages and the lathe threading chart when making set-ups. "Back Position" of a bushing or the screw stub means the position *toward* the headstock. "Front Position" is the position *away from* the headstock. The gear bracket is tightened in position by locking the nut behind large washer on the inside of the "Bracket Adjustment Slot" (Fig. 169.)

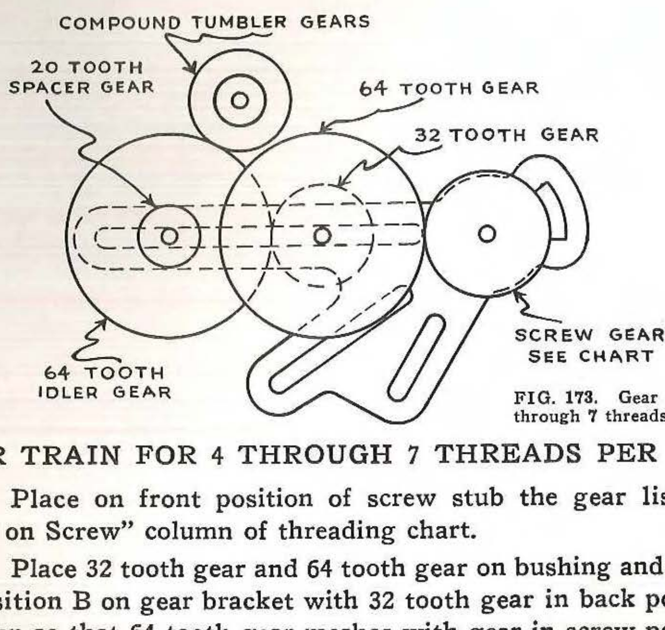


FIG. 173. Gear set-up for through 7 threads per inch.

GEAR TRAIN FOR 4 THROUGH 7 THREADS PER INCH

1. Place on front position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 32 tooth gear and 64 tooth gear on bushing and mount in Position B on gear bracket with 32 tooth gear in back position. Tighten so that 64 tooth gear meshes with gear in screw position.

Exception: When cutting 4 and 4.5 threads per inch, the 32 tooth gear and 64 tooth gear are replaced by 24 and 48 tooth gears respectively. When cutting 7 threads per inch, the 32 and 64 tooth gears are used in Position C instead of Position B.

3. Place 64 tooth gear and spacer on a bushing and mount in Position A with 64 tooth gear in back position. Tighten so that 64 tooth gear meshes with the 32 tooth gear in Position B. The 64 tooth gear is an idler; the 20 tooth gear is a spacer.

4. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with the 32 tooth compound tumbler gear.

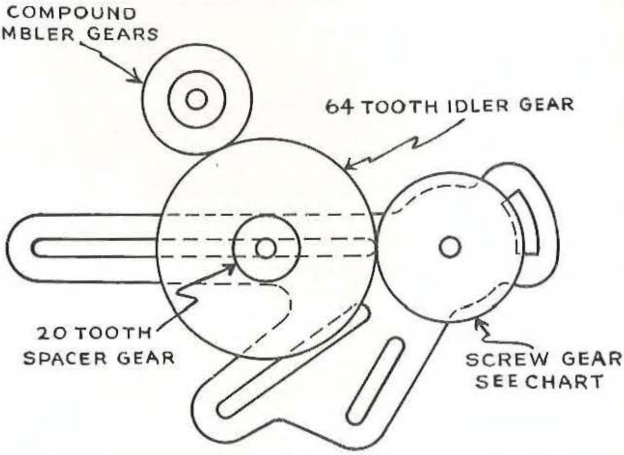


Fig. 174. Gear set-up for 8 through 16 threads per inch.

GEAR TRAIN FOR 8 THROUGH 16 THREADS PER INCH

1. Place on back position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 64 tooth gear and 20 tooth gear on bushing in Position B with 64 tooth gear in back position. Tighten so that 64 tooth gear meshes with gear in screw position. The 64 tooth gear is an idler; the 20 tooth gear is a spacer.

3. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position B meshes with 32 tooth compound tumbler gear.

GEAR TRAIN FOR 18 THROUGH 32 THREADS PER INCH

(See Fig. 175, page 139)

1. Place on front position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 20 tooth gear and 64 tooth gear on bushing and mount in Position C with 20 tooth gear in back position. Tighten so that 64 tooth gear meshes with gear in screw position. The 64 tooth gear is an idler; the 20 tooth gear is a spacer.

Exception: When cutting 32 threads per inch, substitute a 56 tooth gear for the 64 tooth gear..

3. Place 64 tooth gear and 32 tooth gear on bushing and mount in Position A with 64 tooth gear in back position. Tighten so that 32 tooth gear meshes with 64 tooth gear in Position C. (Continued Page 139)

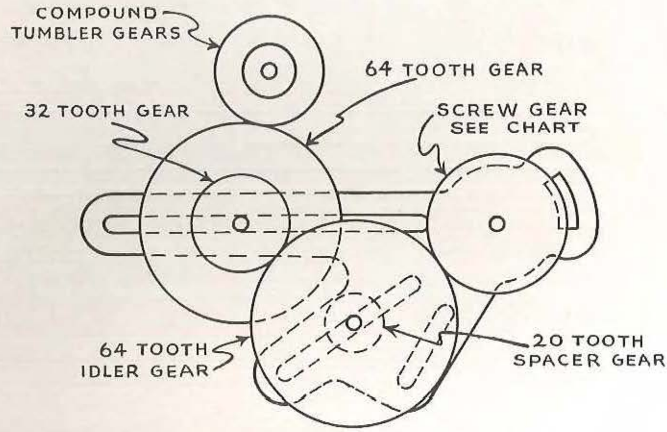


FIG. 175. Gear set-up for 18 through 32 threads per inch.

4. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with the 32 tooth spindle stud gear.

GEAR TRAIN FOR 36 THROUGH 64 THREADS PER INCH

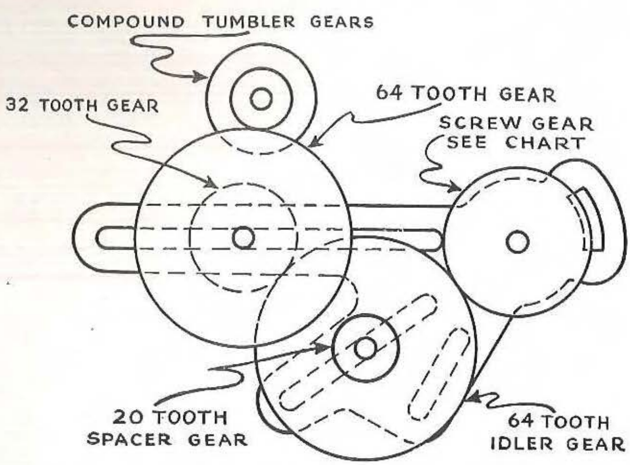


FIG. 176. Gear set-up for 36 through 64 threads per inch (see page 140).

1. Place in back position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 64 tooth gear and 20 tooth gear on bushing and mount in Position C with 64 tooth gear in back position. Tighten so that 64 tooth gear meshes with gear in screw position. The 64 tooth gear is an idler; the 20 tooth is a spacer.

3. Place 32 tooth gear and 64 tooth gear on bushing and mount in Position B with 32 tooth gear in back position. Tighten so that 32 tooth gear meshes with 64 tooth gear in Position C.

Exceptions: (1) When cutting 36 threads per inch substitute 24 tooth gear for 32 tooth gear and 48 tooth gear for 64 tooth gear. (2) When cutting 64 threads per inch omit Position B and mount 48 tooth gear and 24 tooth gear in Position A with 24 tooth gear in back position.

4. Swing entire gear bracket upward so that the 64 tooth gear in Position B meshes with the 16 tooth compound tumbler gear.

GEAR TRAIN FOR 72 THREADS PER INCH

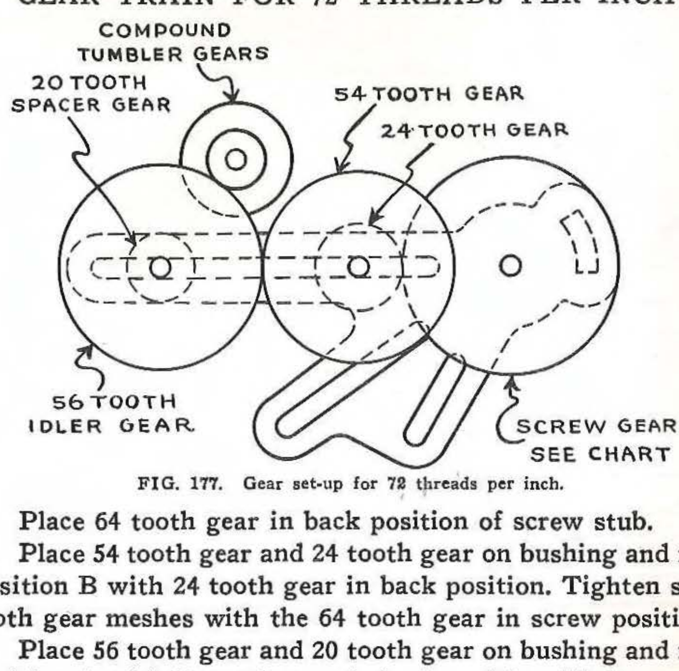


FIG. 177. Gear set-up for 72 threads per inch.

1. Place 64 tooth gear in back position of screw stub.
2. Place 54 tooth gear and 24 tooth gear on bushing and mount in Position B with 24 tooth gear in back position. Tighten so that 24 tooth gear meshes with the 64 tooth gear in screw position.
3. Place 56 tooth gear and 20 tooth gear on bushing and mount in Position A with 20 tooth gear in back position. Tighten so that 56 tooth gear meshes with 54 tooth gear in Position B. The 56 tooth gear is an idler; the 20 tooth gear is a spacer.
4. Swing entire gear bracket upward and tighten so that the 56 tooth gear in Position A meshes with the 16 tooth compound tumbler gear.

GEAR TRAIN FOR 80 AND 96 THREADS PER INCH

1. Place 48 tooth gear on front position of screw stub.
2. Place 40 tooth gear and 24 tooth gear on bushing in Position D with 40 tooth gear in back position. Tighten so that 24 tooth gear meshes with 48 tooth gear on screw stub.

Exception: When cutting 96 threads per inch substitute 20 tooth gear for 24 tooth gear.

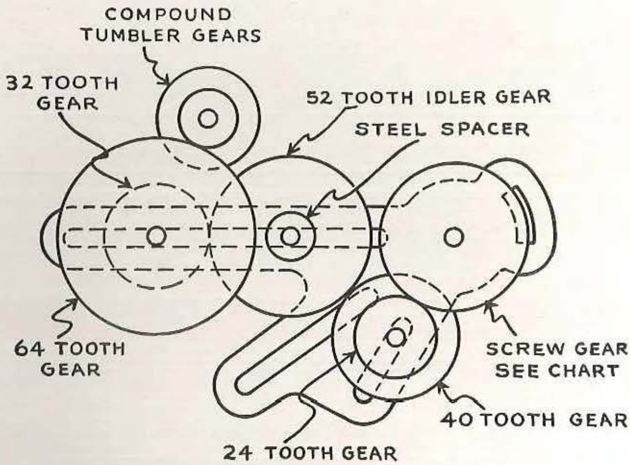


FIG. 178. Gear set-up for 80 and 96 threads per inch.

3. Place 52 tooth gear and steel spacer on bushing in Position B with 52 tooth gear in back position. Tighten so that 52 tooth gear meshes with 40 tooth gear in Position D.

Exception: When cutting 96 threads per inch substitute 56 tooth gear 52 tooth gear and mount in Position C instead of Position B.

4. Place 32 tooth gear and 64 tooth gear on bushing in Position A with 32 tooth gear in back position. Tighten so that 32 tooth gear meshes with 52 tooth gear in Position B.

5. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with the 16 tooth compound tumbler gear.

THREAD CUTTING SPEEDS: The beginner in thread cutting should adjust belts to obtain a speed of 28 R.P.M. (Manual, page 45). This slow speed allows plenty of time to engage and disengage the half-nut lever. After more experience in cutting threads, higher speeds can be used up to approximately 1/3 or 1/2 the speeds recommended for turning the various materials (Manual, Part 4).

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

THE THREADING DIAL

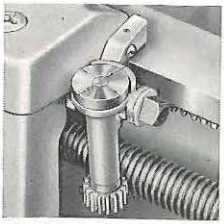


FIG. 179
Threading Dial

The threading dial (Figs. 179 and 180) performs an important function by indicating the proper time to engage the half-nut lever so that the tool will enter the same groove of the thread for each cut. Without the threading dial it would be necessary to reverse the motor at the end of each cut and "wind" the tool out of the thread — a cumbersome method little used except when cutting metric and special fractional threads.

RULES FOR USING THE THREADING DIAL

When cutting an *even-numbered thread* (such as 12, 14, 16, 32, etc. per inch), engage the half-nut lever for the first cut when the stationary mark on the outside of the threading dial is in line with any one of the four marks on the rotating portion of the dial. Any one of the four dial markings may be used for following cuts.

When cutting *odd-numbered threads* (such as 7, 9, 11, 23, 27, etc. per inch), engage the half-nut lever for the first cut when the stationary mark on the threading dial is in line with *either "1" or "2"* on the rotating portion. Either the "1" or "2" dial marking may be used for following cuts.

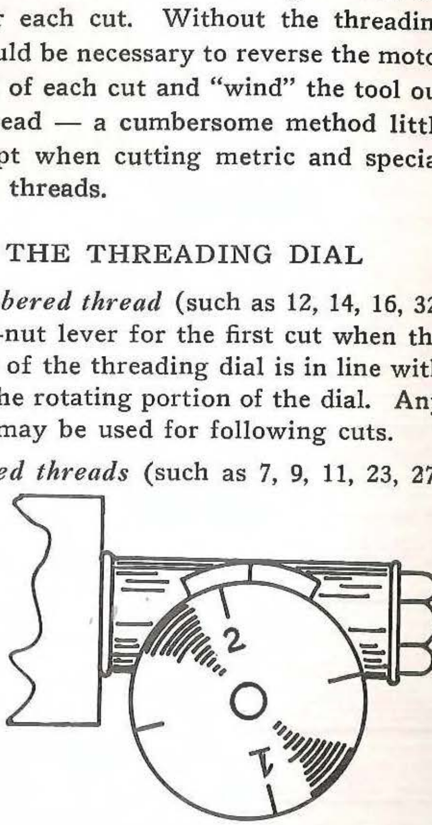


FIG. 180

Threading dial showing mainmarkings. Other lines may be marked in by the operator as needed.

When cutting *half-numbered threads* (such as $4\frac{1}{2}$, $5\frac{1}{2}$, $6\frac{1}{2}$, $11\frac{1}{2}$, etc. per inch), engage half-nut lever *at the same mark* on the threading dial for each cut of the thread.

USING THE THREADING DIAL FOR MULTIPLE THREADS

Although only four marks are cut into the top of the threading dial, there are actually sixteen different positions at which the half-nut lever can be engaged. Figure 181 shows the intermediate points between the four mainmarkings. These points can be marked with pencil, or the positions easily estimated. In the following paragraphs, Lead in Threads Per Inch is equal to 1 divided by Lead in Inches.

CUTTING DOUBLE THREADS WITH LEAD IN THREADS PER INCH DIVISIBLE BY FOUR, BUT NOT BY EIGHT (4, 12, 20, 28, etc.)

A single thread of this lead is cut by engaging the half nuts at any of the four mainmarkings on the threading dial or at any of the four "b" positions. To cut the second groove of a double thread, the half nuts are engaged at any of the "a" or "c" positions.

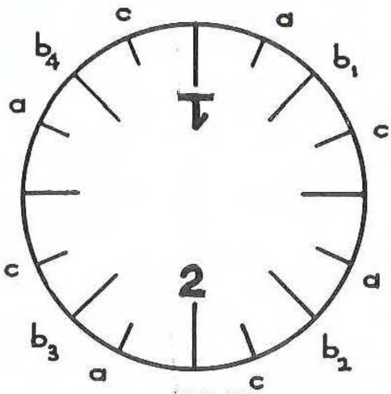


FIG. 181
Intermediate positions on threading dial which can be used for cutting. The numbers "1" and "2" are marked; the lettered positions may be marked as needed.

Example: To Cut a Double Thread with a Pitch of 1/24 inch and a Lead of 1/12 inch. Set up the change gears for the lead in threads per inch (12, not 24). Engage the half nut lever for the first cut when the stationary mark on the outside of the threading dial is in line with any one of the four main marks on the rotating portion of the dial. Then return to the starting point and engage half nuts at any one of the "a" or "c" positions, taking the first cut on the second groove of the thread. The compound rest feed remains *at one setting* until both grooves have been cut to the same depth.

CUTTING DOUBLE AND QUADRUPLE THREADS WITH LEADS IN THREADS PER INCH DIVISIBLE BY TWO, BUT NOT BY FOUR (6, 10, 14, 18, etc.)

A single thread of this lead is cut only by engaging the half nut lever at any one of the four mainmarkings on the threading dial. To cut the second groove of the double thread, the half nuts are engaged at any one of the "b" positions, and the cutting operation is the same as in the preceding paragraph.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

For quadruple threads of this lead engage the half nuts at any of the four mainmarkings for the first groove, at any of the "a" positions for the second groove, at any of the "b" positions for the third groove, and at any of the "c" positions for the fourth groove. The setting of the compound rest feed is changed only after each of the four grooves has been cut to the depth of setting.

CUTTING DOUBLE AND QUADRUPLE THREADS WITH LEAD IN THREADS PER INCH DIVISIBLE BY ONE, BUT NOT BY TWO (ODD NUMBERS)

A single thread of this lead is cut by engaging the half nuts in position "1" or position "2." To cut the second groove of the double thread, the half nuts are engaged at either of the unnumbered marks on the threading dial. The cutting operation is the same as in the preceding paragraph.

For quadruple threads of this lead engage the half nuts at position "1" for the first groove, at position "b_1" for the second groove, at either of the unnumbered lines on the dial for the third groove, and at "b_2" for the fourth groove. The setting of the compound rest feed is changed only after each of the four grooves has been cut to the depth of setting.

CUTTING MULTIPLE THREADS BY SLIPPING TEETH ON THE SPINDLE GEAR

Double and quadruple threads can also be cut by "slipping teeth" on the compound gear. This practice is not so common as the use of the threading dial, but is not complicated.

To cut multiple threads by slipping teeth on the compound gear: cut the complete first groove to a minor diameter dependent upon pitch of the desired thread. The change gear train should be arranged for the desired lead. It is important to use the same 0 point of reference to cut each thread — be sure to remember this point during the cutting operations.

Refer to the table on page 145, then slip the required number of teeth by marking adjacent teeth on the compound gear and the gear meshing with the compound gear. Drop the entire gear bracket low enough to disengage the gears and turn the compound gear forward the proper number of teeth by rotating spindle by hand. Raise the gear bracket so that the previously marked gear tooth meshes with the newly selected compound gear tooth.

To Cut Double Threads:—Slip 16 teeth to cut the second groove.

To Cut Quadruple Threads:—Slip 8 teeth to cut the second groove, 8 teeth more to cut the third groove, and 8 teeth more to cut the fourth groove.

Each thread groove is cut to its complete depth and finished before starting the next groove.

GEAR TRAINS FOR CARRIAGE FEEDS

The automatic longitudinal carriage feed per spindle revolution is obtained by setting up the gear train in the same manner as for thread cutting (pages 134 to 142). The feed in inches is equal to $\frac{1}{\text{threads per inch}}$. For example, a feed of .0087 inch requires the gear set-up as 114.9 threads per inch.

The five most common carriage feeds, as shown in the threading chart (page 135), are .0087, .0071, .0050, .0035, and .0024 inch per spindle revolution. Refer to the threading chart and the following paragraphs when changing these gear set-ups. Table II on page 152 includes gear set-ups for other carriage feeds obtainable with the standard set of gears.

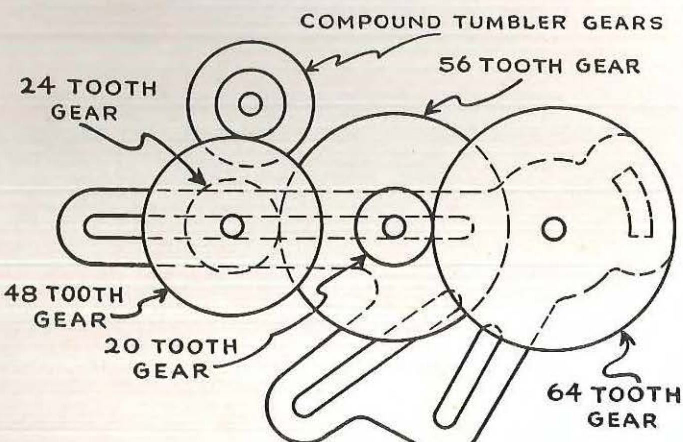


FIG. 182. Gear set-up for .0087 inch carriage feed (see page 146)

GEAR TRAIN FOR .0087 INCH CARRIAGE FEED

(See Fig. 182, page 145)

1. Place 64 tooth gear in front position on screw stub.
2. Place 56 tooth gear and 20 tooth gear on bushing in Position D, with 56 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.
3. Place 24 tooth gear and 48 tooth gear on bushing in Position A, with 24 tooth gear in back position. Tighten so that 24 tooth gear meshes with 56 tooth gear in Position B.
4. Swing entire gear bracket upward and tighten so that 48 tooth gear in Position A meshes with 16 tooth compound tumbler gear.

GEAR TRAIN FOR .0071 INCH CARRIAGE FEED

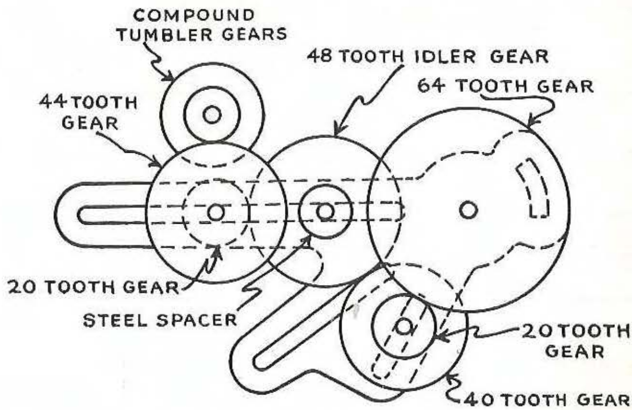


FIG.. 183. Gear set-up for .0071 inch carriage feed.

1. Place 64 tooth gear in front position on screw stub.
2. Place 20 tooth gear and 40 tooth gear on bushing in Position D, with 40 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.
3. Place 48 tooth gear and steel spacer on bushing in Position B, with 48 tooth gear in back position. Tighten so that 48 tooth gear meshes with 40 tooth gear in Position D.
4. Place 44 tooth gear and 20 tooth gear on bushing and mount in Position A with 20 tooth gear in back position. Tighten so that 20 tooth gear meshes with 48 tooth gear in Position B.
5. Swing entire gear bracket upward and tighten so that 44 tooth gear in position A meshes with 16 tooth compound tumbler gear.

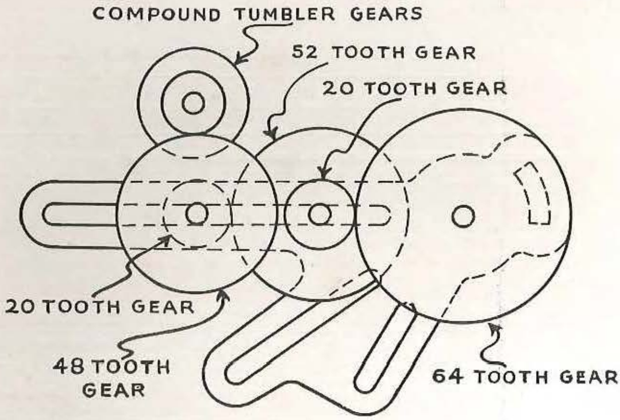


FIG. 184. Gear set-up for .0050 inch carriage feed.

GEAR TRAIN FOR .0050 INCH CARRIAGE FEED

1. Place 64 tooth gear in front position on screw stub.

2. Place 52 tooth gear and 20 tooth gear on bushing in Position B, with 52 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.

3. Place 48 tooth gear and 20 tooth gear on bushing in Position A, with 20 tooth gear in back position. Tighten so that 20 tooth gear meshes with 52 tooth gear in Position B.

4. Swing entire gear bracket upward and tighten so that 48 tooth gear in Position A meshes with 16 tooth compound tumbler gear.

GEAR TRAIN FOR .0060 INCH CARRIAGE FEED

The gear set-up for .0060 inch carriage feed is the same as that for the .0050 inch feed except that a 24 tooth gear is substituted for 20 tooth gear in back position at A.

GEAR TRAIN FOR .0035 INCH CARRIAGE FEED

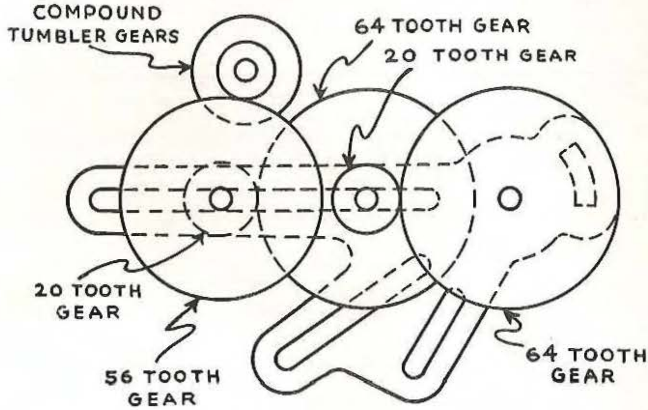


FIG. 185. Gear set-up for .0035 inch carriage feed.

1. Place 64 tooth gear in front position on screw stub.

2. Place 64 tooth gear and 20 tooth gear on bushing in Position B, with 64 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.

3. Place 20 tooth gear and 56 tooth gear on bushing in Position A, with 20 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear in Position B.

4. Swing entire gear bracket upward and tighten so that 56 tooth gear in Position A meshes with 16 tooth compound tumbler gear.

GEAR TRAIN FOR .0024 INCH CARRIAGE FEED

(See Fig. 186, page 149)

1. Place 64 tooth gear in back position on screw stub.

2. Place 20 tooth gear and 56 tooth gear on bushing in Position D, with 20 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.

3. Place 52 tooth gear and 20 tooth gear on bushing in Position C, with 52 tooth gear in back position. Tighten so that 20 tooth gear meshes with 56 tooth gear in Position D.

4. Place 36 tooth gear and 64 tooth gear on bushing in Position A, with 36 tooth gear in back position. Tighten so that 36 tooth gear meshes with 52 tooth gear in Position C.

5. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with 16 tooth spindle stud gear.

GEAR TRAIN FOR .0024 INCH CARRIAGE FEED

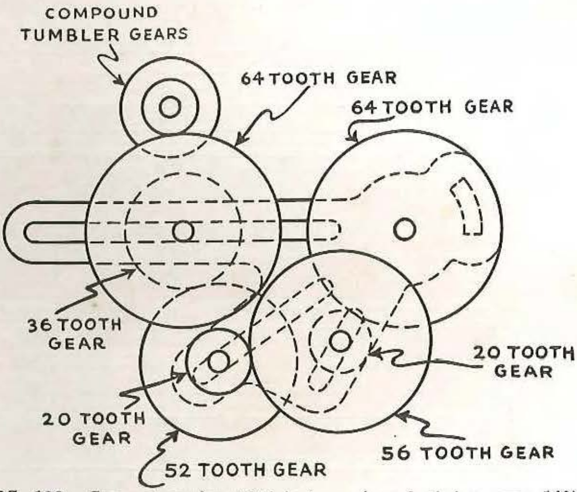


FIG. 186. Gear set-up for .0024 inch carriage feed (see page 148).

SPECIAL THREADS AND FEEDS

Engineers charted over a thousand threads and feeds between the coarsest thread and the finest feed. Tables I and II this section give proper set-ups for a wide variety of special threads and feeds. Most of these set-ups are exact some are accurate to the limits mentioned. Table III gives set-ups for metric threads with pitch between 0.5 and 7.0 millimeters.

TABLE I—GEAR SET-UPS FOR THREADS FORM 1.25 THROUGH 79 PER INCH NOT SHOWN ON THE THREADING CHART

The threading dial can be used when cutting threads below marked "Exact" in the column under "Accuracy per inch", except for the 1.25, 2.25, 2,75, 3.25, and 3.75 threads. These threads and all others must be cut in the same manner as metric threads (See Page 109). Extra gears available from the factory at nominal cost.

Threads per inch	Accuracy per inch	Gear on Screw	Position D B	Position D F	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Spindle Stud Gear	Note
1.25	Exact	20*B*	48	24	—	—	32	64	64*I*	20*S*	32	
1.5	Exact	24*B*	48	24	—	—	32	64	64*I*	20*S*	32	*f*
2	Exact	24*B*	48	32	—	—	32	64	56*I*	20*S*	32	
2.25	1/2250	24*B*	48	36	—	—	32	64	64*I*	20*S*	32	
2.5	Exact	20*B*	48	32	*SS*	44*I*	24	32	64*I*	36*S*	32	*
2.75	Exact	24*B*	48	44	—	—	20	40	64*I*	32*S*	32	
3	Exact	24*F*	—	—	32	64	—	—	64*I*	20*S*	32	
3.25	Exact	52*B*	—	—	40	20	24	48	64*I*	32*S*	32	
3.5	Exact	56*B*	—	—	40	20	24	48	64*I*	32*S*	32	
3.75	Exact	20*F*	48	64	—	—	—	—	64*I*	24*S*	32	
7.5	Exact	40*F*	—	—	—	—	36	48	64*I*	20*S*	32	
8.5	1/3500	44*B*	—	—	36	20	24*S*	48*I*	64	46	32	
9.5	1/1900	52*B*	—	—	40	24	20*S*	44*I*	56	46	32	
10.5	Exact	56*F*	—	—	48	64	—	—	52*I*	20*S*	32	
12.5	Exact	40*F*	—	—	—	—	40	32	64*I*	20*S*	32	*d*
15	Exact	54*F*	—	—	—	—	40	36	64*I*	20*S*	32	
17	1/4250	40*B*	—	—	46	64	—	—	44	36	32	
19	1/1900	40*B*	54	44	—	—	*SS*	52*I*	56	24	32	
21	Exact	56*F*	—	—	54	36	—	—	64*I*	20*S*	32	
25	Exact	40*B*	32	40	—	—	*SS*	56*I*	64	32	32	*d*
29	1/1450	46*B*	40	36	—	—	*SS*	64*I*	56	20	32	
30	Exact	54*B*	—	—	—	—	36	40	20*S*	64*I*	16	
31	Exact	56*B*	20	36	—	—	24*S*	54*I*	64	52	32	
33	Exact	48*B*	—	—	—	—	32	44	24*S*	64*I*	16	
34	1/850	48*B*	20	46	—	—	24*S*	56*I*	64	52	32	
35	Exact	56*B*	—	—	24	48	—	—	40	32	32	
37	1/1850	54*B*	—	—	20	48	—	—	64	56	32	
38	1/1900	52*B*	—	—	20	48	—	—	56	46	32	
39	Exact	48*B*	—	—	—	—	32	52	20*S*	64*I*	16	
41	1/1025	64*B*	—	—	40	64	—	—	32	20	32	
42	Exact	48*B*	—	—	32	56	20*S*	64*I*	—	—	16	
43	1/2100	46*B*	32	46	—	—	*SS*	56*I*	52	20	32	*g*

Table I—Continued

Threads per inch	Accuracy per inch	Gear on Screw	Position D B	Position D F	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Compound Tumbler Gear	Note
45	Exact	54B	—	—	32	64	—	—	40	24	32	
46	Exact	48B	—	—	24	46	20S	64I	—	—	16	
47	1/230	52B	—	—	20	36	SS	44I	48	24	32	
49	Exact	56B	—	—	32	64	—	—	56	32	32	*h*
50	Exact	40F	40	32	—	—	44I	SS	24	48	16	*d*
51	1/1000	48F	36	20	44	52	—	—	24S	54I	16	
52	Exact	48B	—	—	—	—	24	52	20S	64I	16	
53	1/6600	48F	36	20	44	54	—	—	24S	64I	16	
54	Exact	48B	—	—	64I	20S	—	—	24	54	16	
55	1/270	48B	—	—	20	46	24S	64I	—	—	16	
57	1/8000	56B	—	—	20	56	—	—	64	44	32	*h*
58	1/580	46B	20	36	—	—	SS	64I	56	20	32	
59	1/1900	46B	24	44	—	—	SS	54I	56	20	32	
60	Exact	48F	48	24	32	40	—	—	20S	54I	16	*l*
61	1/1500	54F	52	46	—	—	46I	SS	24	48	16	
62	1/620	48F	54	46	—	—	46I	SS	20	44	16	
63	Exact	64B	—	—	—	—	24	54	56	32	32	
65	Exact	48F	40	24	—	—	46I	SS	32	52	16	
66	Exact	64B	—	—	24	54	—	—	44	24	32	*f*
67	1/160	64F	40	24	32	40	—	—	20S	64I	16	*d*
68	1/490	64F	40	36	—	—	56I	SS	24	46	16	
69	Exact	64B	—	—	24	54	—	—	46	24	32	*f*
70	Exact	48F	56	32	24	40	—	—	20S	64I	16	
71	1/118	64F	44	40	—	—	54I	SS	24	48	16	
73	1/730	48F	36	20	52I	24S	—	—	32	54	16	
74	1/180	64F	56	48	—	—	46I	SS	24	48	16	*l*
75	1/370	52B	20	36	—	—	SS.	64I	64	20	32	
76	1/760	44F	46	24	20	36	—	—	32S	64I	16	
77	Exact	48F	44	24	—	—	46I	SS	32	56	16	
78	Exact	48F	40	20	—	—	46I	SS	32	52	16	
79	1/2600	54B	—	—	20	54	—	—	52	24	32	*j*

SYMBOLS:

d—extra 40 tooth gear
f—extra 24 tooth gear
g—extra 46 tooth gear
h—extra 56 tooth gear
j—extra 54 tooth gear
l—extra 48 tooth gear

F—position away from headstock
B—position toward headstock
I—idler gear (page 136)
S—spacer gear (page 136)
SS—Double keyway spacer

*—extra sleeve, bushing and bolt assembly—available at the factory.

TABLE II—GEAR SET-UPS FOR CARRIAGE FEEDS

Six different carriage feeds between .0027 and .0085 inch per spindle revolution are available on the Master Craftsman lathe in addition to the five most common feeds which are pictured and described in detail between pages 135 and 149. When the material or job requires a certain feed, refer to the table below. Extra gears are available from the factory at nominal cost.

Inches Feed in	Threads per inch	Gear on Screw	Position D B	Position D F	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Compound Tumbler Gear	Note
.0085	118.8	48*F*	44	20	—	—	48*I*	*SS*	24	54	16	*I*
.008	124.8	48*F*	48	20	—	—	46*I*	*SS*	24	52	16	*I*
.006	166.4	64*F*	—	—	—	—	52	20	24	48	16	
.004	249.6	64*B*	20	52	48	20	32	40	*SS*	64*I*	16	*
.0032	344.06	64*B*	20	56	48	20	—	—	40	64	16	
.0027	372.7	64*B*	20	56	52	20	—	—	40	64	16	

SYMBOLS:

*—extra sleeve, bushing and bolt assembly

I—extra 48 tooth gear

F—position away from headstock

B—position toward headstock

I—idler gear (page 136)

SS—double keyway spacer

TABLE III — GEAR SET-UPS FOR METRIC THREADS

Two of the standard change gears furnished with the Master Craftsman Lathe, the 52 tooth gear and the 44 tooth gear, combine to give a ratio of 44/52 or .846154, which is an almost exact function of 2.54, the English to Metric ratio. Thus, it is possible to cut metric threads accurate to the extremely close limits of 1 part in 3000.

Refer to page 109 when cutting metric threads.

Pitch in MM.	Gear on Screw	Position D		Position C		Position B		Position A		Compound Tumbler Gear
		B	F	B	F	B	F	B	F	
.5	48*F*	40	44	—	—	52*I*	*SS*	24	56	16
.75	44*F*	40	52	—	—	54*I*	20*S*	24	48	16
1	64*B*	32	40	—	—	*SS*	54*I*	56	44	32
1.25	44*B*	—	—	52	48	—	—	20*S*	64*I*	16
1.5	44*B*	—	—	52	40	—	—	20*S*	64*I*	16
1.75	36*B*	52	56	—	—	20*S*	46*I*	48	32	32
2	40*F*	44	48	—	—	52	36	20*S*	64*I*	16
2.5	44*B*	—	—	—	—	52	24	20*S*	64*I*	16
3	44*B*	—	—	—	—	52	20	24*S*	64*I*	16
3.5	44*F*	48	56	—	—	52	20	24*S*	64*I*	16
4	40*B*	48	44	—	—	36	52	64*I*	20*S*	32
4.5	40*B*	54	44	—	—	36	52	64*I*	20*S*	32
5	24*F*	—	—	—	—	44	52	64*I*	20*S*	32
5.5	20*F*	—	—	—	—	48	52	64*I*	24*S*	32
6	20*F*	—	—	—	—	44	52	64*I*	24*S*	32
7	24*B*	52	44	—	—	40	56	64*I*	20*S*	32

SYMBOLS:

F—position away from headstock

B—position toward headstock

I—idler gear (page 136)

S—spacer gear (page 136)

SS—double keyway spacer

THREAD CUTTING

on the

SIX-INCH LATHE

This section deals with the change gear train and its proper set-up for cutting the various sizes of threads.

The 6″ lathe comes equipped with change gears and threading dial for cutting threads in the following standards: National Coarse (U.S.S.), National Fine (S.A.E.,) Acme, Square, and Whitworth. Gear set-ups for standard threads are shown on the pictorial threading chart on the inside of the change-gear guard. (Fig. 187). Figure 189 is an actual-size reproduction of this threading chart. Gear data for odd-size threads are given in Table I, page 171.

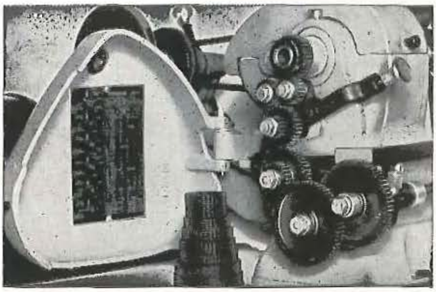


FIG. 187

Left end of lathe with gear guard open, showing change gears, gear train, and location of threading chart.

READING THE GEAR CHARTS

To simplify gear set-ups, the three different gear bracket positions have been assigned letters as shown in Figure 188. These designations will be found on the lathe threading chart as well as in all of the following gear data.

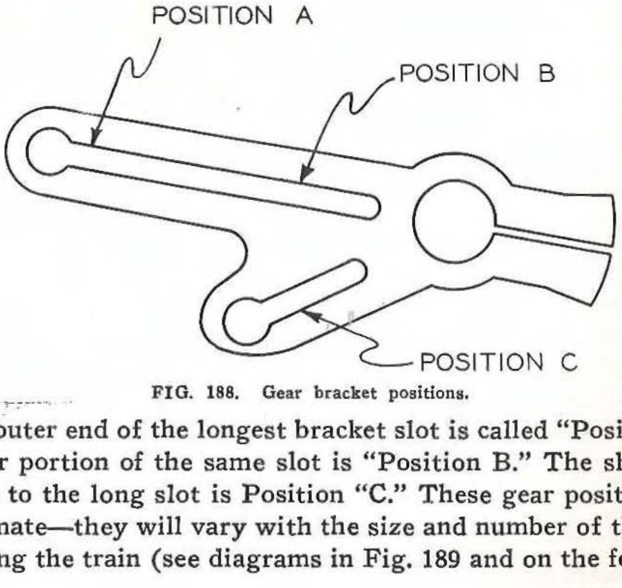


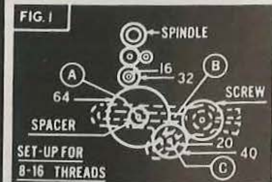
FIG. 188. Gear bracket positions.

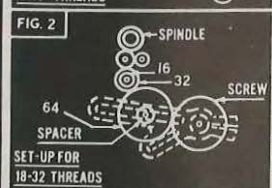
The outer end of the longest bracket slot is called "Position A," the inner portion of the same slot is "Position B." The short slot adjacent to the long slot is Position "C." These gear positions are approximate—they will vary with the size and number of the gears composing the train (see diagrams in Fig. 189 and on the following page).

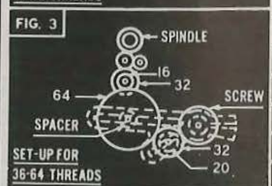
CHANGE GEAR STUD ASSEMBLY

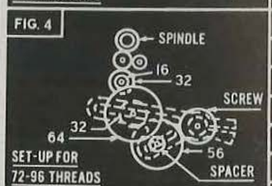
Before setting up a train of change gears, examine one of the change gear stud assemblies which hold the change gears to the gear bracket (Fig. 190). Each stud assembly has an outer gear bushing long enough to accommodate two gears. The gear bushing has a double key which fits into the keyways in the gears. The gear bushing and two gears fit over a stud bushing, and the assembly is bolted to the gear bracket. The washer is a bearing for the outer end of the gear bushing.

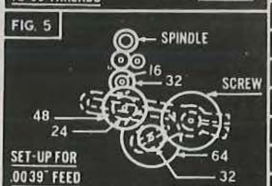
THREADING CHART FOR 6-INCH LATHES

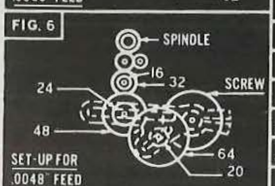












THREADING CHART

A-B-C ARE GEAR STUD POSITIONS
B=BACK POSITION (TOWARD HEADSTOCK)
F=FRONT POSITION (AWAY FROM HEADSTOCK)
I=IDLER GEAR — =BLANK
S=SPACER GEAR XXS=STEEL SPACER

THREADS PER INCH	GEAR ON SCREW	POSITION C B	POSITION C F	POSITION B B	POSITION B F	POSITION A B	POSITION A F	SPINDLE STUD GEAR	FIG.
8	32F	32	64	—	—	64I	XXS	32	1
9	36F	32	64	—	—	64I	XXS	32	1
10	40F	32	64	—	—	64I	XXS	32	1
11	44F	20	40	—	—	64I	XXS	32	1
11.5	46F	20	40	—	—	64I	XXS	32	1
12	48F	20	40	—	—	64I	XXS	32	1
13	52F	20	40	—	—	64I	XXS	32	1
14	56F	20	40	—	—	64I	XXS	32	1
16	64F	20	40	—	—	64I	XXS	32	1
18	36B	—	—	64I	XXS	—	—	32	2
20	40B	—	—	64I	XXS	—	—	32	2
22	44B	—	—	64I	XXS	—	—	32	2
24	48B	—	—	64I	XXS	—	—	32	2
27	54B	—	—	64I	XXS	—	—	32	2
28	56B	—	—	64I	XXS	—	—	32	2
32	64B	—	—	64I	XXS	—	—	32	2
36	36F	20S	32I	—	—	XXS	64I	16	3
40	40F	20S	32I	—	—	XXS	64I	16	3
44	44F	20S	32I	—	—	XXS	64I	16	3
48	48F	20S	32I	—	—	XXS	64I	16	3
56	56F	20S	32I	—	—	XXS	64I	16	3
64	64F	20S	32I	—	—	XXS	64I	16	3
72	36B	56I	XXS	—	—	32	64	16	4
80	40B	56I	XXS	—	—	32	64	16	4
96	48B	56I	XXS	—	—	32	64	16	4
FEED PER REVOLUTION OF SPINDLE									
.0024"	64F	64	20	—	—	24	48	16	5
.0039"	64F	64	32	—	—	24	48	16	5
.0048"	64B	20	64	—	—	48	24	32	6
.0078"	64B	32	64	—	—	48	24	32	6

FIG. 189. Threading chart for cutting standard threads between 8 and 96 per inch. For additional gear train data, refer to Table I, page 173.

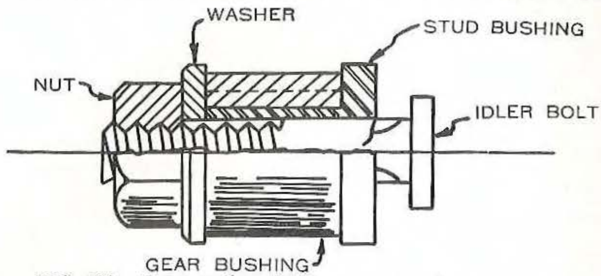


FIG. 190. Cross section of change gear stud assembly.

Notice that in order to make this assembly complete, two gears must be mounted on the gear bushing at one time. When both of the gears on a gear bushing mesh with other gears in the train, they form a "compound" gear assembly. When only one of two gears on a gear bushing meshes with the other gears in the train, it is called an "idler." The smaller gear, which is mounted on the gear bushing with an idler, is called a "spacer" gear and does not mesh with any gear in the train (see Fig. 195).

GEAR CLEARANCE

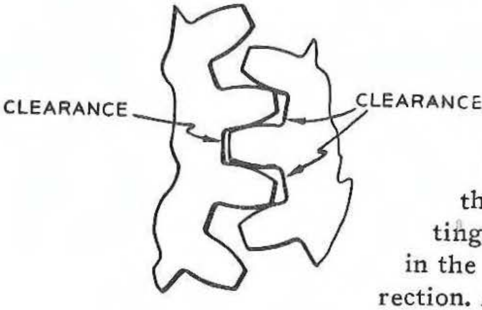


FIG. 191

When setting up the gear train, be sure to allow sufficient clearance between two meshing gears (Fig. 191). Gear clearance does not reduce the accuracy of a thread cutting operation, because all play in the gears is taken up in one direction. A method often used to obtain proper gear clearance is: (1) Place a sheet of thick writing paper between the teeth of the two meshing gears, (2) tighten gears in position, and (3) remove paper. A small amount of grease, preferable graphite grease, applied to gear teeth will often aid in obtaining smoother, more quiet operation.

THE REVERSING MECHANISM

Right hand threads are cut with the carriage traveling toward the headstock. Left hand threads are cut with the carriage traveling toward the tailstock.

Whenever a new gear train has been set up, shift the tumbler gear lever to test the direction of the carriage travel. Because some set-ups are simple-geared and some are compounded, the carriage travel may be right for one set-up and left for another set-up, even though the lever has been shifted to the same position in each case. ***Always test the direction of carriage travel before starting to cut a thread.***

After the tumbler gear lever has been shifted to the proper position, it should not be moved until the thread has been completed. *This is especially important because a shift in the lever position destroys the relation between the threading dial and the lathe spindle and causes splitting of the thread.*

GEAR TRAINS FOR STANDARD THREADS

The following pages give detailed instructions for mounting gears for the more common thread sizes. Refer to these pages and the lathe threading chart when making set-ups. "Back Position" of a sleeve or the screw stub means the position *toward* the headstock. "Front Position" is the position *away from* the headstock. The gear bracket is tightened in position by locking the nut on the front of the gear bracket.

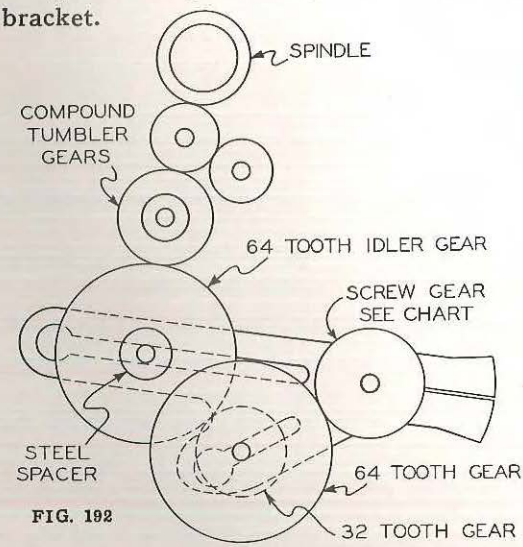


FIG. 192

GEAR TRAIN FOR 8 THROUGH 10 THREADS PER INCH

1. Place on front position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 32 tooth gear and 64 tooth gear on sleeve and mount in Position C on gear bracket with 32 tooth gear in back position. Tighten so that 64 tooth gear meshes with gear in screw position.

3. Place 64 tooth gear and spacer on sleeve and mount in Position A with 64 tooth gear in back position. Tighten so that 64 tooth gear meshes with the 32 tooth gear in Position C. The 64 tooth gear is an idler.

4. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with the 32 tooth compound tumbler gear.

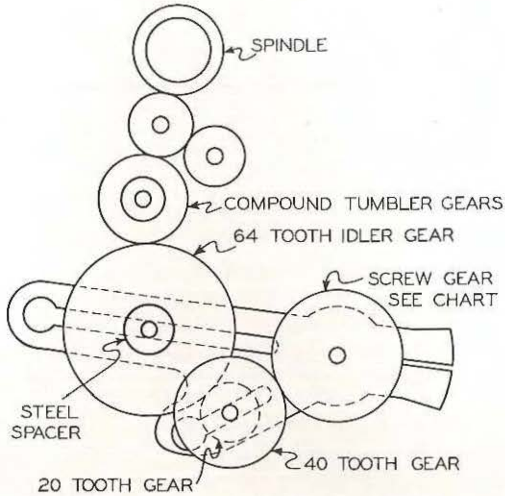


FIG. 193. Gear set-up for 11 through 16 threads per inch.

GEAR TRAIN FOR 11 THROUGH 16 THREADS PER INCH

1. Place in front position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 20 tooth gear and 40 tooth gear on sleeve in Position C with 20 tooth gear in back position. Tighten so that 40 tooth gear meshes with gear in screw position.

3. Place 64 tooth gear and spacer on sleeve and mount in Position A with 64 tooth gear in back position. Tighten so that 64 tooth gear meshes with 20 tooth gear in Position C. The 64 tooth gear is an idler.

4. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position A meshes with 32 tooth compound tumbler gear.

GEAR TRAIN FOR 18 THROUGH 32 THREADS PER INCH

(See Fig. 194, page 161)

1. Place on back position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 64 tooth gear and spacer on sleeve and mount in Position B with 64 tooth gear in back position Tighten so that 64 tooth gear meshes with gear in screw position. The 64 tooth gear is an idler.

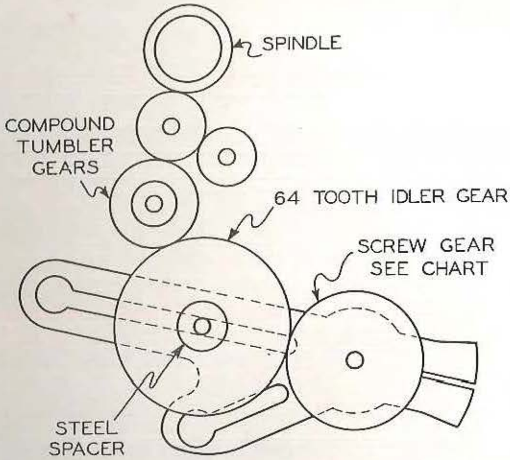


FIG. 194. Gear set-up for 18 through 32 threads per inch.

3. Swing entire gear bracket upward and tighten so that 64 tooth gear in Position B meshes with 32 tooth compound tumbler gear.

GEAR TRAIN FOR 36 THROUGH 64 THREADS PER INCH

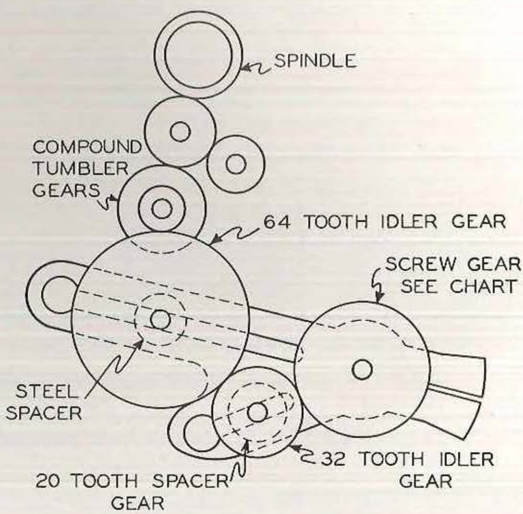


FIG. 195. Gear set-up for 36 through 64 threads per inch. (see page 162).

GEAR TRAIN FOR 36 THROUGH 64 THREADS PER INCH

(See Fig. 195, page 161)

1. Place in front position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 20 tooth gear and 32 tooth gear on sleeve and mount in Position C with 20 tooth gear in back position. Tighten so that 32 tooth gear meshes with gear in screw position. The 32 tooth gear is an idler; the 20 tooth gear is a spacer.

3. Place spacer and 64 tooth gear on sleeve and mount in Position A with spacer in back position. Tighten so that 64 tooth gear meshes with 32 tooth gear in Position C. The 64 tooth gear is an idler.

4. Swing entire gear bracket upward so that the 64 tooth gear in Position A meshes with the 16 tooth compound tumbler gear.

GEAR TRAIN FOR 72 THROUGH 96 THREADS PER INCH

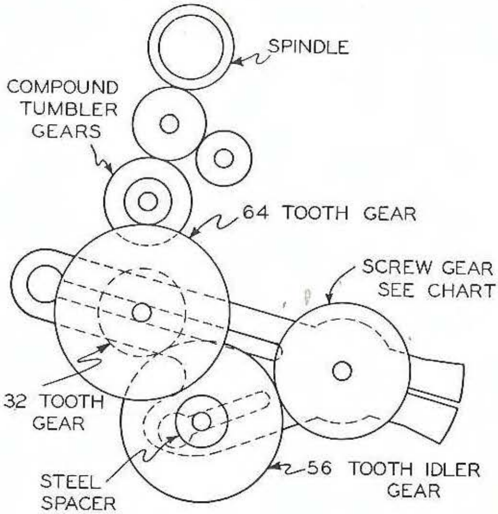


FIG. 196. Gear set-up for 72 and 80 threads per inch.

1. Place in back position of screw stub the gear listed in "Gear on Screw" column of threading chart.

2. Place 56 tooth gear and spacer on sleeve and mount in Position C with 56 tooth gear in back position. Tighten so that 56 tooth gear meshes with the gear in screw position. The 56 tooth gear is an idler.

3. Place 64 tooth gear and 32 tooth gear on sleeve and mount in position A with 32 tooth gear in back position. Tighten so that 32 tooth gear meshes with 56 tooth gear in Position C.

4. Swing entire gear bracket upward and tighten so that the

64 tooth gear in Position A meshes with the 16 tooth compound tumbler stud gear.

THREAD CUTTING SPEEDS: The beginner in thread cutting should adjust belts to obtain a speed of 28 R.P.M. (Manual, page 45). This slow speed allows plenty of time to engage and disengage the half-nut lever. After more experience in cutting threads, higher speeds can be used up to approximately 1/3 or 1/2 the speeds recommended for turning the various materials (Manual, Part 4).

THE THREADING DIAL

FIG. 197
Threading Dial.

The threading dial (Figs. 197 and 198) performs an important function by indicating the proper time to engage the half-nut lever so that the tool will enter the same groove of the thread for each cut. Without the threading dial it would be necessary to reverse the motor at the end of each cut and "wind" the tool out of the thread — a cumbersome method little used except when cutting metric and special fractional threads.

RULES FOR THE USE OF THE THREADING DIAL

When cutting an *even-numbered thread* (such as 12, 14, 16, 32, etc. per inch), engage the half-nut lever for the first cut when the stationary mark on the outside of the threading dial is in line with any one of the four marks on the rotating portion of the dial. The same dial marking, *or the one opposite,* must be used for following cuts.

When cutting *odd-numbered threads* (such as 9, 11, 13, 27, etc. per inch) *and half-numbered threads* (such as 8½, 9½, 10½, etc. per inch), engage the half-nut lever *at the same mark on the threading dial for each cut.*

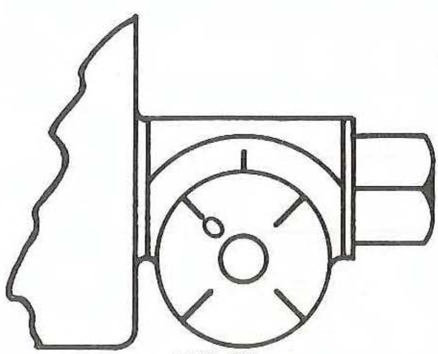


FIG. 198
Threading dial showing mainmarkings. Other markings may be made by the operator as needed.

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
1 - INDEX
12 - NOTES

USING THE THREADING DIAL FOR MULTIPLE THREADS

Although only four marks are cut into the top of the threading dial, there are actually eight different positions at which the half-nut lever can be engaged. Figure 199 shows the intermediate points between the four mainmarkings. These points can be marked with pencil, or the positions easily estimated. In the following paragraphs, "Lead in Threads Per Inch" is equal to 1 divided by Lead in Inches.

CUTTING DOUBLE THREADS WITH LEAD IN THREADS PER INCH DIVISIBLE BY FOUR BUT NOT BY EIGHT (12, 20, 28, 36, etc.)

A single thread of this lead is cut by engaging the half nuts at any of the four main markings on the threading dial (O, A, B or C in Figure 199). To cut the second groove of a double thread, the half nuts are engaged at any of the "b" positions.

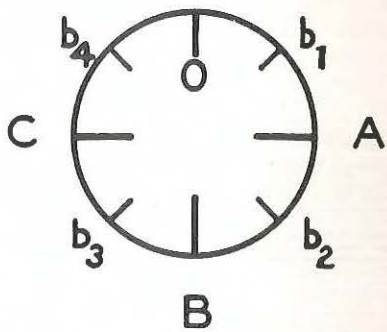


FIG. 199
Threading Dial Positions. The line and letter for the "O" position is marked in the face of the dial. Lines for "A," "B," and "C" positions are marked, but not the letters themselves. The four "b" postions may be marked by the operator as needed.

Example: To Cut a Double Thread with a Pitch of 1/24 inch and a Lead of 1/12 inch. Set up the change gears for the lead in threads per inch (12, not 24). Engage the half nut lever for the first cut when the stationary mark on the outside of the threading dial is in line with any one of the four main marks on the rotating portion of the dial. Then return to the starting point and engage half nuts at any one of the "b" positions, taking the first cut on the second groove of the thread. The compound rest feed remains *at one setting* until both grooves have been cut to the same depth.

CUTTING DOUBLE AND QUADRUPLE THREADS WITH LEAD IN THREADS PER INCH DIVISIBLE BY TWO, BUT NOT BY FOUR (10, 14, 18, 22, etc.)

A single thread of this lead can be cut only by engaging the half nut lever at the "O" or "B" markings, on the threading dial. To cut the second groove of the double thread, the half nuts are engaged at the "A" or "C" markings, and the cutting operation is the same as in the preceding paragraph.

For quadruple threads of this lead, engage the half nut lever at the "O" or "B" markings for the first groove, at the b_1 or b_3 positions for the second groove, at the "A" or "C" markings for the third groove, and at the b_2 or b_4 positions for the fourth groove. The setting of the compound rest feed is changed only after each of the four grooves has been cut to the same depth.

CUTTING DOUBLE AND QUADRUPLE THREADS WITH LEAD IN THREADS PER INCH DIVISIBLE BY ONE, BUT NOT BY TWO (ODD NUMBERS)

A single thread of this lead is cut by engaging the half nut lever at the "O" marking. To cut the second groove of the double thread, the half nuts are engaged at the "B" marking on the threading dial. The cutting operation is the same as in the preceding paragraph.

For quadruple threads of this lead, engage the half nut lever at the "O" marking for the first groove, at "A" for the second groove, at "B" for the third groove, and at "C" for the fourth groove. The setting of the compound rest feed is changed only after each of the four grooves has been cut to the same depth.

CUTTING MULTIPLE THREADS BY SLIPPING TEETH ON THE SPINDLE GEAR

Double and quadruple threads can also be cut by "slipping teeth" on the compound gear. This practice is not so common as the use of the threading dial, but is not complicated.

To cut multiple threads by slipping teeth on the compound gear: cut the complete first groove to a minor diameter dependent upon pitch of the desired thread. The change gear train should be arranged for the desired lead. It is important to use the same 0 point or reference to cut each thread—be sure to remember this point during the cutting operations.

Refer to the table on page 166, then slip the required number of teeth by making adjacent teeth on the compound gear and the gear meshing with the compound gear. Drop the entire gear bracket low enough to disengage the gears and turn the compound gear forward the proper number of teeth by rotating spindle by hand. Raise the gear bracket so that the previously marked gear tooth meshes with the newly selected compound gear tooth.

To Cut Double Threads:—Slip 16 teeth to cut the second groove.

To Cut Quadruple Threads:—Slip 8 teeth to cut the second groove, 8 teeth more to cut the third groove, and 8 teeth more to cut the fourth groove.

Each thread groove is cut to its complete depth and finished before starting the next groove.

GEAR TRAINS FOR CARRIAGE FEEDS

The automatic longitudinal carriage feed per spindle revolution is obtained by setting up the gear train in the same manner as for thread cutting (pages 156 to 163). The feed in inches is equal to

$\frac{1}{\text{threads per inch}}$ For example, a feed of .0078 inch requires the gear set-up as 128 threads per inch.

The four most common carriage feeds, as shown in the threading chart (page 159), are .0078, .0048, .0039, and .0024 inch per spindle revolution. Refer to the threading chart and the four following paragraphs when changing these gear set-ups. Table II on page 173 includes gear set-ups for other carriage feeds.

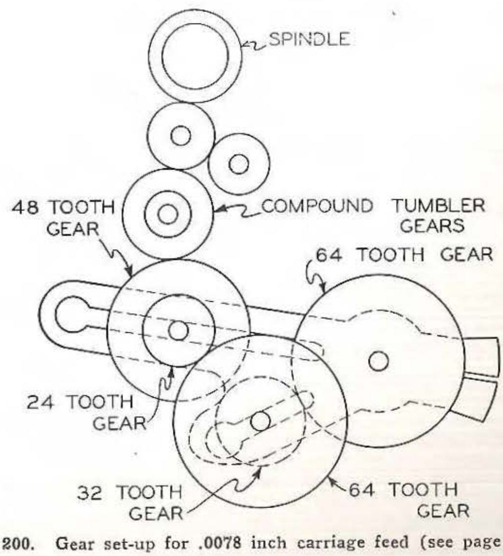


FIG. 200. Gear set-up for .0078 inch carriage feed (see page 167).

GEAR TRAIN FOR .0078 INCH CARRIAGE FEED

(See Fig. 200, page 166)

1. Place 64 tooth gear in back position on screw stub.
2. Place 32 tooth gear and 64 tooth gear on sleeve in Position C, with 32 tooth gear in back position. Tighten so that 32 tooth gear meshes with 64 tooth gear on screw stub.
3. Place 48 tooth gear and 24 tooth gear on sleeve in Position A, with 48 tooth gear in back position. Tighten so that 24 tooth gear meshes with 64 tooth gear in Position C.
4. Swing entire gear bracket upward and tighten so that 48 tooth gear in Position A meshes with 32 tooth compound tumbler gear.

GEAR TRAIN FOR .0048 INCH CARRIAGE FEED

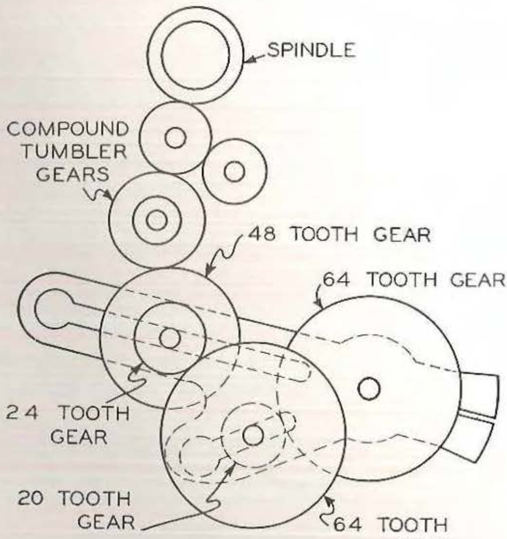


FIG. 201. Gear set-up for .0048 inch carriage feed.

1. Place 64 tooth gear in back position on screw stub.
2. Place 20 tooth gear and 64 tooth gear on sleeve in Position C, with 20 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.
3. Place 48 tooth gear and 24 tooth gear on sleeve in Position A, with 48 tooth gear in back position. Tighten so that 24 tooth gear meshes with 64 tooth gear in Position C.
4. Swing entire gear bracket upward and tighten so that 48 tooth gear in position A meshes with 32 tooth compound tumbler gear.

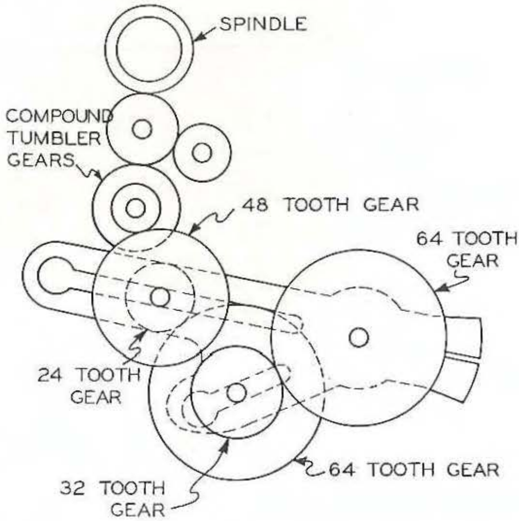


FIG. 202. Gear set-up for .0039 inch carriage feed.

GEAR TRAIN FOR .0039 INCH CARRIAGE FEED

1. Place 64 tooth gear in front position on screw stub.

2. Place 64 tooth gear and 32 tooth gear on sleeve in Position C, with 64 tooth gear in back position. Tighten so that 32 tooth gear meshes with 64 tooth gear on screw stub.

3. Place 24 tooth gear and 48 tooth gear from screw stub on sleeve in Position A, with 24 tooth gear in back position. Tighten so that 24 tooth gear meshes with 64 tooth gear in Position C.

4. Swing entire gear bracket upward and tighten so that 48 tooth gear in Position A meshes with 16 tooth compound tumbler gear.

GEAR TRAIN FOR .0024 INCH CARRIAGE FEED

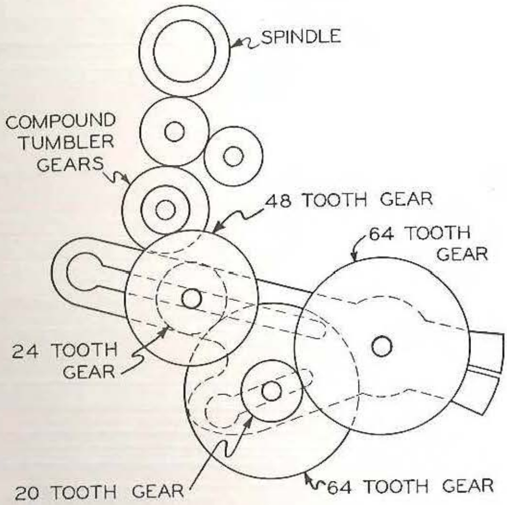


FIG. 203. Gear set-up for .0024 inch carriage feed.

1. Place 64 tooth gear in front position on screw stub.

2. Place 64 tooth gear and 20 tooth gear on sleeve in Position C, with 64 tooth gear in back position. Tighten so that 20 tooth gear meshes with 64 tooth gear on screw stub.

3. Place 24 tooth gear and 48 tooth gear on sleeve in Position A, with 24 tooth gear in back position Tighten so that 24 tooth gear meshes with 64 tooth gear in Position C.

4. Swing entire gear bracket upward and tighten so that 48 tooth gear in Position A meshes with 16 tooth compound tumbler gear.

SPECIAL THREADS AND FEEDS

Engineers have charted over a thousand threads and feeds between the coarsest thread and the finest feed. Tables I and II give proper gear set-ups for a wide variety of special threads and feeds. Most of these set-ups are exact— some are accurate to the limits mentioned. Table III gives set-ups for metric threads with pitch between 0.6 and 3.0 millimeters.

TABLE I—GEAR SET-UPS FOR THREADS FROM 7½ THROUGH 79 PER INCH NOT SHOWN ON THE THREADING CHART

The threading dial can be used when cutting threads below marked "exact" in the column under "Accuracy." All other threads must be cut in the same manner as metric threads (See page 108). Extra gears are also available from the factory at nominal cost.

Threads per inch	Accuracy per inch	Gear on Screw	Position C		Position B		Position A		Position D		Compound Tumbler Gear	Note
			B	F	B	F	B	F	B	F		
7.5	Exact	40*F*	24	64	—	—	64*I*	*xxS*	—	—	32	
8.5	1/470	20*F*	46	54	—	—	64*I*	*xxS*	—	—	32	
10.5	Exact	56*B*	48	36	20	40	44*I*	*xxS*	—	—	32	*
12.5	Exact	40*F*	—	—	20	32	64*I*	*xxS*	—	—	32	
13.5	Exact	54*F*	—	—	20	40	46*I*	*xxS*	—	—	32	
15	Exact	54*B*	36	40	20	40	46*I*	*xxS*	—	—	32	*d**
17	1/560	40*F*	—	—	46	54	52*I*	*xxS*	—	—	32	
21	Exact	56*F*	36	48	—	—	64*I*	*xxS*	—	—	32	
25	Exact	40*F*	—	—	40	32	64*I*	*xxS*	—	—	32	*d*
29	1/780	40*B*	40	56	—	—	56	54	—	—	32	*dh*
30	Exact	48*F*	—	—	40	32	64*I*	*xxS*	—	—	32	
33	Exact	40*B*	32	48	—	—	44	40	—	—	32	*d*
34	1/340	40*B*	32	46	—	—	52	44	—	—	32	
35	Exact	40*F*	*xxS*	54*I*	—	—	56	32	—	—	32	
38	1/1580	52*B*	40	52	—	—	36	32	—	—	32	*p*
39	Exact	54*F*	52	36	—	—	56*I*	*xxS*	—	—	32	
41	1/410	46*B*	40	56	—	—	56	44	—	—	32	*h*
42	Exact	48*F*	56	32	—	—	54*I*	*xxS*	—	—	32	
43	1/2100	44*B*	36	44	*xxS*	40*I*	32	20	—	—	32	*t**
45	Exact	40*F*	*xxS*	52*I*	—	—	54	24	—	—	32	
46	Exact	46*F*	—	—	—	—	*xxS*	64*I*	—	—	16	
47	1/470	54*B*	46	40	—	—	*xxS*	52*I*	—	—	16	
49	Exact	56*F*	56	32	—	—	54*I*	*xxS*	—	—	32	*h*
50	Exact	40*B*	—	—	32	40	*xxS*	64*I*	—	—	16	*d*

8 - ATTACHMENTS
9 - WOODTURNING
10 - TABLES
1 - INDEX
12 - NOTES

Table I—Continued

Threads per inch	Accuracy per inch	Gear on Screw	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Position D B	Position D F	Compound Tumbler Gear	Note
51	1/400	54*B*	36	56	—	—	56	46	—	—	32	*h*
52	Exact	52*F*	—	—	—	—	*xxS*	64*I*	—	—	16	
53	1/3000	54*B*	40	64	—	—	54	44	—	—	32	*j*
54	Exact	54*F*	—	—	—	—	*xxS*	64*I*	—	—	16	
55	1/7000	64*B*	44	64	—	—	52	44	—	—	32	*t*
57	1/320	56*B*	—	—	24	44	40	36	—	—	32	
58	1/580	54*B*	32	54	—	—	56	44	—	—	32	*j*
60	Exact	48*B*	—	—	32	40	*xxS*	64*I*	—	—	16	
61	1/1500	54*B*	46	52	—	—	*xxS*	56*I*	—	—	16	
62	1/620	48*B*	20	44	—	—	54	46	—	—	32	
63	1/2100	64*B*	—	—	24	40	52	44	—	—	32	
65	Exact	48*B*	32	52	—	—	40	24	—	—	32	
66	1/75	44*B*	36	54	—	—	*xxS*	64*I*	—	—	16	
69	Exact	54*B*	36	46	—	—	*xxS*	64*I*	—	—	16	
70	Exact	40*B*	—	—	48*I*	*xxS*	32	56	—	—	16	
71	1/710	64*B*	36	40	—	—	*xxS*	64*I*	—	—	16	
73	1/730	48*B*	20	36	*xxS*	44*I*	54	32	—	—	32	*
75	Exact	40*F*	36	24	32	40	*xxS*	44*I*	—	—	16	*d*
77	Exact	44*B*	32	56	—	—	*xxS*	54*I*	—	—	16	
78	Exact	54*B*	36	52	—	—	*xxS*	64*I*	—	—	16	
79	1/790	54*F*	52	40	—	—	32	36	*xxS*	44*I*	16	(*)

SYMBOLS:

d—extra 40 tooth gear
h—extra 56 tooth gear
j—extra 54 tooth gear
p—extra 52 tooth gear
t—extra 44 tooth gear

F—position away from headstock
B—position toward headstock
I—idler gear
xxS—steel spacer
*—extra sleeve, bushing and bolt assembly

TABLE II—GEAR SET-UPS FOR CARRIAGE FEEDS

Six different carriage feeds between .001046 and .0080 inch per spindle revolution are available on the six-inch lathes in addition to the four most common feeds pictured and described in detail between pages 166 and 169. When the material or job requires a certain carriage feed, refer to the table below.

Feed Inches	Threads per inch	Gear on Screw	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Position D B	Position D F	Compound Tumbler Gear	Note
.008	124.8	64*B*	20	52	—	—	54	36	—	—	32	
.007	143.94	64*B*	20	54	—	—	40	24	—	—	32	
.006	166.4	64*B*	20	52	—	—	*xxS*	56*I*	—	—	16	
.005	199.1	64*F*	64	32	—	—	36	56	—	—	16	
.0021	478	64*F*	64	20	—	—	24	56	—	—	16	

SYMBOLS:

f—extra 24 tooth gear

F—position away from headstock

B—position toward headstock

xxS—steel spacer

I—idler gear

S—Spacer gear

TABLE III—GEAR SET-UPS FOR METRIC THREADS

Two of the standard change gears furnished with the six-inch lathe, the 52 tooth gear and 44 tooth gear, combine to give a ratio of 44/52 or .846154, which is an almost exact function of 2.54, the English to Metric ratio. Thus, it is possible to cut metric threads very close to the standard metric pitches.

Refer to page 108 when cutting metric threads.

Pitch MM.	Gear on Screw	Position C B	Position C F	Position B B	Position B F	Position A B	Position A F	Position D B	Position D F	Compound Tumbler Gear	Note
.6	56*B*	36	64	—	—	44	52	—	—	32	
.7	64*B*	24	32	—	—	44	52	—	—	32	
.75	64*B*	32	40	—	—	44	52	—	—	32	
.8	54*B*	46	64	—	—	44	52	—	—	32	
.9	46*B*	36	52	—	—	44	52	—	—	32	
1.0	40*B*	32	48	—	—	44	52	—	—	32	
1.25	44*F*	48	52	—	—	40*I*	20*S*	—	—	32	
1.50	44*F*	40*S*	52*I*	—	—	46*I*	20*S*	—	—	32	
2.5	44*F*	24	52	—	—	64*I*	20*S*	—	—	32	
3.0	44*F*	20	52	—	—	64*I*	20*S*	—	—	32	

Part 8

LATHE ATTACHMENTS
AND THEIR USES

PART 8

LATHE ATTACHMENTS AND THEIR USES

Lathe attachments fall into two general classes: (1) Those which increase speed and accuracy of general lathe operations. (2) Those which equip the lathe to handle such work as milling, grinding, undercutting etc., which usually require a single purpose machine.

THE STEADY REST

The steady rest (Fig. 204) supports long work during turning, boring or threading operations. The base clamps securely to the bed ways — the adjustable bronze jaws form a bearing for the work and hold it in exact position. The most common methods of mounting work in the steady rest are shown in Figures 205 and 206.

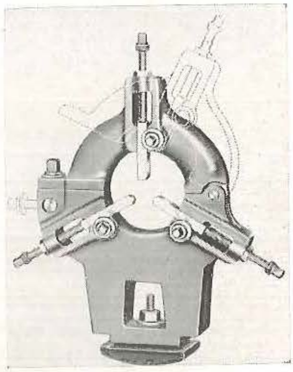


FIG. 204
Steady Rest, or Center Rest.

If the bar is less than ¾ inch in diameter and must be machined near the center or more than 5 or 6 inches from the chuck, the steady rest should be mounted in position near the portion of the work being machined (Fig. 205). To drill, bore, tap or machine the end of a long piece of any diameter up to 3 inches, support the end with the steady rest as shown in Figure 206. The headstock end can be held in a chuck or centered and bound to the face plate (Fig. 207).

MOUNTING WORK IN THE STEADY REST

Accurate positioning of the jaws is essential when mounting work in the steady rest. The bronze jaws must form a true bearing for the work, allowing it to turn freely but without play. The following method is satisfactory for mounting most work: Clean the bed ways. With the work mounted in the lathe, slide the steady rest close to the chuck jaws or lathe dog, tighten the base

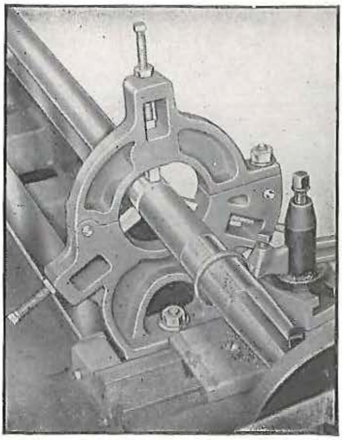


FIG. 205
Steady rest supporting axle for turning

clamp, adjust steady rest jaws and lock them in position on the work. A small piece of cellophane slipped between the jaw and the work is sometimes used to aid in obtaining the proper bearing —advance the jaw until it just touches, then remove cellophane. After tightening both the lock nut and clamp screw on each jaw, loosen the base clamp, slide the steady rest into the proper position and retighten base clamp.

When the work is being held in a chuck, the jaws of the steady rest can be set more accurately if the work is held between lathe centers while the jaws are being adjusted. Take extreme care in locating the tailstock center (see page 67). For jobs requiring maximum accuracy, check trueness of the work with a dial gauge as shown in Figure 75, page 69.

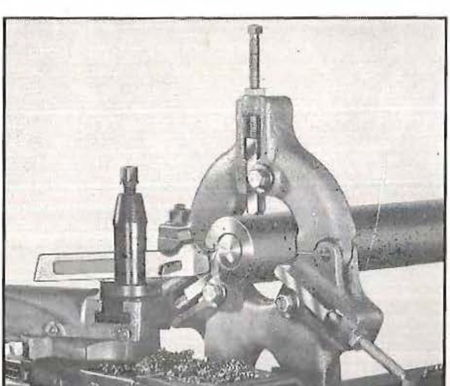


FIG. 206
Steady rest supporting shaft for facing end.

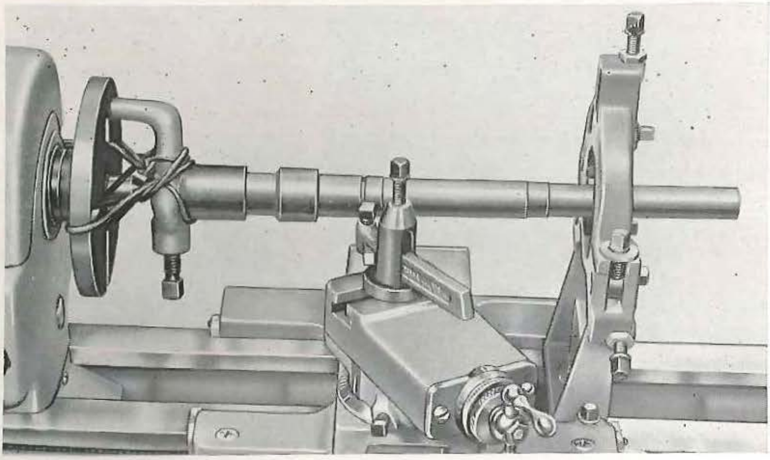


FIG. 207. One end of this piece is being supported by the steady rest; the other is bound to the face plate with a wet leather thong.

During the cutting operation apply plenty of lubricant on the work at the point of bearing with the jaws. When using the steady rest in duplicate work, stock is removed or mounted by loosening and replacing the hinged top.

THE FOLLOWER REST

The follower rest (Fig. 208) is mounted on the back of the carriage dovetail slide and provides support for long slender work mounted between centers. Figure 209 shows a typical application. The two adjustable jaws hold the work in exact position, preventing it from springing away from the tool.

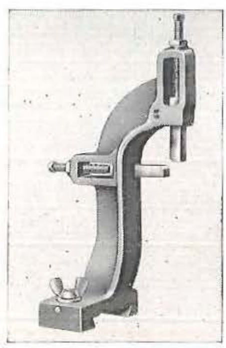


FIG. 208
Follower Rest.

The jaws of the follower rest, like those of the steady rest, must form a true bearing for the work, allowing it to turn with no trace of binding. In setting the follower rest jaws, first remove the guard over the cross feed screw — place a small piece of paper or cardboard over this screw to keep off chips during the cutting operation. The dovetail ways should be wiped clean. Adjust the jaws with the carriage close to the tailstock after a short portion of the work has been turned down at one end. Set the follower rest so that the vertical jaw touches the top of the work. In tightening the follower rest jaws cellophane may be used to determine the proper amount of traction

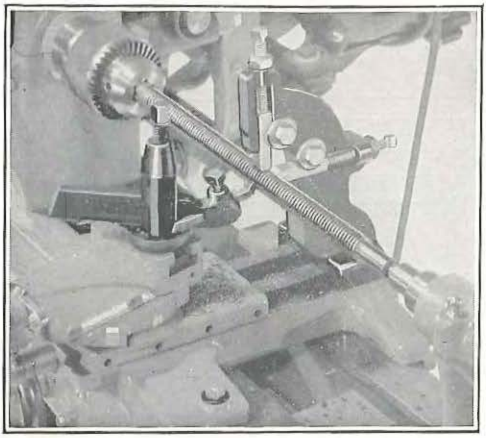


FIG. 209. Threading a long screw with the aid of the follower rest.

in the same manner as with the steady rest jaws (page 176).

During the cutting operation apply plenty of lubricant on the work at the point of bearing with the jaws. After each cut the jaws should be adjusted to retain accuracy. Both the follower rest and steady rest are often used to brace a slender rod (Fig. 210).

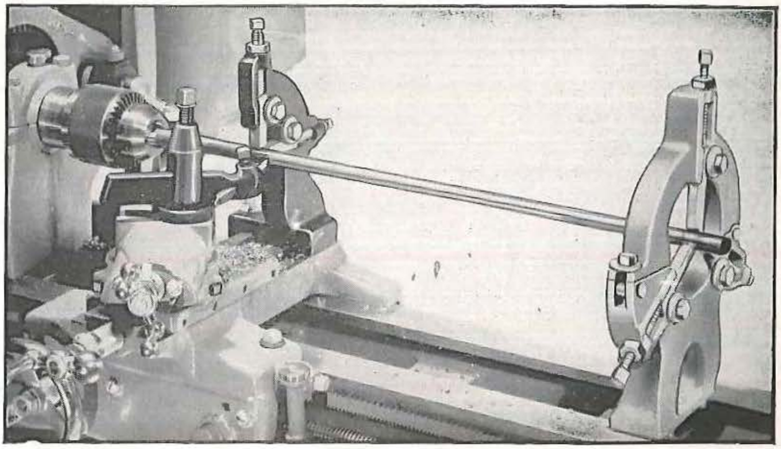


FIG. 210
Using both the steady rest and follower rest to support a long, small-diameter rod.

THE CUT-OFF TOOL

Quick, clean cutting-off requires careful machining and a properly ground tool. The cut-off tool must be set into the work at an exact right angle and with the cutting edge on dead center (see Fig 214).

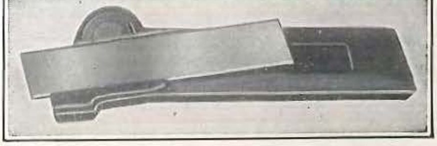


FIG. 211. Cut-off Tool.

Figures 211 and 212 show the tool recommended for most operations. This tool is

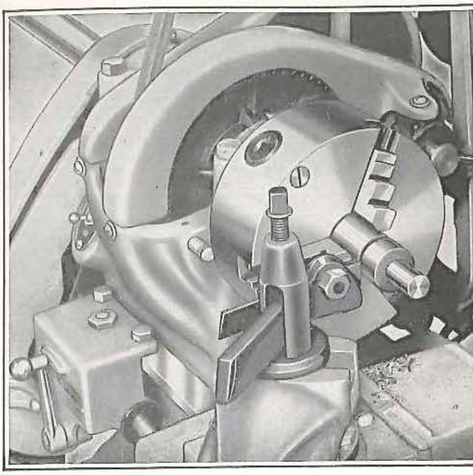


FIG. 212. Cutting off bar stock fed through headstock spindle.

supplied ready - ground with correct top rake, front and side clearance, so that the front face cuts freely without binding (Fig. 213). The cutter blades are replaceable, and the holder is offset to permit cut-off operations close to the headstock spindle.

The two most common troubles in cutting-off are "chatter" and "hogging-in." The following paragraphs tell how these troubles are avoided—follow these rules carefully:

RIGIDITY:—Not only the tool and carriage, but ***every part of the lathe*** must be tight when cutting off—loose fits in the spindle, carriage and compound rest will surely cause trouble. See that the gibs on the rear of the carriage fit snug and that the carriage is locked securely in position on the bed. Tighten the gibs on the cross feed and compound feed. Set the tool holder as far back into the tool post as possible and keep the tool post screw tightened. The tool is fed into the work with the cross feed.

RATE OF FEED: The rate of feeding-in is especially important because the chip is actually wider than the cutting edge of the tool blade. A fast feed tends to cause "hogging," either stopping the lathe or breaking the tool—a slow feed usually produces chatter. Experience aids in "feeling out" the exact rate of feed to avoid both chatter and "hogging-in."

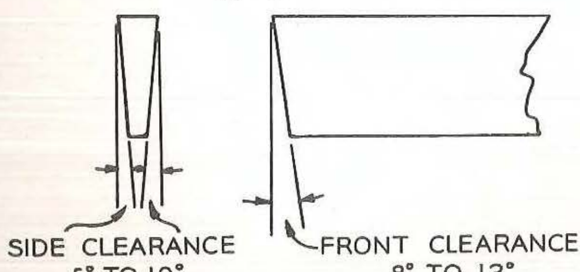


FIG. 213. Clearance angles of cut-off tool.

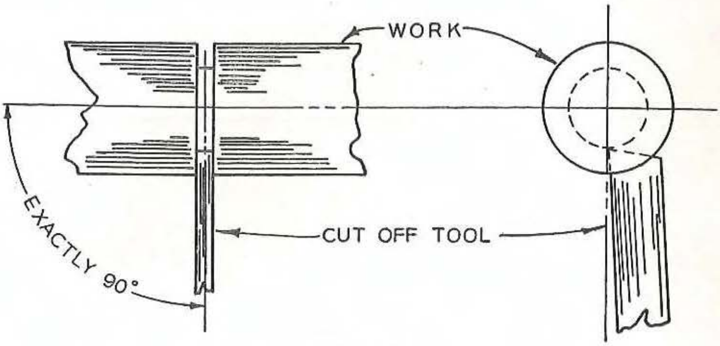


FIG. 214
Setting the cut-off tool into the work. The blade must be at a right angle to the work, and the point should be on the exact center line.

SPINDLE SPEED: The spindle speed should be about 2/3 of the speed recommended for general turning of the material being cut off (Fig. 56, page 47). Do not use too slow a speed.

LUBRICATION: Thorough lubrication is absolutely necessary during the cut-off operation. In large lathe work, a continuous stream of lubricant is directed at the front of the cut-off tool. When cutting off on a small lathe, the lubricant is usually applied with a brush or oil can. Use the same type of lubricant recommended in Part 4 for general turning of the various materials.

FURTHER RULES FOR CUTTING-OFF

1. Set the cutting edge of the tool on the lathe center line—the tool blade should be at an exact right angle with the work (Fig. 214.)

2. If the tool "hogs-in" and stops the spindle rotation, stop the motor and reverse the spindle by hand before backing out the tool with the cross feed. After resetting the tool, feed in slowly and remove the bad spots.

3. Never complete a cut-off of work which does not swing free at one end.

4. Cut off as close to the headstock as possible.

5. When cutting off soft copper or aluminum, refer to page 56 or 60.

6. To resharpen the cut-off tool shown in Figure 211, grind the front edge only, allowing front clearance (see Fig. 213).

7. Figure 58, page 49, shows how grooves can be cut with the cut-off tool to "block-out" the work and indicate the end of a cut. Each groove is slightly less deep than the finish-diameter—this simplifies the turning operation by providing an easy stopping place after each cut.

Experiment to determine the proper spindle speed and rate of feed for the diameter and material being cut off—this is the best way to get the "feel" of the operation.

THE KNURLING TOOL

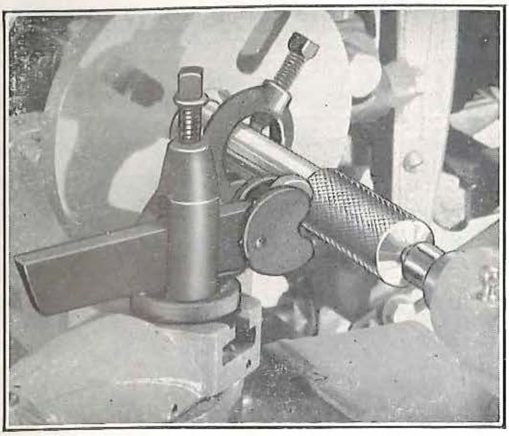


FIG. 215. Knurling a Tool Handle

Figure 217 shows a close-up view of a knurl and the two formed cutters which roll with the work during the knurling operation. A sharp, even, diamond-shaped knurl provides a perfect gripping surface for tool handles, nuts, markers and instruments.

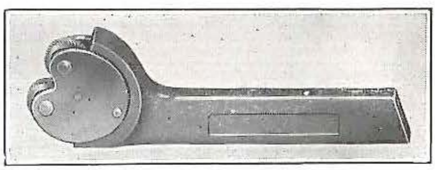


FIG. 216. Knurling Tool.

The type of tool shown in Figure 216 is recommended for knurling operations. The "floating" construction of this tool makes the rollers self-centering, assuring equal pressure on each roller and resulting in two sets of lines of equal depth. The rollers are hardened tool steel.

THE KNURLING OPERATION

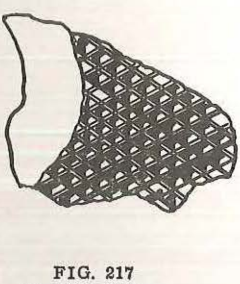


FIG. 217
Knurling cutters and pattern produced.

The knurling tool is set rigidly in the tool post at right angles to the face of the work and as far back in the tool post as possible. Adjust belts for proper spindle speed (see Fig. 55, page 45):

Diameters 1½" and over	83 R.P.M.
Diameters from ½"-1½"	164 R.P.M.
Diameters under ½"	266 R.P.M.

9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

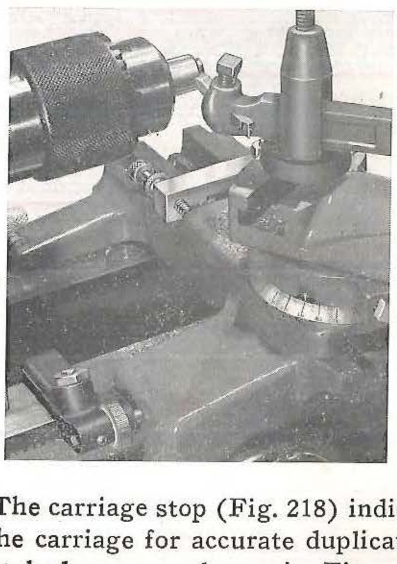
Because of the pressure exerted in knurling, the work should be mounted between centers whenever possible, and small diameters should be supported with the steady rest. When the work is held in a chuck, cut the knurl as close to the headstock as possible.

Advance the tool into the work with the cross feed until the dial reading has been advanced about .050 inch. Stop the lathe and without backing out the tool, check the pattern produced. Usually, with a cut of this depth, a perfect diamond pattern will result. When the pattern is not as desired, back out the tool and take a cut in another place on the work. After the correct design is obtained the test cuts will be rolled into a perfect knurl during the final cutting process.

When a test cut shows the proper pattern, engage carriage feed. Apply plenty of lubricant. *Keep tailstock center well lubricated.*

At the end of the cut shift the *reverse lever* to "Neutral," force the tool .005 or .006 inch deeper and then shift the reversing lever to cut back to the starting point. Continue the knurling operations until the desired depth is reached. *After the knurling process is started, never back out the tool until the knurl is completed.*

THE CARRIAGE STOP



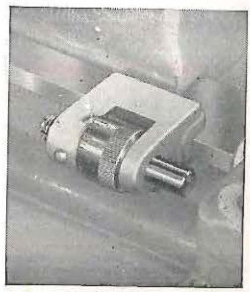


FIG. 218
Carriage Stop.

FIG. 219
A repeat operation requiring both the carriage stop and the cross slide stop. The bar is being fed through the headstock spindle.

The carriage stop (Fig. 218) indicates the proper stopping point of the carriage for accurate duplicate work. It is clamped on the front bed way as shown in Figure 219. Some of the frequent repeat operations which usually require the carriage stop: boring or facing to a given depth, cutting-off at a given point, duplicating longitudinal cuts (for example, mica undercutting) and laying out work on a cylindrical surface. The carriage stop shown in Figure 218 has a micrometer-type screw which permits a very exact setting. Always wipe the front bed way clean at the point where the carriage stop is to be clamped.

The carriage stop cannot automatically stop the power feed—the carriage should always be fed by hand for the last part of a cut. If the automatic feed is allowed to force the carriage into the carriage stop, either the stop or the lead screw bearing will be broken.

THE CROSS SLIDE STOP

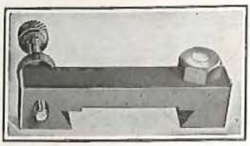


FIG. 220
Cross Slide Stop.

FIG. 221
Another view of the same repeat operation shown in Figure 195. This angle shows how the cross slide stop gauges the depth of the cut. Note carriage stop in background.

The cross slide stop (Fig. 196) indicates the proper depth at which to stop the cross feed, much in the same manner as the carriage stop is used as a guide in longitudinal operations. It is especially valuable for threading and turning down a rough diameter. The cross slide stop is mounted on the cross slide dovetail, either in front of or behind the compound rest. An adjustable screw and lock nut permit accurate setting (see Fig. 221). In mounting the cross slide stop on the cross slide dovetail, first remove the guard. Then clean the dovetail ways and clamp the stop in the approximate position required. Turn the adjusting screw into exact position and lock with the knurled nut. Place a small piece of paper or cardboard over the cross feed screw to keep it free from dirt and chips during the cutting operation.

During threading operations or whenever the tool is fed in with the compound, the cross feed is used only to back the tool out at the end of each cut. The cross slide stop, combined with the micrometer graduations of the cross feed control handle on the lathe, assure an accurate "zero" reading before the compound rest feed is advanced for the next cut. *Do not run the compound rest against the cross slide stop with too much force.*

THE MILLING ATTACHMENT

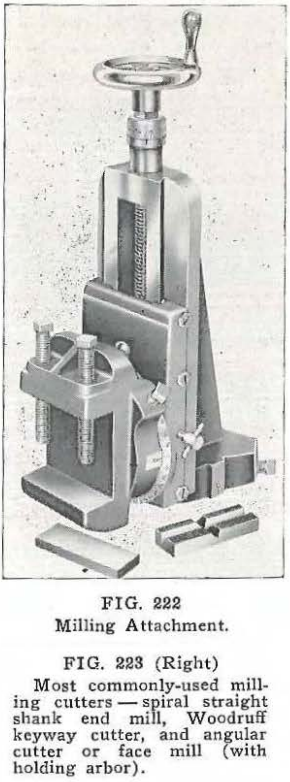


FIG. 222
Milling Attachment.

The versatile milling attachment (Fig. 222) converts the lathe into a small milling machine. Six of the more frequent operations are illustrated on the opposite page. The work is held in the milling vise jaws, and the various types of cutters (Fig. 223) are held in the headstock spindle with the chuck or holding collet shown in Figure 230.

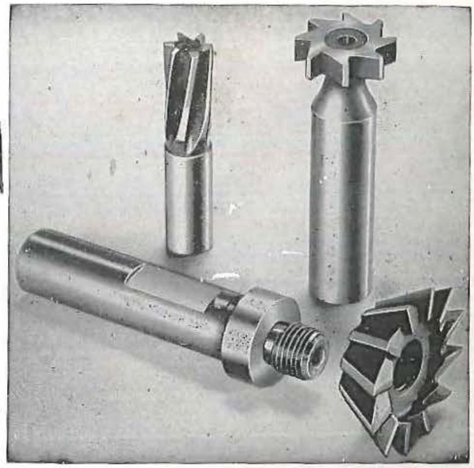


FIG. 223 (Right)
Most commonly-used milling cutters — spiral straight shank end mill, Woodruff keyway cutter, and angular cutter or face mill (with holding arbor).

The end mills are suitable for milling slots, facing and routing small work, squaring or splining shafts, cutting straight keyways, and general milling operations. The primary use of the Woodruff cutters is the cutting of Woodruff keyways—other uses include the cutting of slots, grooves, T slots, etc. Angular cutters cut dovetails and angles less than 90° and are also used for facing operations.

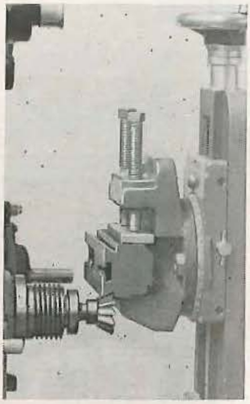


FIG. 224
Milling a Dovetail Slot.

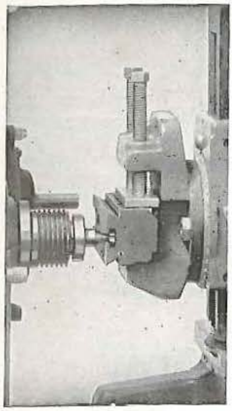


FIG. 225
Milling a Slot with a Woodruff Cutter.

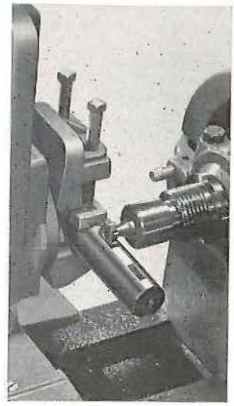


FIG. 226
Milling a Woodruff Keyway.

MOUNTING WORK IN THE MILLING ATTACHMENT

Remove the compound rest from the cross slide swivel.[1] Clean the swivel and the base of the milling attachment. Mount the milling attachment at the desired angle, using the swivel graduations as a guide. Tighten gibs on carriage, cross feed and milling attachment dovetail slide. Loosen milling vise jaws, insert work and tighten. The vise can be swivelled to any desired angle and is adjusted for height with the graduated handwheel.

[1]When the compound lock mechanism is controlled by two clamp screws, loosen these screws about ½ inch only and raise compound rest with a twisting motion. In this way, plunger pins are kept from twisting out of line with bevel of central pilot.

Similar precautions must be taken when mounting the milling attachment and remounting the compound rest: Remove the clamp screws completely and PUSH plunger pins up to bevel of pilot—then reinsert and tighten clamp screws.

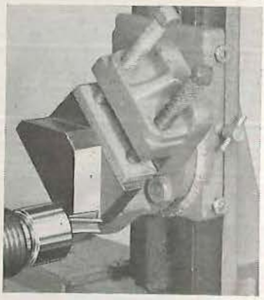


FIG. 227
Angular Milling (Vertical).

FIG. 228
Angular Milling (Horizontal).

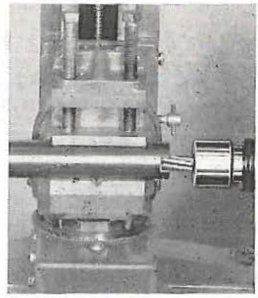


FIG. 229
Milling Slot in End of Shaft.

HOLDING THE MILLING CUTTER

The holding collet shown in Figure 230 is preferred for holding the milling cutter. The headstock spindle chuck is not recommended because the cutter shanks and the chuck jaws are extremely hard and would slip during the milling operation.

The complete collet set includes one arbor for holding straight

FIG. 230

Holding collet set for holding milling cutters. This set consists of: draw bar, sleeve, and an arbor for straight shank cutters. Collet bushings are required to adapt end mills to the arbor of this set.

shank cutters. Two arbors are available for threaded angular cutters. A collet bushing or arbor is also required for all straight shank end mills except the ½ inch diameter. The Woodruff keyway cutters are held directly in the collet arbor without bushings.

Pass the draw bar through the spindle and tighten the arbor into spindle taper by turning handwheel. Tighten cutter in arbor by locking socket-head set screw. The draw bar, arbor, bushing, cutter shank and lathe spindle must be wiped clean and dry. When mounting the milling cutter in the collet arbor, be sure to select the proper size of collet bushing if one is required.

DEPTHS OF CUTS AND FEEDS

When the work is fed across the milling cutter with the cross feed, the depth of cut and rate of feed is determined primarily by the "feel" of the operator. Take light cuts and feed in evenly and slowly until the correct feed can be judged. Never force the work into the cutter too fast. Cuts should be about 1/16" or less.

SPINDLE SPEEDS FOR MILLING

The cutting speed during a milling operation should be approximately 2/3 of the speed recommended for general turning of the material being machined (see Part 4). Figure 231 gives the lathe spindle speed required to obtain a desired surface speed when using the various milling cutters. Thus, knowing 2/3 of the surface speed recommended for a certain metal or plastic (Part 4), first use Figure 208 to find the proper speed for the cutter being used, then refer to Figure 55, page 45 for the belt set-up to obtain that speed.

FIG. 231

TABLE OF CUTTING SPEEDS FOR MILLING

Speeds Obtainable on the Atlas Lathe to Give Approximately the Surface Speeds Sho

	Surface Speed in Feet Per Minute									
	30	*40*	*50*	*60*	*70*	*80*	*100*	*120*	*150*	*200*
ill...	418	685	805	805	805	1270	1270	2072	2072	——
ill...	418	500	685	685	805	805	1270	1270	2072	——
ill...	266	418	500	685	685	805	805	1270	1270	2072
ill...	266	266	418	500	500	685	805	805	1270	2072
ill...	164	266	418	418	500	685	685	805	1270	1270
ill...	164	266	266	418	418	500	685	685	805	1270
tters										
.....	164	266	418	418	500	685	685	805	1270	1270
.....	164	164	266	266	418	418	500	685	805	1270
.....	112	164	164	266	266	266	418	418	500	805
″....	112	112	164	164	266	266	266	418	500	685
.....	83	112	164	164	164	266	266	418	418	685
ters										
″....	83	112	164	164	164	266	266	418	418	685
″....	70	83	112	112	164	164	266	266	266	418

WOODRUFF KEY DIMENSIONS

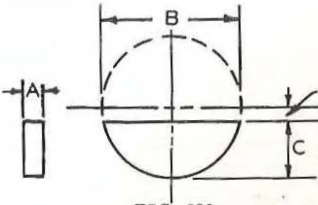


FIG. 232

Nominal Size of Key A × B	Width of Key Min. A	Diam. of Key Min. B	Height of Key C	Distance below Center E
1/8 x 1/2	.1250	.490	.198	3/64
3/16 x 3/4	.1875	.740	.308	1/16
1/4 x 7/8	.2500	.865	.370	1/16
5/16 x 1	.3125	.990	.433	1/16
3/8 x 1 1/4	.3750	1.240	.542	5/64

Note: Allowable oversize of dimensions A and B is .010″—of dimension C is .005″. Optimum key width is equal to one-quarter the shaft diameter. Keys should be chosen to approximate this relation as closely as possible.

DIMENSIONS OF WOODRUFF KEYWAY SLOTS

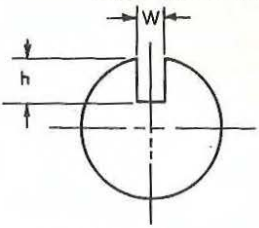


FIG. 233

Nominal Key Size A × B	WIDTH W Max.	WIDTH W Min.	DEPTH H Max.	DEPTH H Min.
1/8 x 1/2	.1255	.1240	.1405	.1355
3/16 x 3/4	.1880	.1863	.2193	.2143
1/4 x 1	.2505	.2487	.3130	.3080
5/16 x 1 1/8	.3130	.3111	.3278	.3228
3/8 x 1 1/4	.3755	.3735	.3595	.3545

Note: To determine distance key projects from slot, subtract depth "h" in Figure 201 from height of key "C" in Figure 200. This should nominally be equal to one-half the width of the key.

SIZE OF STANDARD KEYWAYS FOR MILLING CUTTERS AND SOME SPECIAL SHAFTING

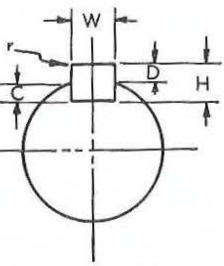


FIG. 234

Shaft Size	Key Size W × H	Corner Radius r	Cutter Keyway Depth D	Shaft Keyway Depth C
1/2	3/32 x 3/32	.020	3/64	3/64
5/8-7/8	1/8 x 1/8	1/32	1/16	1/16
1	1/4 x 1/4	3/64	3/32	5/32
1 1/4	5/16 x 5/16	1/16	1/8	3/16
1 1/2	3/8 x 3/8	1/16	5/32	7/32
1 3/4	7/16 x 7/16	1/16	3/16	1/4
2	1/2 x 1/2	1/16	3/16	5/16
2 1/2	5/8 x 5/8	1/16	7/32	13/32
3	3/4 x 3/4	3/32	1/4	1/2
3 1/2	7/8 x 7/8	3/32	3/8	1/2
4	1 x 1	3/32	3/8	5/8

Note: When a square key is used, dimensions D and C are not equal except on the smaller shafts. If desired, a flat key with a dimension H equal to twice dimension D can be used.

AMERICAN STANDARD (A.S.A.) SQUARE AND FLAT PARALLEL STOCK KEYS

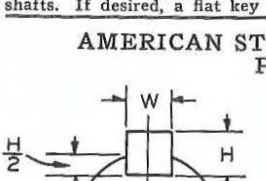


FIG. 235

Shaft Diameter	Square Key W × H	Flat Key W × H
1/2- 9/16	1/8 x 1/8	1/8 x 3/32
5/8- 7/8	3/16 x 3/16	3/16 x 1/8
15/16-1 1/4	1/4 x 1/4	1/4 x 3/16
1 5/16-1 3/8	5/16 x 5/16	5/16 x 1/4
1 7/16-1 3/4	3/8 x 3/8	3/8 x 1/4
1 13/16-2 1/4	1/2 x 1/2	1/2 x 3/8
2 5/16-2 3/4	5/8 x 5/8	5/8 x 7/16
2 7/8-3 1/4	3/4 x 3/4	3/4 x 1/2
3 3/8-3 3/4	7/8 x 7/8	7/8 x 5/8
3 7/8-4 1/2	1 x 1	1 x 3/4

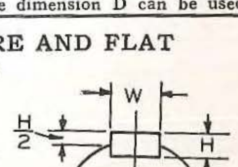
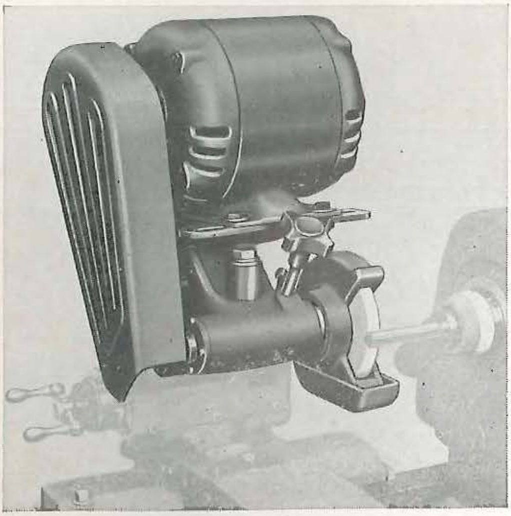


FIG. 236

Note: Dimension H/2 is measured at the side of the keyseat, not in the central plane of the key corresponding to the shaft diameter.



TOOL POST GRINDERS

FIG. 238

The tool post grinder shown in Figure 238 is used for both external and internal finishing whenever precision and a polished surface is required. Some typical grinder jobs: hardened shafts, bushings, tools, dies, lathe centers, arbors, tapered sockets, valves, reseating cutters, milling cutters, valve stems, tappet screws, spiral, taper or straight reamers. A complete diamond dresser for keeping wheels true and sharp is furnished.

MOUNTING THE GRINDER

The grinder is in correct position for most operations when the grinder spindle is on the exact lathe center line. After the tool post has been removed, clean both the grinder base and tool post slot, and clamp the grinder in the slot. Remove the grinder belt guard and loosen clamp on elevating screw. Align center of grinder spindle with lathe tailstock center by adjusting elevating screw. Retighten clamp on elevating screw and replace belt guard.

Swivel and tighten the compound rest at the correct angle. When grinding a surface parallel to the lathe center line, set the compound rest at 0 and feed the carriage back and forth by hand or with the power feed. When grinding at an angle, the compound rest is set at the proper angle and the grinder is fed back and forth with the compound rest feed. The taper attachment (page 204) simplifies taper grinding.

9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

IMPORTANT: *PROTECT THE LATHE*

Grinding dust is a mixture of abrasive dust and fine particles of steel. This dust is extremely harmful when allowed to fall and remain on the lathe bed ways and cross slide. ***Always cover the bed ways and the cross slide during grinding operations.***

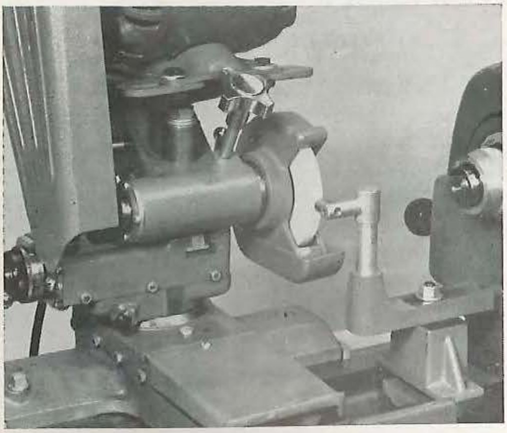
Paper, oilcloth or canvas makes a good cover. When using a cloth, be sure it is closely woven. After the grinding operation, clean the bed ways and carriage dovetails thoroughly. Then apply plenty of clean oil.

DRESSING THE GRINDING WHEEL

When the grinder has been mounted in position and the lathe is properly protected from emery dust, the grinding wheel should be dressed. Figure 242 shows how the dressing tool is mounted in the holder which has been clamped to the lathe bed. The diamond point of the dresser should be at an angle and slightly below center as shown in Figure 243.

Wheel Dresser with Diamond Point.

With the diamond point tightened in the proper position, start the grinder and move the wheel SLOWLY across the diamond with the same feed which will be used in the grinding operation.



Take light cuts and run the wheel back and forth until the diamond cuts evenly and has removed all glazed surface from the wheel.

For a fine accurate finish, the grinding wheel must be dressed before every operation and in exactly the position in which it will be used.

FIG. 242 (Left) Dressing External Grinding Wheel.

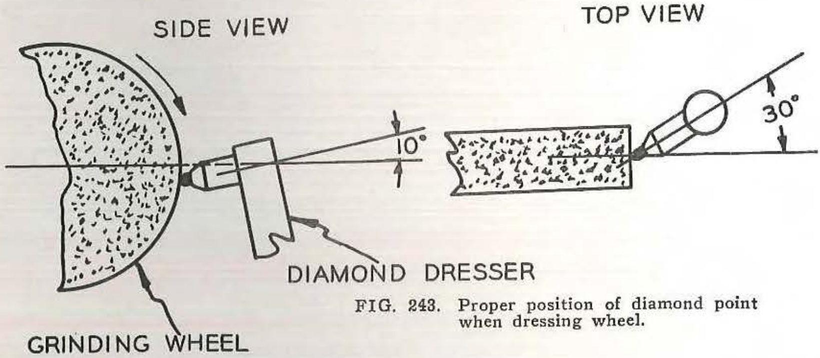


FIG. 243. Proper position of diamond point when dressing wheel.

SPEEDS FOR GRINDING

Two step pulleys from the motor and wheel spindles of the grinder give two spindle speeds. *Always use the lower speed for external grinding and the higher speed for internal grinding.*

DIRECTION OF LATHE SPINDLE ROTATION

In grinding operations the work must turn in a direction opposite that of the grinding wheel. Figure 244 shows how the rotation of the lathe spindle must be clockwise (backward) for external grinding and counterclockwise (forward) for internal grinding.

REVERSING SWITCH:—a dependable reversing switch is essential not only for grinding and polishing but also for such operations as tapping, nut-setting and wood-sanding. The reversing switch shown in Figure 245 is mounted easily in a convenient position on the headstock guard of the horizontal countershaft lathe or cabinet on underneath drive lathe. This switch is satisfactory for practically any motor wired for reversing.

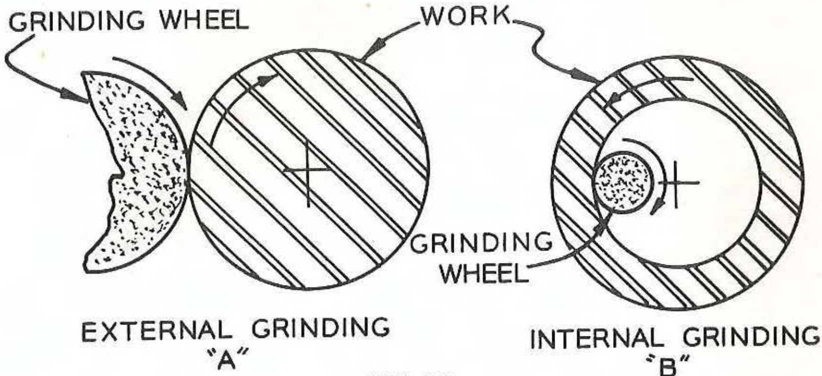


FIG. 244

The work must always turn a direction opposite that of the grinding wheel at the point of contact. At "A" it is necessary to reverse lathe spindle—at "B" the same effect is obtained by running the lathe spindle in a "Forward" direction.

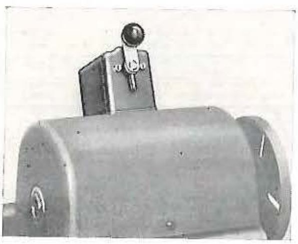
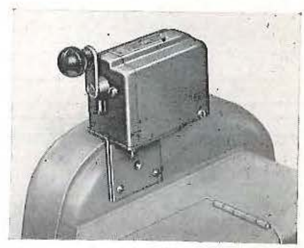


FIG. 245
Drum-type reversing switch mounted on headstock guard.



EXTERNAL GRINDING

Remember that grinding is a finishing operation. The work should be turned as close to the final finish size as possible before the grinding operation is begun.

With the work and the grinder mounted in position and the grinding wheel dressed properly, advance the wheel into one end of the work. Take light cuts across the entire length of the work. If using the automatic carriage feed, set up the change gears for the .0035 inch feed. Hand feeds should be very slow and even. The last finishing cut should be less than .001 inch—very often a last pass is taken without advancing the feed. When hardened stock is being ground, redress the wheel before taking the final cuts.

INTERNAL GRINDING

Figure 246 shows a typical internal grinding operation. The quill which holds the grinding wheel for internal work is threaded and tapered to fit inside the grinder spindle after the external wheel is removed (see Fig. 247). The higher grinder spindle speed should be used, and the lathe spindle must be rotating in a "Forward" direction (see Fig. 244).

To mount the internal grinding wheel: Remove front plate on external wheel guard, large wheel collar, and external wheel. After cleaning threads and tapers, turn quill into grinder spindle — this fit should be snug but not forced. Remove fillister screw at end of quill and screw proper wheel into quill BY HAND (do not use wrench or pliers). Dress wheel at proper angle (see page 190).

FIG. 246. Grinding the inside of a bushing.

During internal grinding operations, it is necessary to take light cuts and feed in very slowly because of the overhang of the grinding wheel. After the last cut allow the wheel to pass back and forth across the work several times without advancing the feed.

Be sure to remove the external wheel before beginning internal grinding operations.

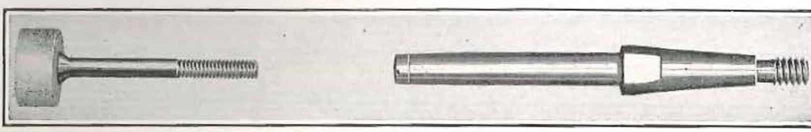


FIG. 247 Arbor and wheel for internal grinding. The threaded portion serves only to pull arbor into tapered socket.

GRINDING VALVES, 15° to 75°

Mount valve in Jacobs headstock chuck as shown in Figure 248. Mount external wheel on grinder spindle, align spindle with tailstock center (page 190), and dress wheel with compound rest set at proper angle for valve. With the lathe spindle turning in a direction opposite that of the grinding wheel, feed in slowly with the compound rest feed, taking light cuts.

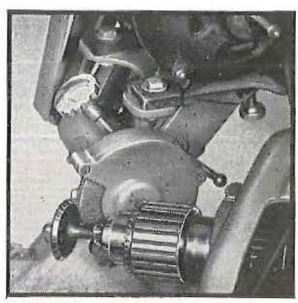


FIG. 248. Grinding 30° Valve.

9 - WOODTURNING
10 - TABLES
11 - INDEX
12 - NOTES

GRINDING FLAT VALVES, 90°

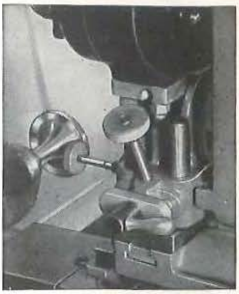


FIG. 249
Grinding 90° Valve.

Mount valve in Jacobs headstock chuck as shown in Figure 249. Mount internal wheel on grinder spindle and align spindle with tailstock center (page 190). With compound rest set at 0, dress grinding wheel, feeding across diamond with compound rest feed. With the lathe spindle turning in a direction opposite that of the grinding wheel, feed in slowly with the compound rest feed, taking light cuts.

GRINDING 60° LATHE CENTERS

With the center and sleeve mounted in the lathe spindle, dress external wheel with the compound rest set at exactly the proper angle. Adjust belts to obtain a slow lathe spindle speed and shift reversing switch lever so that lathe spindle is turning in a direction opposite that of the grinding wheel. Feed up to the center with the carriage handwheel, lock carriage in position and feed across center slowly with compound rest feed. Take light cuts.

NOTES ON GRINDING

A spotty, mottled surface usually means that it is time to dress the grinding wheel.

See that the work is held rigidly—vibration causes poor work.

No lubricant or cutting compound is necessary except for production work.

Keep the tool post grinder clean and well oiled. The bearings are grease packed at the time of assembly and need no further attention.

Always Take Light Cuts When Grinding.

ARMATURE WORK ON THE LATHE

Figures 258 and 259 show two steps in reconditioning armature commutators. First, the commutator segments are "trued" by a light, accurate cut with a carefully ground tool. Second, the mica insulation is undercut, using the motor driven undercutting attachment, (Fig. 260).

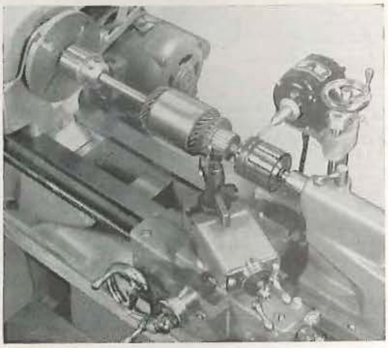
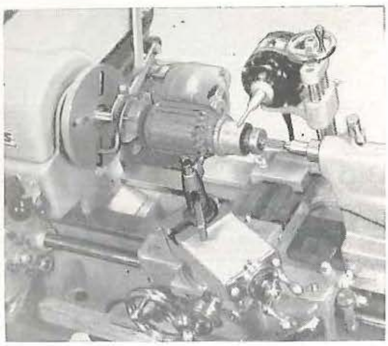


FIG. 258
Trueing a centerless armature before cutting mica. Armature is being supported by Jacobs headstock and center rest chucks. Note undercutting attachment on back of carriage.

FIG. 259
Undercutting mica on armature. This armature shaft has center holes and is driven with a dog during the trueing operation. Note that compound rest can be left on carriage during this operation.



MOUNTING THE ARMATURE

The two methods of mounting the armature are illustrated in Figures 258 and 259. Armature shafts with center holes are mounted between lathe centers and driven with a dog. Be sure that both center holes are free from burrs and dirt. Centerless armatures are mounted as shown in Figure 258—the Jacobs headstock chuck or the universal chuck drives the shaft, and the Jacobs center rest chuck (Fig. 262) supports the tailstock end. Jacobs chucks and arbors are available in a metal case as shown in Figure 261. Tighten the bronze jaws of the center rest chuck just enough to remove looseness but not enough to cause a "drag." Lock the jaws securely in position by turning the collar. IMPORTANT: During the turning operation apply plenty of lubricant on the shaft at the point of bearing with the jaws of the center rest chuck.

THE TRUEING CUT

The trueing cut must be smooth and even, so as to prevent sparking at the armature brushes. There are several methods of grinding tool bits for the trueing cut. The tool shown in Figure 263 has been found satisfactory by many auto repair mechanics.

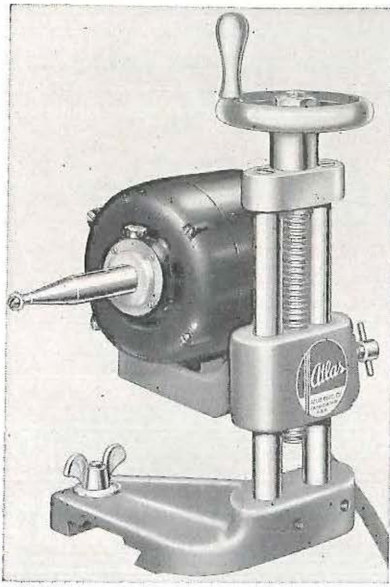


FIG. 260
Motor Driven Mica Undercutter.

FIG. 261
Armature Chuck Kit.

Grind the tool bit to the angles indicated—hone it thoroughly at regular intervals.

Set up the gear train for a fine feed—the three finest feeds shown in the carriage feed table, Section 7, Threading, are recommended. Take light cuts with a surface speed of 200 to 300 feet per minute (pages 45 and 47).

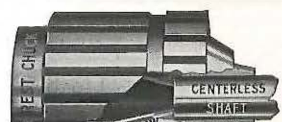


FIG. 262
Jacobs Center Rest Chuck.

UNDERCUTTING MICA

The motor driven undercutting attachment shown in Figure 260 is recommended for the undercutting operation. The base is clamped to the cross slide dovetail and is put into action without removing the tool post.

After the trueing cut, choose an undercutting saw which is the same width as the commutator slot. Mount saw on cutter arbor. Adjust undercutter so that saw is exactly on the lathe center line and in position to take a cut of about the same depth as the width of the slot. Start the undercutter motor and feed the saw through

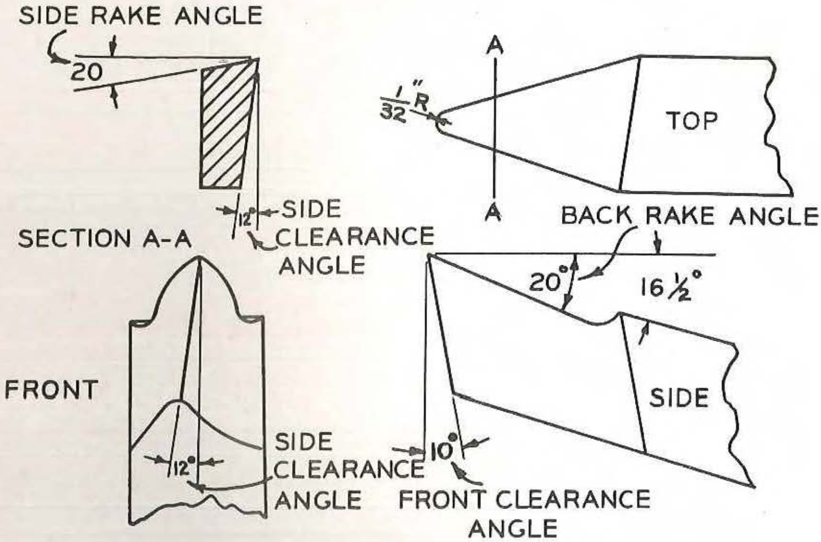


FIG. 263. Approximate tool angles recommended for trueing cut on commutator.

the mica with the carriage handwheel—hold the armature with the left hand and be careful not to cut into the copper segments. The carriage stop (page 182) aids in cutting each slot exactly the same length. Repeat undercutting for each slot, rotating armature by hand.

To polish commutator after undercutting: Take a very light turning cut across the commutator. Then sand with a strip of No. 2/0 or 3/0 sand paper or flint paper about the same width as the commutator (never use emery or carborundum paper). Do not hold the paper against the commutator with the fingers, but use a strip long enough so that an end can be held in each hand.

TAPER TURNING ON THE LATHE

Tapers are cut in one of three ways: (1) with a taper cutting attachment, (2) by setting over the tailstock center and (3) by feeding in at an angle with the compound rest feed. These methods are described in detail in the following paragraphs. A taper cutting attachment is considered the most satisfactory of these methods because it increases accuracy, eliminates computation and simplifies repeat operations.

IMPORTANT: When cutting tapers, always set the point of the tool bit on the exact lathe center line.

TAPER PER FOOT

The difference between the diameters of the two ends of a tapered piece of work, expressed in inches per foot of length, is known as "Taper per Foot."

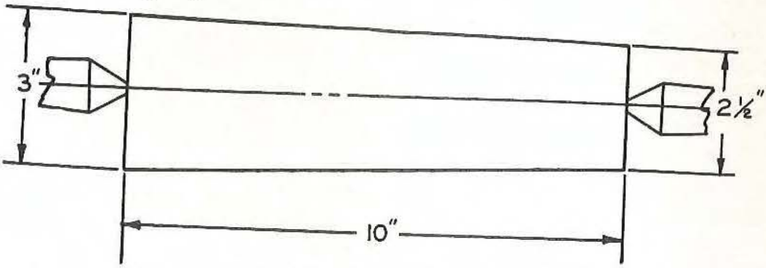


FIG. 264. Taper showing dimensions — see text below.

EXAMPLE: Figure 264 shows a taper with a diameter of 3 inches at one end and 2½ inches at the other.

Difference in diameters = .500 inch.

Length of taper = 10 inches = $\frac{10}{12}$ foot.

Then, taper per foot = .500 ÷ $\frac{10}{12}$ = .600 inch.

FORMULAS:

$$\text{Taper per foot (in inches)} = \frac{\text{Large diameter} - \text{Small diameter}}{\text{Length in feet}}$$

$$= \frac{12\ (\text{Large diameter} - \text{Small diameter})}{\text{Length in inches}}$$

$$\text{Taper per foot (in inches)} = \frac{\text{Large diameter} - \text{Small diameter}}{\text{Length in inches}}$$

CALCULATING ANGLE OF COMPOUND REST

The lathe compound rest can be swivelled and locked at any angle, and is ideal for cutting short tapers, taking angular cuts up to 2¼ inches in length, and boring tapered holes. When the desired taper is expressed in degrees and minutes, the angle is simply transferred to the proper side of the 90° reading on the graduated base of the compound rest (compound rest reading is exactly one-half of total included angle of taper). When the taper is expressed in inches per foot, convert this figure into degrees and minutes, first finding the tangent of the desired angle as follows:

$$\text{Tangent of Angle} = \frac{\text{Taper per Foot in Inches}}{24}$$

Then consult the tangent tables in any machinists handbook (see page 200) to obtain the equivalent of this tangent in degrees and minutes.

EXAMPLE: To set compound rest at proper angle to cut a standard No. 2 Morse Taper.

The No. 2 Morse Taper has a taper per foot of .59941 inch (Table XXX).

The tangent of required angle is $\frac{.59941}{24} = .024975$. Referring to tangent table (page 200): .024975 is the tangent of an angle of 1 degree, 26 minutes or slightly less than $1\frac{1}{2}$ degrees. Therefore, the correct compound rest setting is 1° 26′ from the 90° reading.

CALCULATING TAILSTOCK SETOVER

Figures 265 and 26 show how tapers are cut by setting over the tailstock. Figure 267 shows how to determine the proper direction of tailstock setover. Setting the tailstock forward (toward tool post) results in a taper with the smaller diameter at the tailstock end of the work. Setting the tailstock backward (away from the tool post) results in a smaller diameter at the headstock end of the work.

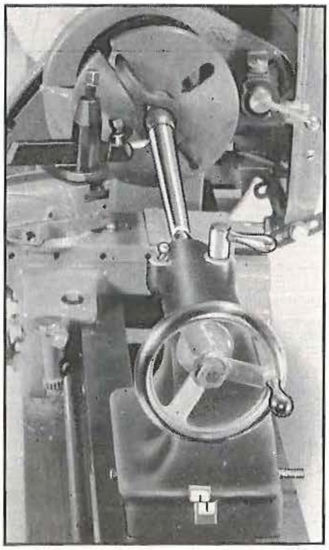


FIG. 265
Tapering with tailstock set over.

In determining the proper amount of tailstock setover, bear in mind that the amount of tailstock setover varies with the length of the piece being tapered — the tailstock must be reset in order to cut the same taper on pieces of different lengths. Use these formulas (S = Setover in inches):

(1) When Taper per Foot is given:—

$$S = \frac{\text{Taper per Foot} \times \text{Length of Taper in Inches}}{24}$$

(2) When entire length of piece is to be tapered and the diameters at ends of tapers are given:—

$$S = \frac{\text{Large Diameter} - \text{Small Diameter}}{2}$$

(3) When a portion of piece is to be tapered and the diameters at ends of the tapered portion are given[1]:—

$$S = \frac{\text{Total Length of Work}}{\text{Length to be Tapered}} \times \frac{\text{Large Diameter} - \text{Small Diameter}}{2}$$

[1] A taper of more than 1 inch per foot on stock 6 inches or less in length is usually cut by tapering only a portion of a longer piece of work and removing the waste stock after the taper is completed. This method avoids a small inaccuracy which would otherwise result, since, after the tailstock has been set over, the angle of the lathe centers does not match exactly the angle of the countersunk holes in the work.

FIGURE 266 TABLE OF TANGENTS

As explained on page 199, a table of tangents is essential in converting taper per foot into degrees and minutes when calculating the proper setting of the compound rest for taper cutting. In the table below, tangents are listed for every 15 minutes of angle. To obtain angles for intermediate tangents, use interpolation. For example: The tangent of the taper in the example on page 185 is .024975, which is between the tangent readings of 1° 15′ and 1° 30′. The exact reading is obtained as follows:

$$.02618 - .02182 = .00436$$

$$.024975 - .02182 = .003155$$

$$\frac{.003155}{.00436} \times 15 \text{ min.} = 11 \text{ min.}$$

1 deg. 15 min + 11 min. = 1 deg. 26 min., *required angle.*

Degrees	Minutes	Tangent	Degrees	Minutes	Tangent	Degrees	Minutes	Tangent
0	0	.00000	13	0	.23087	26	0	.48773
	15	.00436		15	.23547		15	.49314
	30	.00873		30	.24008		30	.49858
	45	.01309		45	.24470		45	.50404
1	0	.01745	14	0	.24933	27	0	.50952
	15	.02182		15	.25397		15	.51503
	30	.02618		30	.25862		30	.52057
	45	.03055		45	.26328		45	.52612
2	0	.03492	15	0	.26795	28	0	.53171
	15	.03929		15	.27263		15	.53732
	30	.04366		30	.27732		30	.54295
	45	.04803		45	.28203		45	.54862
3	0	.05241	16	0	.28674	29	0	.55431
	15	.05678		15	.29147		15	.56003
	30	.06116		30	.29621		30	.56577
	45	.06554		45	.30096		45	.57155
4	0	.06993	17	0	.30573	30	0	.57735
	15	.07431		15	.31051		15	.58318
	30	.07870		30	.31530		30	.58904
	45	.08309		45	.32010		45	.59494
5	0	.08749	18	0	.32492	31	0	.60086
	15	.09189		15	.32975		15	.60681
	30	.09629		30	.33459		30	.61280
	45	.10069		45	.33945		45	.61882
6	0	.10510	19	0	.34433	32	0	.62487
	15	.10952		15	.34921		15	.63095
	30	.11393		30	.35412		30	.63707
	45	.11836		45	.35904		45	.64322
7	0	.12278	20	0	.36397	33	0	.64941
	15	.12722		15	.36892		15	.65563
	30	.13165		30	.37388		30	.66188
	45	.13609		45	.37887		45	.66818
8	0	.14054	21	0	.38386	34	0	.67451
	15	.14499		15	.38888		15	.68087
	30	.14945		30	.39391		30	.68728
	45	.15391		45	.39896		45	.69372
9	0	.15838	22	0	.40403	35	0	.70021
	15	.16286		15	.40911		15	.70673
	30	.16734		30	.41421		30	.71329
	45	.17183		45	.41933		45	.71990
10	0	.17633	23	0	.42447	36	0	.72654
	15	.18083		15	.42963		15	.73323
	30	.18534		30	.43481		30	.73996
	45	.18985		45	.44001		45	.74673
11	0	.19438	24	0	.44523	37	0	.75355
	15	.19891		15	.45047		15	.76042
	30	.20345		30	.45573		30	.76733
	45	.20800		45	.46101		45	.77428
12	0	.21256	25	0	.46631	38	0	.78128
	15	.21712		15	.47163		15	.78834
	30	.22169		30	.47697		30	.79543
	45	.22628		45	.48234		45	.80258

Degrees	Minutes	Tangent
39	0	.80978
	15	.81703
	30	.82434
	45	.83169
40	0	.83910
	15	.84656
	30	.85408
	45	.86165
41	0	.86929
	15	.87698
	30	.88472
	45	.89253
42	0	.90040
	15	.90834
	30	.91633
	45	.92439
43	0	.93251
	15	.94071
	30	.94896
	45	.95729
44	0	.96569
	15	.97416
	30	.98270
	45	.99131
45	0	1.0000
	15	1.0088
	30	1.0176
	45	1.0265
46	0	1.0355
	15	1.0446
	30	1.0538
	45	1.0630
47	0	1.0724
	15	1.0818
	30	1.0913
	45	1.1009
48	0	1.1106
	15	1.1204
	30	1.1303
	45	1.1403
49	0	1.1504
	15	1.1605
	30	1.1708
	45	1.1812
50	0	1.1917
	15	1.2024
	30	1.2131
	45	1.2239
51	0	1.2349
	15	1.2460
	30	1.2572
	45	1.2685
52	0	1.2799
	15	1.2915
	30	1.3032
	45	1.3151
53	0	1.3270
	15	1.3392
	30	1.3514
	45	1.3638
54	0	1.3764
	15	1.3891
	30	1.0419
	45	1.4150
55	0	1.4281
	15	1.4415
	30	1.4550
	45	1.4687
56	0	1.4826
	15	1.4966
	30	1.5108
	45	1.5252
57	0	1.5399
	15	1.5547
	30	1.5697
	45	1.5849
58	0	1.6003
	15	1.6160
	30	1.6318
	45	1.6479
59	0	1.6643
	15	1.6808
	30	1.6977
	45	1.7147
60	0	1.7320
	15	1.7496
	30	1.7675
	45	1.7856
61	0	1.8040
	15	1.8227
	30	1.8418
	45	1.8611
62	0	1.8807
	15	1.9007
	30	1.9210
	45	1.9416
63	0	1.9626
	15	1.9840
	30	2.0057
	45	2.0278
64	0	2.0503
	15	2.0732
	30	2.0965
	45	2.1203
65	0	2.1445
	15	2.1692
	30	2.1943
	45	2.2199
66	0	2.2460
	15	2.2727
	30	2.2998
	45	2.3276
67	0	2.3558
	15	2.3847
	30	2.4142
	45	2.4443
68	0	2.4751
	15	2.5065
	30	2.5386
	45	2.5715
69	0	2.6051
	15	2.6394
	30	2.6746
	45	2.7106
70	0	2.7475
	15	2.7852
	30	2.8239
	45	2.8636
71	0	2.9042
	15	2.9459
	30	2.9887
	45	3.0326
72	0	3.0777
	15	3.1240
	30	3.1716
	45	2.2205
73	0	3.2708
	15	3.3226
	30	3.3759
	45	3.4308
74	0	3.4874
	15	3.5457
	30	3.6059
	45	3.6679
75	0	3.7320
	15	3.7983
	30	3.8667
	45	3.9375
76	0	4.0108
	15	4.0867
	30	4.1653
	45	4.2468
77	0	4.3315
	15	4.4194
	30	4.5107
	45	4.6057
78	0	4.7046
	15	4.8077
	30	4.9151
	45	5.0273
79	0	5.1445
	15	5.2671
	30	5.3955
	45	5.5301
80	0	5.6713
	15	5.8196
	30	5.9758
	45	6.1402
81	0	6.3137
	15	6.4971
	30	6.6911
	45	6.8969
82	0	7.1154
	15	7.3479
	30	7.5957
	45	7.8606
83	0	8.1443
	15	8.4489
	30	8.7769
	45	9.1309
84	0	9.5144
	15	9.9310
	30	10.385
	45	10.883
85	0	11.430
	15	12.035
	30	12.706
	45	13.457
86	0	14.301
	15	15.257
	30	16.350
	45	17.610
87	0	19.081
	15	20.819
	30	22.904
	45	25.452
88	0	28.636
	15	32.730
	30	38.188
	45	45.829
89	0	57.290
	15	76.390
	30	114.59
	45	229.18

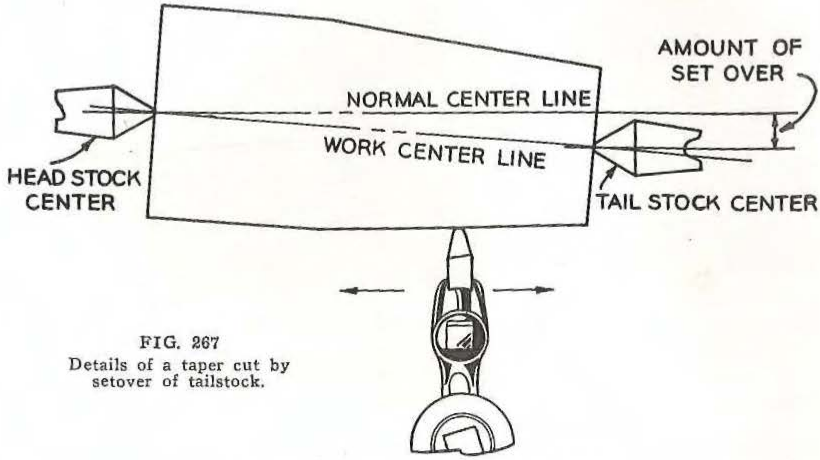


FIG. 267
Details of a taper cut by setover of tailstock.

SETTING OVER THE TAILSTOCK

To set over the tailstock, first loosen the tailstock clamping nut. The upper part of the tailstock is locked in position by tightening two headless set screws, one in the front and one in the back of the base casting (see Fig. 269). Loosen the headless set screw on the side toward which the tailstock will be moved. Then the upper tailstock will move in that direction when the other headless set screw is tightened. The index line at the handwheel end of the tailstock indicates the approximate amount of setover (see Figs. 265 and 269).

Figure 268 shows how to measure the amount of tailstock setover. The tailstock is moved close to the headstock, and a steel

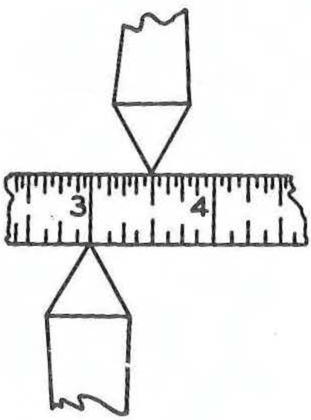


FIG. 268
Measuring amount of tailstock setover.

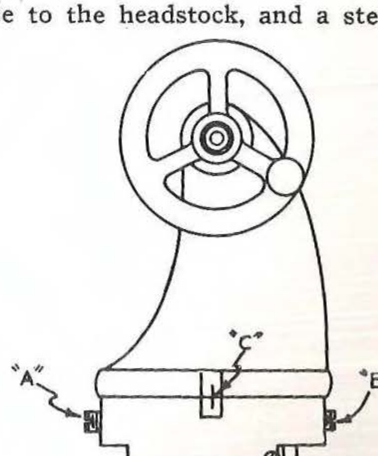


FIG. 269
Handwheel end of tailstock showing index line "C" and headless set screws "A" and "B."

scale, graduated at both edges is held in position for taking the measurement. A proper scale for work of this type is shown in Figure 270. When the tailstock is in position, tighten the tailstock clamping nut.

FIG. 270. Flexible steel rule graduated on two edges. Handy for measuring tailstock setover.

REALIGNING THE TAILSTOCK AFTER TAPER WORK

To restore alignment of tailstock with headstock, loosen tailstock clamping nut and adjust headless set screws until index line indicates that tailstock is in approximate position.

Figure 271 shows how to find the exact position of the tailstock. The check bar mounted between centers is between 12 and 15 inches long and has a shoulder at each end as indicated. A light cut is taken at "A" and "B" and the two diameters are measured with a micrometer. Then:

If the diameter of tailstock shoulder "B" is MORE than the diameter of headstock shoulder "A," the tailstock must be moved FORWARD a distance of half the difference in diameters.

If the diameter of tailstock shoulder "B" is LESS than the diameter of headstock shoulder "A," the tailstock must be moved BACKWARD a distance of half the difference in diameters.

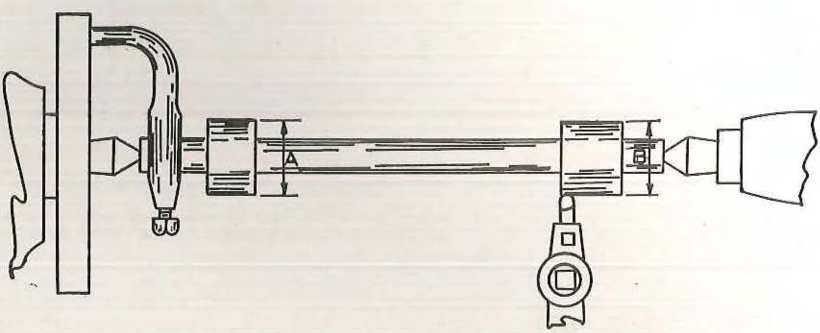


FIG. 271. Check bar for checking alignment of lathe centers. See text above.

The alignment of lathe centers should be checked at regular intervals, so as to maintain accuracy of long cuts. A long cut on a straight bar is sometimes taken instead of the two short cuts on the shoulders of the check bar. A check bar like the one above, however, is a much more rapid method of making the alignment test.

TAPER CUTTING ATTACHMENT

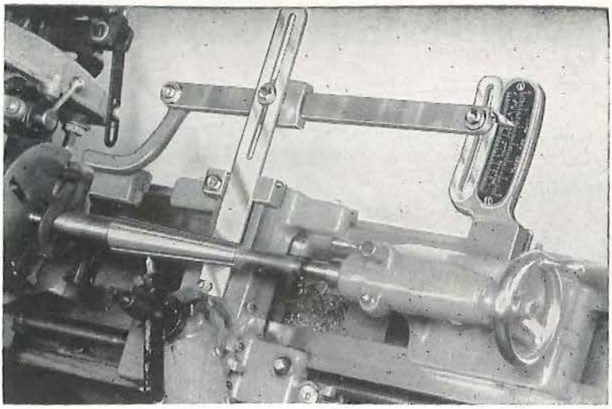


FIG. 272. Taper Cutting Attachment in Operation.

The taper cutting attachment shown in Figure 272 has many advantages over the tailstock setover method. Lathe centers are never taken out of alignment; bearing surfaces of the lathe centers are not affected; duplicate tapers may be cut quickly on pieces of different length; and taper boring, impossible with tailstock setover, is handled quickly and easily. The index plate is graduated in degrees and taper per foot (in inches), simplifying computation and setting.

The slide bar is installed parallel to the lathe bed way and set at the desired angle. A slotted draw bar connects the slide bar and the carriage cross slide, guiding the cutting operation.

ASSEMBLING THE TAPER CUTTING ATTACHMENT

Refer to Figure 274

With the pivot bracket and taper gauge bracket attached to the guide bar fasten the brackets to the lathe bed at the desired position. Attach the pointer to the end of the guide bar with the screw furnished. Make sure that both the top and side surfaces of the guide bar are parallel with the machined ways of the lathe bed when the pointer is set at zero. This adjustment should be made with great care using an indicator to insure accuracy. If the

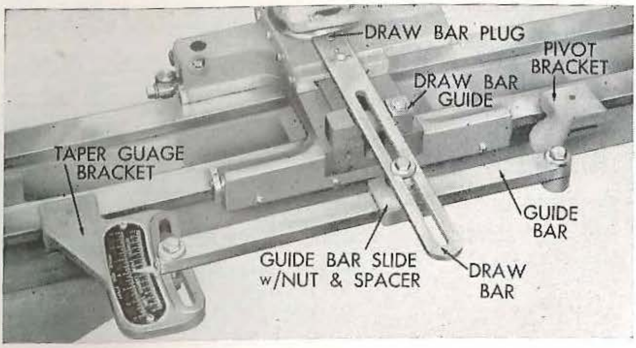


FIG. 274. Details of Taper Cutting Attachment.

guide bar does not check parallel with the side of the lathe bed, set the guide bar parallel with the bed and move the graduated plate until the pointer and the zero mark are in line and fasten in place.

Remove the chip guard from the lathe cross slide and advance cross slide until feed nut is disengaged from cross slide screw. Remove feed nut lock screw and feed nut.

Pull the cross slide back to about the position it is to be used. Insert brass plugs in draw bar guide and clamp guide on the end of the cross slide ways.

Connect draw bar to the cross slide by inserting the draw bar plug in the hole formerly occupied by the brass nut and place the guide bar slide, with feed slide nut inserted, on the guide bar. Adjust the three set screws in the draw bar until they just touch the cross slide then tighten in place by using the round head machine screw furnished. Take up side play between the draw bar and draw bar guide by adjusting the two screws in the guide.

OPERATING THE TAPER CUTTING ATTACHMENT

Position and clamp the guide bar at the desired angle and clamp the draw bar to the guide bar slide by tightening the screw. The angle scales are graduated in degrees and in inches per foot.

After mounting work in the lathe, set the compound rest at right angles to the bedways, with the point of the tool bit on the exact lathe center line. For most operations the small diameter of taper should be toward tailstock, and the cut toward the headstock.

Caution: Do not allow the guide bar slide to strike the cap screws at either end of guide bar especially when using the power feed.

Ordinarily, no computation is necessary in setting a taper cutting attachment. When the large and small diameters are specified for a certain length, apply the formula on page 199.

CHECKING STANDARD TAPERS

In cutting standard tapers such as those shown in Figures 275 and 276, the taper should be checked in a standard socket before the last cuts are taken. In making this check, make a light mark along the entire length of the taper with chalk or Prussian Blue and insert in a standard socket. Twist the taper and remove. If the entire length of the chalk line has been rubbed off, the taper is being cut at the proper angle and the finish cuts may be taken. If a portion of the chalk line has been left untouched, the taper setting is not correct. Make the necessary changes and repeat checking until the entire length of the chalk line is rubbed out.

Sockets to fit standard tapers should be cut to approximate size and reamed with a standard taper reamer. Arbors should be finish-ground with a tool post grinder (page 189) to avoid damaging the taper socket.

BROWN AND SHARPE TAPERS

Figure 275 shows standard Brown and Sharpe taper dimensions. Each of the tapers, except the No. 10, requires a taper per foot of .500 inch. The No. 10 has a taper of .5161 inch per foot. Take cuts until the diameter at the small end is equal to the dimension given in the table on page 207.

MORSE TAPERS

Standard Morse Taper dimensions are shown in Figure 276 The taper per foot varies for each size. When cutting Morse Tapers, it is preferable to set the lathe for the proper taper per foot, rather than to measure the large and small diameters.

IMPORTANT!

When cutting tapers, always have the point of the tool bit on the exact lathe center line. Be sure to check the taper in a standard socket before the final cuts are taken.

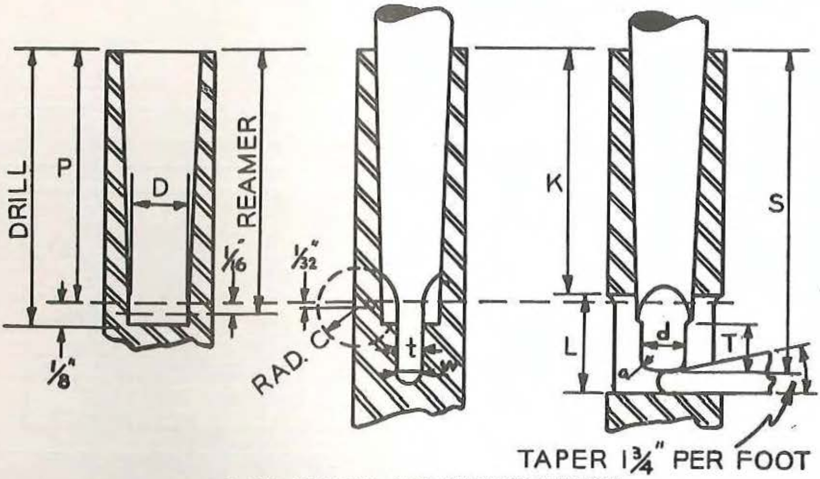


FIG. 275 BROWN AND SHARPE TAPERS
Taper approximates ½" per ft. except No. 10 which is .5161" per ft.

TABLE XXIX — BROWN & SHARPE TAPERS

No. of Taper	Diam. of Plug at Small End	Plug Depth P B & S Standard*	Keyway from End of Spindle	Shank Depth	Length of Keyway†	Width of Keyway	Length of Arbor Tongue	Diameter of Arbor Tongue	Thickness of Arbor Tongue	Radius of Tongue Circle	Radius of Tongue	Limit for Tongue —to project thru Test Tool
	D	*P*	*K*	*S*	*L*	*W*	*T*	*d*	*t*	*C*	*a*	
1	.200	15/16	15/16	1 3/16	3/8	.135	3/16	.170	1/8	3/16	.030	.003
2	.250	1 3/16	1 11/64	1 1/2	1/2	.166	1/4	.220	5/32	3/16	.030	.003
3	.312	1 1/2	1 15/32	1 7/8	5/8	.197	5/16	.282	3/16	3/16	.040	.003
4	.350	1 11/16	1 41/64	2 3/32	11/16	.228	11/32	.320	7/32	5/16	.050	.003
5	.450	2 1/8	2 1/16	2 9/16	3/4	.260	3/8	.420	1/4	5/16	.060	.003
6	.500	2 3/8	2 19/64	2 7/8	7/8	.291	7/16	.460	9/32	5/16	.060	.005
7	.600	2 7/8	2 25/32	3 13/32	15/16	.322	15/32	.560	5/16	3/8	.070	.005
8	.750	3 9/16	3 29/64	4 1/8	1	.353	1/2	.710	11/32	3/8	.080	.005
9	.900	4 1/4	4 1/8	4 7/8	1 1/8	.385	9/16	.860	3/8	7/16	.100	.005
10	1.0446	5	4 27/32	5 23/32	1 5/16	.447	21/32	1.010	7/16	7/16	.110	.005
11	1.250	5 15/16	5 25/32	6 21/32	1 5/16	.447	21/32	1.210	7/16	1/2	.130	.005
12	1.500	7 1/8	6 15/16	7 15/16	1 1/2	.510	3/4	1.460	1/2	1/2	.150	.005
13	1.750	7 3/4	7 9/16	8 9/16	1 1/2	.510	3/4	1.710	1/2	5/8	.170	.010
14	2.00	8 1/4	8 1/32	9 5/32	1 11/16	.572	27/32	1.960	9/16	3/4	.190	.010
15	2.25	8 3/4	8 17/32	9 21/32	1 11/16	.572	27/32	2.210	9/16	7/8	.210	.010
16	2.50	9 1/4	9	10 1/4	1 7/8	.635	15/16	2.450	5/8	1	.230	.010
17	2.75	9 3/4										
18	3.00	10 1/4										

* "B & S Standard" Plug Depths are not used in all cases.
† Special lengths of keyway are used instead of standard lengths in some places. Standard lengths need not be used when keyway is for driving only and not for admitting key to force out tool.

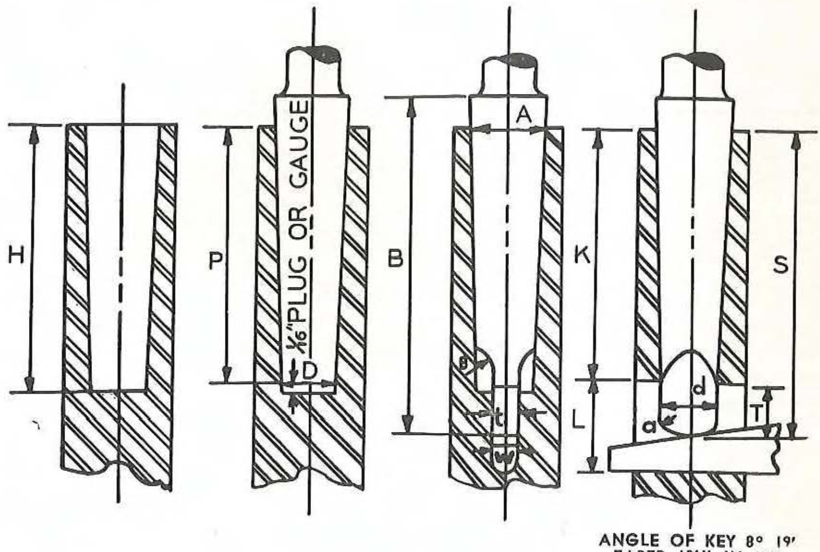


FIG. 276 MORSE TAPERS

TABLE XXX — MORSE TAPERS

Number of Taper	Diam. of Plug at Small End, Inches	Diam at End of Socket, Inches	Whole Length of Shank, Inches	SHANK: Shank Depth, Inches	SHANK: Depth of Hole, Inches	Standard Plug Depth, Inches	TONGUE: Thickness of Tongue, Inches	TONGUE: Length of Tongue, Inches	TONGUE: Rad. of Mill for Tongue, Inches	TONGUE: Diameter of Tongue, Inches	TONGUE: Radius of Tongue, Inches	KEYWAY: Width of Keyway, Inches	KEYWAY: Length of Keyway, Inches	KEYWAY: End of Socket to Keyway, Inches	Taper per Foot	Taper per Inch	Number of Key
	D	A	B	S	H	P	t	T	B	d	a	w	L	K			
0	.252	.3561	2 11/32	2 7/32	2 1/32	2	5/32	1/4	5/32	.235	.04	.160	9/16	1 15/16	.62460	.05205	0
1	.369	.475	2 9/16	2 7/16	2 3/16	2 1/8	13/64	3/8	3/16	.343	.05	.213	3/4	2 1/16	.59858	.04988	1
2	.572	.700	3 1/8	2 15/16	2 5/8	2 9/16	1/4	7/16	1/4	17/32	.06	.260	7/8	2 1/2	.59941	.04995	2
3	.778	.938	3 7/8	3 11/16	3 1/4	3 3/16	5/16	9/16	9/32	23/32	.08	.322	1 3/16	3 1/16	.60235	.05019	3
4	1.020	1.231	4 7/8	4 5/8	4 1/8	4 1/16	15/32	5/8	5/16	31/32	.10	.478	1 1/4	3 7/8	.62326	.05193	4
5	1.475	1.748	6 1/8	5 7/8	5 1/4	5 3/16	5/8	3/4	3/8	1 13/32	.12	.635	1 1/2	4 15/16	.63151	.05262	5
6	2.116	2.494	8 9/16	8 1/4	7 3/8	7 1/4	3/4	1 1/8	1/2	2	.15	.760	1 3/4	7	.62565	.05213	6
7	2.750	3.270	11 5/8	11 1/4	10 1/8	10	1 1/8	1 3/8	3/4	2 5/8	.18	1.135	2 5/8	9 1/2	.62400	.05200	7

TABLE XXXI — TAPERS AND ANGLES

Taper Per Foot	Included Angle Deg.	Included Angle Min.	With Center Line Deg.	With Center Line Min.	Taper Per Inch	Taper Per Inch from Center Line
1/8	0	36	0	18	.010416	.005203
3/16	0	54	0	27	.015625	.007812
1/4	1	12	0	36	.020833	.010416
5/16	1	30	0	45	.026042	.013021
3/8	1	47	0	53	.031250	.015625
7/16	2	05	1	02	.036458	.018229
1/2	2	23	1	11	.041667	.020833
9/16	2	41	1	21	.046875	.023438
5/8	2	59	1	30	.052084	.026042
11/16	3	17	1	39	.057292	.028646
3/4	3	35	1	47	.062500	.031250
13/16	3	53	1	56	.067708	.033854
7/8	4	11	2	06	.072917	.036456
15/16	4	28	2	14	.078125	.039063
1	4	46	2	23	.083330	.041667
1 1/4	5	58	2	59	.104666	.052084
1 1/2	7	09	3	34	.125000	.062500
1 3/4	8	20	4	10	.145833	.072917
2	9	32	4	46	.166666	.083332
2 1/2	11	54	5	57	.208333	.104166
3	14	15	7	08	.250000	.125000
3 1/2	16	36	8	18	.291666	.145833

Part 9

WOODTURNING
ON THE METAL LATHE

PART 9

WOODTURNING ON THE METAL LATHE

The heavy, rigid construction of the lathe headstock, tailstock and bed provides the strength essential for all types of woodturning operations. The higher speeds (pages 45 and 228) are satisfactory for many types of woodworking and are obtained with a 1725 R.P.M. motor.

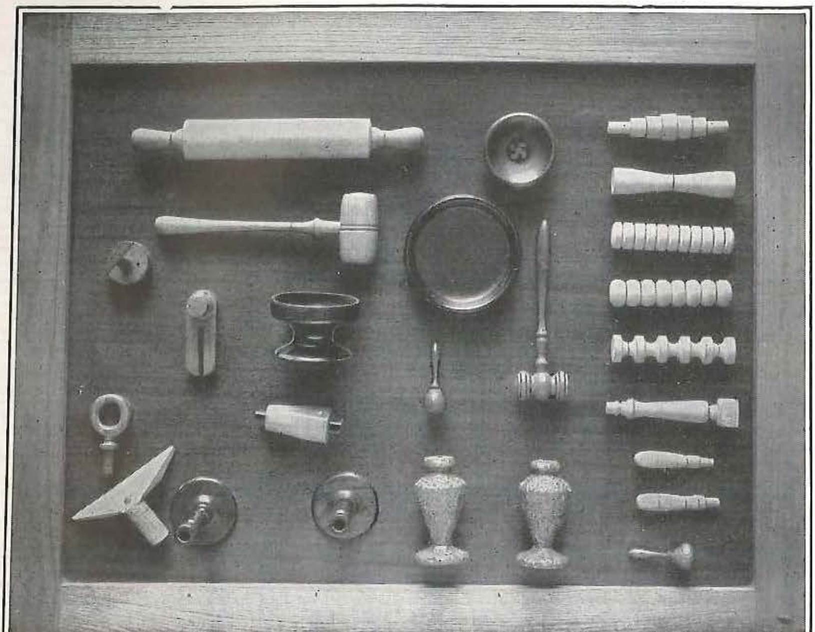


FIG. 283. Woodworking Projects.

The attachments necessary for woodworking operations are described on page 212. Figure 283 shows woodturning projects.

A Timken-equipped lathe (see page 8) is recommended for the shop handling quantities of both wood and metal turning. A Timken-equipped lathe spindle will stand up for long periods without adjustments, even under heavy loads and at high speeds.

WOODWORKING ATTACHMENTS FOR THE METAL LATHE

FIG. 284. Screw Center.

FIG. 285. Spur Center.

FIG. 286. Cup Center.

These three centers are companion pieces for wood turning on the metal lathe. Each center has a No. 2 Morse Taper shank. The screw center (Fig. 284) is mounted in the headstock for facing and hollowing operations. The spur center (Fig. 285) is used in the headstock to drive work mounted between lathe centers. The cup center (Fig. 286) acts as a dead center in the tailstock.

FIG. 290 Chisel Set

This set of wood turning chisels is adequate to fill the needs of most small shops. It includes 3 gouges, 2 skew chisels, one round nose, one spear point and one parting tool.

TYPES OF WOODS

(See pages 229 and 230)

The beginner in woodturning should learn about the various types of woods commonly used in lathe work. Softwoods such as white pine, red gum, and yellow poplar are easily worked; other woods of this type are birch, maple, walnut, mahogany and cherry. Oak, elm and ash are coarse grained woods and require especially careful turning. As a general rule, any medium-hard "decorative" wood is fairly easy to work. Plastics and dense hardwoods, such as ebony, are machined best by following the rules given in Part 4 (see page 62). With extreme care, plastics may be turned "free-hand" with the scraping type of woodworking tool used in face plate turning (page 226).

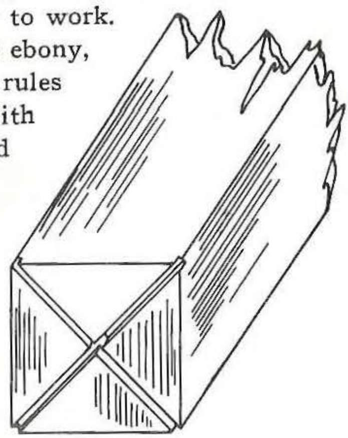


FIG. 292
Saw Cuts for Spur Center.

MOUNTING WORK BETWEEN CENTERS

When turning stock between centers, first locate the center on each end with reasonable accuracy. Square or regular-sized stock is centered by scribing lines across the ends from corner to corner. Round work is centered with the hermaphrodite caliper or center head attachment (see pages 67 and 68.)

After locating the center, make two saw cuts about ⅛ inch deep in the headstock end of the work as shown in Figure 292. These grooves will intersect at the center. When turning especially hard woods, drill holes of about 1/16 inch diameter for center points. With the work resting on the bench, drive the spur center into the saw cuts with a mallet. Many lathe men keep an extra spur center on hand for this purpose. ***Never drive the work against the spur center in the lathe—never force the work into the spur center by advancing the tailstock.***

Mount the spur center in the headstock and the cup center in the tailstock. Set the work in position against the spur center and advance cup center to tailstock end of work. Force point of cup

10 - TABLES
11 - INDEX
12 - NOTES

center into center of work just far enough to provide a firm bearing. Revolve work by hand to see if work turns freely; if it is too tight, back off the cup center slightly. Large pieces of square or rectangular stock will turn much more easily if the corner edges are planed before the work is mounted.

No. 10 motor oil or equivalent is an excellent lubricant for the tailstock center. However, since oil penetrates quickly into the work, it should be used only when there is plenty of waste stock at the tailstock end. With a small amount of waste stock, paraffin or beeswax is the best lubricant.

TURNING BETWEEN CENTERS

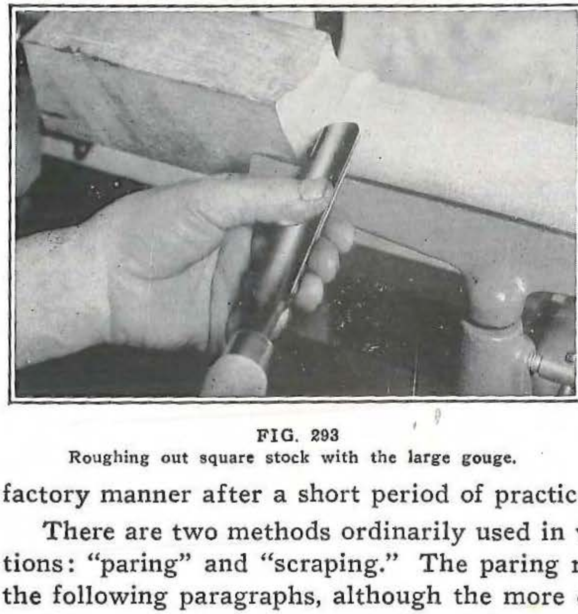


FIG. 293
Roughing out square stock with the large gouge.

The beginner should practice turning scrap stock between centers before attempting more difficult operations with expensive woods. The two most commonly used chisels, the gouge and skew (Figs. 293 and 298, can be handled in a satisfactory manner after a short period of practice and study.

There are two methods ordinarily used in wood turning operations: "paring" and "scraping." The paring method described in the following paragraphs, although the more difficult method, resembles more closely a true cutting action and usually results in a better class of work. The scraping method for face plate work is described on page 226.

TAKING THE CUT

As a general rule, the proper tool motion is *parallel with the grain of the work.* Always cut from the center toward the ends of the work — never start cuts at an end. When turning a taper, always cut toward the smaller diameter.

When taking the first cuts on rough stock, set the tool rest about 3/16 inch from the work and about 1/8 inch above the lathe center line. Move the tool rest forward for each cut as the diameter of the work decreases. When using the skew chisels, set the tool rest 1/8 inch or less from the work and enough above center to permit a paring cut. After setting the tool rest, always revolve the work by hand to make sure there is sufficient clearance. *Never adjust the tool rest while the lathe is running.*

The top of the tool rest should be kept smooth and straight so that tools always slide easily. Dress it occasionally with a file.

THE LARGE GOUGE

The large gouge chisel (Fig. 294) is used to "rough down" the stock to what will be approximately its largest diameter when completed. Notice that, unlike the cabinet maker's "firmer gouge," the point is rounded. The ground face ("A" in Fig. 294) is always ground flat or with a slight curve outward, never hollow ground. When honing the gouge, always hold the chisel and the stone free in the hands. Continue honing until a feather edge is felt, then remove this edge by honing lightly on the inside with a slip stone.

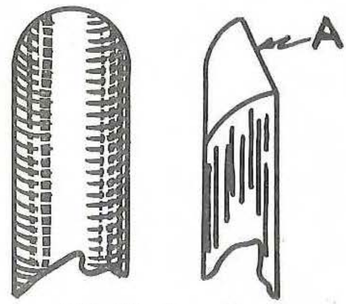


FIG. 294. Large Gouge.

Figures 295 and 296 show the proper method of holding the large gouge against the work. Grasp the chisel firmly, with one hand guiding the handle and the other holding the blade just behind the tool rest (Fig. 293). Move the gouge evenly along the work with the point of the tool ahead of the handle end. Take light cuts, and use slower lathe speeds when roughing out.

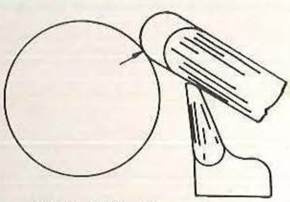


FIG. 295. Correct.

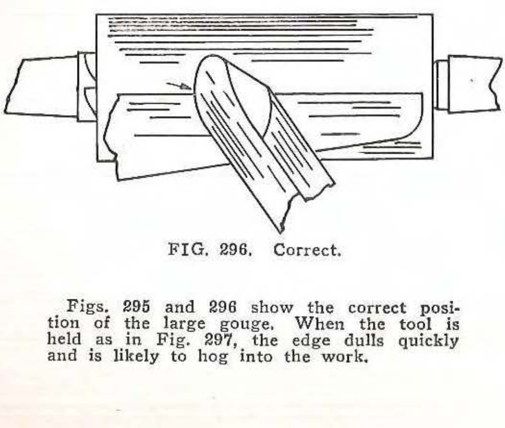


FIG. 296. Correct.

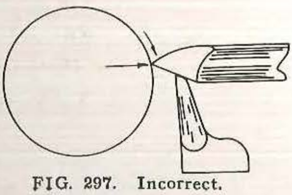


FIG. 297. Incorrect.

Figs. 295 and 296 show the correct position of the large gouge. When the tool is held as in Fig. 297, the edge dulls quickly and is likely to hog into the work.

Experiment to find the best chisel position. Figure 295 shows how the gouge is held just past the point where the ground side rubs on the work. Raise the handle to obtain this position. Never hold the gouge as shown in Figure 297. This position quickly dulls the cutting edge, produces rough work, and is very likely to split the stock.

Refer to Figure 337 for proper speeds for roughing cuts.

THE LARGE SKEW CHISEL

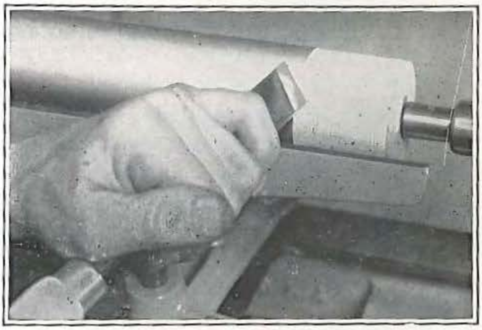


FIG. 298
Taking a finish cut with the large skew chisel.

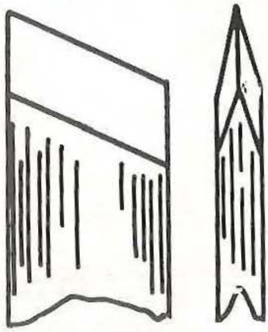


FIG. 299
Large Skew Chisel.

The large skew chisel (Fig. 299) is used for smooth cutting on cylindrical or long taper work. The cutting edge is "askew," or at an angle, and both side faces are ground to permit cuts to both the right and left.

When sharpening the large skew, the sides are ground off equally to an angle approximately equal to that shown in Figure 299. A finer cutting edge results in a cleaner cut, but becomes dull quicker than a large cutting angle. The ground surfaces should be flat—hollow grinding makes it difficult to hold the tool in the

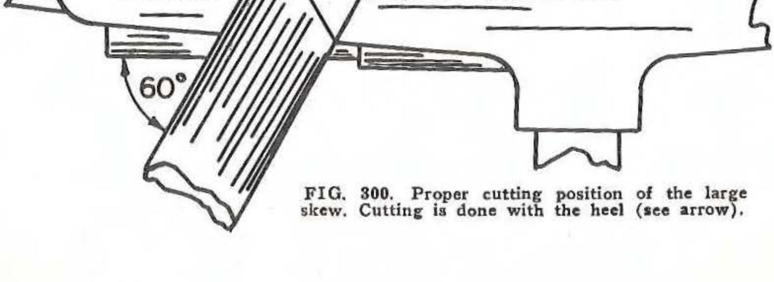


FIG. 300. Proper cutting position of the large skew. Cutting is done with the heel (see arrow).

correct position. Hone carefully to a sharp point with no "wire edge."

Figures 300 and 301 show the proper way to hold the large skew when taking a cut. Lift the handle just enough to allow the edge to cut and so that the lower ground side of the chisel rubs lightly on the work and prevents "hogging-in."

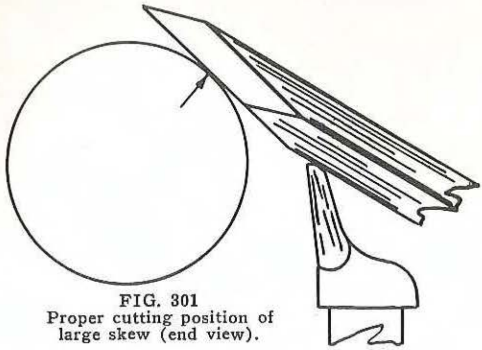


FIG. 301
Proper cutting position of large skew (end view).

Properly used, the large skew gives a smooth finish requiring very little sanding. Always cut from the center toward an end of the work—never start cutting at an end.

USING THE BLOCK PLANE ON CYLINDRICAL WORK

Long cylinders or tapers can be smoothed down accurately with a small block plane. The plane is held at an angle of approximately 45 degrees with the axis of the work as shown in Figure 302 Light cuts give a clean, continuous shaving and a smooth surface.

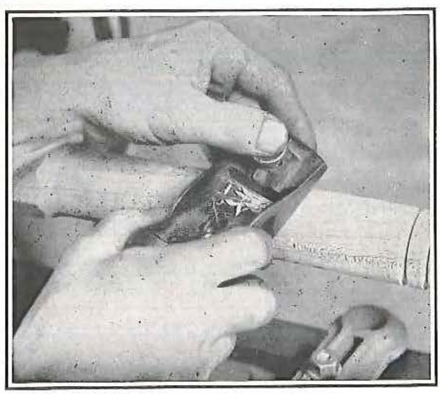


FIG. 302. Smoothing down work with the small block plane.

THE PARTING TOOL

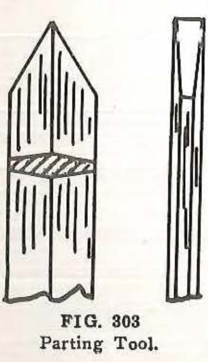


FIG. 303
Parting Tool.

The parting tool is used for two purposes: (1) for taking sizing cuts which serve as a guide in turning to size, and (2) for cutting off operations. Figure 303 shows the proper tool shape. The parting tool is a double wedge, wider at the center to provide clearance. The point is ground so that the cutting edge is on the *exact* center line of the tool. Grind the tool so that the sides are straight or slightly hollow-ground, then hone carefully. The cutting angle should be about 60°.

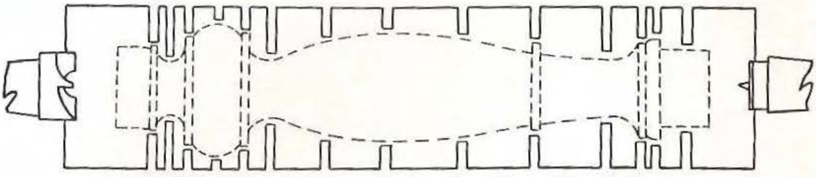


FIG. 304. Sizing grooves made with parting tool.

Figure 304 shows how the parting tool is used to cut grooves at various points of work which is to be turned to a required shape. Each groove is cut nearly as deep as the finish-diameter, allowing between 1/16 and 3/32 inch for finishing. The proper tool position is shown in Figure 306. Beginners should hold the tool with both hands and remove the tool when taking measurements with the caliper. ***Do not cut too deep.*** Experienced wood turners cut and caliper the sizing grooves at the same time (Fig. 305), a method which saves time and reduces the possibility of cutting the groove too deep.

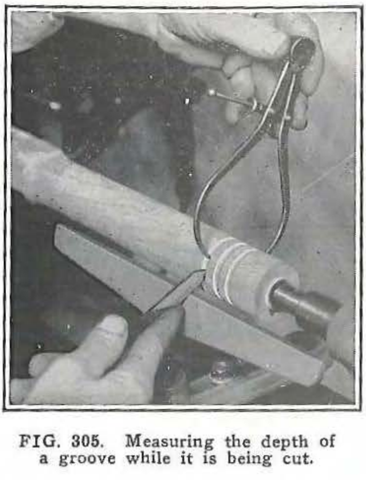


FIG. 305. Measuring the depth of a groove while it is being cut.

When using the parting tool for cutting off waste ends of the work, cut grooves at both ends so that the diameter is about ¼ inch at those points. Then cut the tailstock groove another ⅛ inch or slightly deeper. Now cut entirely through the headstock end and catch the work as it drops. The small skew chisel gives a smoother cutoff than the parting tool (see page 224). Some operators prefer to stop the lathe when the groove diameters are about ¼ inch and finish the cut-off with a saw.

The parting tool must be held carefully to prevent binding or "hogging-in." Figure 306 shows the proper tool position Hold the chisel firmly and advance it into the work at a right angle to the center line. After a few

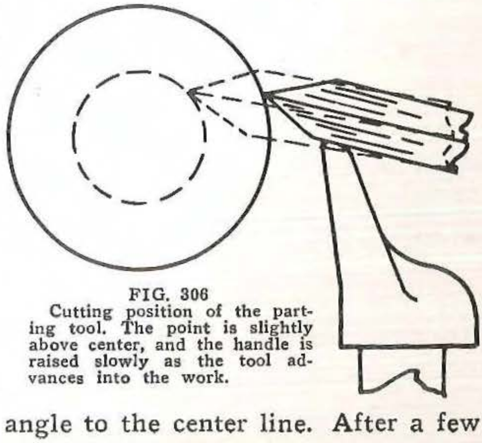


FIG. 306
Cutting position of the parting tool. The point is slightly above center, and the handle is raised slowly as the tool advances into the work.

practice cuts, the operator will get the proper "feel" of the parting tool and keep it automatically in the correct cutting position.

When taking deep cuts or cutting hardwood, the sides of the parting tool may rub enough to wedge and generate heat. Such jobs are simplified by taking two cuts and making a wider groove. Very high speeds are dangerous. ***Never force the tool in too fast —hard woods may burn the point of the tool and spoil its temper.***

TYPES OF CUTS IN WOOD TURNING

In addition to cylindrical turning and taper work, there are three other common types of cuts in wood turning: V cuts, convex cuts (or beads), and concave or cove cuts. V cuts are made with the small skew or the "spear point" chisel. Convex cuts are made with the skew chisel; concave cuts with the gouge.

These three cuts are described in the following paragraphs. Experiment and practice until these cuts can be taken properly—they are used in almost every wood turning job.

TAKING VEE CUTS

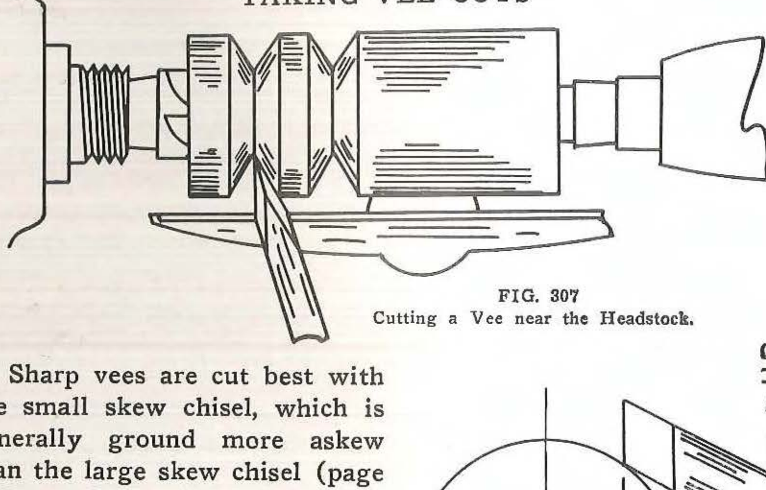


FIG. 307
Cutting a Vee near the Headstock.

Sharp vees are cut best with the small skew chisel, which is generally ground more askew than the large skew chisel (page 216). Figure 308 shows the proper method for starting the cut. After the work has been marked out, start the first groove with the heel of the tool. Then push down on the tool rest and raise the chisel handle, forcing the heel of the skew into the work. As the groove becomes deeper, be

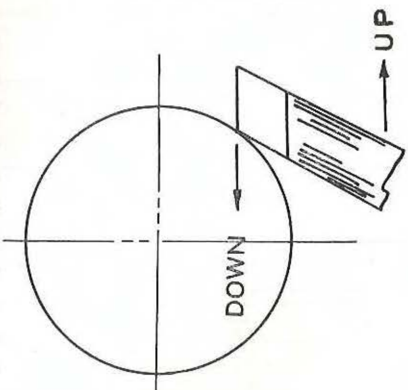


FIG. 308
Starting the groove for a vee cut.

careful to keep the unengaged portion of the cutting edge from "hogging-in." After cutting the groove, turn the chisel blade to the position shown in Figure 307 and cut the sides of the vee, first one side then the other, until reaching the proper depth. Never feed in the tool fast enough the burn the tool edge—be especially careful when turning hardwoods.

TAKING CONVEX CUTS

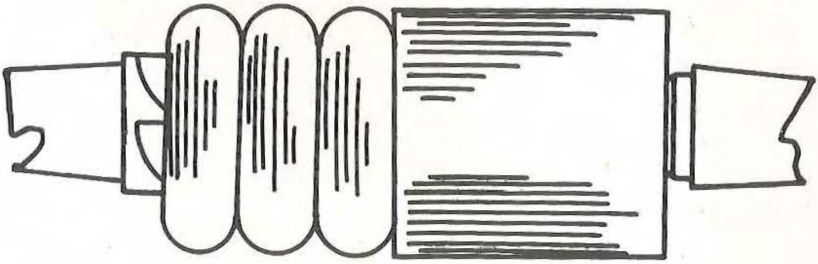


FIG. 309. Convex Cut or "Bead"

The best convex cuts, or "beads," (Fig. 309), are made with the small skew chisel. Four stages in the cutting process are illustrated in Figure 310. The first grooving is made exactly the same as for a V cut and at the points which will be the ends of the bead (Fig. 310). Then start the cut from the center of the bead as shown in Figure 311. Hold the blade fairly flat and high enough on the work so that only the heel will cut. Work the tool toward the side (Fig. 312), and at the same time draw it forward to avoid cutting the adjacent stock. As the groove becomes deeper, keep the back edge of the blade from spoiling the slope of the bead.

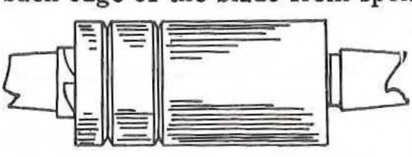


FIG. 310
First grooves for convex cut.

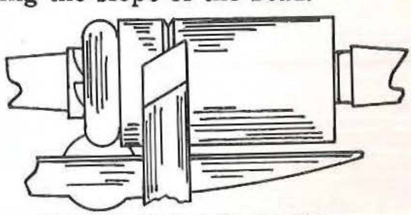


FIG. 311. Start of Shaping Cut.

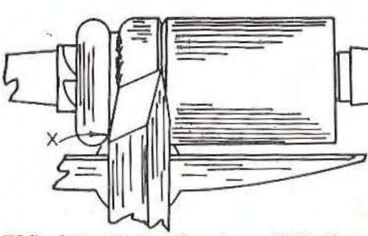


FIG. 312. Tool will rub at "X" if tool is not drawn forward as work progresses.

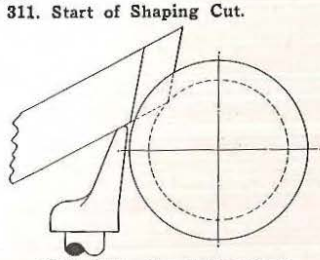


FIG. 313. Completing bead. Note chisel position.

CONCAVE CUTS

Refer to page 204.

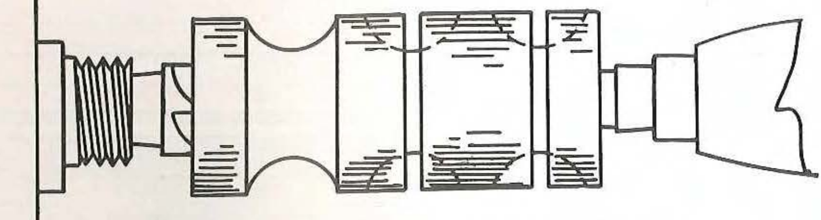


FIG. 314. Concave or "Cove" Cut. Note Sizing Groove.

The small gouge is generally preferred for concave cuts, although a round-nose scraping tool may also be used. Figure 315 shows the proper methods for sharpening the large and small gouge. The sharpened surface should be flat or with a slight outward curve, never hollow-ground, and the cutting edge should be honed very carefully.

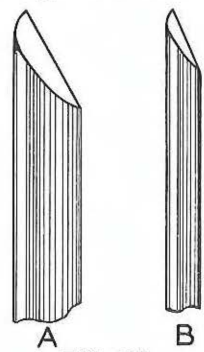


FIG. 315
The small gouge "B" is ground at more of an angle than the large gouge "A."

In making concave cuts with the small gouge, first make sizing grooves with a parting tool as shown in Figure 314. Then cut a groove to approximate shape by pushing the tool directly into the work with a scraping cut (Fig. 317). Then start the tool with the ground portion at a right angle to the side of the groove (Fig. 318) and roll the tool down to the groove center (fig. 319), cutting with the side lip of the tool. Then reverse the blade and repeat for the other side of the groove. Never attempt to take a cut from the bottom to the top of the groove.

Figure 316 shows how the round-nose scraping tool is used in making a concave curve. This type of chisel is pushed into the work with a scraping cut. The handle is then shifted so that the groove is gradually widened and deepened until finished.

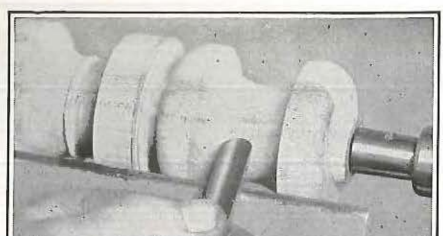
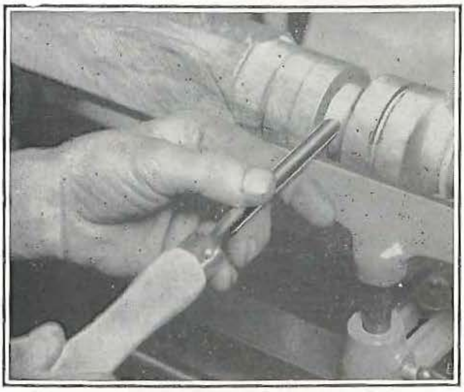


FIG. 316
Taking concave cut with the round nose scraping tool.

CUTTING A CONCAVE CURVE WITH THE SMALL GOUGE



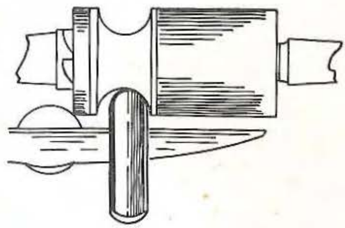
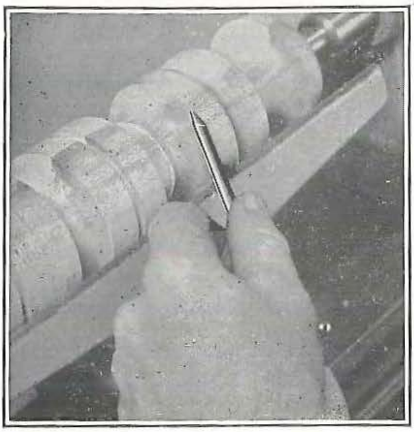


FIG. 317
Roughing out the curve. The tool is pushed directly into the work with a scraping cut.



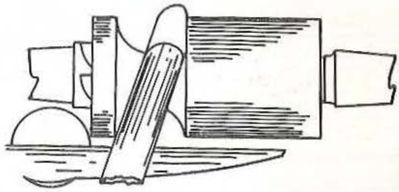
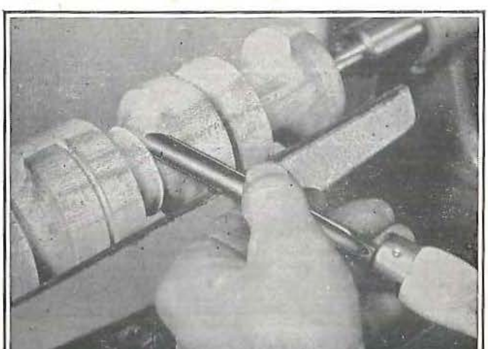


FIG. 318
Proper tool position at end of "rolling" cut. The ground portion of the tool is at a right angle to the side of the groove.



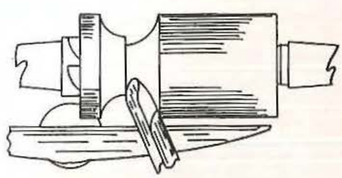


FIG. 319
Tool position at end of "rolling" cut. The cutting has been done with the side lip of the chisel.

CUTTING SQUARE CORNERS

Figure 320 shows how the point of the small skew chisel is held when cutting square corners. The surface of the ground portion is parallel to the shoulder—when the tool is swung farther toward the shoulder, the cutting action is likely to be stopped—when the tool is swung too far away from the shoulder, the chisel point "hogs in."

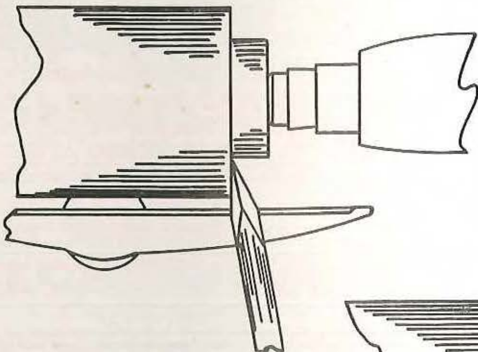


FIG. 320
Using the small skew chisel to cut a square corner. The ground portion at the left is held parallel to the shoulder.

Cutting up to a shoulder on a cylinder requires considerable practice. Hold the point of the small skew as shown in Figure 321 so that the heel of the tool takes a light cut.

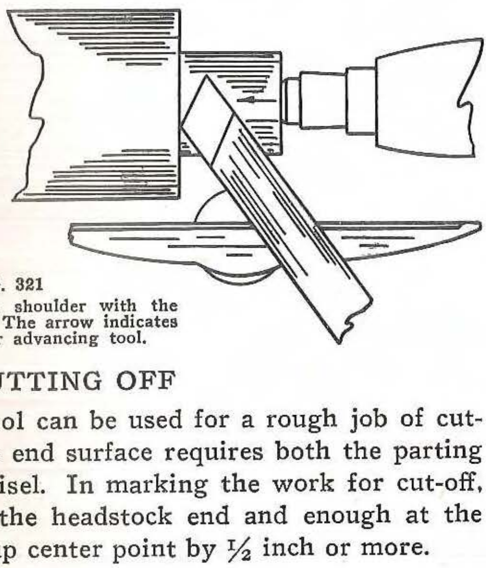


FIG. 321
Cutting up to a shoulder with the small skew chisel. The arrow indicates proper direction for advancing tool.

CUTTING OFF

Although the parting tool can be used for a rough job of cutting off, a smooth, accurate end surface requires both the parting tool and the small skew chisel. In marking the work for cut-off, allow at least 2 inches at the headstock end and enough at the tailstock end to clear the cup center point by ½ inch or more.

Illustrations and detailed instructions for cutting off are given on page 224.

THE STEADY REST

The steady rest is required to support long, slender work, exactly as in metal turning (page 175). The same steady rest can be used for both wood and metal operations. Whenever possible place the steady rest so that the jaws bear on an unfinished portion of the work.

THREE STEPS IN A CLEAN CUT-OFF JOB

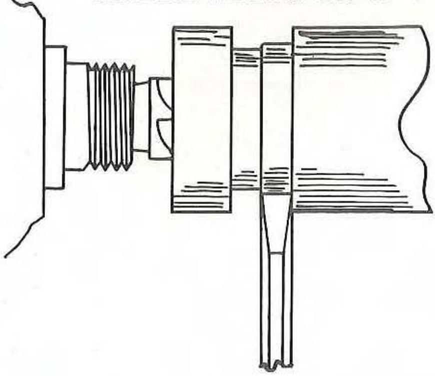


FIG. 322

Use the parting tool to cut a groove about 3/8 inch wide and deep enough to leave a diameter of about 3/4 inch at each end of the work. As shown at the left, this groove is made by taking two separate cuts with the parting tool.

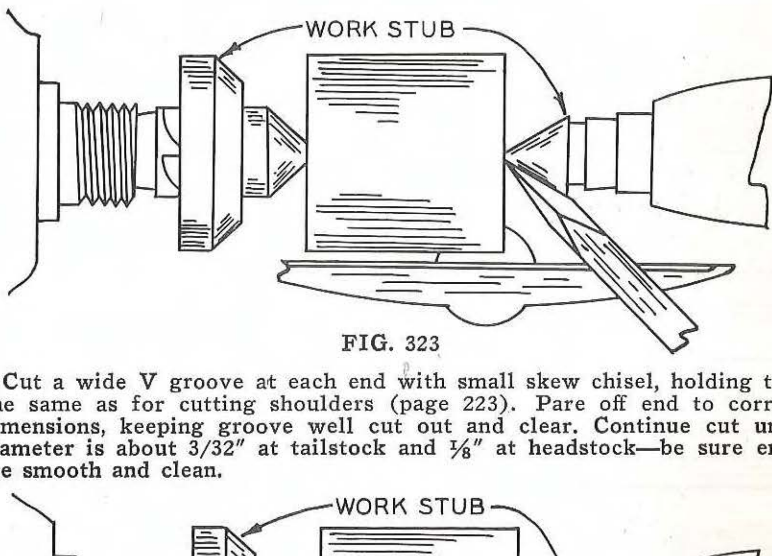


FIG. 323

Cut a wide V groove at each end with small skew chisel, holding tool the same as for cutting shoulders (page 223). Pare off end to correct dimensions, keeping groove well cut out and clear. Continue cut until diameter is about 3/32″ at tailstock and 1/8″ at headstock—be sure ends are smooth and clean.

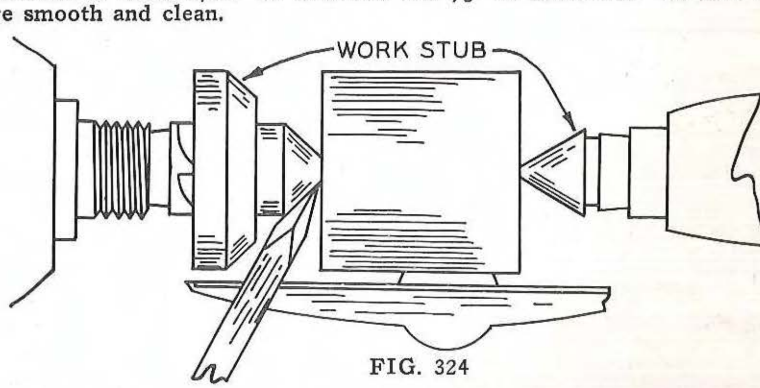


FIG. 324

Cut off the *headstock* end with the chisel point, catching the work with the left hand as it falls. Cut off the small tailstock end with the skew or a sharp knife. Some operators prefer to stop the lathe when the groove diameters are about 1/4 inch and finish the cut-off with a saw.

FACE PLATE TURNING

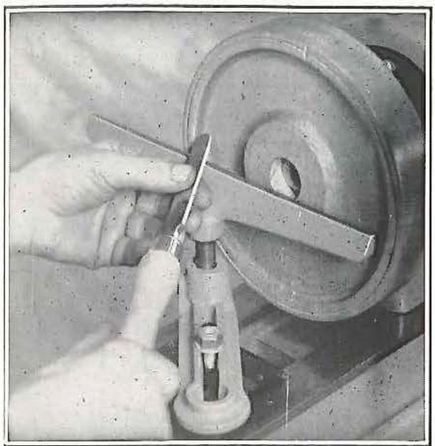


FIG. 325. A Facing Cut.

Work which cannot be turned between centers is fastened to a face plate or held in a chuck. The screw center (Fig. 284) is considered a special type of face plate for small work. Figure 325 shows a typical face plate operation.

Two methods of fastening the work to the face plate are shown in Figures 326 and 327. Three short, heavy screws are inserted in the holes of the face plate and can be turned directly into the stock if the screw holes will not be objectionable in the finished work. When screw holes must be avoided, the work is glued to a piece of scrap stock, and the screws tightened into the scrap as shown in Figure 327. A piece of newspaper placed between the two pieces simplifies removal of the work from the scrap after the operation is completed.

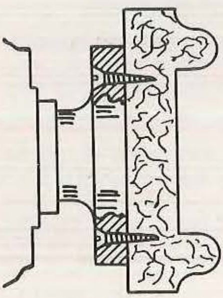


FIG. 326

When mounting small work on the screw center (Fig. 328), first drill a lead hole for the screw. Apply soap to the screw when mounting hard woods.

Work can be held in any of the standard metal working chucks or in easily turned

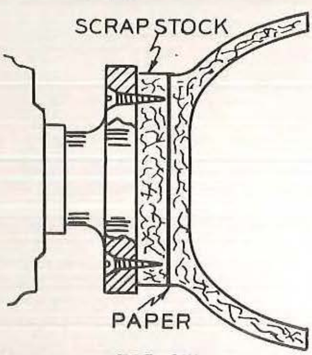


FIG. 327

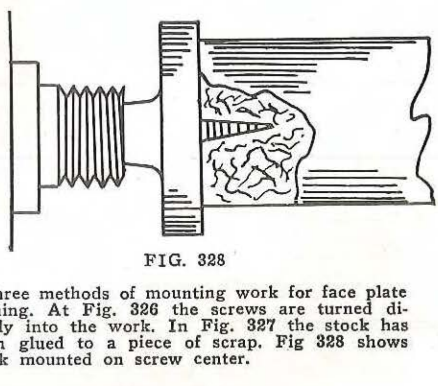


FIG. 328

Three methods of mounting work for face plate turning. At Fig. 326 the screws are turned directly into the work. In Fig. 327 the stock has been glued to a piece of scrap. Fig 328 shows work mounted on screw center.

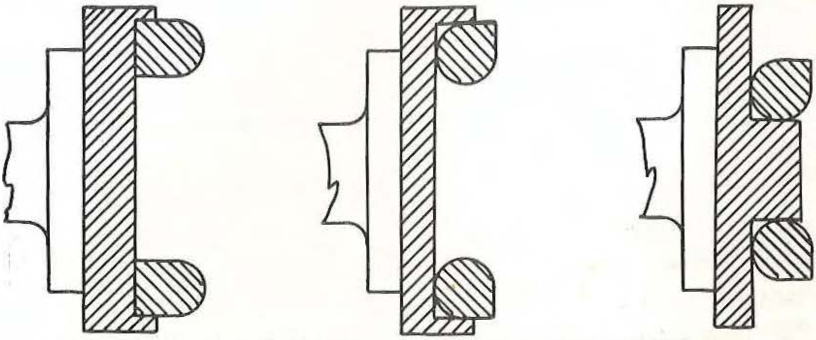


FIG. 329. Using chucks made from scrap stock. Take light cuts during these operations.

chucks made from pieces of wood scrap. Figure 329 shows how a doughnut ring can be turned while held in two simple chucks made especially for the job. The contact between the work and the chuck must be a snug push fit.

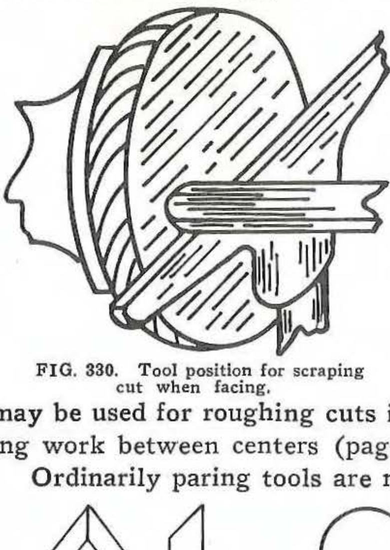


FIG. 330. Tool position for scraping cut when facing.

"SCRAPING" METHOD OF FACE PLATE TURNING

Most face plate work requires a scraping cut with the scraping chisel held horizontally and flat into the work (Figs. 330 and 336). Because face plate turning requires cutting the grain of the wood in all directions, the paring or cutting method used on work between centers is not satisfactory. The large gouge, however, may be used for roughing cuts in the same manner as when turning work between centers (page 224).

Ordinarily paring tools are not used for scraping cuts because

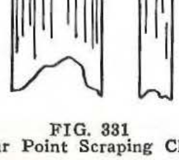


FIG. 331
Spear Point Scraping Chisel

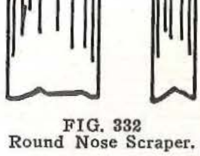


FIG. 332
Round Nose Scraper.

FIG. 333
Small Skew.

the larger clearance or bevel of the paring tool causes the cutting edge to dull quickly. Sometimes the small skew is used for taking convex scraping cuts. The three tool shapes shown in Figures 331 to 333 are recommended for scraping operations. Notice how the clearance angle has been decreased to provide additional strength at the cutting edge.

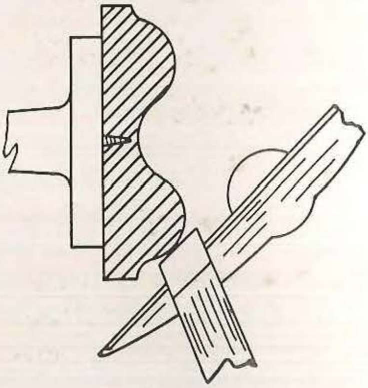


FIG. 334
Taking a convex scraping cut with the small skew chisel.

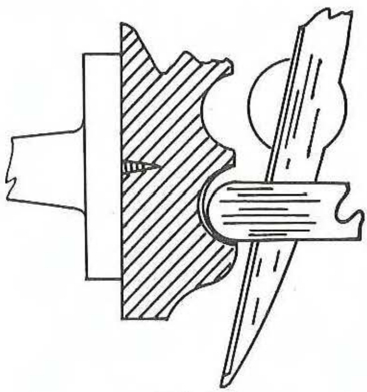


FIG. 335
Taking a concave scraping cut with the round nose scraper.

Before turning down irregular-shaped work on the face plate, saw off the corners or cut them down to approximate shape with a jig saw. Take light cuts to avoid splitting off corners or tearing the work from the face plate. Figures 334, 335, and 336 show the use of scraping tools in three typical facing operations.

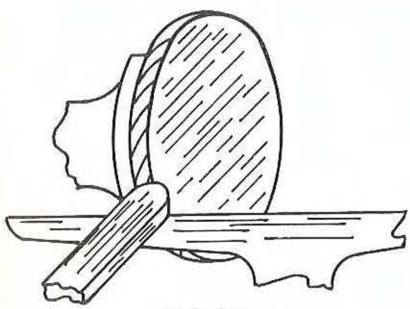


FIG. 336
Using a square end scraper on the outer edge of face plate work.

FINISHING THE WORK

A speed one step faster than that used for general turning is usually fast enough for sanding operations (see Fig. 337). The lathe speed should not be so fast as to burn the wood or paper. Never wrap the sandpaper around the work and grasp it with your fingers. Tear the sandpaper in strips and hold the ends only, or hold an end in one hand and the other end against the bottom of the work with the fingers of the other hand. Hold the paper *lightly* against the work.

Always remove the entire tool rest before sanding a piece of work.

If the work has been turned properly, only a moderate amount of sanding with No. 4/0 paper will be necessary. A short preliminary sanding with No. 1/0 paper is required only for face plate work or when tool marks must be removed.

There are several methods commonly used to give additional finish to the work. To French polish a piece of work, use a pad of cloth to apply a solution of ½ shellac and ½ wood alcohol to the piece while stationary. Then put a few drops of machine oil on the same pad and hold it against the revolving work, moving the pad from side to side. Apply additional shellac solution and oil to the pad as it dries. Continue polishing to desired finish.

Turnings can be waxed by holding a lump of beeswax, paraffin or Carnauba wax against the work as it revolves. The heat will leave a layer of wax on the work, which can be polished to a fine luster with a soft cloth. An attractive finish can be obtained by applying several coats of boiled linseed oil and polishing each coat thoroughly while the work revolves.

When the work is to be stained or filled and later finished with either a French polish or wax, it is best to leave the ends uncut so that the work can be remounted in the lathe for polishing.

WOODTURNING SPEEDS

Figure 337 gives recommended speeds for roughing, general finish cutting, and fine finish cuts and sanding for various work diameters. ***The largest diameter of the work determines the lathe spindle speed.***

These speeds are obtained with 1725 R.P.M. motor (Fig. 55, page 45). They are approximate only. The 2,072 R.P.M. speed is satisfactory for the general cutting and finishing of small work less than 2 inches in diameter.

FIGURE 337

WOOD TURNING SPEEDS IN R.P.M.

Diameter	Roughing Cuts	General Finish Cuts	Fine Finish Cuts and Sanding
Up to 2″	2,072	2,072	2,072
2″ to 3″	1,270	1,270	2,072
3″ to 4″	805	1,270	2,072
4″ to 5″	685	1,270	1,270
5″ to 6″	685	805	1,270
6″ to 7″	500	805	1,270
7″ to 8″	500	685	805
8″ to 9″	418	500	805
9″ to 10″	418	500	685

SAFETY FIRST!

Never wear a necktie when working on the lathe. Keep your sleeves rolled up or wear a work shirt with no sleeves. Industrial safety codes demand these precautions.

Select solid wood, free from slivers. Always lock tailstock before beginning operation—don't take deep cuts at the start.

PROPERTIES OF THE COMMON WOODS

WHITE PINE has been one of the most useful of all trees in the United States. The wood is one of the easiest and most satisfactory to work, due to the uniformity of its grain. White pine is unequalled for all purposes requiring a wood that checks and shrinks very little and holds its shape well. It is very light and soft, and of medium strength, elasticity and durability. It splits easily but nails well. In color it is light brown, almost a cream color. The grain is not noticeable and has no particular beauty.

MAPLE is hard, strong and ideal for general turning operations. It is close-grained and, when finished properly, makes a fine wood for furniture and decorative work.

CYPRESS is a soft, easily worked wood which does not warp badly, but is likely to contain many fine checks. It nails well and is very durable. It has a reddish brown color and no resin ducts. Its beauty makes it a desirable wood for interior finish and some types of furniture.

WHITE OAK is commonly used for furniture and interior finish. It is very strong, quite heavy, elastic, and hard. It is rather hard to work and nail, and checks and warps considerably unless carefully seasoned. However oak is extremely attractive when carefully worked. The color is a light brown. The rings are plainly defined by pores which, when stained, make a pleasing contrast with the "summer" wood, or pith rays.

WHITE ASH is a heavy, strong, elastic hardwood, often used for such articles as tool handles, oars, barrels, etc. It splits badly in nailing. It is commonly used for furniture and inside finish because of its strength and beauty when slash sawn.

YELLOW POPLAR is a general utility wood which has largely taken the place of white pine. It is light, brittle, soft, easy to work, nails very well, has medium strength, and does not warp badly when handled properly. The pith rays are quite noticeable, but not commonly used for decorative purposes. The color is greenish, or yellow brown. Yellow poplar is a good wood to keep in stock for all sorts of purposes, and an ideal wood for carving.

GUM has a beautiful chocolate color, varied with uneven deposits of coloring matter. It has an even texture, takes a fine finish, polishes well; it is comparatively easy to work, a good wood for carving, and nails fairly easily. However, gum twists and warps more than any other common wood. It is often used in making small articles for household use.

BLACK WALNUT is a rather expensive coarse-grained wood, very easily worked. It is chocolate in color and used very commonly for fine furniture.

MAHOGANY is a general name covering a number of species, all of which are imported. The different varieties are somewhat alike in color, reddish brown. The annual rings vary considerably in hardness, difficulty of nailing and shade of color. The grain is likely to be variable, causing an attractive reflection of light. Few woods take glue better than a mahogany.

Part 10

MACHINISTS TABLES

PART 10

MACHINISTS TABLES

Page — Table

CIRCUMFERENCES AND AREAS OF CIRCLES

Dia.	Circ.	Area	Dia.	Circ.	Area	Dia.	Circ.	Area
1	3.1416	.7854	35	109.96	962.11	68	213.63	3631.68
2	6.2832	3.1416	36	113.10	1017.88	69	216.77	3739.28
3	9.4248	7.0686	37	116.24	1075.21	70	219.91	3848.45
4	12.5664	12.5664	38	119.38	1134.11	71	223.05	3959.19
5	15.7080	19.635	39	122.52	1194.59	72	226.19	4071.50
6	18.850	28.274	40	125.66	1256.64	73	229.34	4185.39
7	21.991	38.485	41	128.81	1320.25	74	232.48	4300.84
8	25.133	50.266	42	131.95	1385.44	75	235.62	4417.86
9	28.274	63.617	43	135.09	1452.20	76	238.76	4536.46
10	31.416	78.540	44	138.23	1520.53	77	241.90	4656.63
11	34.558	95.033	45	141.37	1590.43	78	245.04	4778.36
12	37.699	113.1	46	144.51	1661.90	79	248.19	4901.67
13	40.841	132.73	47	147.65	1734.94	80	251.33	5026.55
14	43.982	153.94	48	150.80	1809.56	81	254.47	5153.00
15	47.124	176.71	49	153.94	1885.74	82	257.61	5281.02
16	50.265	201.06	50	157.08	1963.50	83	260.75	5410.61
17	53.407	226.98	51	160.22	2042.82	84	263.89	5541.77
18	56.549	254.47	52	163.36	2123.72	85	267.04	5674.50
19	59.690	283.53	53	166.50	2206.18	86	270.18	5808.80
20	62.832	314.16	54	169.65	2290.22	87	273.32	5944.68
21	65.973	346.36	55	172.79	2375.83	88	276.46	6082.12
22	69.115	380.13	56	175.93	2463.01	89	279.60	6221.14
23	72.257	415.48	57	179.07	2551.76	90	282.74	6361.73
24	75.398	452.39	58	182.21	2642.08	91	285.88	6503.88
25	78.540	490.87	59	185.35	2733.97	92	289.03	6647.61
26	81.681	530.93	60	188.50	2827.43	93	292.17	6792.91
27	84.823	572.56	61	191.64	2922.47	94	295.31	6939.78
28	87.965	615.75	62	194.78	3019.07	95	298.45	7088.22
29	91.106	660.52	63	197.92	3117.25	96	301.59	7238.23
30	94.248	706.86	64	201.06	3216.99	97	304.73	7339.81
31	97.389	754.77	65	204.20	3318.31	98	307.88	7542.96
32	100.53	804.25	66	207.34	3421.19	99	311.02	7697.69
33	103.67	855.30	67	210.49	3525.65	100	314.16	7853.98
34	106.81	907.92						

TABLE OF DECIMAL EQUIVALENTS
Millimeters and Fractions of an Inch

Fraction of an Inch "N"	Milli-meters	Decimal Equiv-alent Inches	Circle with Diameter "N" Circum.	Circle with Diameter "N" Area
	.1	.0039		
	.2	.0079		
	.3	.0118		
1/64		.0156	.04909	.00019
	.4	.0157		
	.5	.0197		
	.6	.0236		
	.7	.0276		
1/32		.0312	.09818	.00077
	.8	.0315		
	.9	.0354		
	1.0	.0394		
	1.1	.0433		
3/64		.0469	.14726	.00173
	1.2	.0472		
	1.25	.0492		
	1.3	.0512		
	1.4	.0551		
	1.5	.0590		
1/16		.0625	.19635	.00307
	1.6	.0630		
	1.7	.0669		
	1.75	.0689		
	1.8	.0709		
	1.9	.0748		
5/64		.0781	.24544	.00479
	2.0	.0787		
	2.1	.0827		
	2.2	.0866		
	2.25	.0886		
	2.3	.0905		
3/32		.0937	.29452	.00690
	2.4	.0945		
	2.5	.0984		
	2.6	.1024		
	2.7	.1063		
	2.75	.1083		
7/64		.1094	.34361	.00939
	2.8	.1102		
	2.9	.1142		
	3.0	.1181		
	3.1	.1220		
1/8		*.1250*	*.39270*	*.01227*
	3.2	.1260		
	3.25	.1280		
	3.3	.1299		
	3.4	.1339		
	3.5	.1378		
9/64		.1406	.44179	.01554
	3.6	.1417		
	3.7	.1457		
	3.75	.1476		
	3.8	.1496		
	3.9	.1535		
5/32		.1562	.49087	.01917
	4.0	.1575		
	4.1	.1614		
	4.2	.1654		
	4.25	.1673		
	4.3	.1693		
11/64		.1719	.53996	.02320
	4.4	.1732		
	4.5	.1772		
	4.6	.1811		
	4.7	.1850		
	4.75	.1870		
3/16		.1875	.58905	.02761
	4.8	.1890		
	4.9	.1929		
	5.0	.1968		
	5.1	.2008		
13/64		.2031	.63814	.03241
	5.2	.2047		
	5.25	.2067		
	5.3	.2087		
	5.4	.2126		
	5.5	.2165		
7/32		.2187	.68722	.03258
	5.6	.2205		
	5.7	.2244		
	5.75	.2264		
	5.8	.2283		
	5.9	.2323		
15/64		.2344	.73631	.04314
	6.0	.2362		
	6.1	.2402		
	6.2	.2441		
	6.25	.2461		
	6.3	.2480		
1/4		*.2500*	*.78540*	*.04909*
	6.4	.2520		
	6.5	.2559		
	6.6	.2598		
	6.7	.2638		
17/64		.2656	.83448	.05542
	6.75	.2657		
	6.8	.2677		
	6.9	.2717		
	7.0	.2756		
	7.1	.2795		
9/32		.2812	.88357	.06213
	7.2	.2835		
	7.25	.2854		
	7.3	.2874		
	7.4	.2913		
	7.5	.2953		
19/64		.2969	.93266	.06922
	7.6	.2992		

Fraction of an Inch "N"	Milli-meters	Decimal Equiv-alent Inches	Circle with Diameter "N" Circum.	Area
	7.7	.3031		
	7.75	.3051		
	7.8	.3071		
	7.9	.3110		
5/16		.3125	.98175	.07670
	8.0	.3150		
	8.1	.3189		
	8.2	.3228		
	8.25	.3248		
	8.3	.3268		
21/64		.3281	1.0308	.08456
	8.4	.3307		
	8.5	.3346		
	8.6	.3386		
	8.7	.3425		
11/32		.3437	1.0799	.09281
	8.75	.3445		
	8.8	.3465		
	8.9	.3504		
	9.0	.3543		
	9.1	.3583		
23/64		.3594	1.1290	.10143
	9.2	.3622		
	9.25	.3642		
	9.3	.3661		
	9.4	.3701		
	9.5	.3740		
3/8		*.3750*	*1.1781*	*.11045*
	9.6	.3780		
	9.7	.3819		
	9.75	.3839		
	9.8	.3858		
	9.9	.3898		
25/64		.3906	1.2272	.11984
	10.0	.3937		
	10.25	.4035		
13/32		.4062	1.2763	.12962
	10.5	.4134		
27/64		.4219	1.3254	.13979
	10.75	.4232		
	11.0	.4331		
7/16		.4375	1.3744	.15033
	11.25	.4429		
	11.5	.4528		
29/64		.4531	1.4235	.16126
	11.75	.4626		
15/32		.4687	1.4726	.17257
	12.0	.4724		
	12.25	.4823		
31/64		.4844	1.5217	.18427
	12.5	.4921		
1/2		*.5000*	*1.5708*	*.19635*
	12.75	.5020		
	13.0	.5118		
33/64		.5156	1.6199	.20881

Fraction of an Inch "N"	Milli-meters	Decimal Equiv-alent Inches	Circle with Diameter "N" Circum.	Area
17/32		.5312	1.6690	.22166
	13.5	.5315		
35/64		.5469	1.7181	.23489
	14.0	.5512		
9/16		.5625	1.7671	.24850
	14.5	.5709		
37/64		.5781	1.8162	.26250
	15.0	.5906		
19/32		.5937	1.8653	.27688
39/64		.6094	1.9144	.29165
	15.5	.6102		
5/8		*.6250*	*1.9635*	*.30680*
	16.0	.6299		
41/64		.6406	2.0126	.32233
	16.5	.6496		
21/32		.6562	2.0617	.33824
	17.0	.6693		
43/64		.6719	2.1108	.35454
11/16		.6875	2.1598	.37122
	17.5	.6890		
45/64		.7031	2.2089	.38829
	18.0	.7087		
23/32		.7187	2.2580	.40574
	18.5	.7283		
47/64		.7344	2.3071	.42357
	19.0	.7480		
3/4		*.7500*	*2.3562*	*.44179*
49/64		.7656	2.4053	.46038
	19.5	.7677		
25/32		.7812	2.4544	.47937
	20.0	.7874		
51/64		.7969	2.5035	.49874
	20.5	.8071		
13/16		.8125	2.5525	.51849
	21.0	.8268		
53/64		.8281	2.6016	.53862
27/32		.8437	2.6507	.55914
	21.5	.8465		
55/64		.8594	2.6998	.58004
	22.0	.8661		
7/8		*.8750*	*2.7489*	*.60132*
	22.5	.8858		
57/64		.8906	2.7980	.62299
	23.0	.9055		
29/32		.9062	2.8471	.64504
59/64		.9219	2.8962	.66747
	23.5	.9252		
15/16		.9375	2.9452	.69029
	24.0	.9449		
61/64		.9531	2.9943	.71349
	24.5	.9646		
31/32		.9687	3.0434	.73708
	25.0	.9842		
63/64		.9844	3.0925	.76104
1 Inch	*25.4*	*1.0000*	*3.1416*	*.7854*

POWERS, ROOTS AND RECIPROCALS

No.	Square	Cube	Sq. Root	Cube Root	Reciprocal	No.
1	1	1	1.00000	1.00000	1.0000000	1
2	4	8	1.41421	1.25992	0.5000000	2
3	9	27	1.73205	1.44225	0.3333333	3
4	16	64	2.00000	1.58740	0.2500000	4
5	25	125	2.23607	1.70998	0.2000000	5
6	36	216	2.44949	1.81712	0.1666667	6
7	49	343	2.64575	1.91293	0.1428571	7
8	64	512	2.82843	2.00000	0.1250000	8
9	81	729	3.00000	2.08008	0.1111111	9
10	100	1,000	3.16228	2.15443	0.1000000	10
11	121	1,331	3.31662	2.22398	0.0909091	11
12	144	1,728	3.46410	2.28943	0.0833333	12
13	169	2,197	3.60555	2.35133	0.0769231	13
14	196	2,744	3.74166	2.41014	0.0714286	14
15	225	3,375	3.87298	2.46621	0.0666667	15
16	256	4,096	4.00000	2.51984	0.0625000	16
17	289	4,913	4.12311	2.57128	0.0588235	17
18	324	5,832	4.24264	2.62074	0.0555556	18
19	361	6,859	4.35890	2.66840	0.0526316	19
20	400	8,000	4.47214	2.71442	0.0500000	20
21	441	9,261	4.58258	2.75892	0.0476190	21
22	484	10,648	4.69042	2.80204	0.0454545	22
23	529	12,167	4.79583	2.84387	0.0434783	23
24	576	13,824	4.89898	2.88450	0.0416667	24
25	625	15,625	5.00000	2.92402	0.0400000	25
26	676	17,576	5.09902	2.96250	0.0384615	26
27	729	19,683	5.19615	3.00000	0.0370370	27
28	784	21,952	5.29150	3.03659	0.0357143	28
29	841	24,389	5.38516	3.07232	0.0344828	29
30	900	27,000	5.47723	3.10723	0.0333333	30
31	961	29,791	5.56776	3.14138	0.0322581	31
32	1,024	32,768	5.65685	3.17480	0.0312500	32
33	1,089	35,937	5.74456	3.20753	0.0303030	33
34	1,156	39,304	5.83095	3.23961	0.0294118	34
35	1,225	42,875	5.91608	3.27107	0.0285714	35
36	1,296	46,656	6.00000	3.30193	0.0277778	36
37	1,369	50,653	6.08276	3.33222	0.0270270	37
38	1,444	54,872	6.16441	3.36198	0.0263158	38
39	1,521	59,319	6.24500	3.39121	0.0256410	39
40	1,600	64,000	6.32456	3.41995	0.0250000	40
41	1,681	68,921	6.40312	3.44822	0.0243902	41
42	1,764	74,088	6.48074	3.47603	0.0238095	42
43	1,849	79,507	6.55744	3.50340	0.0232558	43
44	1,936	85,184	6.63325	3.53035	0.0227273	44
45	2,025	91,125	6.70820	3.55689	0.0222222	45
46	2,116	97,336	6.78233	3.58305	0.0217391	46
47	2,209	103,823	6.85565	3.60883	0.0212766	47
48	2,304	110,592	6.92820	3.63424	0.0208333	48
49	2,401	117,649	7.00000	3.65931	0.0204082	49
50	2,500	125,000	7.07107	3.68403	0.0200000	50

POWERS, ROOTS AND RECIPROCALS—(Continued)

No.	Square	Cube	Sq. Root	Cube Root	Reciprocal	No.
51	2,601	132,651	7.14143	3.70843	0.0196078	51
52	2,704	140,608	7.21110	3.73251	0.0192308	52
53	2,809	148,877	7.28011	3.75629	0.0188679	53
54	2,916	157,464	7.34847	3,77976	0.0185185	54
55	3,025	166,375	7.41620	3.80295	0.0181818	55
56	3,136	175,616	7.48331	3.82586	0.0178571	56
57	3,249	185,193	7.54983	3.84850	0.0175439	57
58	3,364	195,112	7.61577	3.87088	0.0172414	58
59	3,481	205,379	7.68115	3.89300	0.0169492	59
60	3,600	216,000	7.74597	3.91487	0.0166667	60
61	3,721	226,981	7.81025	3.93650	0.0163934	61
62	3,844	238,328	7.87401	3.95789	0.0161290	62
63	3,969	250,047	7.93725	3.97906	0.0158730	63
64	4,096	262,144	8.00000	4.00000	0.0156250	64
65	4,225	274,625	8.06226	4.02073	0.0153846	65
66	4,356	287,496	8.12404	4.04124	0.0151515	66
67	4,489	300,763	8.18535	4.06155	0.0149254	67
68	4,624	314,432	8.24621	4.08166	0.0147059	68
69	4,761	328,509	8.30662	4.10157	0.0144928	69
70	4,900	343,000	8.36660	4.12129	0.0142857	70
71	5,041	357,911	8.42615	4.14082	0.0140845	71
72	5,184	373,248	8.48528	4.16017	0.0138889	72
73	5,329	389,017	8.54400	4.17934	0.0136986	73
74	5,476	405,224	8.60233	4.19834	0.0135135	74
75	5,625	421,875	8.66025	4.21716	0.0133333	75
76	5,776	438,976	8.71780	4.23582	0.0131579	76
77	5,929	456,533	8.77496	4.25432	0.0129870	77
78	6,084	474,552	8.83176	4.27266	0.0128205	78
79	6,241	493,039	8.88819	4.29084	0.0126582	79
80	6,400	512,000	8,94427	4.30887	0.0125000	80
81	6,561	531,441	9.00000	4.32675	0.0123457	81
82	6,724	551,368	9.05539	4.34448	0.0121951	82
83	6,889	571,787	9.11043	4.36207	0.0120482	83
84	7,056	592,704	9.16515	4.37952	0.0119048	84
85	7,225	614,125	9.21954	4.39683	0.0117647	85
86	7,396	636,056	9.27362	4.41400	0.0116279	86
87	7,569	658,503	9.32738	4.43105	0.0114943	87
88	7,744	681,472	9.38083	4.44797	0.0113636	88
89	7,921	704,969	9.43398	4.46475	0.0112360	89
90	8,100	729,000	9.48683	4.48140	0.0111111	90
91	8,281	753,571	9.53939	4.49794	0.0109890	91
92	8,464	778,688	9.59166	4.51436	0.0108696	92
93	8.649	804,357	9.64365	4.53065	0.0107527	93
94	8,836	830,584	9.69536	4.54684	0.0106383	94
95	9,025	857,375	9.74679	4.56290	0.0105263	95
96	9,216	884,736	9.79796	4.57886	0.0104167	96
97	9,409	912,673	9.84886	4.59470	0.0103093	97
98	9,604	941,192	9.89949	4.61044	0.0102041	98
99	9,801	970,299	9.94987	4.62607	0.0101010	99
100	10,000	1,000,000	10.00000	4.64159	0.0100000	100

SQUARE AND CUBE ROOTS OF DECIMAL NUMBERS

Decimal	Square Root	Cube Root	Decimal	Square Root	Cube Root	Decimal	Square Root	Cube Root
0.01	0.1000	0.2154	0.34	0.5831	0.6980	0.67	0.8185	0.8750
0.02	0.1414	0.2714	0.35	0.5916	0.7047	0.68	0.8246	0.8794
0.03	0.1732	0.3107	0.36	0.6000	0.7114	0.69	0.8307	0.8837
0.04	0.2000	0.3420	0.37	0.6083	0.7179	0.70	0.8367	0.8879
0.05	0.2236	0.3684	0.38	0.6164	0.7243	0.71	0.8426	0.8921
0.06	0.2449	0.3915	0.39	0.6245	0.7306	0.72	0.8485	0.8963
0.07	0.2646	0.4121	0.40	0.6325	0.7368	0.73	0.8544	0.9004
0.08	0.2828	0.4309	0.41	0.6403	0.7429	0.74	0.8602	0.9045
0.09	0.3000	0.4481	0.42	0.6481	0.7489	0.75	0.8660	0.9086
0.10	0.3162	0.4642	0.43	0.6557	0.7548	0.76	0.8718	0.9126
0.11	0.3317	0.4791	0.44	0.6633	0.7606	0.77	0.8775	0.9166
0.12	0.3464	0.4932	0.45	0.6708	0.7663	0.78	0.8832	0.9205
0.13	0.3606	0.5066	0.46	0.6782	0.7719	0.79	0.8888	0.9244
0.14	0.3742	0.5192	0.47	0.6856	0.7775	0.80	0.8944	0.9283
0.15	0.3873	0.5313	0.48	0.6928	0.7830	0.81	0.9000	0.9322
0.16	0.4000	0.5429	0.49	0.7000	0.7884	0.82	0.9055	0.9360
0.17	0.4123	0.5540	0.50	0.7071	0.7937	0.83	0.9110	0.9398
0.18	0.4243	0.5646	0.51	0.7141	0.7990	0.84	0.9165	0.9435
0.19	0.4359	0.5749	0.52	0.7211	0.8041	0.85	0.9220	0.9473
0.20	0.4472	0.5848	0.53	0.7280	0.8093	0.86	0.9274	0.9510
0.21	0.4583	0.5944	0.54	0.7348	0.8143	0.87	0.9327	0.9546
0.22	0.4690	0.6037	0.55	0.7416	0.8193	0.88	0.9381	0.9583
0.23	0.4796	0.6127	0.56	0.7483	0.8243	0.89	0.9434	0.9619
0.24	0.4899	0.6214	0.57	0.7550	0.8291	0.90	0.9487	0.9655
0.25	0.5000	0.6300	0.58	0.7616	0.8340	0.91	0.9539	0.9691
0.26	0.5099	0.6383	0.59	0.7681	0.8387	0.92	0.9592	0.9726
0.27	0.5196	0.6463	0.60	0.7746	0.8434	0.93	0.9644	0.9761
0.28	0.5292	0.6542	0.61	0.7810	0.8481	0.94	0.9695	0.9796
0.29	0.5385	0.6619	0.62	0.7874	0.8527	0.95	0.9747	0.9830
0.30	0.5477	0.6694	0.63	0.7937	0.8573	0.96	0.9798	0.9865
0.31	0.5568	0.6768	0.64	0.8000	0.8618	0.97	0.9849	0.9899
0.32	0.5657	0.6840	0.65	0.8062	0.8662	0.98	0.9899	0.9933
0.33	0.5745	0.6910	0.66	0.8124	0.8707	0.99	0.9950	0.9967

WEIGHTS AND MEASURES

LINEAR MEASURE

12 inches (in) .... = 1 foot .... ft.
3 feet .... = 1 yard .... yd.
5.5 yards .... = 1 rod .... rd.
40 rods .... = 1 furlong .... fur.
8 furlongs .... = 1 mile .... mi.

in.	ft.	yd.	rd.	fur.	mi.
36 =	3 =	1			
198 =	16.5 =	5.5 =	1		
7,920 =	660 =	220 =	40 =	1	
63.360 =	5,280 =	1,760 =	320 =	8 =	1

SQUARE MEASURE

144 square inches (sq. in.) .... = 1 square foot .... sq. ft.
9 Square feet .... = 1 square yard .... sq. yd.
30¼ square yards .... = 1 square rod ....sq. rd.
160 square rods .... = 1 acre .... A.
640 acres .... = 1 square mile .... sq. mi.

sq. mi.	A.	sq. rd.	sq. yd.	sq. ft.	sq. in.
1 =	640 =	102,400 =	3,097,600 =	27,878,400 =	4,014,489,600

CUBIC MEASURE

1,728 cubic inches (cu. in.) ..... = 1 cubic foot .... cu. ft.
27 cubic feet .... = cubic yard .... cu. yd.
128 cubic feet .... = 1 cord .... cd.
24¾ cubic feet .... = 1 perch .... P.

1 cu. yd. = 27 cu. ft. = 46,656 cu. in.

MEASURE OF ANGLES OR ARCS

60 seconds (″)	=	1 minute	′
60 minutes	=	1 degree	°
90 degrees	=	1 rt. angle or quadrant	□
360 degrees	=	1 circle	cir.

1 cr. = 360° = 21,600′ = 1,296,000″

AVOIRDUPOIS WEIGHT

437.5 grains (gr.)	=	1 ounce	oz.
16 ounces	=	1 pound	lb.
100 pounds	=	1 hundredweight	cwt.
20 cwt., or 2,000	=	1 ton	T.

1 T. = 20 cwt. = 2,000 lb. = 32,000 oz. = 14,000,000 gr.

The avoirdupois pound contains 7,000 grains.

DRY MEASURE

2 pints (pt.)	=	1 quart	qt.
8 quarts	=	1 peck	pk.
4 pecks	=	1 bushel	bu.

1 bu. = 4 pk. = 32 qt. = 64 pt.

The U. S. struck bushel contains 2,150.42 cu. in. = 1.2444 cu. ft. By law, its dimensions are those of a cylinder 18½ in. in diameter and 8 in. deep. The heaped bushel is equal to 1¼ struck bushels, the cone being 6 in. high. The dry gallon contains 268.8 cu. in., being ⅛ of a struck bushel.

For approximations, the bushel may be taken at 1¼ cu. ft.; or a cubic foot may be considered 4/5 of a bushel.

The British bushel contains 2,218.19 cu. in. = 1.2837 cu ft. = 1.032 U. S. bushels.

LIQUID MEASURE

4 gills (gi.)	=	1 pint	pt.
2 pints	=	1 quart	qt.
4 quarts	=	1 gallon	gal.
31½ gallons	=	1 barrel	bbl.
2 barrels, or 63 gallons	=	1 hogshead	hhd.

1 hhd. = 2 bbl. = 63 gal. = 252 qt. = 504 pt. = 2,016 gi.

The U. S. gallon contains 231 cu. in. = .134 cu. ft., nearly; or 1 cu. ft. contains 7.841 gal. The following cylinders contain the given measures very closely:

	Diam.	Height		Diam.	Height
Gill	1¾ in.	3 in.	Gallon	7 in.	6 in.
Pint	3½ in.	3 in.	8 Gallons	14 in.	12 in.
Quart	3½ in.	6 in.	10 Gallons	14 in.	15 in.

When water is at its maximum density, 1 cu. ft. weighs 62,425 lb. and 1 gallon weighs 8.345 lb.

For approximations, 1 cu. ft. of water is considered equal to 7½ gal., and 1 gal. as weighing 8⅓ lb.

The British imperial gallon, both liquid and dry, contains 277.274 cu. in. = .16046 cu. ft., and is equivalent to the volume of 10 lb. of pure water at 62° F. To reduce British to U. S. liquid gallons, multiply by 1.2 Conversely, to convert U. S. into British liquid gallons, divide by 1.2; or, increase the number of gallons 1/5.

THE METRIC SYSTEM

MEASURES OF LENGTH

10 millimeters (mm.)	= 1 centimeter	cm.
10 centimeters	= 1 decimeter	dm.
10 decimeters	= 1 meter	m.
10 meters	= 1 decameter	Dm.
10 decameters	= 1 hectometer	Hm.
10 hectometers	= 1 kilometer	Km.
1 millimeter	= .03937 inches	
1 inch	= 25.40 millimeters	

MEASURES OF AREA (NOT LAND)

100 square millimeters (mm^2.)	= 1 square centimeter	cm^2.
100 square centimeters	= 1 square decimeter	dm^2.
100 square decimeters	= 1 square meter	m^2.
1 square meter	= 10.7639 square feet	
1 square millimeter	= .00155 square inches	
1 square inch	= 645.163 square millimeters	
1 square foot	= .0929 square meters	

MEASURES OF VOLUME

1,000 cubic millimeters (mm^3.)	= 1 cubic centimeter	cm^3.
1,000 cubic centimeters	= 1 cubic decimeter	dm^3.
1,000 cubic decimeters	= 1 cubic meter	m^3.
1 cubic meter	= 35.3133 cubic feet	
1 cubic inch	= 16387.17 cubic millimeters	

MEASURES OF CAPACITY

10 milliliters (ml.)	= 1 centiliter	cl.
10 centiliters	= 1 deciliter	dl.
10 deciliters	= 1 liter	l.
10 liters	= 1 decaliter	Dl.
10 decaliters	= 1 hectoliter	Hl.
10 hectoliters	= 1 kiloliter	Kl.

NOTE.—The liter is equal to the volume occupied by 1 cubic decimeter.

1 liter	= 61.02398 cubic inches
1 cubic foot	= 28.3169 liters

MEASURES OF WEIGHT

10 milligrams (mg.)	= 1 centigram	cg.
10 centigrams	= 1 decigram	dg.
10 decigrams	= 1 gram	g.
10 grams	= 1 decagram	Dg.
10 decagrams	= 1 hectogram	Hg.
10 hectograms	= 1 kilogram	Kg.
1,000 kilograms	= 1 ton	T.
1 kilogram	= 2.2046 pounds avoir.	
1 pound avoir	= .4536 kilograms	

NOTE.—The gram is the weight of 1 cubic centimeter of pure distilled water at a temperature of 39.2° F.; the kilogram is the weight of 1 liter of water; the ton is the weight of 1 cubic meter of water.

PROPERTIES OF CIRCLES

Circumference of circle = diameter $\times$ 3.1416.
Diameter of circle = circumference $\times$ 0.3183.
Side of square inscribed in circle = diameter of circle $\times$ 0.7071.
Diameter of circle circumscribed about square = side of square $\times$ 1.4142.
Length of arc = number of degrees $\times$ diameter $\times$ 0.008727.
Circumference of circle whose diameter is 1 = 3.14159265.
Length of chord = diameter of circle $\times$ sine of $\frac{1}{2}$ included angle.

AREA OR SURFACE

Triangle = base $\times$ half perpendicular height.
Parallelogram = base $\times$ perpendicular height.
Trapezoid = half the sum of the parallel sides $\times$ perpendicular height.
Trapezium, found by dividing into two triangles.
Circle = diameter squared $\times$ 0.7854; or radius squared $\times$ 3.1416.
Sector of circle = length of arc $\times$ half radius. (See above for length of arc.)
Segment of circle less than semi-circle = area of sector minus area of triangle.
Segment of circle greater than semi-circle = area of sector + area of triangle.
Side of square of equal area as circle = diameter of circle $\times$ 0.8862.
Diameter of circle of equal area as square = side of square $\times$ 1.1284.
Parabola = base $\times$ $\frac{2}{3}$ perpendicular height.
Ellipse = long diameter $\times$ short diameter $\times$ 0.7854.
Regular polygon = sum of sides $\times$ half perpendicular distance from center to sides.
Cylinder = (circumference $\times$ height) + area of both ends.
Sphere = diameter squared $\times$ 3.1416.
Segment of sphere = (height of segment $\times$ circumference of sphere of which it is a part) + area of base.
Right pyramid or cone = periphery or circumference of base $\times$ half slant height + area of base.
Frustum of a regular right pyramid or cone = (sum of peripheries or circumferences of the two ends $\times$ half slant height) + area of both ends.

SOLID CONTENTS OR VOLUME

Prism, right or oblique, = area of base $\times$ perpendicular height.

Cylinder, right or oblique, = area of section at right angles to sides $\times$ length of side.

Sphere = diameter cubed $\times$ 0.5236.

Segment of sphere = (height of segment squared + three times the square of radius of base of segment) $\times$ height of segment $\times$ 0.5236.

Side of cube having equal volume as sphere = diameter of sphere $\times$ 0.806.

Length of cylinder having equal volume and same diameter as sphere = diameter of sphere $\times$ 0.6667.

Pyramid or cone, right or oblique, regular or irregular, = area of base $\times$ $\frac{1}{3}$ perpendicular height.

Frustum of cone = multiply area of two ends together and extract the square root; add to this square root the sum of the areas of both ends and then multiply the total sum by $\frac{1}{3}$ the perpendicular distance between the ends.

The Prismoidal Formula can be used for obtaining the volume of any solid with irregular or regular shaped parallel ends and with straight sides (prismatoids). First add together the areas of the two parallel ends. To this add four times the area of a section parallel to the ends and midway between them. Now multiply the total sum by $\frac{1}{6}$ the perpendicular distance between the parallel ends and you have the volume.

DIAMETERS OF NUMBERED DRILLS

Drill No.	Diameter Inches	Drill No.	Diameter Inches	Drill No.	Diameter Inches
80	.0135	53	.0595	26	.1470
79	.0145	52	.0635	25	.1495
78	.0160	51	.0670	24	.1520
77	.0180	50	.0700	23	.1540
76	.0200	49	.0730	22	.1570
75	.0210	48	.0760	21	.1590
74	.0225	47	.0785	20	.1610
73	.0240	46	.0810	19	.1660
72	.0250	45	.0820	18	.1695
71	.0260	44	.0860	17	.1730
70	.0280	43	.0890	16	.1770
69	.0292	42	.0935	15	.1800
68	.0310	41	.0960	14	.1820
67	.0320	40	.0980	13	.1850
66	.0330	39	.0995	12	.1890
65	.0350	38	.1015	11	.1910
64	.0360	37	.1040	10	.1935
63	.0370	36	.1065	9	.1960
62	.0380	35	.1100	8	.1990
61	.0390	34	.1110	7	.2010
60	.0400	33	.1130	6	.2040
59	.0410	32	.1160	5	.2055
58	.0420	31	.1200	4	.2090
57	.0430	30	.1285	3	.2130
56	.0465	29	.1360	2	.2210
55	.0520	28	.1405	1	.2280
54	.0550	27	.1440		

DIAMETERS OF LETTERED DRILLS

Drill No.	Diameter Inches
A	.2340
B	.2380
C	.2420
D	.2460
E	.2500
F	.2570
G	.2610
H	.2660
I	.2720
J	.2770
K	.2810
L	.2900
M	.2950
N	.3020
O	.3160
P	.3230
Q	.3320
R	.3390
S	.3480
T	.3580
U	.3680
V	.3770
W	.3860
X	.3970
Y	.4040
Z	.4130

ALLOWANCES FOR MACHINE FITS

In all work requiring running, push, drive or forced fits, the diameter of the hole should be exact as specified within the limits given, while the diameter of the shaft shall be such that it will fit the hole according to the given allowances for various fits.

Nominal diameter Inches	Hole Tolerance	Allowances for Different Fits			
		Running Fit	Push Fit	Drive Fit	Forced Fit
Up to ½″	+.0005 −.0005	−.001 −.002	−.0003 −.0008	+.0005 +.0003	+.001 +.0005
½″ to 1″	+.001 −.0005	−.0015 −.003	−.0003 −.0008	+.001 +.0008	+.002 +.0015
1″ to 2″	+.001 −.0005	−.002 −.004	−.0003 −.0008	+.0015 +.001	+.004 +.003
2″ to 3″	+.0015 −.001	−.0025 −.0045	−.0005 −.001	+.0025 +.0015	+.006 +.0045
3″ to 4″	+.0015 −.001	−.003 −.005	−.0005 −.001	+.003 +.002	+.008 +.006

WIRE AND SHEET METAL GAUGES IN APPROXIMATE DECIMALS OF AN INCH

No. of Wire Gauge	American or Brown & Sharpe	Birmingham or Stub's Iron Wire	Washburn & Moen, Am. Steel & Wire Co., and Roebling	Stub's Steel Wire	Trenton Iron Co.	British Imperial Wire	U. S. Standard for Plate
0000000	....	...	.4900	...	....	.5000	.5000
000000	.5800	...	.4615	...	....	.4640	.4688
00000	.5165	.500	.4305	...	.4500	.4320	.4375
0000	.4600	.454	.3938	...	.4000	.4000	.4063
000	.4096	.425	.3625	...	.3600	.3720	.3750
00	.3648	.380	.3310	...	.3300	.3480	.3438
0	.3249	.340	.3065	...	.3050	.3240	.3125
1	.2893	.300	.2830	.227	.2850	.3000	.2813
2	.2576	.284	.2625	.219	.2650	.2760	.2656
3	.2294	.259	.2437	.212	.2450	.2520	.2500
4	.2043	.238	.2253	.207	.2250	.2320	.2344
5	.1819	.220	.2070	.204	.2050	.2120	.2188
6	.1620	.203	.1920	.201	.1900	.1920	.2031
7	.1443	.180	.1770	.199	.1750	.1760	.1875
8	.1285	.165	.1620	.197	.1600	.1600	.1719
9	.1144	.148	.1483	.194	.1450	.1440	.1563
10	.1019	.134	.1350	.191	.1300	.1280	.1406
11	.0907	.120	.1205	.188	.1175	.1160	.1250
12	.0808	.109	.1055	.185	.1050	.1040	.1094
13	.0720	.095	.0915	.182	.0925	.0920	.0938
14	.0641	.083	.0800	.180	.0800	.0800	.0781
15	.0571	.072	.0720	.178	.0700	.0720	.0703
16	.0508	.065	.0625	.175	.0610	.0640	.0625
17	.0453	.058	.0540	.172	.0525	.0560	.0563
18	.0403	.049	.0475	.168	.0450	.0480	.0500
19	.0359	.042	.0410	.164	.0400	.0400	.0438
20	.0320	.035	.0348	.161	.0350	.0360	.0375
21	.0285	.032	.0317	.157	.0310	.0320	.0344
22	.0253	.028	.0286	.155	.0280	.0280	.0313
23	.0226	.025	.0258	.153	.0250	.0240	.0281
24	.0201	.022	.0230	.151	.0225	.0220	.0250
25	.0179	.020	.0204	.148	.0200	.0200	.0219
26	.0159	.018	.0181	.146	.0180	.0180	.0188
27	.0142	.016	.0173	.143	.0170	.0164	.0172
28	.0126	.014	.0162	.139	.0160	.0148	.0156
29	.0113	.013	.0150	.134	.0150	.0136	.0141
30	.0100	.012	.0140	.127	.0140	.0124	.0125
31	.0089	.010	.0132	.120	.0130	.0116	.0109
32	.0080	.009	.0128	.115	.0120	.0108	.0102
33	.0071	.008	.0118	.112	.0110	.0100	.0094
34	.0063	.007	.0104	.110	.0100	.0092	.0086
35	.0056	.005	.0095	.108	.0095	.0084	.0078
36	.0050	.004	.0090	.106	.0090	.0076	.0070
37	.0045	...	.0085	.103	.0085	.0068	.0066
38	.0040	...	.0080	.101	.0080	.0060	.0063
39	.0035	...	.0075	.099	.0075	.0052	....
40	.0031	...	.0070	.097	.0070	.0048	....

DESCRIPTION AND DATA ON STEELS

Commonly Warehoused in the United States

— S.A.E. 1112 —

PHYSICAL PROPERTIES

As Cold Drawn:—Tensile Strength 80,000 lbs/in^2. Brinnel Hardness 185. Elongation in two inches 16%.
As Hot Rolled:—Tensile Strength 67,000 lbs/in^2. Brinnell Hardness 140. Elongation in two inches 27%.

A free cutting, Bessemer Screw Stock, easy to machine. Used in automatic screw machine work for parts not requiring great strength or ductility. Not used for parts requiring bending, expanding, rivetting or deforming operations. It is not suitable for simple heat treatment, but can be carburized and case hardened, although a rather weak core results. S.A.E. 1112 is somewhat brittle, which is emphasized by case hardening. For simple carburizing to produce maximum surface hardness, carburize at 1600° F. and quench in water or oil.

— S.A.E. X1112 —

PHYSICAL PROPERTIES

As Cold Drawn:—Tensile Strength 100,000 lbs/in^2. Brinnel Hardness 193. Elongation in two inches 15%.

An exceptionally free cutting, Bessemer Screw Stock, very easy to machine. Used for automatic screw machine work and screw machine work for close-tolerance parts requiring a smooth finish. Will withstand deforming better than 1112 and is a little stronger. Recommended in place of 1112 for most types of work. It does not respond well to simple heat treatment, but can be carburized or cyanided and will have a little better core strength than 1112. Carburizing treatment is the same as given for 1112.

— AISI C1117 AND AISI C1118 —

PHYSICAL PROPERTIES

As Cold Drawn:—Tensile Strength 82,000 lbs/in^2. Brinnel Hardness 162. Elongation in two inches 18%
As Hot Rolled:—Tensile Strength 71,000 lbs/in^2. Brinnel Hardness 135. Elongation in two inches 28%.

Open hearth steel, high manganese screw stocks, easy to machine, with good ductility and mechanical strength. There is little difference between X1314 and X1315 although is is claimed that X1314 is better for thin wall case hardened parts such as piston pins, etc. Used for shafting, steering gear cams, etc. Both can be simple heat treated, although X1315 is recommended for that purpose. Both can be case hardened successfully. These steels can be substituted for 1015 and 1020 except for parts requiring heavy deformation, and will be found to give faster machining, brighter and smoother finish and longer tool life. They are used both cold drawn and hot rolled. For simple heat treatment, heat to 1650° F., quench in water, and draw to required hardness between 400° and 1200° F., however, for such treatment it is recommended that X1335 be considered. For simple carburizing, carburize at 1650° F. and quench in oil or water.

— AISI C1137 —

PHYSICAL PROPERTIES

As Cold Drawn:—Tensile Strength 105,000 lbs/in^2. Brinnel Hardness 212. Elongation in two inches 15%.

As Hot Rolled:—Tensile Strength 95,000 lbs/in^2. Brinnel Hardness 185. Elongation in two inches 18%.

An excellent steel for machining and automatic or hand screw machine work and often used as a substitute for 1025, 1030, 1035, 1040 and 1045, due to its strength and heat treating properties. It is a high manganese screw stock, open hearth steel, used for gears, worms, bolts, shafting, moving mechanisms, etc. Machines with a better finish than carbon steel and produces much better threads. Ideal for simple heat treatment, but not suitable for cyanide or case hardening. An ideal steel for the experiental machinist. For simple heat treatment—Heat to 1525° F., quench in oil, and draw to the required hardness between 600° and 1200° F. (Drawing at 600° F. produces a Tensile Strength of 197,000 lbs/in^2, and a Brinnel hardness of 461).

—— S.A.E. 1020 ——

PHYSICAL PROPERTIES

As Cold Drawn:—Tensile Strength 77,000 lbs/in^2. Brinnel Hardness 163. Elongation in two inches 19%.
As Hot Rolled:—Tensile Strength 62,000 lbs/in^2. Brinnel Hardness 120. Elongation in two inches 35%.

Commonly referred to as plain carbon carburizing steel or case hardening steel. 1020 is usually supplied when simple "Cold Rolled" steel is ordered. It is not an easily machined stock as it tears badly when threaded and leaves a rough finish when turned. It is used when a ductile steel is necessary, and can be bent, punched, or deformed more than the high sulphur screw stocks. For general lathe work it is not recommended—1112, 1315 and 1335 are more suitable unless great ductility is required. Simple heat treatment is not recommended for 1020, a higher content carbon steel being more suitable. Case hardening of 1020 steel is quite common, giving a hard case and a ductile core. For case hardening, carburize at 1650° F., and quench in water or oil.

—— S.A.E. 3135 AND 3140 ——

PHYSICAL PROPERTIES OF 3140

As Cold Drawn:—Tensile Strength 105,000 lbs/in^2. Brinnel Hardness 202. Elongation in two inches 17%.
As Hot Rolled:—Tensile Strength 115,000 lbs/in^2. Brinnel Hardness 240. Elongation in two inches 22%.

Medium carbon types of the low chromium nickel steels used for parts requiring greater strength than obtainable with 3130. The ductility and toughness depend upon the heat treatment used but for the same conditions of treatment will be less than 3130. These steels, 3135 and 3140, are widely used and well known, and are used for shafting, forgings and machined parts of high strength. They are suitable only for simple heat treatment and should never be carburized. For machining purposes, 3135 when cold drawn should be specified as "Annealed" but when Hot Rolled should be used in its natural state; 3140 should be specified as "Annealed" when either the cold drawn or the hot rolled is used. For simple heat treatment—Heat to 1500° F. and quench in oil. Draw at the required hardness between 800° and 1300° F. (Drawn at 800° F. gives a Tensile Strength of 175,000 lbs/in^2 and a Brinnel Hardness of 341.)

—— 18-8 STAINLESS STEEL No. 303 ——

PHYSICAL PROPERTIES AS ANNEALED

Tensile Strength 94,000 lbs/in^2. Brinnel Hardness 165. Elongation in two inches 61%.

This steel is commonly spoken of as 18-8 Free Machining Stainless Steel, and contains 18% Chromium and 8% Nickel. This steel cannot be heat treated for hardening and is generally furnished in the annealed state. In general, the 18-8 group of Stainless Steels are the best for corrosion resistance and is used for hotel and clinical equipment, food and dairy equipment, fruit cannery, meat packing, and soda fountain equipment, etc. As produced by the Allegheny Steel Company this metal is called "Allegheny Metal." Beauty of surface, with a permanent finish resistant to many acids and alkalies, makes it a popular steel for decorative purposes. It can be used in non-corrosive atmosphere up to 1700° F. Machining is not difficult with proper tools and tool shapes, but the above grade number (No. 303) should be specified when this metal is to be used on the lathe.

TEMPERATURES OF STEEL JUDGED BY COLOR

The modern heat treatment of steel does not depend upon the color for an indication of temperature. Alloy steels are critical as to their hardening temperature, and gas heated or electrical ovens with accurate temperature indicators are used.

Judging temperature, especially high temperatures, by color depends entirely too much upon the lighting of the room and the eyes of the operator. For this reason, these listed colors should not be relied upon for accurate heat treatment work.

COLORS FOR TEMPERING

Degrees Centigrade	Degrees Fahrenheit	Color
221.1	430	Very Pale Yellow
226.7	440	Light Yellow
232.2	450	Pale Straw Yellow
237.8	460	Straw Yellow
243.3	470	Deep Straw Yellow
248.9	480	Dark Yellow
254.4	490	Yellow Brown
260.0	500	Brown Yellow
265.6	510	Spotted Red-Brown
271.1	520	Brown Purple
276.7	530	Light Purple
282.2	540	Full Purple
287.8	550	Dark Purple
293.3	560	Full Blue
298.9	570	Dark Blue

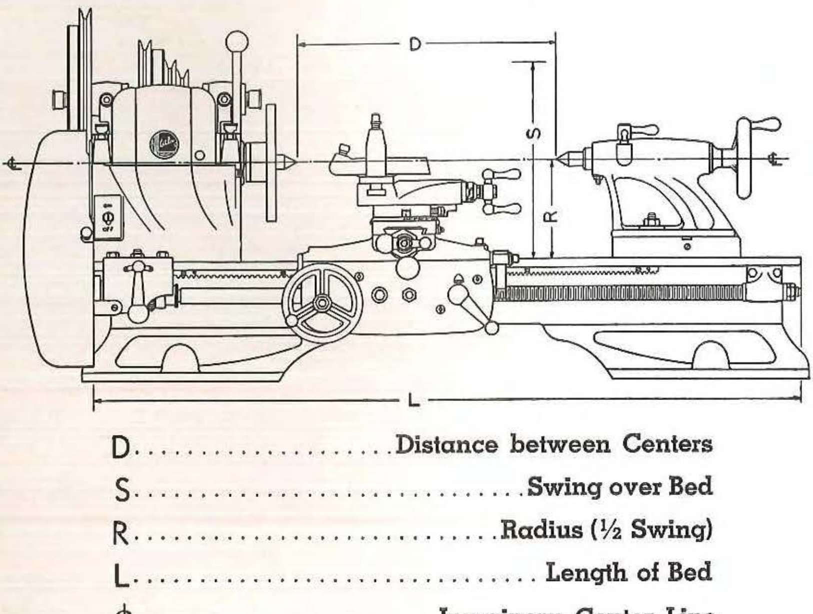
HIGH TEMPERATURES JUDGED BY COLOR

Degrees Centigrade	Degrees Fahrenheit	Color
400	752	Red heat—Visible in the dark
474	885	Red heat in twilight
525	975	Red heat in daylight
581	1077	Red heat in sunlight
700	1292	Dark Red
800	1472	Dull Cherry Red
900	1652	Cherry Red
1000	1832	Bright Cherry Red
1100	2012	Orange Red
1200	2192	Orange Yellow
1300	2372	Yellow White
1400	2552	White Welding Heat
1500	2732	Brilliant White
1600	2912	Dazzling Bluish White

Courtesy Machinery's Handbook

THE BACK-GEARED, SCREW-CUTTING LATHE

DEFINITION OF TERMS



D.......................Distance between Centers

S.............................Swing over Bed

R............................Radius (½ Swing)

L...............................Length of Bed

¢.........................Imaginery Center Line

Part 11

INDEX

11 - INDEX

12 - NOTES

INDEX

12 - NOTES

Part 12

PAGES FOR YOUR SHOP NOTES

PAGES FOR YOUR SHOP NOTES

Your own shop notes, entered on the following pages, will make a valuable supplement to the facts presented in this Manual. Every experienced shop man keeps such a record. It should include miscellaneous "dos" and "don'ts," reading notes, formulas, page references and "pointers" from other lathe men.

The notes on these first two pages are reprinted through the courtesy of Popular Science Monthly. They will give you a good start toward a complete "glossary" of your own.

* * *

A 3/4" brass bar about 12" long makes the best "bumper" for ejecting headstock center or reducing sleeve -- a smaller bar may bruise tapers. Face the end of the bar occasionally.

* * *

A piece of paper pasted over the face of a dial indicator and exposing ten thousandths each side of zero will keep the eyes from wandering.

* * *

For precise measuring with a steel scale, become accustomed to make measurements from the one inch graduation. The ends of scales wear and cause inaccuracies when used as the initial gauging point.

* * *

Geared scroll chucks must operate freely and should be given a good cleaning every five or six weeks. They will give better results and last much longer when cleaned regularly.

* * *

Characters for identification purposes can be placed on steel and metal bars with rubber stamps and printer's ink, and protected with a thin coat of shellac or lacquer.

* * *

When using a tool post grinder to grind diameters to close tolerances, swivel the compound rest 80 degrees from center. Each movement of 0.001 inch on the micrometer dial will then move the grinding wheel forward 0.00017 inch. When taking minute cuts of known value, this method has many advantages.

Stand aside before setting any grinding wheel into motion. The centrifugal force of a wheel increases as the square of the velocity.

* * *

When you are through with a tool, return it immediately to its proper place in as good condition as when you took it.

* * *

Never withdraw a hand reamer from a hole by turning it counter-clockwise.

* * *

Never attempt to drive or press a bushing into a hole if it is possible to draw it into place by inserting a threaded bolt through the bushing and the part to be bushed and taking up on a nut at the opposite end.

* * *

When a milling cutter chatters, try slowing down the spindle speed. The teeth of the cutter may be synchronized with the lathe gear teeth at the speed the spindle is running.

* * *

The cutting edges of reamers may be protected by wrapping them in several places with gummed tape before storing.

* * *

If a correctly ground wire drill cuts large, the first thing to do is to check the concentricity of the chuck. If there is no trouble there, shorten the drill 1/4 to 1/2 inch and repoint it.

* * *

An occasional oilstoning of the cutting edges of a drill or reamer will extend the time between grinds. Stoned cutting edges also stand more feed and speed.

* * *

A milling cutter will last much longer between grinds if the sharpening is finished by taking a light cut with a freshly dressed wheel and oilstoning the cutting edges.

Printing and Cerlox Binding
By
IHLING BROS. EVERARD CO.
Kalamazoo, Michigan