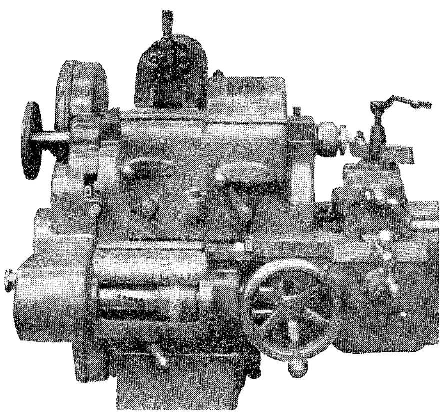


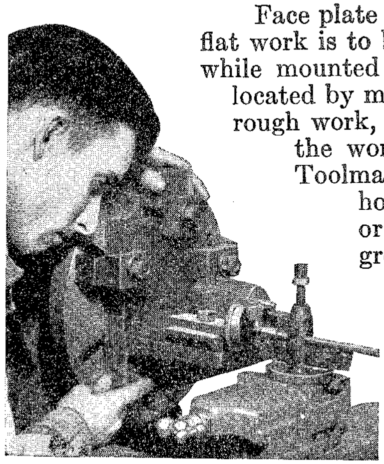
A Draw-In Attachment on a Lathe

The collet or draw-in chuck is an extremely accurate type of chuck used to hold parts, on finished surfaces, for additional machining operations.

The collet is a split cylinder with a male taper on the projecting end. The male taper is pulled into a female taper, thus closing the collet. The collet has a hole of standard size or slightly larger. To clamp or grip a piece of work, insert it in the collet and draw the collet back into the closer bush by means of the handwheel on the end of the spindle. The illustration shows a draw-in chuck on a lathe.

Many variations of chucking work are possible. The illustrations will give some idea of the universal nature of chucking possibilities.

Work may also be bolted to the carriage or the carriage slide while the hole is machined with a fly cutter mounted in a boring bar held between centers and driven by a lathe dog. This method is used largely where the piece to be bored is too large to swing in the lathe, or is too bulky or awkward to handle properly, see page 74.



Centering a Button

Face plate boring is used in many cases where flat work is to be bored. Small jigs may be bored while mounted on the face plate and the holes located by means of the center punch marks for rough work, or by using toolmakers' buttons if the work must be performed accurately. Toolmakers' buttons are merely small hollow cylinders, generally .300″, .400″ or .500″ in diameter by $\frac{7}{16}$″ long, ground round and straight on the outside diameter and having the ends square with the outside surface.

A small hole is tapped in the jig or part to be machined in the approximate location of the hole to be bored to receive the screw that holds the button. The hole through the cylindrical button is larger than the diameter of the screw allowing for movement of the button to locate it accurately. The outside diameter of the button is accurately located over the hole to be bored, by the use of straight edges and mikes, and then the button is positioned over the axis of the lathe, after which the hole is bored. The accuracy of the positions of the holes is thus limited only by the accuracy of the button positions and the accuracy with which the buttons are centered on the lathe preparatory to boring. See illustration on opposite page.

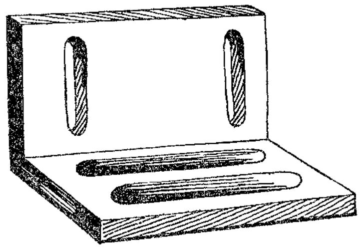
Work on Face Plate

Many jobs are of such a nature that due to physical dimensions they are most conveniently mounted on the face plate. They may be fastened to the face plate by tee bolts, clamp dogs, or planer type stops, as best suited to the job.

The small face plate (page 41) is used mainly to provide a drive to a lathe dog clamped on work. However, a stud may be bolted to the face plate to engage a projection on the work which then acts as its own dog.

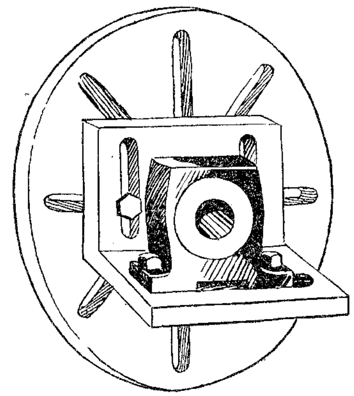
The large plate is usually used in a different manner.

Fundamentally, it is used to provide a plane surface at right angles to the axis of spindle, on which the work is mounted for turning, boring, drilling, etc.



By means of a 90° angle plate, as illustrated, many additional jobs may be mounted for lathe work.

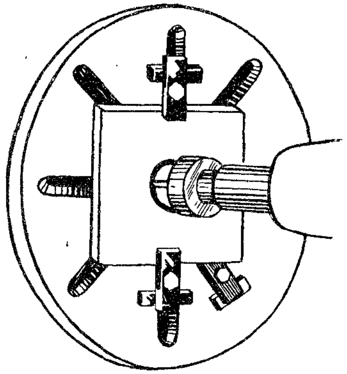
For the purpose of fastening work to the face plate, radial slots are provided to engage bolts used to clamp the work.



An outline of a few typical face plate jobs illustrates both the types of job and manner of fastening.

1. A motor base would be clamped to face plate, with the under side of base against face plate, to enable the surfacing of upper side of base by a facing cut in lathe, as illustrated.
2. A bracket, with axis of the hole perpendicular to face of bearing surface, could be bored very easily by means of face plate mounting.

3. Trepanning may be easily accomplished by bolting material to face plate, as illustrated.



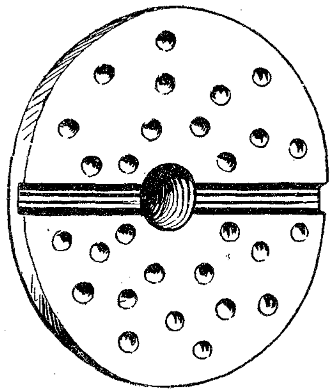
4. Trepanning is the term used to denote cutting out a washer or some such similar piece from a flat, relatively thin piece of stock which could not be bored and held on a mandrel as could a regular turning job such as a bushing.

5. A part, such as a round pipe, with a large hole with flanged lids, is readily faced on both ends by face plate mounting.

Other jobs will be easily handled if mounted on the face plate for drive and location.

Boring jobs where several holes must be bored of specific dimensions, can be well handled with this mounting.

Face plates, as illustrated, may be specially prepared for certain classes of work. For instance, the face plate may be cast solid, that is without slots, and may be provided with a large number of tapped bolt holes arranged in circles so as to provide bolting surface for many kinds of odd-shaped pieces. Also a solid face plate may be provided with a groove similar to a keyway to facilitate mounting angle plates similarly provided with slots by means of a key and appropriate clamping.



An odd jobs plate such as described above is recommended for every lathe installation. Such a plate should be kept ready with various size clamps and clamping bolts to facilitate mounting of odd-sized pieces.

A small milling machine vise may be clamped to the plate and used to hold odd-sized parts for flat facing, etc. This method is extremely useful in facing small odd-shaped pieces if a chuck is not available.

Facing Work Held in Chuck or on Face Plate

Work that is large in diameter in proportion to length, or has no hole for mandrel mounting, is usually held in the three or four-jawed chuck for facing off end.

Other pieces of odd shapes are sometimes held most advantageously on a face plate and machined in this position.

Work should be chucked with as little overhang as possible to provide a rigid mounting and eliminate chatter.

If a cored or machined hole is present, it is advisable to start cut at inside and feed to outer edge when facing, as this direction of feed gives much better results than feeding from outside edge in. Various reasons are assigned for this result, but it is essential only to know that this method works and is so justified.

In facing as above, a left-hand turning bit should be used in a right-hand or straight tool holder, as facing in this manner requires a tool similar to a tool used in turning toward headstock. The finish cuts are best taken with the right-hand side facing tool sharpened for the material being cut.

If no center hole is in work, cuts must be started at outer edge and worked to center. In this case a straight or left-hand holder should be used with a right-hand turning bit.

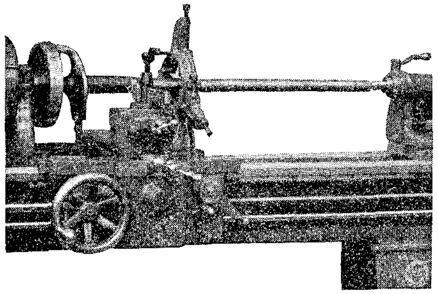
For finishing, use right-hand finishing tool as before, except that tool should be fed into work at center so as to leave no point at exact center of work. The tool is then fed outward as before.

For roughing cuts, tool should be set a bit above center, as in turning large diameters. In finishing, tool should be exactly on center.

Steady Rest

The steady rest should be placed where it will give the greatest support to the piece to be turned, which is generally about the middle of the piece. However, the best position is obviously determined by the design of the piece.

If the part to be supported has a diameter of short width to be turned, this can be finished with a fine feed and slow speed and

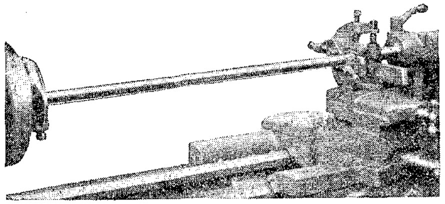


used as the supporting position in the steady rest.

If the construction does not determine the position of the steady rest, turn a "spot" about the middle of the piece or pieces to be turned. Place the part between centers, as illustrated, with the steady rest in position at the turned "spot", close and clamp the upper part of the steady rest, and bring the jaws to bear lightly on the finished "spot" with a running clearance. The clearance is set by means of the adjusting screws. To prevent scoring, oil the jaws each time a piece is clamped in the steady rest. To remove the piece, loosen the clamp bolt and swing the upper part of the steady rest back. Thus the pieces can be changed without changing the adjustment of the jaws.

The best way to align a steady rest to hold the unsupported end of a piece, as illustrated on page 47, is to place a bar of the same diameter, as the piece to be machined, between centers and adjust the jaws to it. Then remove the bar and place the piece in position. This method insures the proper centering of the steady rest.

Follow Rest



The follow rest differs from the steady rest in that it moves with the carriage and provides support against the forces of the cut only. The tool should be set to the diameter selected and a "spot" turned about ⅝" to ¾" wide. Then the follow rest jaws should be adjusted to the finished diameter to follow the tool along the entire length to be turned.

The follow rest jaws should be kept oiled while in use to prevent scoring or picking up.

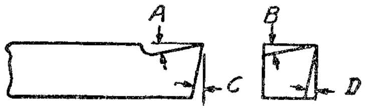
The follow rest is indispensable when chasing threads on long screws, as it allows the cutting of a screw with a uniform pitch diameter. Without the follow rest the screw would be inaccurate, due to the springing away from the tool, perhaps impossible to cut it at all.

HOW TO GRIND LATHE TOOLS

The successful operation of a lathe and the class of work turned out depend to a great extent on the skill of the operator in grinding his tools. Dull and improperly ground tools throw a heavy strain on the feed mechanism, cause the work to spring and the lathe to chatter.

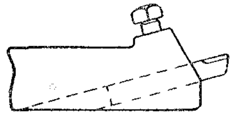
Lathe tools are made of carbon steel, high-speed steel and alloys such as stellite and cemented carbide. The stellite and cemented carbide tools are becoming more generally used as their cost is reduced. There are but a few carbon steel tools used, the general practice is to use high-speed steel tool bits in holders. Determine the kind of a tool you are grinding, as carbon and high-speed steel requires different treatment. Tools should be marked to show the kind of material from which they are made. A quick and simple way to tell whether a tool is carbon or high-speed steel is to grind the end and watch the sparks. If carbon steel, the wheel will throw a light colored spark and if high-speed steel, the sparks will be a dark red.

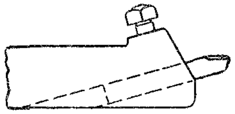
Carbon steel tools should be ground on a grindstone or wet emery wheel or dipped into water often to keep the tool cool while grinding to prevent the drawing of the temper. On the other hand, high-speed steel tools should not be dipped into water when hot as it will crack the tool and crumble the cutting edge.



127

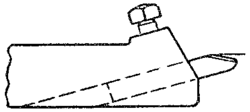
Four angles are important in grinding tools and these vary with the material being machined. The top rake (see illustration 127) (and page 38) "A", the side rake "B", the front clearance "C" and the side clearance "D".


128

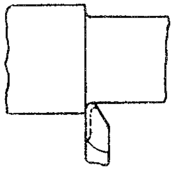

129

The top rake is usually provided for in the tool holder by the tool being set on an angle (128), which is correct for the machining of steel and cast-iron. On solid steel tools it is necessary to grind the top rake in the tool. By adjusting the tool in the tool post through wedge or rocker, this top rake can be varied somewhat to suit the material being turned. The softer the material the less the top rake should be as there is a tendency for the tool to dig in if the rake is too great.

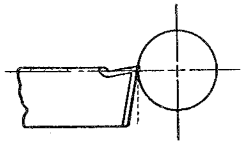
For turning brass, there should be no top rake (129) and the cutting edge of the



130



131



132

tool should be about horizontal. For turning soft copper, babbitt, and some die casting alloys, a negative rake (130) is often used.

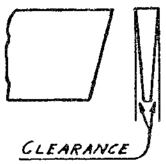
The side rake "B" (127) also varies with the material being machined. If this angle is made great enough, the tool will drag the carriage along by feeding into the work of its own accord, especially if the material is soft. On the other hand, without side rake, the tool would not cut and the feed mechanism would be under excessive strain. The proper angle is from 6 degrees for soft material to 15 degrees for steel.

The tool is ground with the side clearance "D" (127) to take care of the feed advance and to prevent the dragging of the tool on the shoulder formed by the cut (131). This angle is usually about 6 degrees from the vertical and is constant.

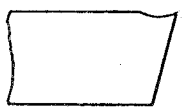
The front clearance "C" (127) depends somewhat on the diameter work to be turned. To turn cast-iron or steel it is advisable to set the tool above center. If the tool was ground square without any front clearance, it would not cut, but would rub on the material to be turned below the cutting edge of the tool (132). The front clearance is necessary for this reason. This clearance should be less for small diameters than for large diameters. The clearance should range from 8 to 15 degrees. Do not grind more front clearance than is necessary as this takes away the support from the cutting edge of the tool.

Tool bits can be ground best in their own holders. To prevent grinding the holder, extend the tool beyond its regular cutting position.

After a tool has been ground on the emery wheel, it will produce better work and last longer if the cutting edge is stoned with an oil stone. This takes out the wheel marks and gives a smooth cutting edge. Care must be taken in grinding cut-off tools to see that both sides of the tool have the necessary side clearance (133). A tool of this kind also cuts better if a lip is ground back of the cutting edge to curl the chip as it comes off the piece (134).

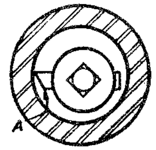


133



134

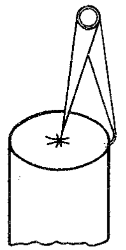
In grinding boring tools, see that the front clearance is sufficient to prevent the tool from rubbing in the hole and dragging at the point "A" (135).



135

CENTERING WORK

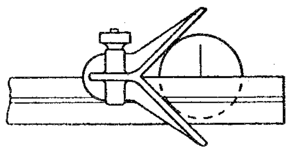
Lathe work may be divided into two principal classes, namely, work machined between centers and work machined in chucks. Bar or shaft work is done between the centers, the piece to be turned having been previously centered.



136

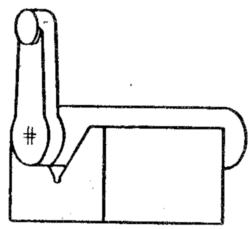
There are many ways of centering a piece of material. In large production shops this work is done in a centering machine. In small shops the lathe operator usually centers his own work. The first thing to do is to find the center on each end of the pieces. This can be done by using hermaphrodite calipers. Set the caliper to about one-half the diameter of the piece, chalk the end of the piece so the scribe marks can be seen, and scribe four arcs, one from each quarter of the circumference (136). The center of the piece lies between the four arcs. Mark the center thus located with a center punch. Perform the same operation on the other end of the piece and it is ready to have the centers drilled.

The center of a piece can also be located by the use of a center head on a combination square. Draw two lines at about right angles to each other. Where they bisect will be the center of the piece (137).



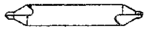
137

To center irregularly shaped pieces such as a drop forged brake lever, lay the piece in a V-block (138) on a plane surface and use a surface gauge for scribing the lines on each end. First set the gauge to the approximate height of the center and scribe a line. Turning the piece, scribe three more lines, each at about 90 degrees. The center of the piece is in the center of the square formed by the scribed lines.



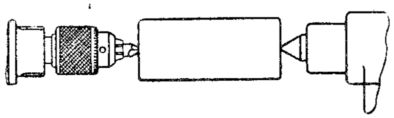
138

After the center has been found on each end of the piece and it has been center-punched, the actual drilling of the center can be done under a drill press or in the lathe itself.



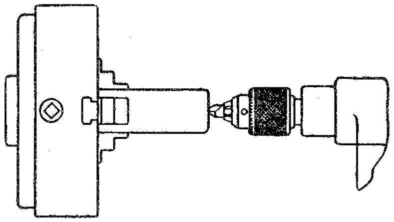
139

Center Drills (139) are usually used for centering. The drill is held in a drill chuck (as shown in Fig. 125 or 126, Page 29), having a shank that fits the taper hole in the spindle. The tailstock is set on the bed the proper distance from the headstock to permit holding the piece in the center punch marks between the center and the centering drill (140). To keep the piece from rotating, hold it with the left hand, and advance the tailstock spindle with the tailstock screw handle (page 26). The piece is fed into the drill and the one end center drilled. The piece is then reversed and the other end is drilled. This method is used for centering rough bar stock where the entire length is to be turned.



140

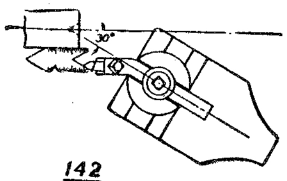
To center finish ground stock such as drill rod or cold-rolled steel, where the ends are to be turned and must be concentric with the unturned body, other methods must be used.



141

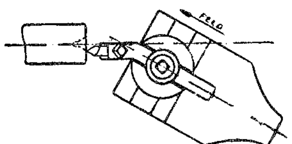
If the piece is small enough in diameter to pass through the spindle, it can be held in a universal chuck on the spindle nose (141), or a draw-in collet (Page 58). Hold the center drill in a drill chuck in the tailstock spindle and center drill one end; reverse the piece, and proceed as before.

If a piece must be centered very accurately, the tapered center should be rebored after center drilling, to correct any run-out of the center drill. This is done by grinding a tool bit to a center gauge at a 60-degree angle. Then with the tool holder held in the tool post, set the compound rest at 30 degrees with line of centers (142).

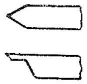


142

Set the tool exactly on the center for height and adjust to the proper angle with a center gauge. By feeding the tool in on this angle, any run-out of the center is corrected (143).



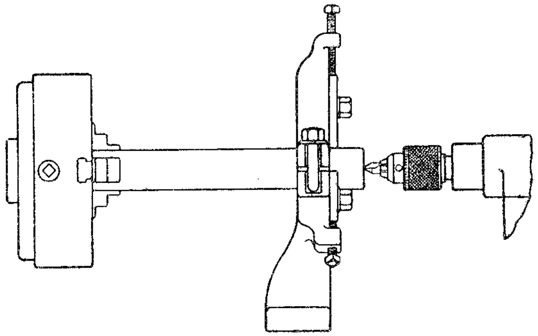
143



144

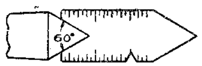
The tool bit should be relieved under the cutting edge to prevent the tool dragging or rubbing in the hole (144).

To center long pieces that will not pass through the spindle, a steady rest is used to support the outside end of the piece as near the end as possible (145). The drill and chuck are held in the tailstock spindle and the drill fed to depth by advancing the tailstock spindle. To correct any run-out of the center use the same methods as described before.

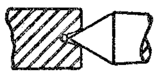


145

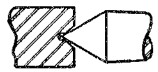
It is absolutely essential that the centers in the piece to be machined conform to the centers in the lathe so that the sides of the center in the piece have uniform bearing on the lathe center. Lathe centers are ground on a 60-degree included angle and can be tested with a center gauge (146). To do good work the lathe centers must fit the drilled centers If the center bearing is not uniform, the work will not be round but it is likely to be tapered due to the rapid wear of the center hole. Another result of improperly drilled centers is chatter marks due to the piece being loose between the centers. Avoid conditions such as are shown in Figures 147, 148 and 149. This is especially true of the tailstock end or dead center. To test for the kind of bearing the lathe center is taking in the work, rub a little red lead on the lathe center and spin the piece on the centers by hand. The bearing surface will be bright. If the bearing is incomplete, correct it as directed above.



146



147

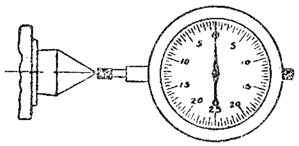


148

149

Testing Lathe Centers

The center in the lathe should run true. To test this an indicator is necessary. The indicators are usually of the dial type and read in thousandths of an inch. When testing the headstock center try the indicator first on the point of the center to see that the spindle has no end movement (150). If the indicator is tried on the angle of the center when the headstock spindle has end play, the reading on the indicator dial will be inaccurate and misleading. However, if there is no end play in the spindle and the indicator shows run-out on the headstock center, then the three possible points of error should be checked and correction made.



150

The first and most probable source of trouble is the presence of dirt or chips in the taper hole between the spindle and the center bushing, or between the center bushing and the center itself. To correct this trouble, remove the center and the center bush, clean the taper holes in the spindle and the center bush and the outside of both the center bush and the center. Replace the bush and the center and again test with the indicator.

The second source of trouble may be the presence of a burr or scratch on the surface of the spindle tapered hole or on the surface of the center bush. Since the center bush is hardened (about file hard), it is very seldom that trouble from this source occurs. It happens at times, however, that there is a burr in the spindle hole, caused by a drill chuck shank turning under cut or some similar occurrence. Should this be the case, carefully remove the burr with a scraper, being careful to remove only the high spot, or use a Morse taper reamer of appropriate size as outlined on Page 26.

Le Blond Regal Lathes are tested before shipping, and the taper holes in both headstock and tailstock spindle have been reamed clean and true. Operators should make every effort to retain the original accuracy of the lathe, being careful to preserve smooth clean taper holes in the head and tailstock spindles, as the accuracy of the lathe is largely dependent on the bearing of the centers in the tapered center holes.

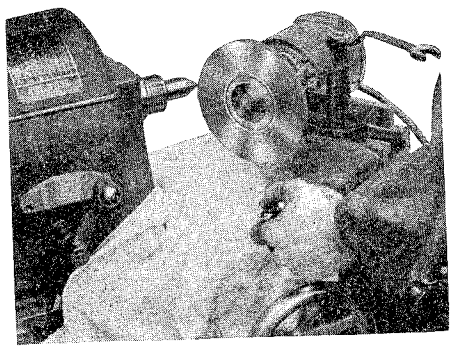
The third cause of center run-out is inaccuracy of the center point itself. Since both hard and soft centers may be used in the headstock spindle, two methods of correction are outlined on **next page.**

For soft centers it is only necessary to set the compound rest at 30 degrees with the axis of the lathe and take a skim cut over the point with a lathe tool sharpened to cut smoothly.

When repointing a hard center, it is necessary to grind it, as the center is harder than the turning tools.

The procedure for grinding is as follows:

As before, the first step is to set the compound rest over to 30 degrees with the axis of the lathe. Next, mount a tool post grinder or our grinding attachment in the tool post slide. Third, cover the exposed ways of the lathe with cloth or paper to prevent the grinding grit reaching the bearing surfaces of the bed and cross slides. Fourth, put the headstock in gear to give approximately 200 R. P. M. to the spindle, and take a light cut over the center point, feeding the wheel across the point by means of the compound rest feed handle. Continue to feed the wheel back and forth until it is cutting evenly all around and on the entire length of the center point, and then check the angle with a standard center gauge. Reset the compound rest if necessary and continue grinding until the center fits the center gauge accurately. The accuracy of the fit can be observed by placing an electric light bulb beneath the center and looking for light between the center point surface and the edge of the center point gauge.



Grinding Center

The operator should be able to make the center run true by checking the three trouble-giving points outlined on page 68.

OPERATIONS

Facing to Length

One of the operations done most frequently on a lathe is facing the ends of pieces to length. The procedure is as follows:

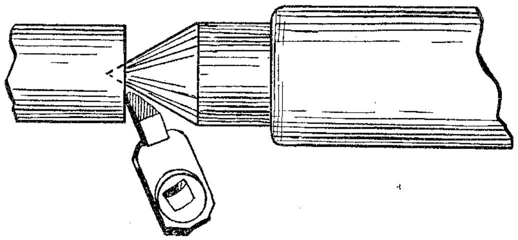
1. Measure piece to determine how much stock is to be taken off.

2. Be sure center holes are free of dirt, chips, etc., and have no burr on edge.

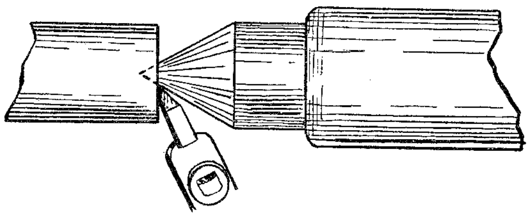
Put red or white lead or other center lubricant in center on tailstock end.

3. Tighten driving dog on work and place work on live or headstock center and then run dead center into place. Hold work with left hand and adjust tailstock center with right hand until work can be turned easily on centers but has no end play. Clamp tailstock spindle.

Another method is to put work between centers, run up tail center until tight and back off until just free. Start lathe. Lathe dog tail will click as face plate and work revolve. Tighten tailstock until clicking just stops and clamp tailstock spindle. After a few trials the operator will learn to judge the proper setting of the tailstock center by feel.



4. If stock to be turned off is slight, say $\frac{1}{64}''$ to $\frac{1}{32}''$, the right-hand side finishing tool may be used. If the stock is about $\frac{1}{16}''$ to $\frac{1}{8}''$ or more, however, the end should first be roughed off with the right-hand corner tool as far as possible toward center. Then the finish tool should be used to remove burr around center and a skim face cut taken for a smooth finish.



The corner tool is as above stated used for roughing off to length and cuts from outside toward center of work.

The finishing tool, however, is only used for skim cuts and cuts from the center hole out to the outside diameter.

For roughing cut, set edge of corner tool exactly on center of work and run the cross slide in until tool just misses the OD of work; measure work with scale and set tool edge to cut length so as to leave a skim cut for finishing. If excess length cannot be removed in one cut, split the excess amount into about equal parts and rough down to length.

5. Remove corner tool bit and put right side facing tool in holder. Clamp bit, set holder so tool edge is exactly on center line of work and tool edge is at slight angle with end of work, the point of tool doing the cutting.

(The selector bush should be set in proper position for cross feed and the head reverse to feed gears set for "out" cross feed before taking the finish cut.)

6. Feed tool into ridge around center hole until ridge is cut off flush with end of work. Use both hand longitudinal and cross feeds for this operation.

7. Set point of tool toward headstock, right next to center, clamp carriage and start tool out by hand feed. After about ¼ turn of cross feed handwheel, throw in feed clutch and take rest of cut out to edge of work power.

8. The surface of the cut above taken should be very smooth and without tool marks.

If tool marks are evident, the following points should be checked:

The feed should not be too great, from 3 to 8 thousandths of an inch is the usual range.

The tool edge should not be at too great an angle to the line of the surface being faced.

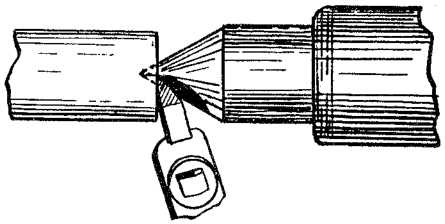
There should be no burr on the tool point such as would be caused by running point of tool into tailstock center when removing ridge around center hole.

To insure a smooth finish cut, sharpen tool with a small hand stone just before the finish skim cut.

Steel finishing side tools are ground with a 10° to 12° side clearance, about 10° to 15° side rake, no top rake, and about 8° front clearance.

Cast iron, brass, etc., the same clearance except no side rake is used on tool.

9. When facing to length it is possible to use the regular tailstock center. It is, however, much handier and quicker to have a center with a groove or flat milled or ground on the taper portion so as to provide clearance for the tool edge. This clearance space allows the use of standard shaped tools when facing to length instead of sharp pointed ones necessary to get into space between end of work and regular center.



The necessary flat may be ground by hand on a bench grinder. When grinding flat, be sure to grind less than half way through point as otherwise the center would become in effect a center reamer and would cut the center larger and allow the work to loosen between centers and wobble.

Chatter marks on finish cut are caused by one of the following:

1. Speed of work is too high—remedy, slow work down.
2. Feed is too high—remedy, slow feed down.
3. Tool edge is set too nearly parallel with edge of work being cut—remedy, increase angle of tool slightly.

DRILLING, BORING AND REAMING

Holes made be drilled with the lathe by driving the drill in the headstock spindle, holding the work against a pad on the tailstock spindle and using the tailstock spindle motion for feed. Small pieces may be held on the face plate or in a chuck while the drill is held and fed by the tailstock spindle.

Holes and counterbores are easily machined in a lathe. The advantage of boring is that a perfectly round true hole is obtained. Two or more holes of the same or different diameters may be bored at one setting, thus insuring absolute alignment of the axis of the holes.

Work to be bored may be held in a chuck, bolted to the face plate, or bolted to the carriage slide. The tailstock end of long pieces may be supported in a steady rest as shown in the illustration on page 62.

Single point boring is the process of rotating the work either in a chuck or on a face plate while a tool is fed into a drilled or cored hole in the work.

Special boring bars of single and multiple tool types may also be used, provided the amount of work justifies the extra expense of the special tools.

However, for the most part, single point boring is used on a single piece or small lot jobs.

Boring tools may be either of the solid forged type or of the inserted cutting bit type as shown on pages 35 and 38. The use of either depends upon the operator's choice or preference with reference to the respective costs.

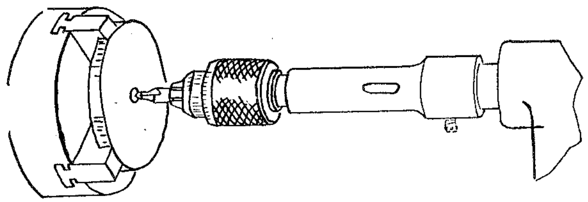
It is advisable, wherever possible, to bore the hole within a few thousandths of the finished size and then to finish ream the hole to exact size.

The three above operations, while distinct items of individual operations, are often performed in sequence on a single piece of work at the same work setting, that they will be considered together.

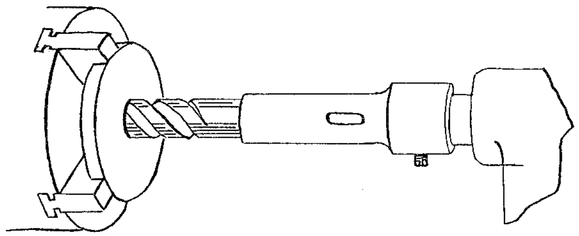
We will assume piece to be drilled, etc., has been properly mounted on face plate or in chuck to proceed with drilling. (See articles on chucking and face plate mounting, page 51.)

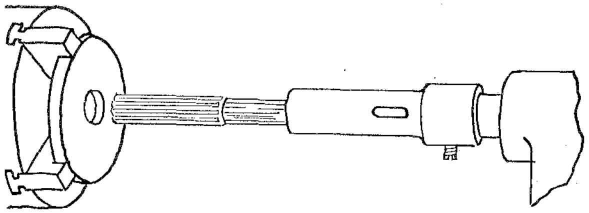
Spot center with center drill to provide true point of entry for small drill.

Drill through piece with $\frac{1}{8}''$—$\frac{1}{4}''$ drill to clear flat tip of larger drill used to remove bulk of metal and to permit entry of boring tool if used.

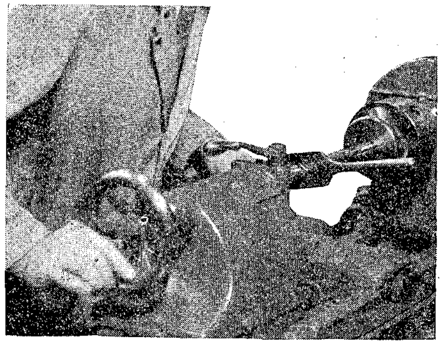


If hole is to be just a rough job both as to size and straightness, use large drill $\frac{1}{64}''$ or $\frac{1}{32}''$ smaller than finish size, and run through piece. Drill is held in taper hole in tailstock spindle if taper shank drill is used, or in drill chuck mounted in tailstock spindle if straight shank drill is used.



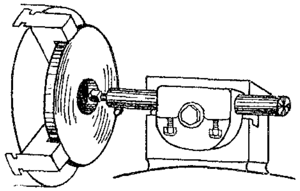


A reamer can be held either on the taper shank in the tailstock bore, in a drill chuck mounted in the tailstock spindle, or held in line with the tailstock center of the lathe. When held on tailstock center, the reamer is kept from turning with a wrench or clamping dog while it is fed into the work by means of the tailstock screw. This latter method of holding a reamer is shown in the illustration.



Hold center hole in reamer against tailstock center and guide the cutting of reamer end into hole, as right hand turns tailstock handwheel to bring reamer up to work. Run lathe spindle at about 40 R. P. M. Reamer should be prevented from rotating by clamping wrench on reamer and allowing handle to stop on carriage wing or bar held in tool post. Start lathe and feed reamer in by tailstock handwheel (as illustrated). On cast iron or brass no lubrication is necessary. On steel and wrought iron, it is advisable to lubricate with lard oil or some similar lubricant.

Above method produces a reamed hole, but hole may not be exactly straight or parallel with axis of lathe, due to fact that large drill previously used may have run slightly off the straight path. The reamer in this case follows the crooked hole.



To produce absolutely straight and true holes, drill used to rough out stock should leave hole about $\frac{3}{64}''$ to $\frac{1}{16}''$ small.

Use boring bar mounted in tool post (as illustrated), and bore hole until about eight to ten thousandths stock are left.

To act as true guide for reamer, bore hole at outer end—or end reamer enters for $\frac{1}{8}''$—to about three thousandths small.

Proceed with rough and finish reamers if both are available.

If roughing reamer is not available, leave only 3 to 5 thousandths stock for finish reaming after boring.

Variations in the procedure outlined above are numerous, and will suggest themselves to the operator, but the general procedure is the same in all cases.

Points to be remembered when performing the above operation include the following:

Straighter holes are made with a drill held from rotating and fed into the revolving work, than with a revolving drill in a stationary piece . . . A small pilot hole to clear the flat tip of a larger drill to follow allows the drilling of the larger hole quicker and with less power . . . To insure a straight hole, singlepoint bore the hole after drilling and before reaming . . . Boring tools generally used are the single point type for rotating work, and bar type for fixed work where bar rotates between centers and work is fed past the cutter bit . . . Steel requires lubricant, it is advantageous on brass or cast iron, but not necessary. Boring tool tips should have clearance and rake similar to turning tools used on the respective materials . . . When boring to finished size, if no reamer is available, take last cut with feed entirely through piece, then reverse feed without changing boring tool setting and feed tool out again. The slight natural spring of the tool and the crossed feed line will assist in producing a smooth finished hole. In this connection it should be pointed out that the rounder nose the tool has the smoother the job if chatter does not occur. In case of chatter, slow work up. If this does not eliminate the trouble, try a more pointed nose on tool. Also try changing center height of tool.

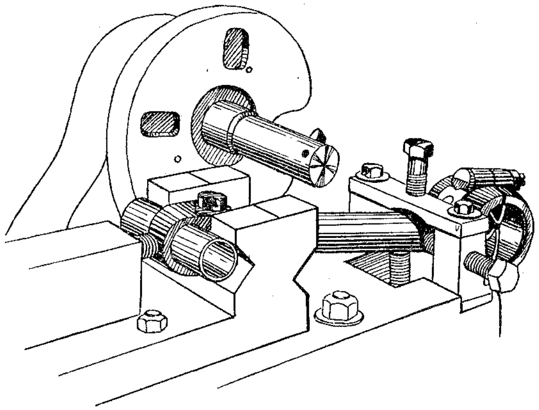
Boring is very accurate with work mounted on a face plate and offers possibilities in boring a number of holes at accurate center dimensions by use of buttons (see page 58), disks or some similar method of location.

DRILLING, BORING AND REAMING
Work Held on Cross Slide or Carriage

In this particular type of drilling and boring, the work remains stationary and the cutting tool rotates.

The feed of work relative to tool is obtained by moving the carriage on which the work is mounted.

The tool may be held either in a boring bar mounted only in the spindle, as illustrated, or if hole extends entirely through work it may be held in a boring bar placed between centers of lathe and driven by lathe dog.



Boring two holes on the same axis may be done very accurately by this method. If the work is mounted on the compound rest, two sets of holes may be bored with axis parallel and to very close limits. By mounting the drill in the spindle and feeding the work onto the drill clearance holes can be opened to allow the boring bar to pass through, to finish bore the hole. If the holes are cored in the casting, the boring bar may clear the cored hole and the rough and finish bore operations may be proceeded with directly. Otherwise open up cored hole with drill or single point boring tool, then bore as before.

Boring is a simple and accurate method of machining a true straight hole, for both single hole jobs, which must be produced quickly and economically without jigs, and very accurate work where hole must be bored true with relation to other holes or surfaces. After boring hole to within a few thousandths of finished size, a reamer is generally passed through holes to produce the exact size required.

The actual boring procedure is simple but is influenced considerably by the skill of the operator.

TAPPING

Taps may be held on the tailstock center in the same manner as the reamer shown on page 74, to secure a tapped thread in line with the drilled hole.

A die holder may also be fitted to the tailstock spindle or to the compound rest to facilitate cutting threads on a turned pin or bolt held in a chuck. On work of this class a hand feed or floating connection is preferred so that the tap or die may travel along the cut thread at its own rate of travel without influence by the feed of the tailstock spindle which may not be fed correctly by hand. The proper unit to use for this work is a turret mounted on the bed of the lathe. However, this unit is special and unless there is a large quantity of work of the type mentioned, the cost will be excessive for the benefit derived.

Self-opening dies and self-closing taps are available which automatically release the cutting edges from the threads at the end of the cut. These units, also, are useful only when the lots of pieces to be machined are large.

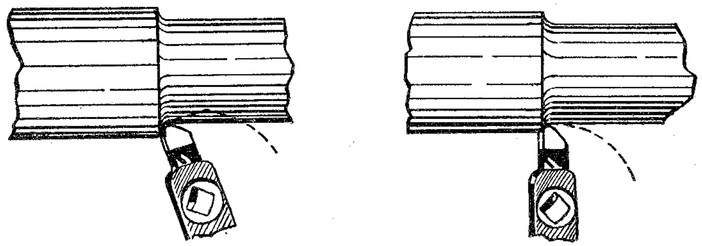
Straight Turning on Centers

Be sure lathe centers are in alignment and that live center runs true.

Put dog on work, clean all dirt, chips or burrs from center holes and adjust between centers, putting center lubricant in tailstock center hole.

Use tool bit properly ground to shape for the operation and material being worked. Tighten bit in holder so that bit does not project more than 1½ times the thickness of bit nor less than the thickness of bit.

Set tool holder square with axis of work so that any movement of tool will lessen the depth of cut and not dig into the work, as



illustrated. Tool holder should be as close to tool post as possible for rigidity and still allow tool bit clamp screw to be turned by its wrench. Be sure the compound rest is as far back on bottom slide as is necessary to avoid overhang of the top slide. See page 39.

Adjust tool bit cutting edge by rocking tool holder on tilting wedge until cutting edge is at proper height, then clamp tight.

A tool properly ground should have the point set on the center line of the work. Some operators make a practice of setting the tool above the center line, about $\frac{1}{64}$ of an inch for each 1″ clearance in diameter. This, however, necessitates changing the clearance and rake angles of the tool and for this reason we do not recommend it for the student or beginner.

Move carriage left and right to be sure tool will traverse the length of cut without obstruction when lathe is running.

Be sure tool edge is free of work, then shift gears to give proper speed of work (see article under "Cutting Speeds", pages 97 and 107) and start lathe.

Start cut at tailstock end, whenever possible, as the thrust of the cut is then taken by the thrust bearing in the headstock instead of by the small center hole in the work.

If only one roughing cut is necessary, adjust tool* to cut piece one sixty-fourth inch over finished size of piece.

Many times it is not possible to remove all the excess stock in one roughing cut. In this case take as many roughing cuts as are necessary to bring work down to finish turning size. Endeavor to split up roughing cuts into equal depths of cuts and take each cut at the maximum capacity of the tool and the machine.

When tool is set to proper depth of cut, throw in feed and cut to required dimension, or, if work is to be turned entire length, to a little over half the length.

Have gibs on cross-slide tight enough so a positive top raked tool will not draw cross-slide in deeper when cutting steel.

Throw out feed, stop lathe, and remove work from centers.

Return carriage to starting position and reverse work between centers, or put in another piece for same operation.

After all roughing work is completed, put round nose finishing tool in tool holder, set lathe for correct finishing speed (see Cutting Speed article, page 97) and set feed for finish cut.

Set tool for a light cut to true up all around; turn about ¼" and mike diameter of part thus turned. Loosen crossfeed dial screw, set dial at "0" and retighten. Set tool in one-half amount work is oversize and again turn ¼". Stop lathe but do not disengage feed. Mike diameter and if correct start lathe again and continue across length of cut. If incorrect, disengage feed and repeat setting until diameter is correct, then proceed as above.

Turn only far enough from end to properly measure diameter, both to save time and avoid spoilage of entire piece.

When stopping lathe to try end, if feed is disengaged, a slight hollow will be made in the finish cut, due to the tool dwelling in this one position for several revolutions.

Note*—When adjusting tool to depth of cut, always move tool towards work so as to remove any backlash present between crossfeed screw and nut.

Finish Allowance

On work to be finish ground, make the finish turned diameter 15 thousandths over actual finish ground diameter.

On work to be file finished and polished, leave 2 to 3 thousandths of an inch over finish size to provide stock to file and remove slight tool marks and give polish finish.

When finish turning several pieces to same diameter, do not change crossfeed setting after once correctly set. Turn diameter required length, stop feed, stop lathe, back off tail center, remove work, return carriage to starting position, insert new piece and turn as before. This insures duplication of size.

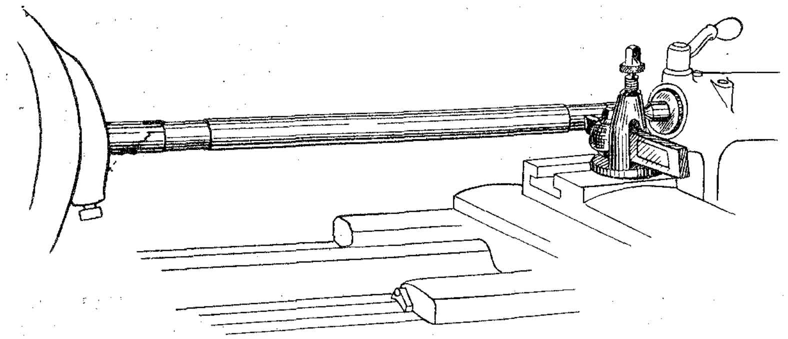
The finish turned surface of steel is usually considerably smoother if cutting oil or soapy water is dropped slowly over the work and tool point on this cut. The liquid may be dropped from an oil can, saturated sponge or waste squeezed in the hand, or by a regular drip spout connected to a can of solution and mounted on the carriage so as to follow the tool.

When clamping any driving device to a finished surface, be sure to protect the surface by placing a copper or brass sheet under the clamping screw.

Alignment of Centers

When turning straight work, try alignment of tailstock center with headstock center to be sure work will not be tapered.

When zero marks are in line on tailstock, top and bottom, centers are approximately in line, but due to the impossibility of seeing an error of .001" misalignment, it is probable that the work will be tapered if a no more sensitive test is applied to the center alignment.

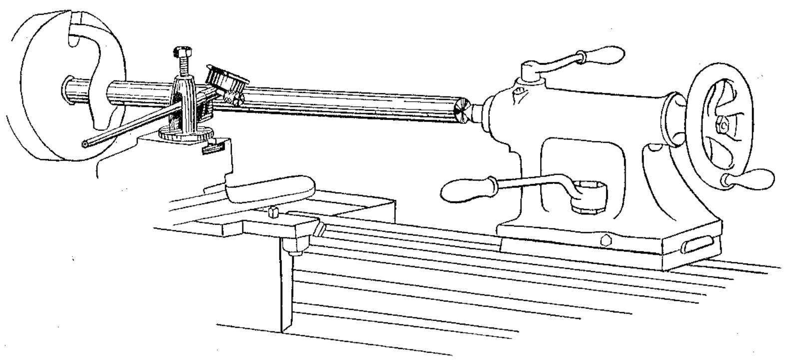


A test bar as shown in illustration is easy to make and use and gives positive results.

The tool is set to just miss one end and then the carriage is run to the other end to see if tool registers the same as the end first tried. If adjustment is as close as can be made by eye, a trial cut of about 2 thousandths depth is taken on one end and then a similar cut is taken on the other end. The two cuts are checked with micrometers and any error corrected by moving the tailstock one-half the error. If work is small at tailstock end, move tailstock away from tool edge. If tailstock end is large, move tailstock toward tool edge.

With this test the work can be held to any desirable limit of straightness.

Another method is to have perfectly straight test piece and try carriage alignment with this piece held between centers. A dial type indicator mounted on the compound rest will show up any error in alignment. Before testing with a straight piece, check live center in head to see that it runs true.



When shifting tailstock to line up centers: Unscrew handwheel, thus removing center from work; loosen hold-down clamp screws; loosen adjusting screw on side opposite to motion of upper tailstock slide; tighten adjusting screw on side towards motion of upper tailstock slide; reclamp tailstock hold-down clamp bolts; readjust tailstock center in work and make trial for alignment.

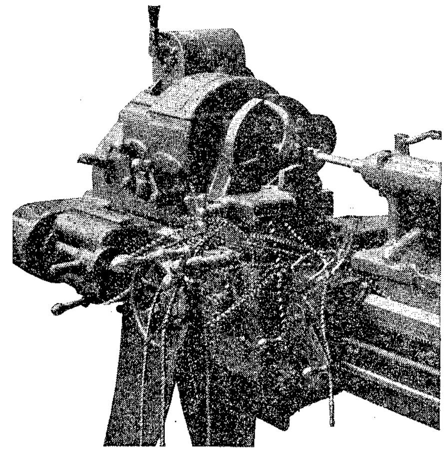
Taper Turning and Boring

There are two methods of taper turning, one, by means of a taper attachment, the other by means of the "set-over tailstock" method.

Both these methods will be described:

Setover Tailstock Method

The oldest and probably most used method of taper turning, shown in the illustration, is the setover tailstock method. The tailstock is made in two pieces, the lower fitted to the bed, while the upper part is fitted to a cross keyway machined on the lower section. To turn straight diameters, the tailstock spindle is set exactly in line with the headstock center, indicated by the zero mark on the graduated boss on the rear of the tailstock. To turn taper diameters the upper half of the tailstock is set over the amount necessary to produce the taper required.



To do this—

Loosen spindle clamp, unscrew handwheel, loosen tailstock clamp, adjust set-over screws to move tailstock stop in proper direction, reclamp tailstock clamps, readjust center, reclamp spindle clamp and take another cut and try taper.

When turning a number of tapered pieces by the above method, it is essential that all pieces be the same length within about 5 thousandths of an inch and should be centered to the same depth. When using the taper attachment, however, the length of the work has no more effect on the taper cut if the taper attachment remains set at the same taper.

Tapers are usually given as the included angle in degrees or as a certain taper per foot. For instance, a taper of one-half inch per foot means that a bar one foot long, having such a taper, would be one-half inch smaller in diameter at the small end than at the large end. However, to turn such a taper, it is only necessary to offset the tailstock one-half the taper specified. In this case, for a piece one foot long, one-half inch taper per foot, set the tailstock over one-quarter of an inch.

If the piece were only six inches long, the tailstock top offset necessary to secure one-half inch taper per foot would be one-eighth of an inch.

From the foregoing examples it will be seen that the setover necessary would be:

$$S = \frac{T}{2} \times \frac{L}{12}$$

where
S = setover in inches.
T = taper per foot in inches.
L = length of piece in inches.

Taper Attachment Method of Taper Turning

The most economical way to produce a number of pieces of duplicate taper, especially where the length of pieces varies, is by the use of the taper attachment. It has been shown by the formula for the amount of tailstock setover, that the setover varies as the length of work, even when turning a taper of the same degree. It can be seen therefore, that a taper attachment is almost indispensable for taper work. The advantage of a taper attachment will readily be seen when it is necessary to turn a male and female taper to fit each other as it will not be necessary to change the taper attachment setup to machine the tapers. It is only necessary to turn the male taper, then change from the turning to the boring tool in the tool post, and put on the chuck or other accessory to hold the part in which the female taper is to be reproduced. The point is that the sensitive part of the setup, the setting of the taper attachment, is not disturbed.

A. Clamp regular rough turning tool bit in tool holder and set cutting edge exactly on center of work. This is important, since if the tool was above or below center, the angle of the cut would not be the same as the setting of the taper attachment.

B. Mount work between centers in lathe as in straight turning.

C. Move carriage so taper attachment shoe is in center of guide bar, etc., as illustrated and described on page 42.

D. Any depth adjustment of tool is made by means of compound rest top slide screw.

E. Run carriage toward tailstock until tool is about ½″ to 1″ past end of work. Next, run carriage back in direction of cut by hand until tool is just clearing end of work. Set tool for sufficient depth of cut to cut taper about ⅓ of finished taper length. Try in taper gauge. Turn down until taper centers about ⅓ of length into taper gauge.

F. Take light skim cut over surface just turned, about .005″ depth to give smooth surface to test taper.

G. Test taper just turned in taper test sleeve or standard female taper gauge. Try to rock taper piece in taper gauge. If taper is too steep, the piece will be tight at the big end of gauge hole and will rock in the back. If the taper is too slight the work will be tight at the back of the hole and will rock at the big end of hole.

H. Loosen clamp No. 158 (page 42), and adjust taper attachment to correct error in taper of work. Tighten clamp No. 158 and proceed to make another cut of just sufficient depth to turn surface for trial. Take a second skim cut to be sure surface tested is cut by taper attachment without spring.

Test as above until nearly correct and then put chalk lines on work lengthwise in four positions equally spaced. Twist gauge on work and see whether chalk marks are rubbed off evenly over entire length. If marks are rubbed off more on small end of taper, set taper attachment for more taper, and vice versa.

I. After work is proved to be of correct taper, rough work down until tapered piece goes into gauge all but ¾″.

J. Put in finishing tool and take cut 8 thousandths deep to leave taper stick out from gauge ⅜″ after cut. This ⅜″ is left to provide grinding stock.

If turning and polishing to size take 8 thousandths deep cut as before, use fine feed and check taper by using Prussian Blue instead of chalk in rub test as described in "H". If taper is correct, take cut of about 6 thousandths for finishing cut and try gauge. Gauge should fit about $\frac{1}{16}$″ to ⅛″ away from final position before turning is completed. The last $\frac{1}{16}$″ or ⅛″ is left for filing and polishing which should be done slowly and carefully to insure a good fit in the gauge.

While filing and polishing try work in gauge and use rub test with Prussian Blue to insure proper taper and fit.

A cut of 8 thousandths will allow the gauge to go approximately ⅜″ farther on the work on either Jarno or Morse taper and approximately $\frac{5}{16}$″ on a Brown & Sharpe taper.

The turning cut must be as straight and smooth as possible since the less filing and polishing necessary the more accurate the taper job will be.

Taper turning and boring may also be done by setting the compound rest at the desired angle and using the compound rest slide screw for the feed motion. This class of taper work is usually of the short steep taper type, as the compound rest may be set at any angle but has a relatively short travel.

Points To Observe When Taper Turning With Taper Attachment

Have gibs in taper attachment adjusted to eliminate unnecessary play in taper attachment slides.

Always run carriage past end of work when returning tool and then come back in direction of cut by hand so as to remove any back lash in slide and connecting mechanism.

Have tool cutting edge exactly on center of work.

Be sure to remove screw S before trying to use taper attachment as otherwise taper attachment and crossfeed screw would be bucking each other and something would break.

With Tailstock Setover Method

Have tool edge set exactly on center of work.

Have ends of work faced off square with centers.

Do not adjust centers too tight as center point in tailstock burns easily in this set-up.

Setting Lathe To Duplicate Taper Piece, With Taper Attachment

1. Clean centers of sample and mount in regular manner between centers.

2. If dial or pointer indicator is available, mount it in tool post so that actuating contact is on line with centers of the lathe.

3. Move carriage so indicator is at about the center of the trial taper and set taper attachment to bring shoe in center of slide; tighten bed clamp and remove crossfeed nut screw (S), (page 42).

4. Adjust taper slide to approximately the estimated or measured angle of sample, clamp the slide and the shoe and try indicator on taper surface by moving carriage about an inch or so . . . when error is in range of full scale reading on indicator, move carriage until indicator is at one end . . . set indicator at zero and run to the other end of taper, noting error.

5. Loosen clamp and taper slide and move taper slide until indicator moves back one-half previously noted error. Repeat test over full length and again correct taper attachment until error is small enough to be within limits of accuracy desired.

6. Mount tool in tool post and proceed to rough and finish **work to same dimensions as sample.**

With Tailstock Setover Method

1. Measure large and small diameters of taper and measure length of taper between points measured **along axis of work.** Do not measure length along taper surface as this length is longer than the axial length.

Taper per foot = **difference in diameter at large and small end in inches x 12, divided by length of taper in inches.**

2. The next step is to prepare the work on which taper is to be turned. Center both ends and face ends square and to correct length. Measure length accurately for use in following formula to obtain setover.

$$\text{Setover in inches} = \frac{\frac{1}{2} \text{ of required taper per ft. x length in inches.}}{12}$$

3. To measure the setover of the tailstock two methods may be used.

Set indicator in tool post and while tailstock is still in line with headstock, bring contact point of indicator up to barrel of tailstock spindle which should be extended about 2″ for convenience.

With indicator set at zero in this position, set index dial on cross feed screw also at zero, and then move cross slide away from tailstock spindle amount of offset as above computed.

Loosen tailstock clamp bolts and move tailstock top slide toward indicator by means of setover screws until indicator again reads zero. Reclamp tailstock and proceed with turning work to same measurements as sample.

If no indicator is available, mount a tool holder in the tool post, wrong end to, and bring the blunt end up to the tailstock spindle with a piece of tissue paper between them. Make light contact until tissue paper can be pulled between tool holder and tailstock spindle with slight but perceptible tension.

Move cross slide away from tailstock spindle more than amount of setover as before, then return to correct setting and bring tailstock spindle up to tool holder by means of setover screws until tissue paper has same feel as in last setting.

When setting cross slide to zero, be sure cross slide is going toward center of lathe so pressure of contact between tailstock spindle and tool holder backs up against cross feed screw without slack.

When moving cross slide out for setting, move more than necessary and then return towards centerline for proper setting so as to again keep pressure of test solidly against cross feed screw.

When tailstock is properly offset, proceed as in regular taper turning, machining down to dimensions of sample.

Tools should be ground and set approximately the same for boring as for turning, as described in the section devoted to tools and tool setting, pages 63 and 64.

Boring Taper Holes

The procedure used in boring taper holes follows closely the methods of straight hole boring.

If a taper attachment is used, the only variation is in actually connecting the taper attachment to the cross slide and setting the attachment for the proper taper. In setting the taper attachment, the same precautions should be taken as outlined in section devoted to external taper turning.

The tool point should be set exactly on center, because if this setting is not made the taper hole produced will be incorrect.

In many instances it will give a smoother finish if the work is run backwards and the tool point inverted. The reaction to the cut is thus downward and a smoother finish results.

If a lathe with a taper attachment is not available, the taper may be secured by swiveling the compound rest around to the proper angle and feeding the tool by hand with the compound rest top slide screw adjustment.

With this method of taper boring it is, of course, impossible to use the automatic feed to tool. However, by feeding slowly and uniformly by hand a creditable finish will be secured.

Tapers of any angle may be bored with the use of the compound rest and as long as the length of cut is less than the length of travel of compound rest slide there will be no difficulty. On surfaces larger than can be made in one cut, two cuts may be made if care is used in matching them up.

Taper parts like conical valve seats, dies with clearances, etc., are readily machined by means of this arrangement.

In respects other than noted in the foregoing, the regular boring procedure given in another section should be followed.

Boring with a Bar Between Centers

The boring described in other sections has been devoted to the types where the tool was mounted in a bar either held in the tool post or on the spindle of the lathe as illustrated on page 76.

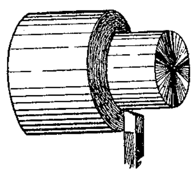
A third variation is possible. The work may be mounted on the carriage or cross slide and the hole bored out by a tool bit mounted in a boring bar held between centers of the lathe. This method is very useful when boring long holes or where the piece is mounted easier by bolting down flat than by holding in a rotary chuck.

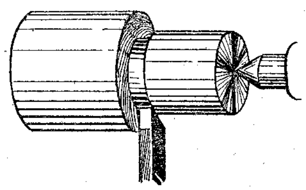
The tool bit may be held in any practicable manner, such as clamping by set screws, by screw nuts or by taper clamping plugs. Bars of this kind are easily made and present no difficulty, since all that is necessary is to take a bar as large as can be passed through rough hole with ⅛″ clearance—long enough to suit width of work plus travel of work for cut. Center this bar on both ends and place between centers with work in position at start of cut. Mark location for cutting bit, remove bar, drill for bit and clamp screw and bar is ready for use. Drive boring bar with lathe dog.

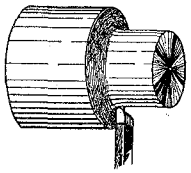
Shoulder and Radius Forming

The corners in turned work are at various times finished square, necked, either square or round, or with a small radius. Also with square or radius tool at an angle of 45° to allow clearance for grinding both diameter and face.

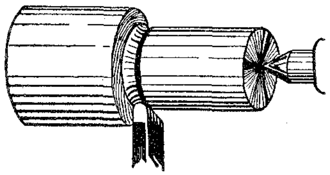
The square corner is the simplest type and is used where the piece is not subject to excess stress at the corner section. The necked corner is used as above, also where grinding allowance is left in turning. The undercut neck prevents undue wear on the corner of the grinding wheel. This design is weaker than the square corner due to the undercut.







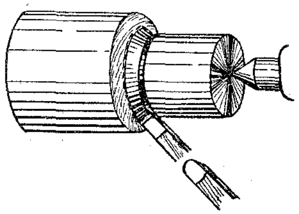
From the design standpoint, the rounded fillet corner is the best due to its strength. It is more costly in production, however, and so is used only in certain classes of work (where vibration is present) to prevent crystallization of steel at sharp corners.



For square corner work, the tool is sharpened as shown and the corner roughed out to within about $\frac{1}{64}$ or less before the final finishing cut is taken on the straight diameter. The tool is then adjusted, as explained under straight turning, to secure the proper finished diameter of work, power feed engaged and cut taken to within about $\frac{1}{64}$ of shoulder. At this point disengage power feed and read cross dial—feed corner tool into shoulder the approximate amount by hand—lock carriage clamp—and feed tool out by hand cross feed handle. Check length of shoulder, and if too long, run tool in to micrometer dial reading noted above. Loosen carriage clamp, advance tool to side into shoulder for next cut—reclamp carriage and feed outward with hand cross feed. Repeat until dimension length of shoulder is obtained. The tool used in the above is a side and front cutting corner tool.

A quick method of squaring the corner is to rough to within $\frac{1}{64}$ as before. Finish turn to rough shoulder with round nose turning tool. Put in front turning corner tool as illustrated. Run in until just barely skimming finish turned diameter. Note cross feed dial reading. Back away from work and move carriage so point of tool is in proper position to cut shoulder the right length. Lock carriage in this position and feed tool in with slow hand feed until cross feed dial reading is again reached. Release carriage clamp and feed carriage to right by hand until cut of corner tool and turned surface merge.

The side cutting corner tool fed outward produces the smoothest finish, but is not as fast in removing stock and finishing length to size as the front cutting corner tool, which leaves a finish good enough for most jobs.



When roughing and finishing a diameter which ends against a round fillet corner, it is advisable to sharpen the tool bit to approximately the radius of the fillet on the cutting edge. The final operation of forming the fillet is then easily accomplished by using a tool

with radius ground to a fillet gauge corresponding to the fillet to be produced.

With a tool set at a 45 deg. angle, the undercut allows clearance for the wheel in grinding the diameter and the face of the larger diameter.

The fillet forming tool is mounted in the tool post and the tool fed in until a very light skim is taken off the turned diameter. The tool is then fed by hand longitudinal feed till cutting a slight amount off face of shoulder. The tool is next fed by hand after clamping the carriage and the length of shoulder checked. If shoulder is large, repeat above facing operation until shoulder is reduced to correct length.

When forming a fillet in steel, it is advisable to lubricate the work with lard oil for a very smooth finish. Proper height of tool edge varies from exactly on center of work to as much as $\frac{1}{8}''$ above center on large work, depending on job and material. This height is best found by trial as no set rule governs every case.

When special forms or beads are required in turned work it is usually advisable, not to say necessary, to grind a tool to the proper shape and form the work by advancing tool straight in to work.

In such straight forming the tool edge is set exactly on center so as to produce the correct contour of the finish formed surface. The tool should be ground and stoned to a smooth finish as any marks in the tool will be reproduced in the work.

A form tool for brass or cast iron should have a flat top, while one for steel should have a slight lip around the contour of the cutting edge to enable cutting and not tearing the material.

Form tools should be kept as narrow as possible, since a wide form tool is much more prone to chatter than a narrow one.

Filing in the Lathe

No matter how much care is used in turning, it is usually impossible to secure a finish smooth and polished enough to be used directly in service.

It is unquestionably better to grind the finished surface whenever possible but many times a grinding machine is not available.

Under such circumstances work is usually filed to size and polished.

Procedure

1. Clean centers and see that there are no burrs around center holes.

2. Put center lubricant in tailstock center.

3. Put dog on work, protecting surface under clamp screw with small piece of copper or brass.

4. Place work on headstock center and run up tail center with right hand until tight. Loosen tail center slightly and start lathe, adjusting center to be just loose enough to allow dog to click in face plate but not loose enough to rattle or for work to have end shake.

5. Shift gears in headstock to give about twice finish turning speed on surface to be filed.

6. Select twelve-inch mill file and be sure handle is properly driven on tang. If file is clean, it is ready for use on brass, cast iron, etc., but for use on steel it should be rubbed with chalk until evenly covered. (The chalk prevents the steel filings clogging the teeth.)

This is Important

7. Be sure your shirt sleeve is rolled up above elbow or jumper is buttoned tightly around wrist so that no loose edges fly around to get caught in dog or work.

8. Start lathe and file work with slow even strokes, lapping the strokes from side to side.

When filing, use a long slow forward stroke and press firmly and evenly on the revolving piece being filed.

Relieve pressure on return stroke.

9. Stop lathe and try diameter with micrometers frequently until whole surface is filed as straight and smooth as possible, and leave about 5 to 8 ten-thousandths (.0005 to .0008) of an inch for polishing.

Use file card to keep file teeth clean.

Note—A file card is a short bristled wire brush which is brushed along the valleys of teeth to remove clogging material. A small metal pick is usually found in the handle. This pick is used to remove chips of metal which are stuck in teeth.

10. Polishing work.

After filing is completed the finished surface should be polished.

On work that is well balanced, set change levers to give highest speed. On unbalanced work run at highest possible speed without causing undue vibration.

On straight shafts, etc., 1″ or under in diameter, work should be polished in speed lathe if possible.

Use strip of emery or carborundum paper of fine grade and press against work, moving abrasive cloth from side to side to cross lines and bring work to a rough polish and to avoid cutting rings in work.

Use **very fine** abrasive cloth or a wornout piece of **fine** grade and use oil on work to bring to final polish. Use crocus paper for very fine finish polishing.

If a piece of wood can be pivoted on the tool rest and used to press abrasive cloth on work a quicker and higher-lustred job is secured due to the higher pressure possible.

When polishing, check size and straightness frequently with mike to be sure dimensions are correct on finished piece.

The best polishing requirements are:

High speed of work;

Fine grade abrasive cloth;

Use of oil on abrasive cloth (preferably lard oil);

Greatest possible pressure on work (a polishing lever is recommended).

Note—Polishing and filing heat the work. When measuring diameter with micrometer, either cool work by immersing in water or make allowance of one or two ten-thousandths for cooling of work.

It is recommended that less experienced operators cool work to room temperature before measuring.

Chasing Threads

An important function of an engine lathe is to chase threads. To cut a thread requires first, that the work rotate; and, secondly, that the tool advance along the axis of the work at a predetermined constant rate to cut the thread desired.

Threads are commonly designated in the English system by giving the number of complete revolutions of the thread per inch length of the screw. If, for instance, the chasing tool travels one inch along the screw while it rotates twice, there will be two revolutions of thread in one inch, commonly called 2 threads per inch.

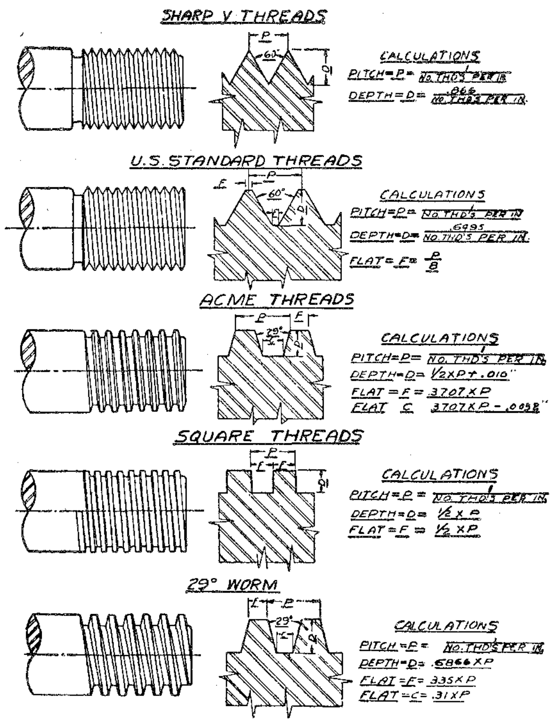
The lead-screw unit on a lathe is rotated by means of a gear train connection between the lead-screw and the spindle. A nut mounted in the apron engages the lead-screw to move the carriage. If, for instance, the lead-screw is 6 threads per inch, then for each revolution of the lead-screw the carriage is moved $\frac{1}{6}''$ along the bed. If the spindle and the lead-screw are geared so that the spindle rotates once while the lead-screw rotates once, the carriage will move $\frac{1}{6}''$ per revolution of the spindle, and the thread cut will be 6 threads per inch. If the lead-screw rotates twice as fast as the spindle, the carriage will move $2/6''$ or $\frac{1}{3}''$ per revolution of the spindle, and three threads per inch will be cut. If, on the other hand, the lead-screw rotates one-half as fast as the spindle, the carriage will move $\frac{1}{12}''$ per revolution of the spindle; thus twelve threads per inch will be cut.

It is thus seen that threads of any desirable pitch can be cut, if an appropriate connection between the spindle and the lead-screw is provided.

On standard change gear lathes, a quadrant and loose change gears are provided to cut various threads, and a chart placed on the gear cover indicates the proper gears and positions to set them to cut each thread within the range of the gears.

On the quick feed change lathes, the change gears are mounted in a gear box, with a gear train between the spindle and gear box, providing the various ratios needed to cut different threads. The changes are made merely by shifting levers on the box and the headstock. An index plate on the quick change box is plainly marked so that changes are quickly and accurately made.

Several different thread forms are used in practice and all may be cut in a lathe. Forms commonly used include sharp vee, U. S. S., Whitworth, Acme and 29-degree worm thread, illustrated on next page.



The setting of the change gears, in the quick change box, is the same, regardless of the form of thread to be cut. The only change is in the actual tool form used to cut the thread.

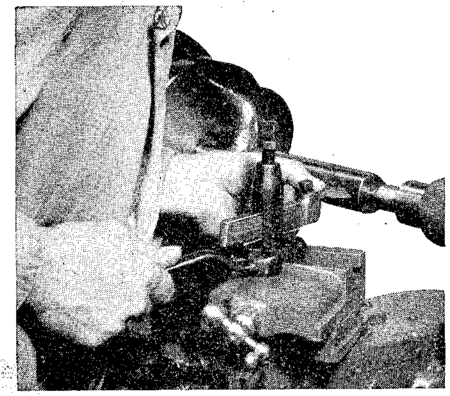
Tools used to cut threads are flat on top with no top rake. A slight lip may be ground in the side edges when cutting steel, but for other materials the top is usually absolutely flat. The front and side clearances on thread-cutting tools are very important. The side clearances must also be adjusted for the helix angle of the thread being cut. On threads 26 and finer, this helix angle is negligible, but on coarse threads the amount of the helix angle is quite appreciable and must be taken into consideration when grinding clearance for the lead on the side of the tool.

To measure the number of threads per inch a thread gauge is generally used, but it is not absolutely necessary. A thread gauge is merely a cluster of individual gauges, each one of which is cut with a thread tooth on a thin section of strip steel; so the teeth of only one gauge will properly mesh with the threads on the screw tested. Each gauge is labeled with the pitch of the teeth cut on it.

Another method is to lay a scale along the tops of the threads parallel to the axis of the screw with the end of the scale opposite the top point of a thread. Then, skipping the thread top directly below the end of the scale, count the number of tops until one falls directly below an inch mark on the scale. The number of thread tops thus counted, divided by the number of inches, gives the number of threads per inch. For example, suppose there were 27 thread tops under two inches of the scale (not counting the one under the end of the scale). Then $27 \div 2 = 13\frac{1}{2}$ threads per inch, or the thread is $\frac{1}{13\frac{1}{2}}$, or .074 of an inch pitch.

When cutting threads, it is necessary to set the tools at a right angle to the piece to be turned; that is, the axis of the thread tool should be exactly 90 degrees from the axis of the work. This is easily done by use of a thread-setting gauge shown below and on page 66, No. 142. Edges of the gauge are ground square with the male and female angles ground on the ends and the various notches in the sides.

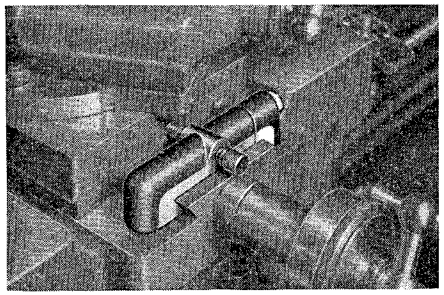
Hold the gauge against the diameter of the work, as illustrated, and adjust the tool until it fits in the notch accurately, thus insuring the proper setting of the tool square with the work. Next, set the cutting edge of the tool exactly on the dead center. The depth of the thread and the thread angle will not be cut correctly if the tool is set in any other position. When cutting threads on cast-iron or brass, no cutting lubricant is necessary, but on steel it should be used. A good quality of cutting oil should be applied to the tool, especially on the finish cuts; a smoother surface is thus obtained.



On threads of fine lead, about 30 and finer, the tool may be fed straight into the work in successive cuts. However, on coarser leads it is better to set the compound rest at one-half the included angle of the thread, and feed in along the side of the thread, so that the tool cuts on one side only during roughing. On the last two or three cuts the tool should be fed straight in to remove all lines caused by feeding along the side of the thread.

Since chasing requires a number of cuts and all must be in the same line of the cut of the thread, it is necessary either to keep the half-nut engaged on the leadscrew at all times and return the carriage by reversing the spindle rotation through the motor drive, or to use an indicator which meshes with the lead-screw and shows when the half-nut can be engaged so that the tool will cut along the same thread. The device used for this purpose is called a chasing dial or thread indicator, and consists of a worm wheel meshing with the leadscrew, and connected by a short shaft to the indicating dial. The dial is calibrated with four numbered lines and four others midway between them as shown on page 41. For even threads the half-nut may be closed when the index mark is opposite any line of the dial; for odd threads at any numbered line and for half threads at any even numbered line.

The advantage of the chasing dial is that the tool may be drawn back and the half-nut disengaged at the end of the cut, thereby permitting the quick hand return of the carriage to the starting point. When ready for the next cut, set the tool to the proper depth and engage the half-nut when the proper line on the chasing dial is opposite the index mark and take another cut across the thread. The threading stop shown below may be used in several ways to aid the operator when cutting threads. When the tool is set for the first chasing cut, set the stop and clamp it to the carriage dovetail. Place the stop screw through the clearance in clamping piece over dovetail slide and start the screw thread in the cross slide. Move the cross slide until the underside of the head of the screw is against the stop clamp and take the cut. At the end of the cut the tool must be backed off to clear the thread. This should be done with the cross feed screw. For the next cut, move the cross slide in to its previous position against the stop, then use the compound rest screw to move the tool in to the desired depth of cut. For the last few cuts, the stop screw should be run out (when the compound is set on an angle) and, reading the cross feed dial, the tool fed straight in to clean up both sides of the thread.



Another use of the chasing stop is to leave it set for the depth of the finished thread. Then on the succeeding pieces each successive cut can be fed in by the cross feed screw until the chasing stop prevents further motion. The screw will then be the same size as the previous screws turned with this setting.

The points just outlined are also true for taper and internal threading.

We would again point out that the top edge of the tool should always be set on the lathe center line and the proper side and front clearance must be allowed to clear the sides of the threads.

The following table conveys some idea of the number of cuts necessary to chase vee threads in common use:

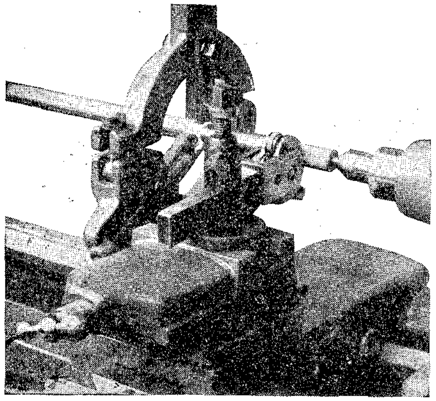
Thread-Cutting Data

No. of Threads per Inch	No. of Chasing Cuts
8	18
10	14
11	13
12	11
13	10
16	9
20	8

This table is based on .005″ per cut allowing an extra cut for finish which is the actual practice in our shop.

Knurling

Many pieces used for handles or control knobs require a rough but finished appearing surface so that a good grip may be obtained without impairing the appearance of the part. This effect is obtained by raising the surface of the piece in symmetrical or cross lines and it is called knurling. Knurling is a form of pressure indentation by a continuous process. The tool employed, as illustrated, consists of small rolls containing serrations in the periphery which squeeze the metal in the work piece to form a reproduction of the knurling rolls. When using the knurling tool, a slow speed is required. Adjust the tool to mark the work lightly. After it



is seen that the tool is working properly, increase the pressure on the rolls by means of the cross feed screw, liberally oil the surface to be knurled, engage the power feed, allowing the tool to move across the work until the leading edge of the roll is just flush with the other end of the work. Reverse the feed, increase the pressure on the rolls slightly and feed back to the starting position. Repeat this procedure until the indentation is deep enough to suit the purpose for which the part is to be used. To produce the desired result a few trials on a sample piece will show the amount of pressure necessary.

Cutting Speeds*

For efficient operation of a lathe, the proper surface speed of work being machined must be maintained. If the speed is too slow, the job takes longer than necessary, and often the work produced is unsatisfactory. On the other hand, if the speed is too great, the tool edge will be worn down too rapidly, and frequent grinding will be necessary, which is also wasteful. For ordinary production work the speed should be as great as the tool will stand without requiring sharpening more often than every two to three hours when cutting continuously.

APPROXIMATE CUTTING SPEEDS

Turning and Boring

MATERIAL	Roughing Cutting Speed, Feet per Minute	Finishing Cutting Speed, Feet per Minute	Chasing Cutting Speed, Feet per Minute
Cast-Iron	60	120	50
Mild Machine Steel	80	150	60
Alloy Steel*	50	90	40
Bronze	100	150	70
Brass	200	300	80
Aluminum	250	400	90

*Data for average alloy steel annealed.

When chasing threads on small diameters the limitation will be the ability of the operator to handle the lathe, rather than the cutting limit of the tool. We have found that 200 R. P. M. is practically the limit at which threads can be chased.

The table shown gives the approximate speeds which can be maintained with various materials for rough and finish cuts. The surface speed is found by multiplying the length of the periphery in feet by the revolutions per minute of the work. Thus the cutting speed for a 4″ diameter rotating at 60 R. P. M., will be $\frac{4 \times 3.1416}{12} \times 60$, or 62.83′ per minute. The formula for surface speed is diameter

*For Cemented Carbide Tools, see page 107.

in inches times "π" (3.1416) times the R. P. M. divided by twelve inches, giving the answer in feet per minute.

The cutting speeds possible are greatly affected by the use or absence of a suitable cutting fluid. Thus steel, which can be rough turned dry at 60′ per minute, can be rough turned at about 80′ or 90′ per minute when flooded with good cutting lubricant.

When roughing parts down to size, use the greatest depth of cut and feed per revolution that the work, the machine and the tool will stand at the highest practicable speed. On many pieces where tool failure is the limiting factor in the size of roughing cut, it is usually possible to reduce the speed slightly and increase the feed to a point where the metal removed is much greater with longer tool life. For example: Where the depth of cut is ¼″, the feed 20 thousandths of an inch per revolution and the speed 80 feet per minute. If the tool will not permit additional feed at this speed, it is usually possible to drop the speed to 60′ per minute and increase the feed to about 40 thousandths of an inch per revolution without having tool trouble.

In this case, the speed is reduced 25% but the feed increased 100%, so that the actual time required to complete the work is less.

On the finish turning operation, a very light cut is taken since most of the stock has been removed on the roughing cut. Due to requirements of the finish a fine feed can usually be used and still make it possible to run at a high surface speed. A 50% increase in speed over the roughing speed is commonly used. In particular cases the finishing speed may be twice the roughing speed. In any event, to secure the maximum speed in this operation, the work should be run as fast as the tool will reasonably stand up without excessive regrinding. A sharp tool should be used when finish turning. The tool should be rewhetted to a keen edge if the same tool is used for roughing and finishing.

Chatter

Briefly, chatter is vibration in either the tool or the work, producing a finished work surface that has a grooved or lined finish instead of the smooth surface that is to be expected. The vibration is set up by a weakness in the work, work support, tool or tool support, and is about the most elusive thing to find in the entire field of machine work. As a general rule, strengthening the various parts of the tool support train will help, also supporting the work by a steady or follow rest.

Possibly the fault may be in the machine adjustments. Gibs may be too loose; bearings may, after a long period of heavy service, be worn; the tool may be sharpened improperly, etc. If the machine is in perfect condition, the fault may be in the tool or tool setup. Grind the tool with a point or as near a point as the finish specified will permit; avoid a rounded leading edge on the tool. Reduce the overhang of the tool as much as possible and be sure that all the gib and bearing adjustments are properly made. See that the work receives proper support for the cut, and, above all, do not try to turn at a surface speed that is too high. Excess speed is probably the greatest cause of chatter, and the first thing an operator should do when chatter occurs is to reduce the speed.

On large thin sections such as cups or brake drums, a coiled spring stretched around the piece may dampen the vibration sufficiently to prevent chatter. Often, packing the inside with a wood disc cut to fit the cup will permit a smooth finish to be obtained on the outside surface.

Lapping

Where holes are to be finished to an exact size or to a maximum straightness, it is advisable to leave the hole a few ten thousandths under size and remove this metal by lapping.

Lapping can be employed on both flat and cylindrical surfaces.

The procedure to be followed varies slightly, depending upon the reason for lapping; whether it is to finish a hole to an exact straight size, polish the surface for high finish, or merely to remove a bit of material from hardened metal.

When the requirement is to slightly enlarge a hole, a piece of carborundum cloth wrapped around a rotating rod held in the lathe chuck will provide the quickest but not the most accurate way.

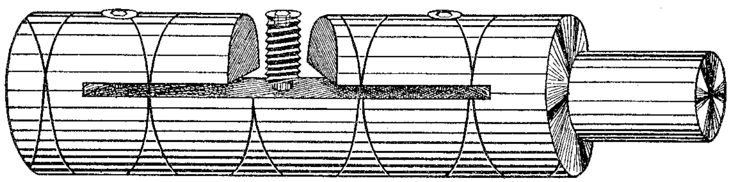
Laps are made in both the solid and the expanding types. The expanding type is preferred to the solid type because it can be expanded, trued up, and recharged when the cutting surface of the lap is worn down.

A lap may be charged with cutting grit so that it will cut along its entire length or it may be charged to cut in one spot only.

It is not advisable to crowd a lap, since the process is only used as a finishing operation and not to remove a large amount of stock.

The maximum stock allowance for lapping should not exceed one to two thousandths of an inch and is preferably about three to five ten-thousandths of an inch.

A good serviceable lap for general use is illustrated on this page. The construction of such a lap is simple. Turn the cast-iron piece all over to rough size—turn handle end to size—turn lap surface to standard size minus one thousandth of an inch or two thousandths if the holes you lap are apt to come that small. With work turning at a slow spindle speed, turn slight line in lap surface at very coarse lead about one inch per revolution. This feed can be done by hand as there is nothing particular about it. Make two such cuts, one right-hand lead, the other left hand. The purpose of these grooves is simply to act as grit and oil distributor troughs. They should be about $\frac{1}{64}''$ to $\frac{1}{32}''$ wide and $\frac{1}{64}''$ to $\frac{1}{32}''$ deep, depending on diameter of the lap. Cut with a sharp vee tool.



The lap surface is next split with a milling cutter within about ¼″ to ⅜″ from the end of the lapped surface as shown above. It is then drilled and tapped at right angles to the split for the expansion screws. This slit through lap permits expansion by means of set screws in one-half acting on other half through the slot as shown.

The best grits to employ are Arkansas grit, of the correct grain for the work to be tapped, and Bon Ami cleaning powder. These grits are not as fast cutting as some but produce good accurate work with a high finish. The grit used is mixed with machine oil to a light paste consistency and applied to the lap evenly.

The lap is then pushed into the hole with a combination push and twist drill going in and pull and twist in opposite direction coming out. The lap is rotated slightly in the hole after every complete stroke to avoid lapping too much in a position which might keep the hole from being lapped cylindrically round.

Sufficient take-up should always be given to the adjusting screw to insure the lap fitting the hole snugly. If this is not done the hole may be lapped bell mouthed.

The above-mentioned grits and procedure may be used when lapping holes in steel, either hard or soft, cast iron and bronze, and are advantageous in that the work is not charged with the cutting grit as may occur when emery or carborundum is employed.

On some very hard materials the lap may be made of copper and diamond grit employed. In this case, however, the grit paste is rolled into the surface of the lap with a roller or by rotating the lap on a flat plate smeared with the grit paste as one handles a rolling pin. The excess loose grit is then washed off the lap with gasoline or turpentine and regular lapping procedure followed. The amount of cutting done and the finish left are dependent on the size grit used and we would suggest looking up this subject in any good toolmaker's book which will give complete information on diamond lapping.

Lead taps are used for rough lapping where the main consideration is to remove material without extreme requirements as to hole accuracy or bell mouth.

A flat lapping disc is also useful on a lathe.

The rough finish type is merely a plate with a taper shank which fits in the headstock spindle on the face of which a disc of emery cloth is attached with beeswax.

A fine type of lap consists of the same type of plate provided with a lead or copper face which can be charged with suitable abrasive grit. Flat valve seats and other parts of this nature are thus easily provided with a proper flat surface of high finish.

It should be remembered that lapping is a sensitive and essentially slow operation. Lapping is not primarily a metal removing but a finishing or polishing operation.

All work should be finished as smooth as possible with the cutting tool or in the grinding operation so the lapping operation will have as little metal as possible to remove.

Use as little lapping compound as possible since a thin layer will cut according to the pressure or contact of lap while a thick layer may cut even over a low spot in the hole.

Metric Threads

Metric threads can be cut through the quick change gear box by the addition of compound gears between the drive gear on the head and the gear on the feed box.

Size Lathe	Compound Gears	Range of Threads
10″	127T— 90T	¼ M. M. Pitch to 3¾ M. M. Pitch
13″—15″	127T—120T	⅛ M. M. Pitch to 7 M. M. Pitch
17″—19″	127T—120T	$\frac{3}{16}$ M. M. Pitch to 16 M. M. Pitch
21″—24″	127T—120T	203 M. M. Pitch to 24 M. M. Pitch

Rules for Figuring Tapers

Given	To Find	Rule
The taper per foot........	The taper per inch.........	Divide the taper per foot by 12.
The taper per inch........	The taper per foot.........	Multiply the taper per inch by 12.
End diameters and length of taper in inches.......	The taper per foot.........	Subtract small diameter from large; divide by length of taper, and multiply quotient by 12.
Large diameter and length of taper in inches and taper per foot..........	Diameter at small end in inches..................	Divide taper per foot by 12; multiply by length of taper, and subtract result from lar diameter.
Small diameter and length of taper in inches, and taper per foot.........	Diameter at large end in inches..................	Divide taper per foot by 12; multiply by length of taper; and add result to small diameter.
The taper per foot and two diameters in inches.....	Distance between two given diameters in inches......	Subtract small diameter from large; divide remainder by taper per foot, and multiply quotient by 12.
The taper per foot........	Amount of taper in a certain length given in inches....	Divide taper per foot by 12; multiply by given length of tapered part.

Diameters of Numbered and Lettered Drills

Drill No.	Diameter Inches	Drill No.	Diameter Inches	Drill No.	Diameter Inches	Drill No.	Diameter Inches
1	.2280	28	.1405	55	.0520	A	.2340
2	.2210	29	.1360	56	.0465	B	.2380
3	.2130	30	.1285	57	.0430	C	.2420
4	.2090	31	.1200	58	.0420	D	.2460
5	.2055	32	.1160	59	.0410	E	.2500
6	.2040	33	.1130	60	.0400	F	.2570
7	.2010	34	.1110	61	.0390	G	.2610
8	.1990	35	.1100	62	.0380	H	.2660
9	.1960	36	.1065	63	.0370	I	.2720
10	.1935	37	.1040	64	.0360	J	.2770
11	.1910	38	.1015	65	.0350	K	.2810
12	.1890	39	.0995	66	.0340	L	.2900
13	.1850	40	.0980	67	.0320	M	.2950
14	.1820	41	.0960	68	.0310	N	.3020
15	.1800	42	.0935	69	.0292	O	.3160
16	.1770	43	.0890	70	.0280	P	.3230
17	.1730	44	.0860	71	.0260	Q	.3320
18	.1695	45	.0820	72	.0250	R	.3390
19	.1660	46	.0810	73	.0240	S	.3480
20	.1610	47	.0785	74	.0225	T	.3580
21	.1590	48	.0760	75	.0210	U	.3680
22	.1570	49	.0730	76	.0200	V	.3770
23	.1540	50	.0700	77	.0180	W	.3860
24	.1520	51	.0670	78	.0160	X	.3970
25	.1495	52	.0635	79	.0145	Y	.4040
26	.1470	53	.0595	80	.0135	Z	.4130
27	.1440	54	.0550				

Table of Decimal Equivalents

1/64″ to 1″ in 64ths

Fraction	Decimal Equivalent	Fraction	Decimal Equivalent	Fraction	Decimal Equivalent
1/64	0.015 625	11/32	0.343 75	43/64	0.671 875
1/32	0.031 25	23/64	0.359 375	11/16	0.687 5
3/64	0.046 875	3/8	0.375	45/64	0.703 125
1/16	0.062 5	25/64	0.390 625	23/32	0.718 75
5/64	0.078 125	13/32	0.406 25	47/64	0.734 375
3/32	0.093 75	27/64	0.421 875	3/4	0.750
7/64	0.109 375	7/16	0.437 5	49/64	0.765 625
1/8	0.125	29/64	0.453 125	25/32	0.781 25
9/64	0.140 625	15/32	0.468 75	51/64	0.796 875
5/32	0.156 25	31/64	0.484 375	13/16	0.812 5
11/64	0.171 875	1/2	0.500	53/64	0.828 125
3/16	0.187 5	33/64	0.515 625	27/32	0.843 75
13/64	0.203 125	17/32	0.531 25	55/64	0.859 375
7/32	0.218 75	35/64	0.546 875	7/8	0.875
15/64	0.234 375	9/16	0.562 5	57/64	0.890 625
1/4	0.250	37/64	0.578 125	29/32	0.906 25
17/64	0.265 625	19/32	0.593 75	59/64	0.921 875
9/32	0.281 25	39/64	0.609 375	15/16	0.937 5
19/64	0.296 875	5/8	0.625	61/64	0.953 125
5/16	0.312 5	41/64	0.640 625	31/32	0.968 75
21/64	0.328 125	21/32	0.656 25	63/64	0.984 375

Millimeters into Inches

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

10 Millimeters = 1 Centimeter
10 Centimeters = 1 Decimeter
10 Decimeters = 1 Meter

1 Kilometer = .6214 mile
1 Meter = 39.37 inches; 3.2808 feet; 1.0936 yard
1 Centimeter = .3937 inch
1 Millimeter = .03937 inch

1 Centimeter = .3937 inch
1 Decimeter = 3.937 inches
1 Meter = 39.37 inches

1 Mile = 1.609 kilometers
1 Yard = .9144 meter
1 Foot = .3048 meter
1 Foot = 304.8 millimeters
1 Inch = 2.54 centimeters
1 Inch = 25.4 millimeters

Limits for Turning and Grinding—The limits given in the table below are recommended for use in the manufacture of machine parts, to produce satisfactory commercial work. These limits should only be followed under ordinary conditions. For special cases, it may be necessary to increase or decrease the limits given in the table. The allowance to be used when rough turning parts to be ground varies from 0.010 to 0.030 inch; that is, a part to be ground to a diameter of 1 inch would be rough turned in the lathe to a diameter of from 1.010 to 1.015 inch, while a 3-inch shaft may have an allowance of from 0.015 to 0.025 inch. The allowance depends largely on the class of work.

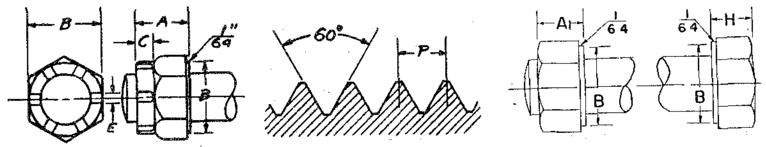
Allowances for Fits

Grinding Limits for Cylindrical Parts

(+ Designates larger than nominal size; — Smaller than nominal size.)

Diameter, Inches	Limits, Inches	Diameter, Inches	Limits, Inches
Running Fits—Ordinary Speed		**Driving Fits—Ordinary**	
Up to ½	— 0.00025 to — 0.00075	Up to ½	+ 0.00075 to + 0.0015
½ to 1	— 0.00075 to — 0.0015	½ to 1	+ 0.001 to + 0.002
1 to 2	— 0.0015 to — 0.0025	1 to 2	+ 0.002 to + 0.003
2 to 3½	— 0.0025 to — 0.0035	2 to 3½	+ 0.003 to + 0.004
3½ to 6	— 0.0035 to — 0.005	3½ to 6	+ 0.004 to + 0.005
Running Fits—High-Speed, Heavy Pressure and Rocker Shafts		**Forced Fits**	
Up to ½	— 0.0005 to — 0.001	Up to ½	+ 0.00025 to + 0.0005
½ to 1	— 0.001 to — 0.002	½ to 1	+ 0.0015 to + 0.0025
1 to 2	— 0.002 to — 0.003	1 to 2	+ 0.0025 to + 0.004
2 to 3½	— 0.003 to — 0.0045	2 to 3½	+ 0.004 to + 0.006
3½ to 6	— 0.0045 to — 0.0065	3½ to 6	+ 0.006 to + 0.009
Sliding Fits		**Driving Fits—For such Pieces as are Required to be Readily Taken Apart**	
Up to ½	— 0.00025 to — 0.0005	Up to ½	0 to + 0.00025
½ to 1	— 0.0005 to — 0.001	½ to 1	+ 0.00025 to + 0.0005
1 to 2	— 0.001 to — 0.002	1 to 2	+ 0.0005 to + 0.00075
2 to 3½	— 0.002 to — 0.0035	2 to 3½	+ 0.00075 to + 0.001
3½ to 6	— 0.003 to — 0.005	3½ to 6	+ 0.001 to + 0.0015

S. A. E. Standard Screws and Nuts



D (in table) = Diameter of screw.
N = Number of threads per inch.
B applies to all nuts and screw heads.
d = Diameter of cotter pin.

S = Tap drill size.
D x 2 + ¼" = Length of thread.
P/8 = Width of flat.

D	¼	5/16	⅜	7/16	½	9/16	⅝	¾	⅞	1	1⅛	1¼	1⅜	1½
N	28	24	24	20	20	18	18	16	14	14	12	12	12	12
A	9/32	21/64	13/32	29/64	9/16	39/64	23/32	13/16	29/32	1	1 5/32	1¼	1⅜	1½
A1	7/32	17/64	21/64	⅜	7/16	31/64	35/64	21/32	49/64	⅞	63/64	1 3/32	1 13/64	1 5/16
B	7/16	½	9/16	⅝	¾	⅞	15/16	1 1/16	1¼	1 7/16	1⅝	1 13/16	2	2 3/16
C	3/32	3/32	⅛	5/32	5/32	3/16	7/32	¼	¼	9/32	11/32	⅜	⅜	7/16
E	5/64	3/32	⅛	⅛	5/32	5/32	3/16	3/16	3/16	¼	¼	5/16	5/16	⅜
H	3/16	15/64	9/32	21/64	⅜	27/64	[illegible]	9/16	21/32	¾	27/32	15/16	1 1/32	1⅛
d	1/16	1/16	3/32	3/32	⅛	⅛	5/32	5/32	5/32	7/32	7/32	¼	¼	5/16
S	7/32	17/64	21/64	⅜	7/16	½	35/64	43/64	25/32	29/32	1 3/64	1 9/64	1 17/64	1 25/64

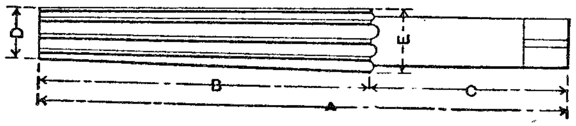
American Standard Threads, Regular Bolt Head and Regular Nuts

SEMI-FINISHED HEXAGON

The tap drill diameters in the Table provide for a 75% full depth of thread engagement. For complete bolt head, nut and thread data, see American Standard Data Bulletins.

Diameter	No. of Threads per Inch	Diameter at Root of Thread	Diameter of Tap Drill	Area in Sq. In. — Of Bolt	Area in Sq. In. — At Root of Thrd.	Tensile Strength at Stress of 6,000 Pounds per Square Inch	Dimensions of Nuts and Bolt Heads — Nuts — Across Flats	Nuts — Across Corners	Nuts — Height	Bolts — Height	Bolts — Across Flats	Bolts — Across Corners
¼	20	0.185	13/64	0.049	0.026	160	7/16	0.485	13/64	5/32	⅜	0.413
5/16	18	0.240	¼	0.076	0.045	270	9/16	0.624	¼	3/16	½	0.552
⅜	16	0.294	5/16	0.110	0.068	410	⅝	0.691	5/16	15/64	9/16	0.620
7/16	14	0.345	23/64	0.150	0.093	560	¾	0.830	23/64	9/32	⅝	0.687
½	13	0.400	27/64	0.196	0.126	760	13/16	0.898	27/64	19/64	¾	0.826
9/16	12	0.454	15/32	0.248	0.162	1,000	⅞	0.966	31/64	11/32	⅞	0.966
⅝	11	0.507	17/32	0.307	0.202	1,210	1	1.104	17/32	25/64	15/16	1.033
¾	10	0.620	41/64	0.442	0.302	1,810	1⅛	1.240	41/64	15/32	1⅛	1.240
⅞	9	0.731	¾	0.601	0.419	2,520	1 5/16	1.447	¾	9/16	1 5/16	1.447
1	8	0.838	55/64	0.785	0.551	3,300	1½	1.653	55/64	19/32	1½	1.653
1⅛	7	0.939	31/32	0.994	0.694	4,160	1 11/16	1.859	31/32	11/16	1 11/16	1.859
1¼	7	1.064	1 [illegible]	1.227	0.893	5,350	1⅞	2.066	1 1/16	25/32	1⅞	2.066
1⅜	6	1.158	1 7/32	1.485	1.057	6,340	2 1/16	2.273	1 11/64	27/32	2 1/16	2.273
1½	6	1.283	1 11/32	1.767	1.295	7,770	2¼	2.480	1 9/32	15/16	2¼	2.480

Standard Taper Pin Reamers*

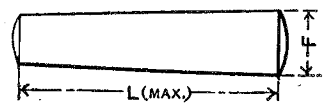


Taper, ¼ inch per foot.

No. of Taper Pin Reamer	Diameter at Large End of Reamer E	Diameter at Small End of Reamer D	Total Length of Reamer A	Length of Cutting Edges B	Length of Shank C	No. of Flutes
00000	0.0984	0.075	1-5/8	1-1/8	1/2	4
0000	0.1140	0.088	1-3/4	1-1/4	1/2	4
000	0.1296	0.101	2	1-3/8	5/8	4
00	0.1452	0.114	2-1/4	1-1/2	3/4	4
0	0.1608	0.127	2-3/8	1-5/8	3/4	6
1	0.1824	0.146	2-1/2	1-3/4	3/4	6
2	0.2036	0.162	3	2	1	6
3	0.2298	0.183	3-1/2	2-1/4	1-1/4	6
4	0.2600	0.208	4	2-1/2	1-1/2	6
5	0.3024	0.240	4-1/2	3	1-1/2	6
6	0.3544	0.279	5	3-5/8	1-1/2	6
7	0.4246	0.331	6	4-1/2	1-1/2	6
8	0.5072	0.398	6-3/4	5-1/4	1-1/2	8
9	0.6094	0.482	8	6-1/8	1-7/8	8
10	0.7266	0.581	9	7	2	8
11	0.8776	0.706	11-1/4	8-1/4	3	8
12	1.050	0.842	13-3/8	10	3-3/8	....
13	1.2586	1.009	16	12	4	....
14	1.5412	1.250	18-1/4	14	4-1/4	....

*Adopted by manufacturers of reamers and taper pins.

Standard Taper Pins



Taper, ¼ inch per foot

No. of Taper Pin	Diameter F at Large End of Pin	Approx. Fractional Size F	Maximum Length L
00000	0.094	3/32	3/4
0000	0.109	7/64	7/8
000	0.125	1/8	1
00	0.141	9/64	1-1/8
0	0.156	5/32	1-1/4
1	0.172	11/64	1-1/4
2	0.193	3/16	1-1/2
3	0.219	7/32	1-3/4
4	0.250	1/4	2
5	0.289	19/64	2-1/4
6	0.341	11/32	3
7	0.409	13/32	3-3/4
8	0.492	1/2	4-1/2
9	0.591	19/32	5-1/4
10	0.706	23/32	6
11	0.860	55/64	7-1/4
12	1.032	1-1/32	9
13	1.241	1-15/64	11
14	1.523	1-33/64	13

Cutting Speeds and Feeds for Cemented-Carbide Tools

Examples from Practice: Tantalum-carbide tools were used for materials marked with asterisks (*), Column 2. Tungsten carbide was used for all other examples. These speeds and feeds are intended as a general guide only. The rigidity of the machine and tool support, interrupted cuts and other factors result in wide variation in practice.

Machining Operation	Kinds of Material	Cutting Speed Ft. per Min.	Depth of cut, Inch	Rate of Feed Per Rev.
Turning	Cast Iron	250	1/4	0.070
	Cast Iron	210	3/8	0.062
	Cast Iron (1)	260	3/16	0.015
	Cast Iron (2)	150	1/4—3/4	0.050
	Semi-Steel (3)	280	1/8	0.012
	Semi-Steel	300	....	.....
	*Semi-Steel	225	....	0.028
	*Semi-Steel	286	5/32	0.015
	*Chilled C. I.	62	5/8	.....
	*S. A. E. 52100	185	1/8—5/16	0.020
	*Tool Steel 1.10%	140	1/2	0.020
	Silicon Steel	500	....	.....
	Carbon Steel, 0.90%	500	....	.....
	Bronze	400	....	0.020
	Bronze	425	3/16	0.024
	Bronze	550	1/8	0.031
	Brass Casting (4)	458	3/32	0.108
	Cast Aluminum	1000	....	.....
	Aluminum Alloy	570	1/8	0.031
Boring	Cast Iron	250	....	.....
	Brass	350	....	.....
	Aluminum	1500	....	.....

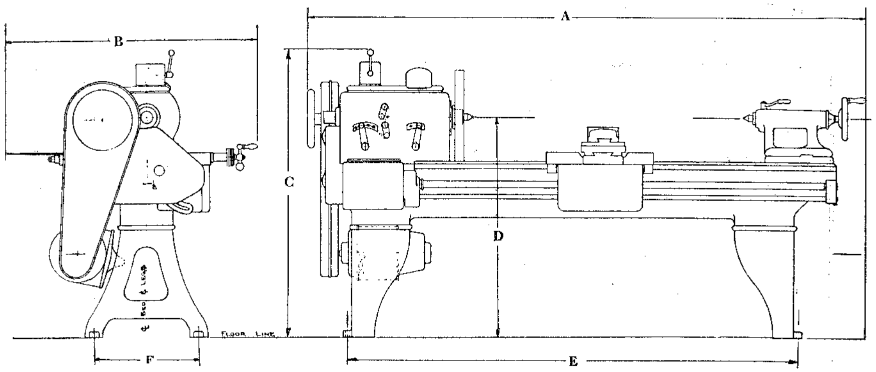
(1) Interrupted cut.

(2) Much higher speed possible for lighter cut.

(3) 9 to 17 hours between tool grindings.

(4) Six days between tool grindings.

Dimensions of Regal Lathes and Approximate Space Required



Size of Lathe, Inches	Center Distance*, Inches	Bed Length, Feet	FLOOR SPACE REQUIRED A Inches	FLOOR SPACE REQUIRED B With Taper Att., Inches	FLOOR SPACE REQUIRED B Without Taper Att., Inches	Total Height C, Inches	Height to Center D, Inches	APPROX. DISTANCE BETWEEN BOLT HOLES E, Inches	APPROX. DISTANCE BETWEEN BOLT HOLES F, Inches
FLOOR TYPE									
13	18	4′ 1″	58¼°	34⅜	30¾	54⅝	42½	44½	21½
15	18	4′ 3″	60¾°	35¼	32¼	54⅝	42½	46½	21
17	30	5′ 11″	82⅛°	46⅝	38½	53¾	42¾	64½	19¾
19	30	6′ 0″	84⅝°	48⅝	40½	55¾	42¾	64	20½
21	36	7′ 3″	107 7/16	56	45¼	53	42½	78¼	21
24	36	7′ 3″	107 7/16	56	45¼	56¼	44¼	78¼	21
BENCH TYPE									
13	18	4′ 1″	58¼	38	38	30⅞	18¾	46¾	9½

*Base Lengths—For longer beds, add additional length.
°With disc clutch and brake add for 13″ and 15″—5¾″ and 17″ and 19″—6⅝″.

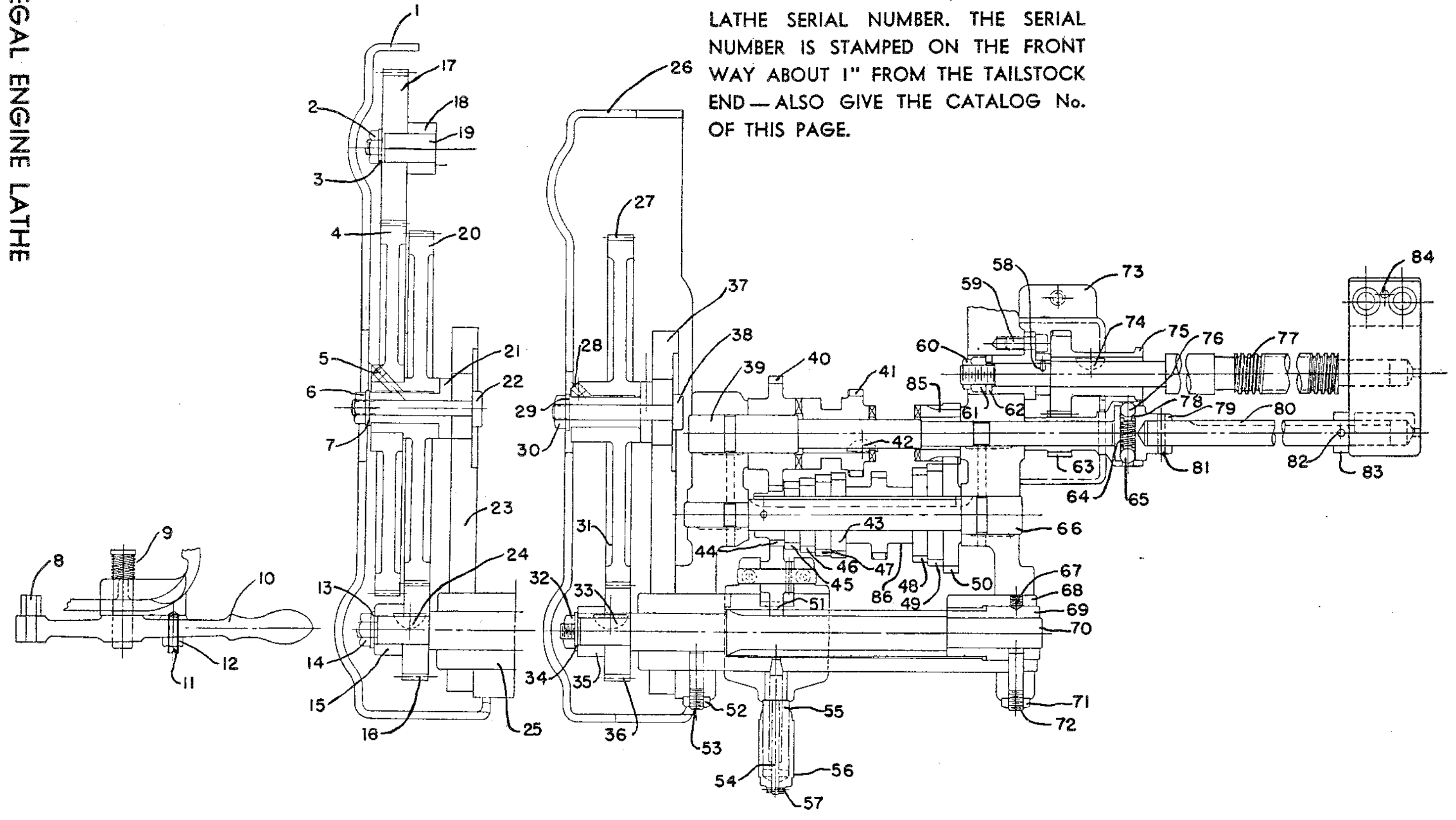
Repair Parts Section

When ordering repair parts it is IMPORTANT for *prompt service* to give the following information in your order:

1. Size of lathe.
2. Serial number.
3. Number of part as listed.
4. Quantity wanted.

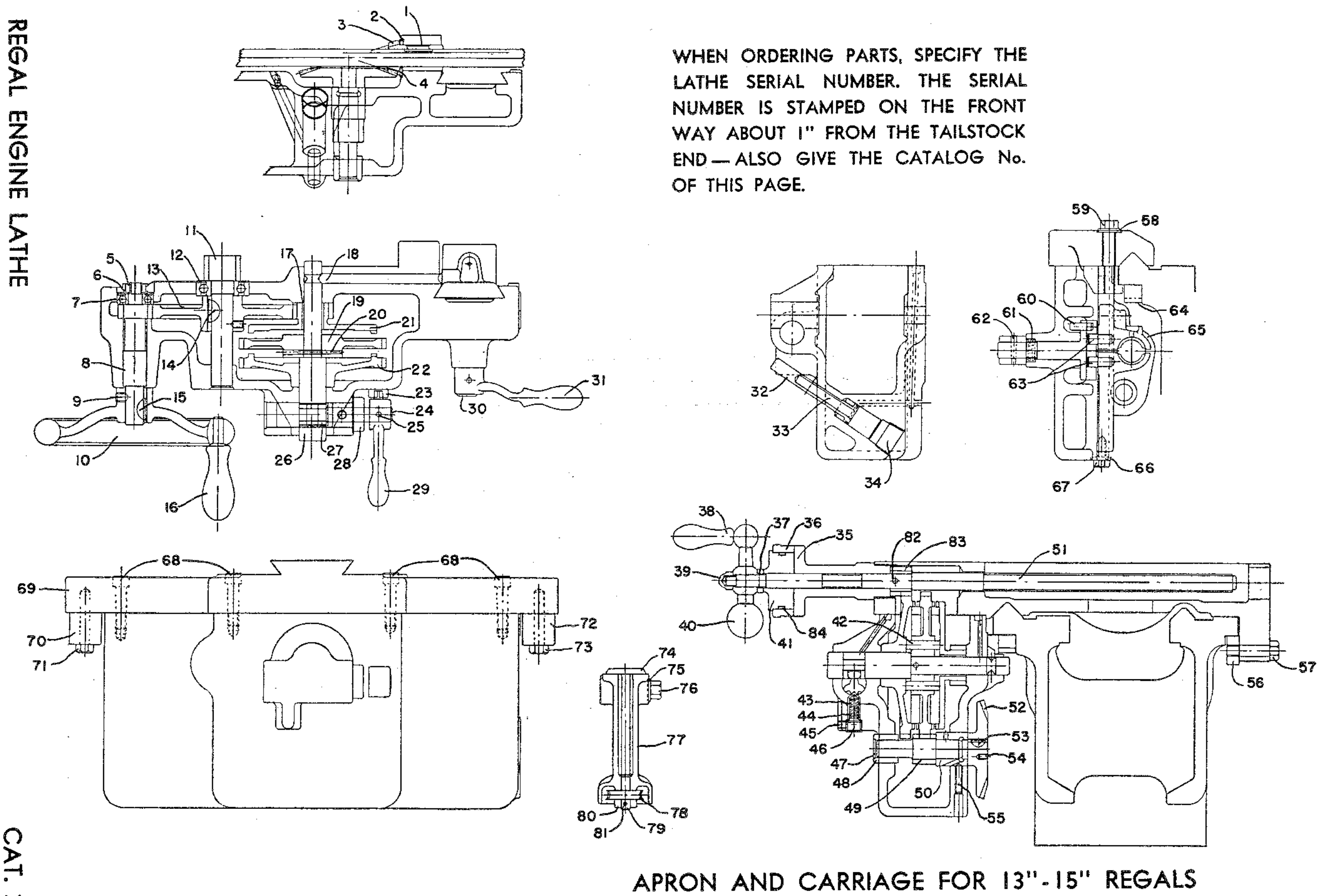
The serial number is absolutely necessary. It is stamped on the flat surface on the front way at the tailstock end of the bed on models built since 1940; prior to that time the number was stamped on the cross girth of the bed at the tailstock end. The numbers indicating the parts are *not* stock numbers, therefore, the information requested is of prime importance for the prompt handling of your order.

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



QUICK CHANGE BOX AND TRANSLATING GEARS ON 13"-15" REGALS

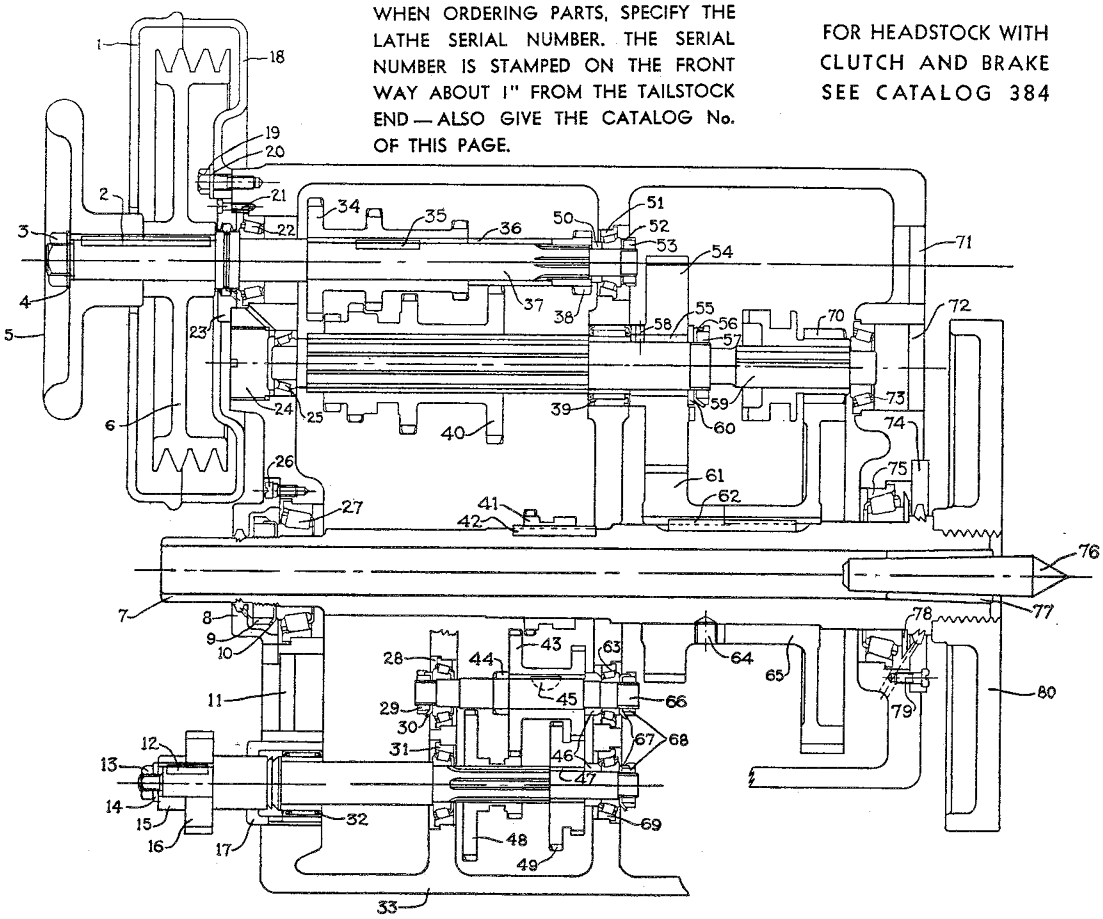
WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



APRON AND CARRIAGE FOR 13"-15" REGALS

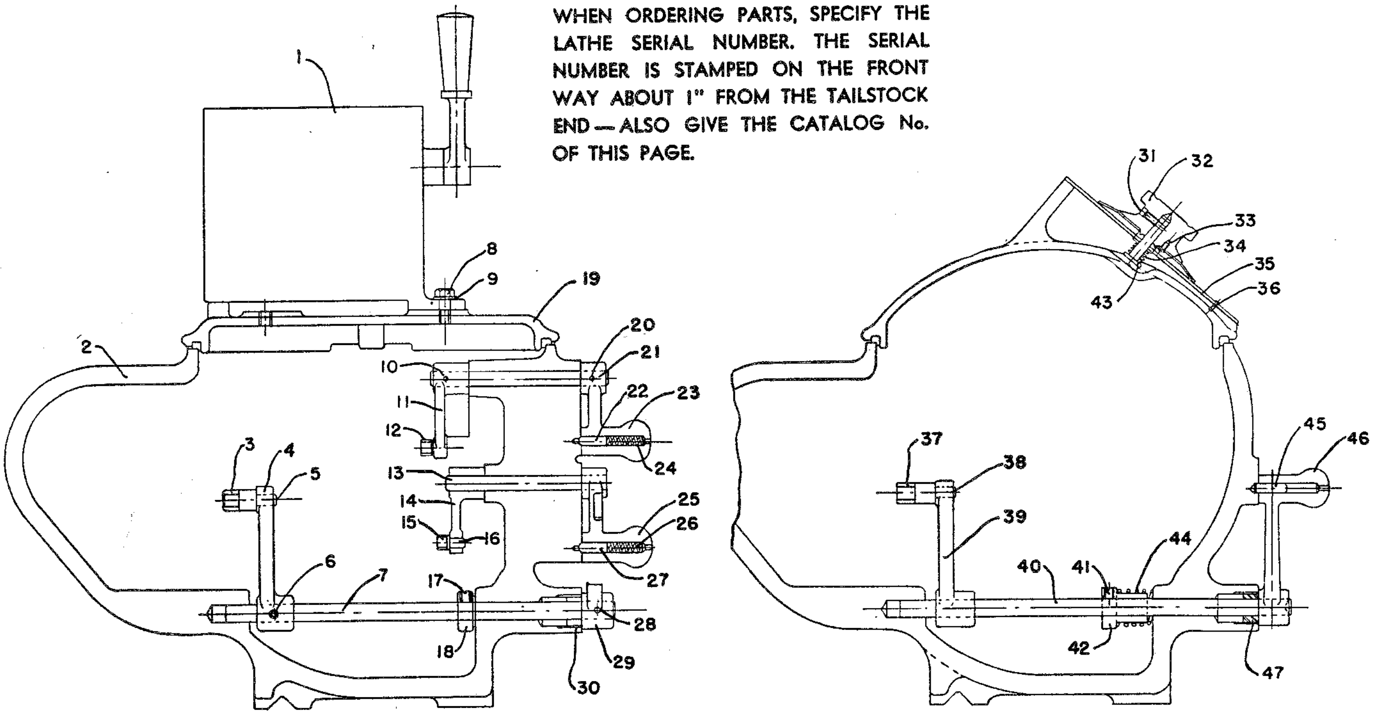
WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.

FOR HEADSTOCK WITH CLUTCH AND BRAKE SEE CATALOG 384



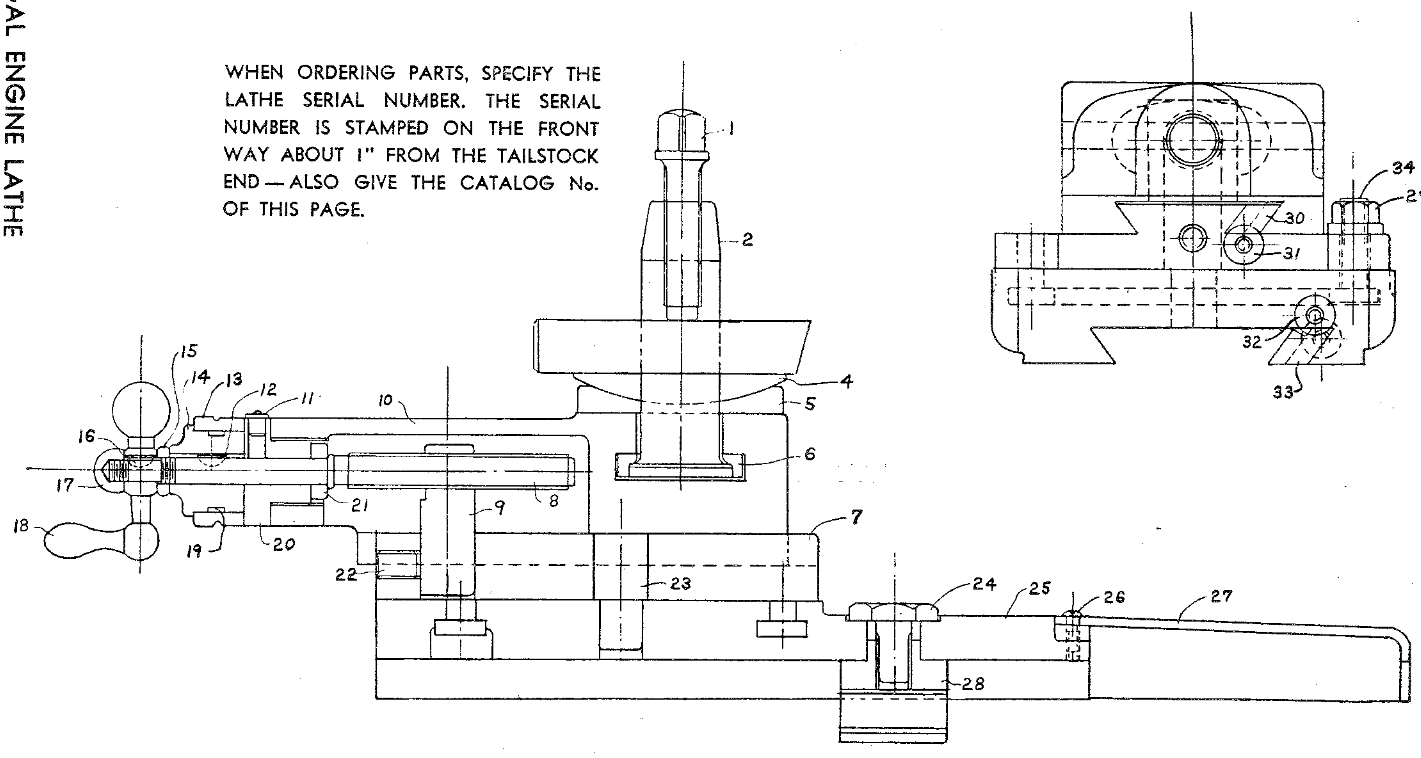
HEAD SECTION COMMON ON 13"-15"-17"-19" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.

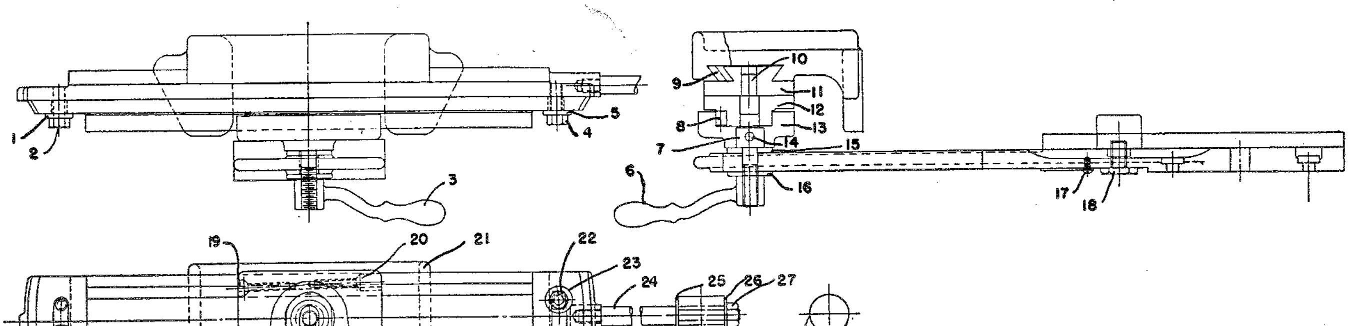
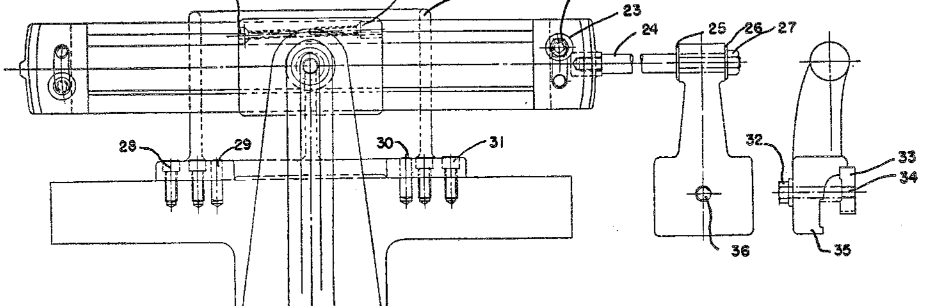
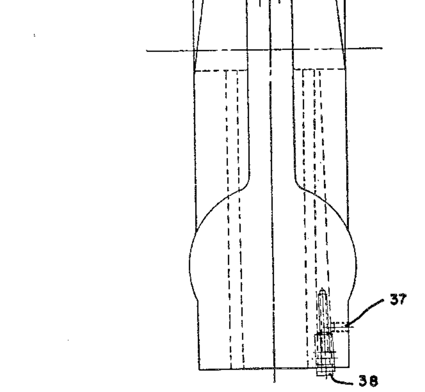


HEAD SHIFTERS COMMON ON 13"-15"-17"-19" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



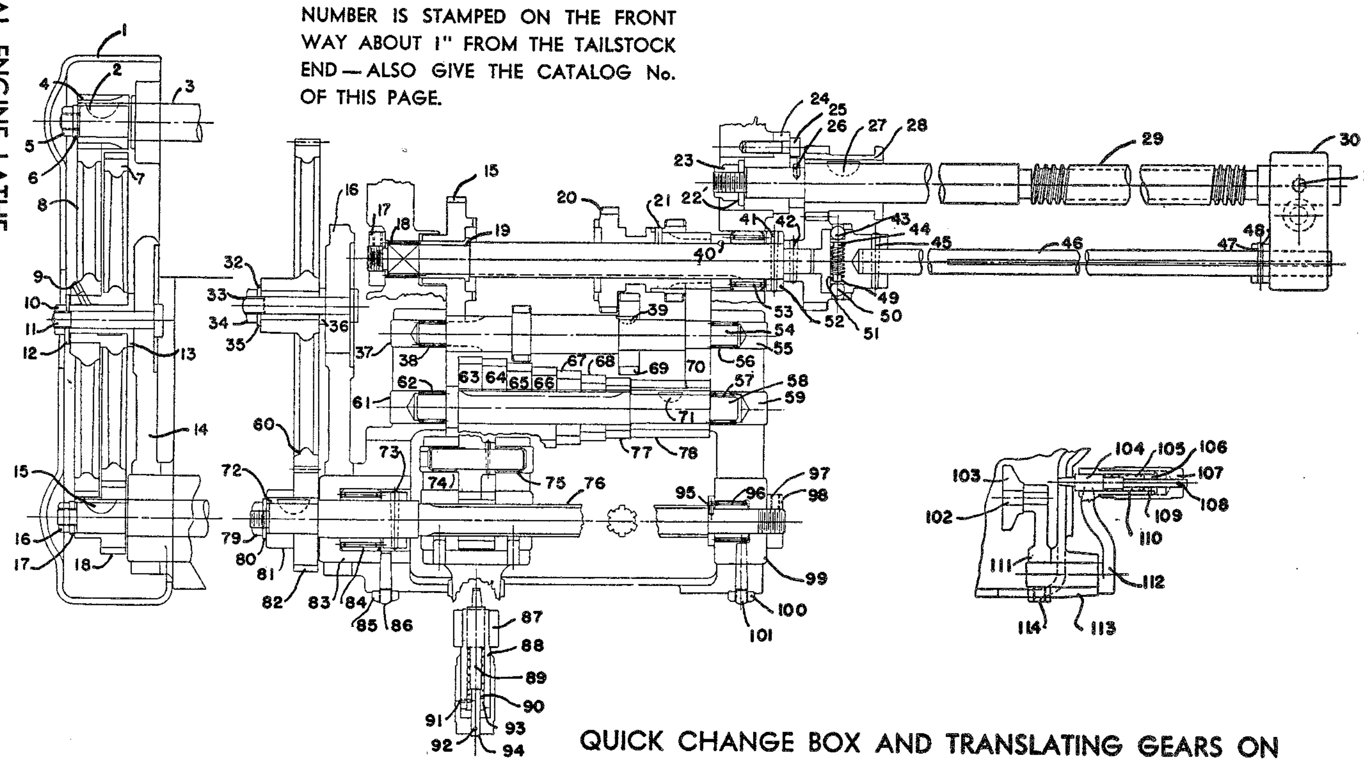
COMPOUND REST ON 13"-15"-17"-19"-21"-24" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.

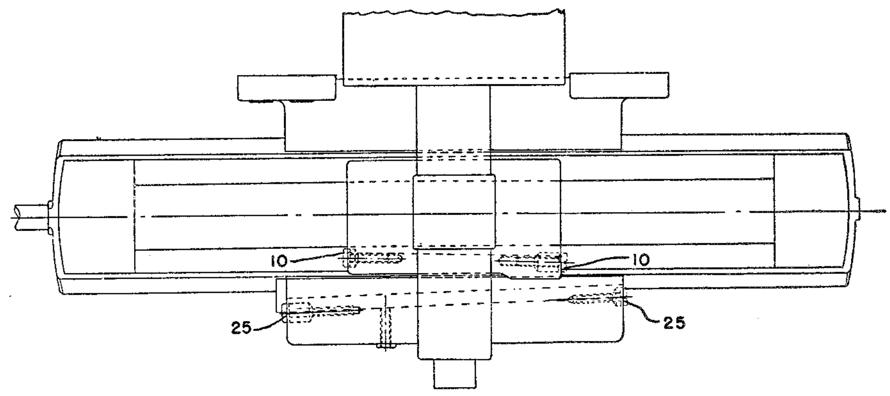
PLAIN TAPER ATTACHMENT FOR 13"-15"-17"-19" REGALS

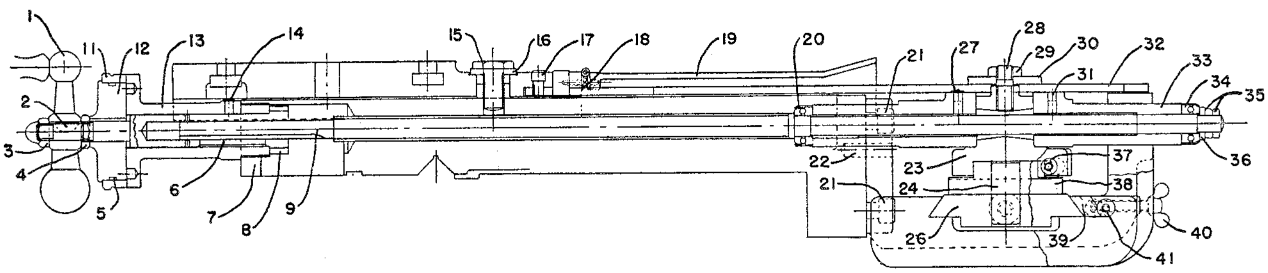
WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



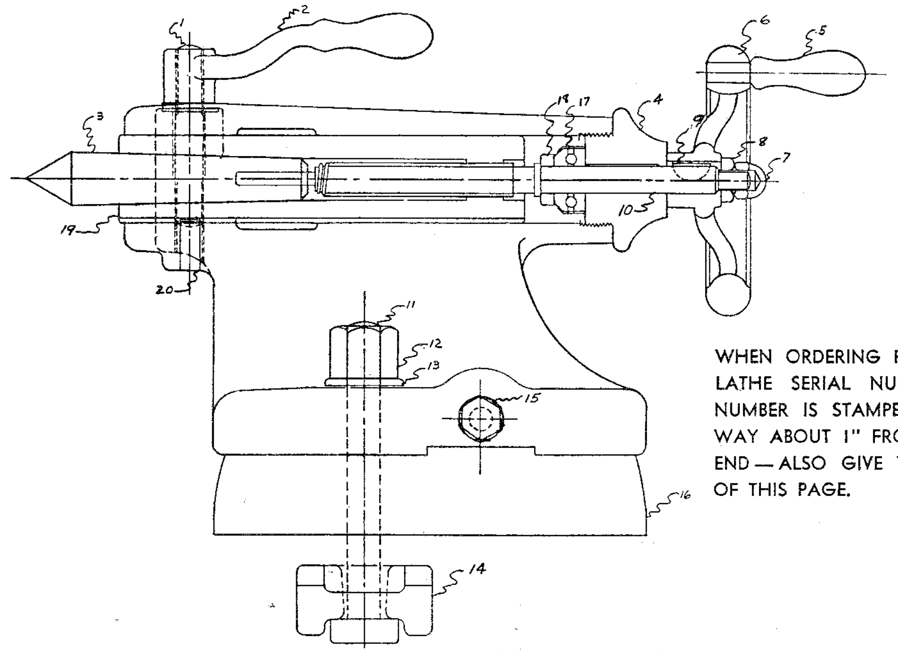
QUICK CHANGE BOX AND TRANSLATING GEARS ON 17" - 19" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.





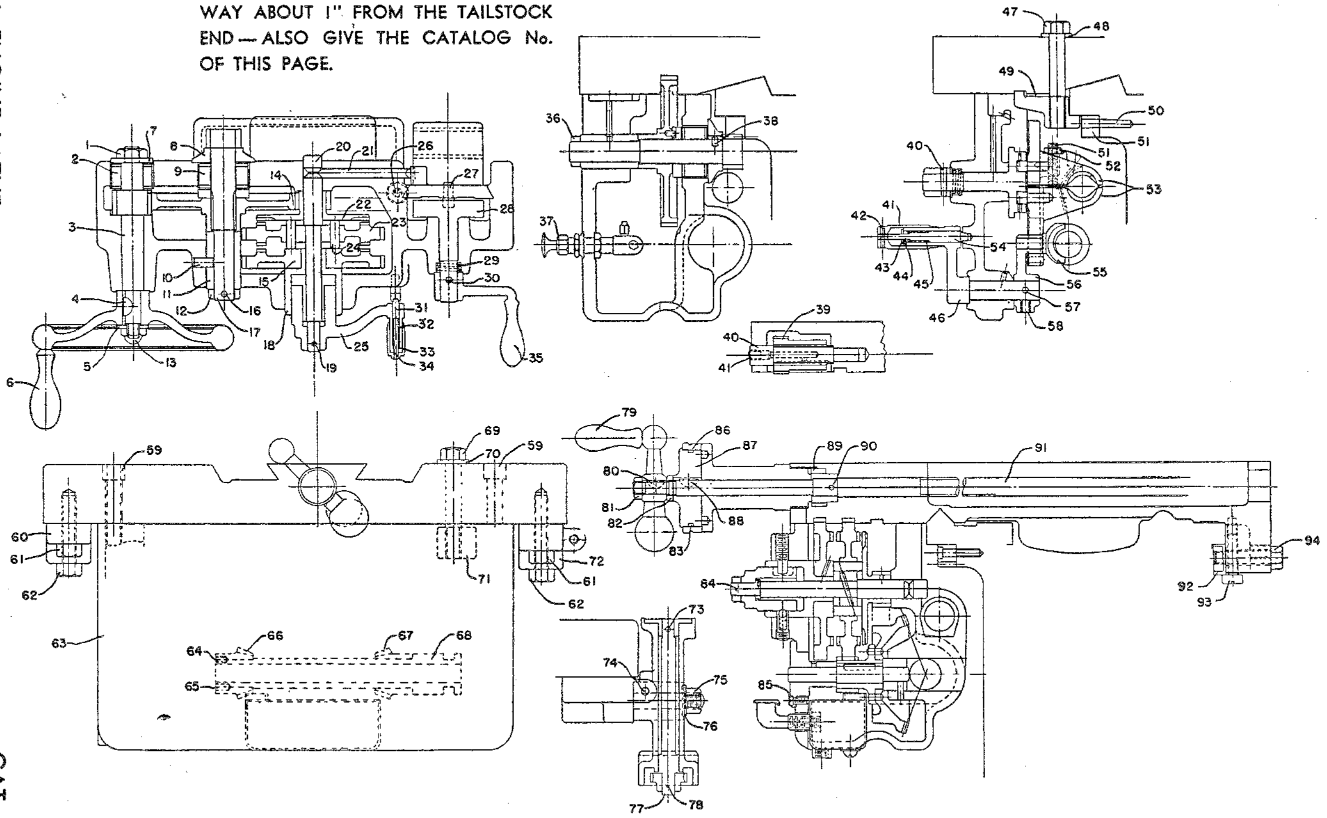
TELESCOPIC TAPER ATTACHMENT FOR 13"-15"-17"-19"-21"-24" REGALS



WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.

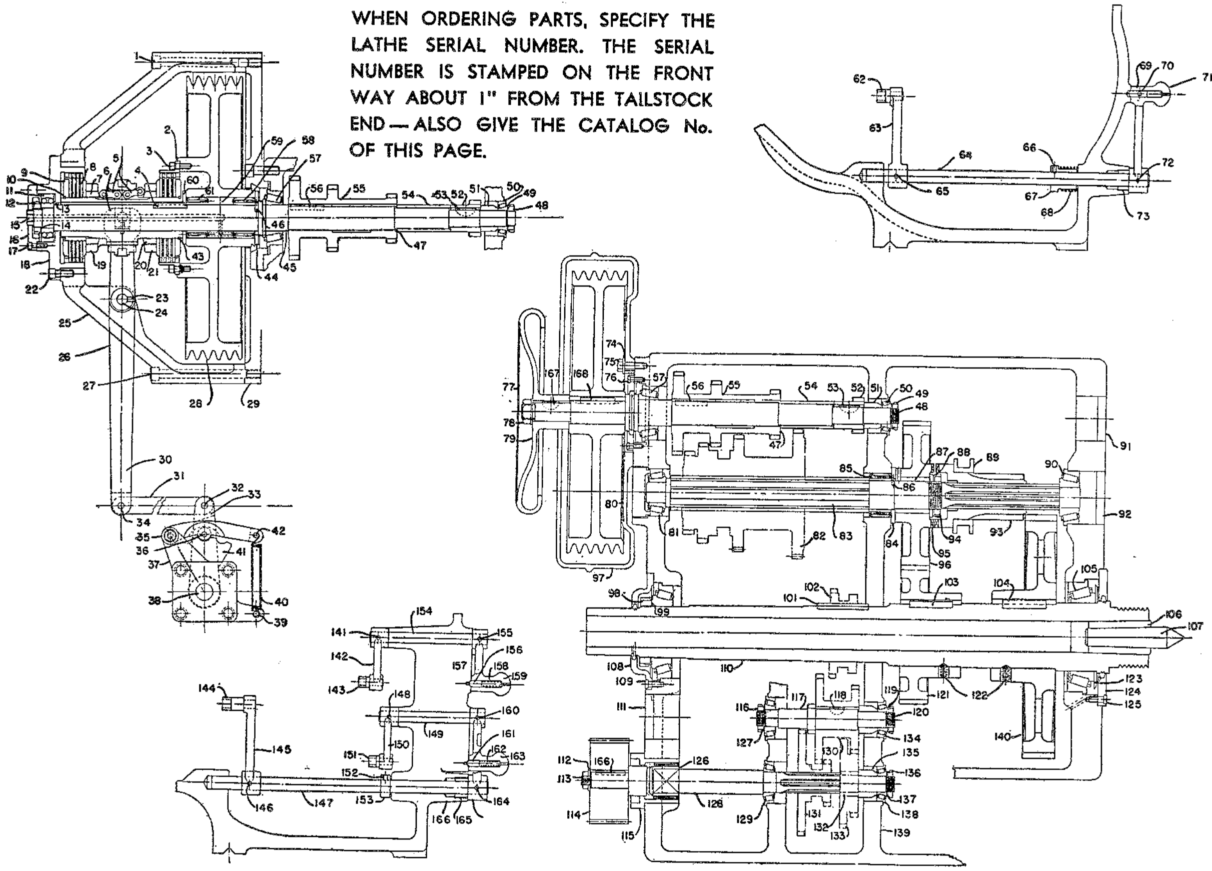
TAILSTOCK FOR
13"-15"-17"-19"-21"-24" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



12" AND 14" HEAVY DUTY APRON WITH
REGAL CARRIAGE USED ON 17"-19"-21"-24" REGALS

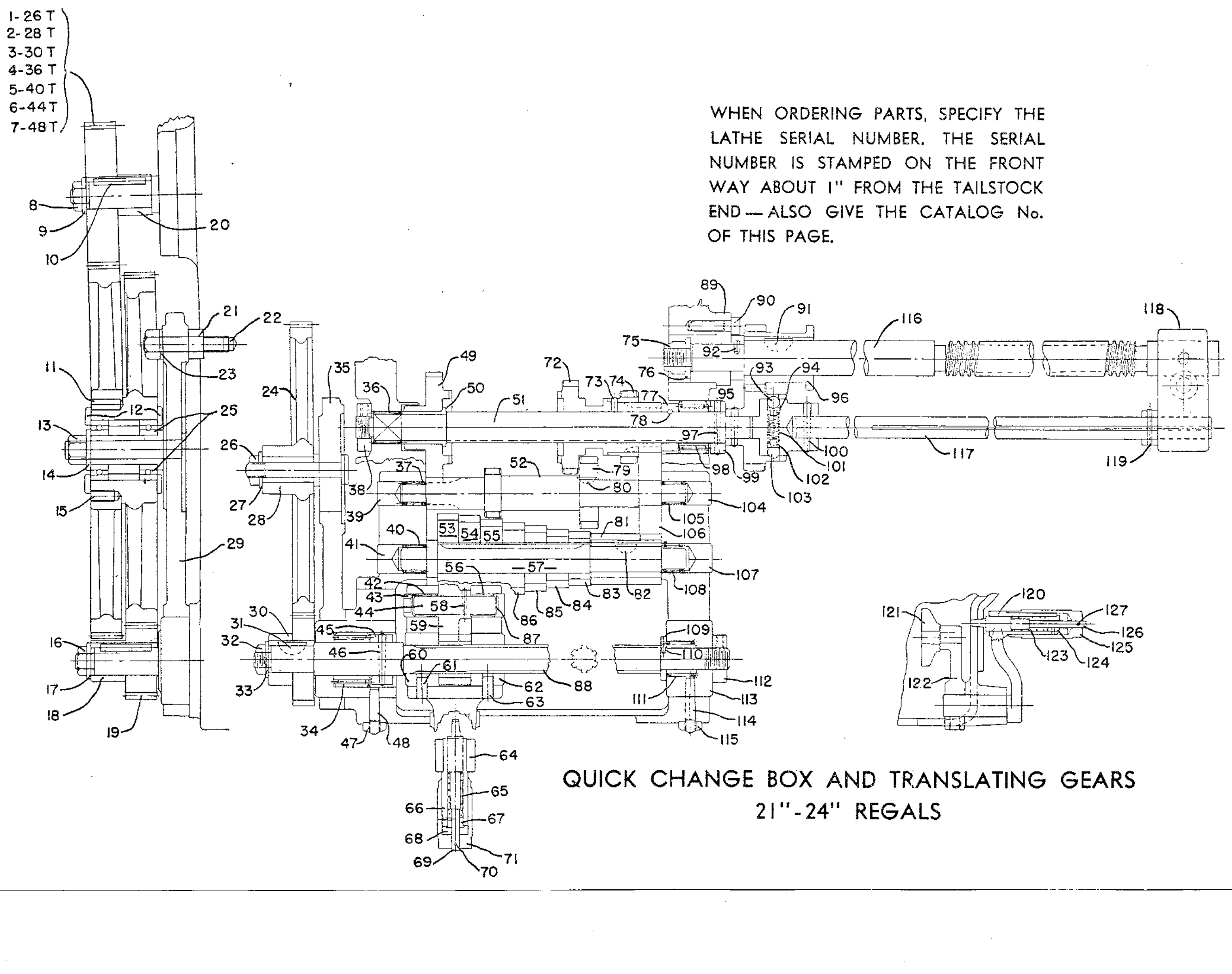
WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



HEAD SECTION AND SHIFTERS — CLUTCH AND BRAKE ON 21" - 24" REGALS

CLUTCH AND BRAKE OPTIONAL ON 13", 15", 17" AND 19" REGALS

WHEN ORDERING PARTS, SPECIFY THE LATHE SERIAL NUMBER. THE SERIAL NUMBER IS STAMPED ON THE FRONT WAY ABOUT 1" FROM THE TAILSTOCK END — ALSO GIVE THE CATALOG No. OF THIS PAGE.



QUICK CHANGE BOX AND TRANSLATING GEARS 21" - 24" REGALS

CHAPTER INDEX

ALPHABETICAL INDEX

REGAL

A Line of

Modern Geared Head Precision Lathes at Popular Prices

Built in Sizes

13-15-17-19-21-24-inch

Sold on the Le Blond Deferred Payment Plan

A qualified representative in your vicinity will gladly show you this outstanding lathe value. Write for his name.

Printed in United States of America

Printed by The Bohnett Co.