RUNNING A REGAL

THE R. K. LE BLOND MACHINE TOOL CO.
CINCINNATI, OHIO, U. S. A.

When ordering repair parts always specify size of lathe and serial number. The serial number is stamped on the flat surface of the front way at the tailstock end of the bed. Prior to 1940 it was stamped on the tailstock end, cross girth of the bed.

RUNNING
A
REGAL

A MANUAL

of Lathe Operations and Maintenance of a modern

GEARED HEAD ENGINE LATHE

Price 25 Cents Postpaid

THE R. K. LE BLOND MACHINE TOOL CO.
CINCINNATI, OHIO

PREFACE

THIS manual is intended to present the basic principles of lathe operation to the student, the apprentice and others mechanically inclined.

The Engine Lathe is the basic, fundamental tool of industry. Henry Maudslay's crude lathe of the late Eighteenth Century was the beginning of the world's most prosperous era, and it is still the basic machine of industrial manufacturing. Without it the present industrial era would not exist. Our luxuries and conveniences would be only a dream. It has made possible our various modes of transportation—steamships, electric and steam railways, automobiles and aeroplanes. Without lathes the production machines used for manufacturing these conveniences would not be possible. Practically every modern mechanical invention and improvement is developed through the use of engine lathes which produce the majority of the component parts of the new machine.

In view of its importance to industry, and as a servant to mankind, it is remarkable that so few people know just what an engine lathe is, what it will do and how it is operated.

To make this booklet as practical as possible, we have assigned one of our men, who has not only operated, but designed and built engine lathes for many years, to give you in plain language the benefit of his knowledge and experience on the subject.

It is almost impossible to cover all phases of operation which might come up in connection with running a lathe; however, we are always glad to assist you in solving your individual problems if you will write us, giving complete details of what you are trying to accomplish.

Running a Regal will be a valuable guide to the beginner. This manual will familiarize him with the various parts of a modern engine lathe, with procedure and operation, and with the kinds of work that can be performed on it. If this book is helpful, we will be more than compensated for its publication.

The R. K. LeBlond Machine Tool Co.
CINCINNATI, OHIO

March, 1942. 6th Edition.

IMPORTANT

Read Carefully

When you order a Le Blond Regal Lathe you will receive an acknowledgment of the order. Acknowledgment specifies a date of shipment, also the probable amount of freight charges for transporting the lathe from the factory to your nearest shipping point.

When the lathe is shipped from the factory, the railroad company or other transporting agency issues a bill of lading. This is a receipt that the machine has been accepted in good order by the railroad.

The railroad freight agent will advise you when the lathe reaches its destination. The lathe becomes your property upon payment of the freight charges and surrender of the bill of lading.

Before accepting the shipment, be sure that the lathe has not been damaged in transit. Regal Lathes are carefully crated to protect them against the usual handling. Sometimes, however, in switching or transferring from one car to another, they are subjected to rough handling, causing breakage for which the railroad company is responsible. Therefore, inspect the machine carefully. If damaged in any way, do not accept it, but write at once to our Traffic Department, giving complete information on the nature of the damage. We will immediately take up the matter with the railroad company and have it adjusted to your satisfaction.

How To Set Up The Lathe

The lathe should be placed in a well lighted area. If the space available is dark, artificial illumination should be provided. The shop should be comparatively dry to keep the machine free from rust.

Remove the crating carefully and leave the skids under the lathe until you have skidded the machine to its approximate location.

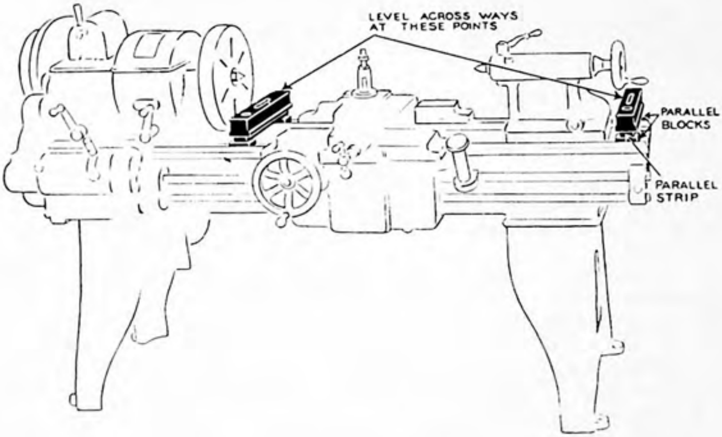
The floor on which the lathe sets should be absolutely firm. A lathe must set level and solid in order to perform accurately. It will be impossible to keep the machine level and in alignment if the floor is springy. Therefore, a solid foundation for the tool is of the utmost importance.

Next remove the lag screws which hold the legs to the skids and remove the skids from under the machine, taking care that the machine does not upset.

Remove all slush oil from the various parts of the machine. This can best be done with a rag or waste saturated with kerosene. Next wipe off all the bright or bearing parts with a dry rag or waste, following with a rag saturated with clean machine oil to cover all these parts with a protecting film of oil.

The lathe is then ready to be leveled. Even some of the best mechanics do not realize how important it is to level the lathe properly. Although the bed is heavy it can easily be sprung if not properly leveled. All of our care and inspection is wasted if the machine is not set up properly.

First of all, secure a precision ground bulb level for this work, such as made by Pratt & Whitney, Starret, or Queen & Company. An ordinary carpenter's level or a combination square level is not sensitive enough for this important operation. Next drive one of the four hardwood wedges sent with the machine under each of the legs to compensate for any depressions in the floor. Place the level on two short parallel strips between the front and back ways just as near to the headstock as possible, and drive the wedges under the headstock legs until the bubble is in the center of the bulb. Then take the level and parallel strips to the tailstock end of the lathe and wedge up under the tailstock legs until the level registers the same as at the headstock end. By repeating this several times, both the head and tailstock ends of the bed


LEVEL ACROSS WAYS
AT THESE POINTS
PARALLEL
BLOCKS
PARALLEL
STRIP


LEVEL

will be brought to a perfect level. The lathe, when properly leveled, should show the same degree of accuracy of alignment as noted on the test card which accompanies each machine.

If the lathe sets on a wood floor, the same lag screws taken from the skids can be used for lagging the machine to the floor. These, however, should not be pulled down so tight that they draw the bed out of level, but only tight enough to keep the lathe from "walking".

If set on a concrete floor, expansion bolts should be used for this purpose. Do not bed the legs in concrete because it will be necessary from time to time to check and correct the machine for level.

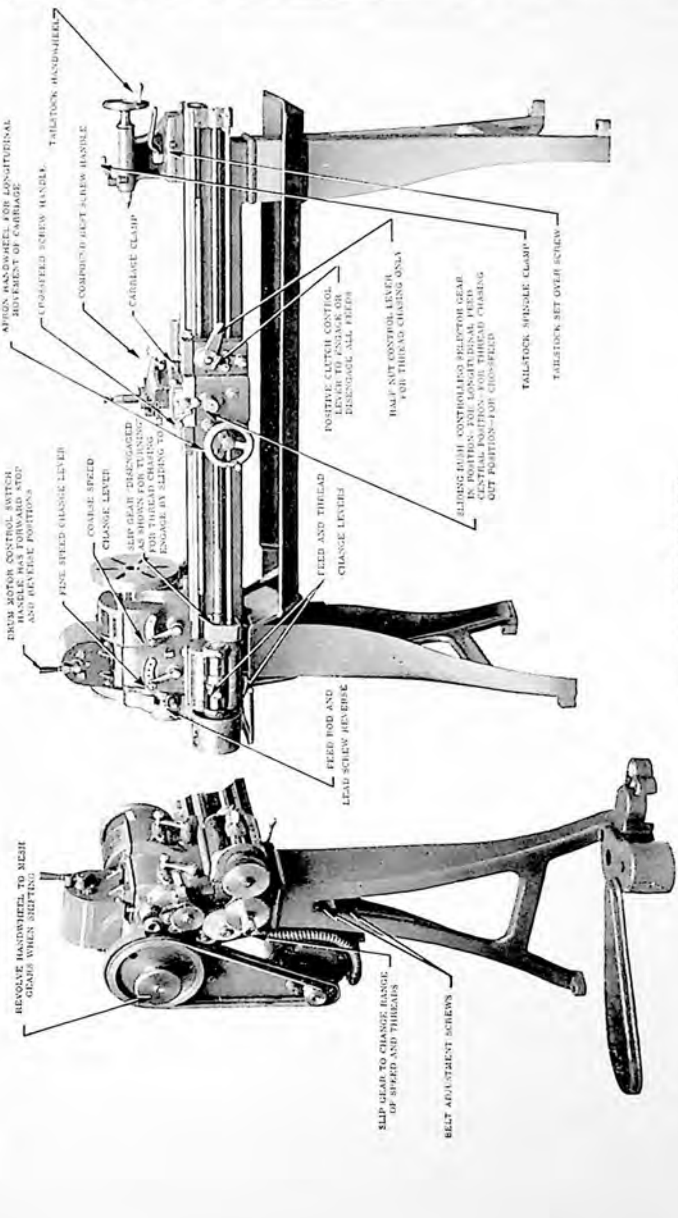
The next step is to connect the service lines to the motor. It is important that the voltage and the other specifications of the motor are the same as those of your service lines. The data plate on the motor specifies whether the current should be direct (D. C.) or alternating (A. C.). If direct current is specified, the voltage is shown. If alternating current is specified, the voltage, frequency (cycles) and number of phases are shown. If there is any doubt about the current and voltage, call your local power and light company and verify the supply. If there is a difference, advise us before connecting, and avoid burning out or otherwise damaging the motor.

Before you start the lathe consult the lubrication chart on the following pages, which shows the location of various oil inlets on the machine. Fill the headstock with a medium grade of machine or engine oil (SAE 20) to the oil level line indicated on the chart and squirt oil in all oil holes. It is important to use only the best grade of lubricating oil. All of the bearings fit closely and it is absolutely essential that the machine is properly lubricated before it is operated.

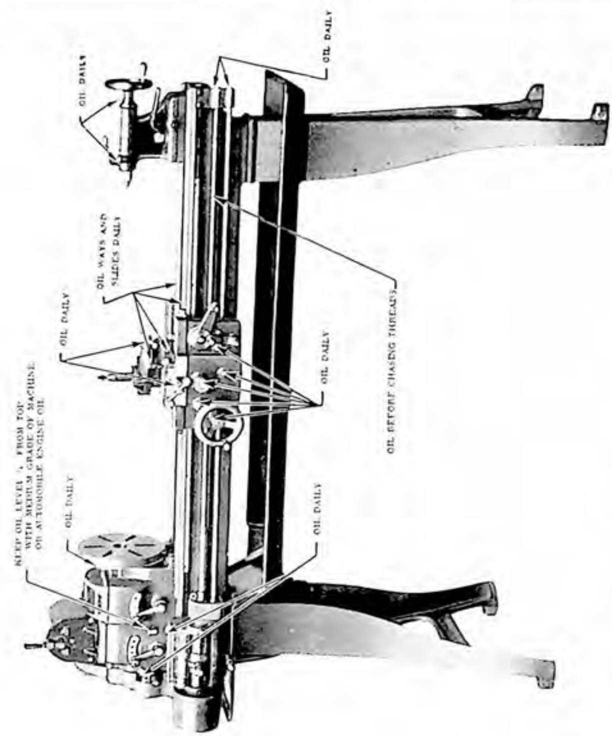
A lathe, like an automobile or any other piece of mechanical equipment, depends on the attention it receives during the first three or four days' use—"the breaking-in period". See that all bearings are carefully oiled and watch that none run hot.

Get Acquainted With Your Lathe

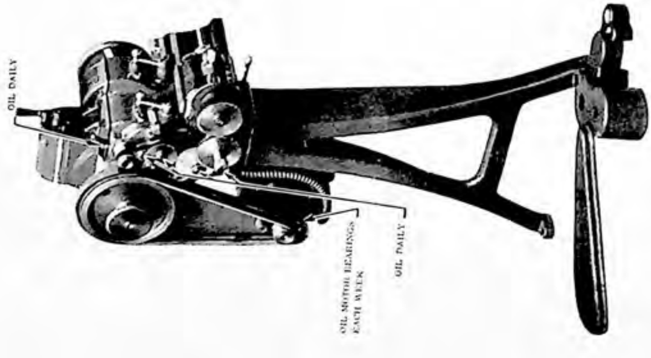
Before trying to do any work on the machine, the operator should familiarize himself with the names of the various working parts from the charts on pages 8 and 10, as the parts are referred to throughout the book by these names. He should also know the functions of the various parts.

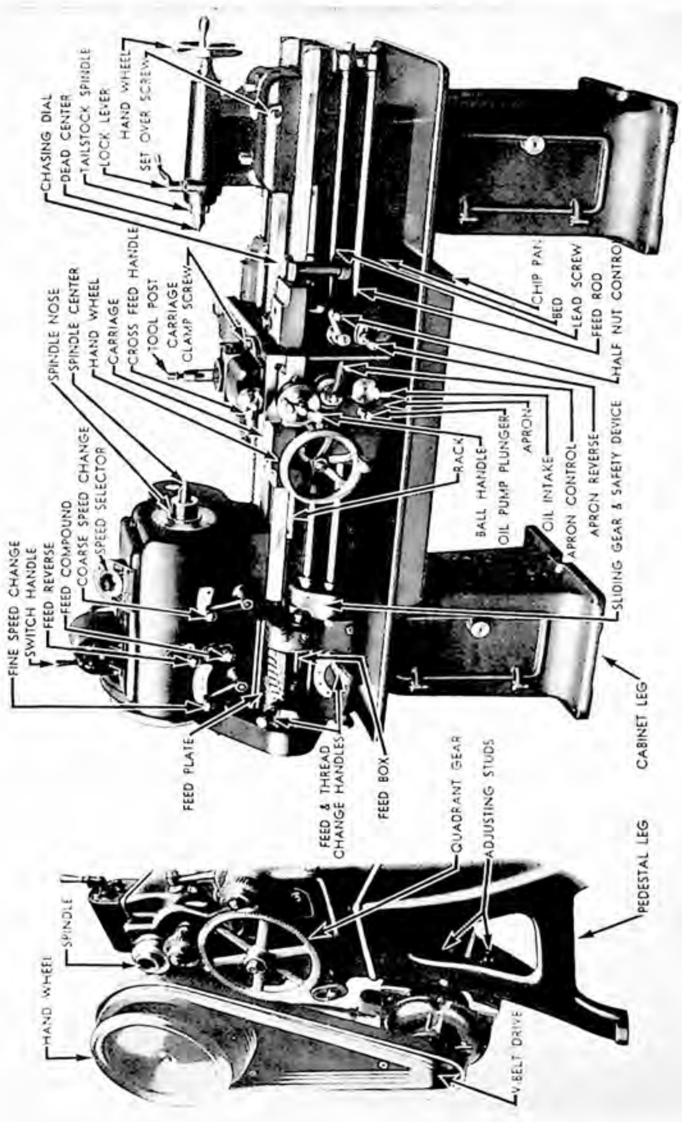


OPERATION CHART
10" REGAL LATHE

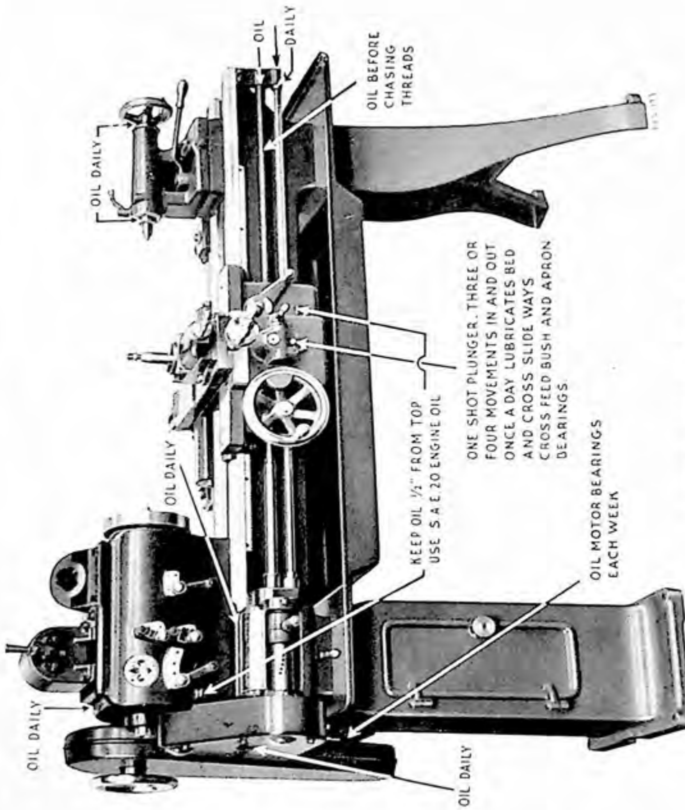


LUBRICATION CHART
10" REGAL LATHE

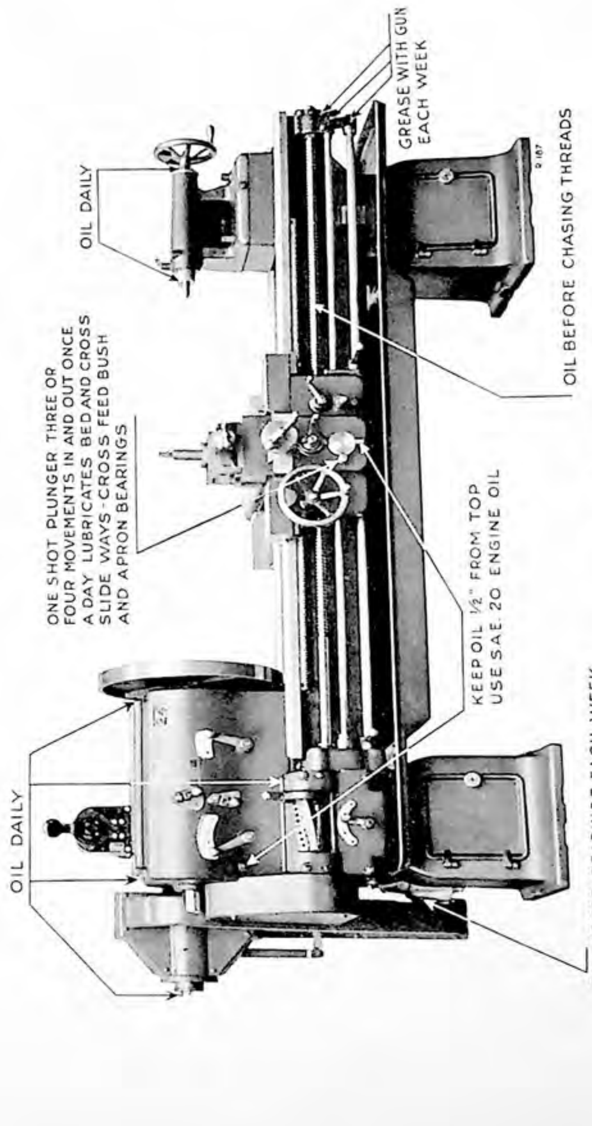




OPERATION CHART—13" to 24" REGAL LATHES



LUBRICATION CHART
13" to 19" REGAL LATHES



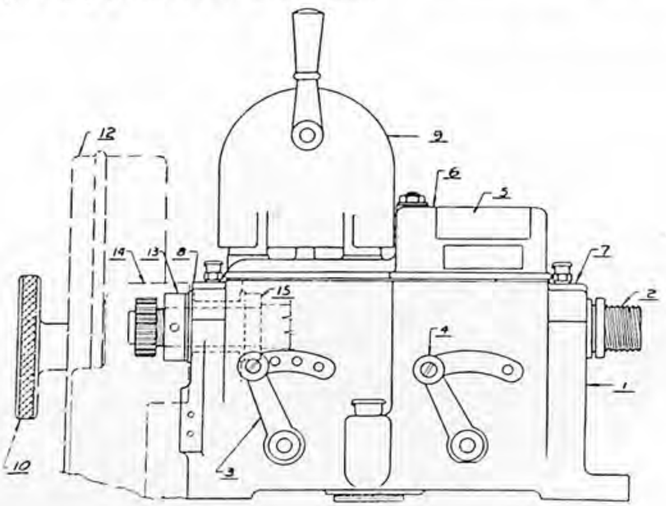
LUBRICATION CHART
21" to 24" REGAL LATHES

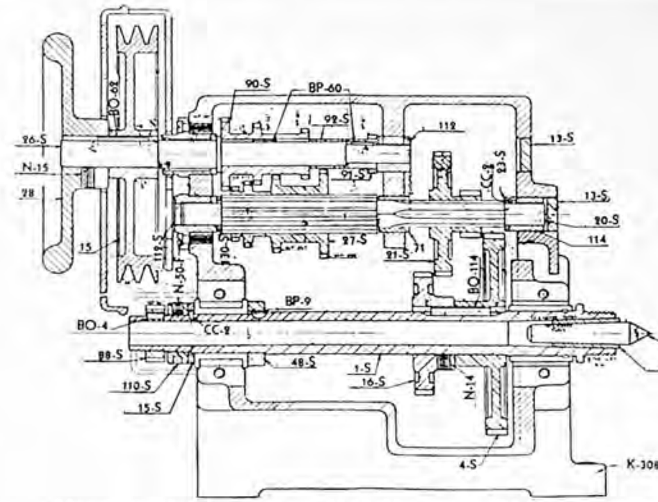
Headstock

10-Inch

The headstock (1) below carries the spindle (2), intermediate shaft (20-S), page 13, and drive shaft (26-S). On these shafts are mounted the sliding gears for obtaining the different spindle speeds. The speeds are selected by means of the speed change lever (3) to get the four finer gradations of speeds, and the back gear lever (4) to get the wider gradation of speeds—the high and low back gear ratio. The speed plate (5) shows the position of the levers to obtain the various spindle speeds. The headstock cover (6) allows convenient access to the interior of the headstock. The headstock should be filled to the oil level line with a good grade of medium machine or automobile engine oil. This not only lubricates the driving gears and shifter yokes but also the bearings for the intermediate and drive shafts by means of the splash system. The oil in the headstock should be changed occasionally by draining when it becomes dirty or gummy. The oil can be drained by removing the plug in the bottom of the headstock.

Allowance for adjustment is made on the front spindle bearing cap (7) and rear spindle bearing cap (8). Each cap is provided with a separate oiler which should be filled at least once a day to insure adequate lubrication of the bearings. The caps are properly adjusted when the lathe leaves the factory and they should not require any attention for a long time.





Many mechanics believe that the main spindle bearings must be taken up when a lathe chatters. However, as there are so many things that can cause chatter (see separate chapter on "Chatter," page 103), it is best to look for other reasons before attempting correction by spindle bearing adjustment.

Spindle Adjustment

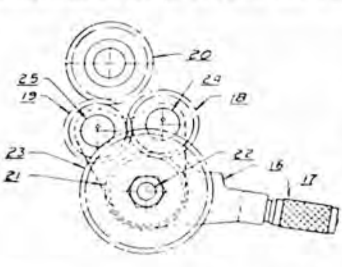
The spindle adjusting nut (13), page 12, located under the reverse plate cover (14), is used to take up the end play, or longitudinal movement of the spindle. This nut is also properly adjusted before the lathe leaves the shop, but if adjustment is necessary proceed as follows:

Remove reverse plate cover (14), which is secured to the headstock and quick change box by three screws. Set the speed change gear lever (3) and the back gear lever (4) in neutral positions where no gears are in mesh, and the spindle can be revolved by turning the small face plate. Loosen the two set screws in the adjusting nut and tighten up the nut (turn to right) with a spanner wrench. This pulls the thrust collar (15) on the spindle against the back bearing of the head. When properly adjusted, there should be no end play in the spindle, yet it should revolve freely by hand. When the adjustment is completed, it is important to lock the nut on the spindle with the two set screws and replace the gear cover. It is important that the bearing between the nut and the spindle bearing be lubricated occasionally.

Reverse Plate

10-inch

The reverse plate (16) is used to reverse the direction of rotation of the feed rod and the lead-screw. There are three positions of the reverse plate handle locking plunger (17). When the handle is in the upper position, longitudinal movement of the carriage is toward the headstock, and cross movement of the compound rest is toward the center of the lathe. This position is also used when cutting right-hand threads. With the handle in the lower position, a longitudinal feed toward the tailstock and the cross feed is from the center. In the lower position the lead-screw revolves in the proper direction for cutting left-hand threads. With the handle in the center position, the entire feed and thread-cutting mechanism is disengaged. This is an advantage when using the lathe on the high speeds for filing or for turning wood.



The reverse plate should not be shifted with the lathe running or under cut. The lathe should be stopped and then the tumbler gears (18) and (19) above, are rolled into engagement by turning the speed change hand wheel, page 16.

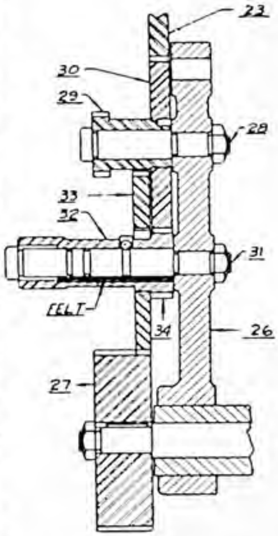
The reverse plate gears get their movement by engaging either of the tumbler gears on the plate with the spindle feed gear (20). One of the tumbler gears engages the reverse plate gear (21) mounted on reverse plate shaft (22). The entire reverse plate swings on a hub pivoted in the headstock casting. A second and larger diameter gear, the reverse plate driving gear (23), is mounted on the reverse plate shaft and this gear drives the gears on the quadrant. All the reverse plate parts should be oiled at least once a day, preferably before starting each morning. The reverse plate shaft is oiled through the headstock oil hole, and tumbler gears are oiled through the holes in the tumbler gear studs (24) and (25).

ON THE 13″, 15″, 17″, 19″, 21″ AND 24″ REGALS THE GEARS FOR THE REVERSE TO THE FEED AND THE COMPOUNDING OF THE FEED ARE BUILT IN THE HEADSTOCK AND NO REVERSE PLATE, QUADRANT OR SLIP GEAR IS REQUIRED.

Quadrant

10-inch

The quadrant (26) is a swinging plate pivoted around a hub on the quick change box and serves as a mounting plate, carrying the connecting gears between the reverse plate driving gear (23) and the quick change box gear (27). On the first stud of the quadrant (28) two gears (29 and 30) are mounted and revolve on a common hub, the larger of which (30) engages the reverse plate driving gear (23). The second stud on the quadrant (31) carries a double slip gear (32), and by pulling out the slip gear handle the large gear (33) can be engaged with gear (29) for the finer feeds, or the small gear (34) can be engaged with gear (30) for the coarser feeds. Quick change box gear (27) has a wide face, so that slip gear (33) is always in mesh whether slip gear is in or out. These two ratios of gearing to the quick change box can be made without removing the gear cover. The quadrant can be swung into proper engagement and locked in place with a nut.



All quadrant gears revolve on studs. Care must be taken to lubricate the gear bearings regularly. The quadrant studs have oil holes which should be filled once a day when the reverse plate is oiled.

Special quadrant and gears can be provided to cut metric or special threads that cannot be obtained through the quick change box.

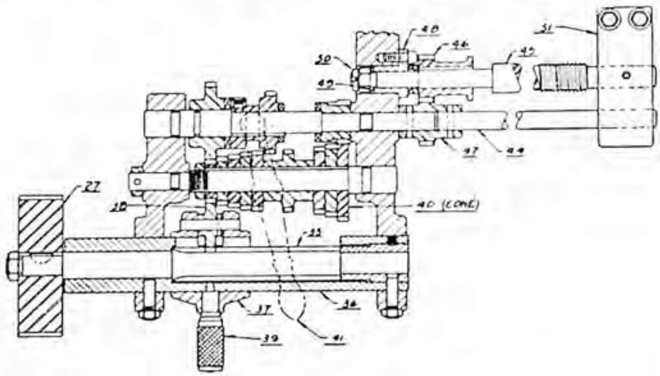
ON THE 13″, 15″, 17″, 19″, 21″ AND 24″ REGALS THE GEARS FOR THE REVERSE TO THE FEED AND THE COMPOUNDING OF THE FEED ARE BUILT IN THE HEADSTOCK AND NO REVERSE PLATE, QUADRANT OR SLIP GEAR IS REQUIRED.

Quick Change Mechanism

10-inch

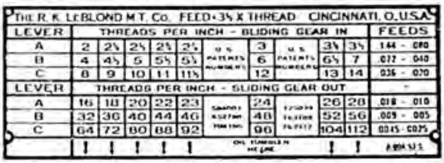
The quick change box is the mechanism by which various gear ratios are obtained by sliding gears to provide the proper combination for the various changes of feed and thread-cutting. On the old style engine lathes thread and feed changes were made by changing the gears on the end of the lathe. On the modern lathe these gears are all contained in the gear box. Thread and feed changes can be made quickly; hence the name, quick change box.

The construction of the feed box is as follows: The quick change box gear (27), (below) is driven from the quadrant slip gear (33), (page 15). It (27) is mounted on the end of the sleeve gear (35). This sleeve gear runs inside of the quick change box sleeve (36). The quick change box tumbler (37) is mounted on the sleeve (36). The back of the sleeve is cut out to permit



the tumbler gear (38) to engage the sleeve gear. By means of the tumbler gear handle (39) the tumbler gear can be rocked into engagement with any of the change gears (40) in cone formation. This is done by pulling out the handle plunger and bearing down on the handle to disengage the tumbler gear, then sliding the tumbler along the sleeve to the proper location and lifting the handle up to bring the gears into engagement. When the gears are in engagement the plunger in the handle locks the tumbler in place so that it cannot kick out when cutting left-hand threads. A series of slots milled in the quick change box sleeve and a pin engaging these slots prevent the tumbler from engaging two change gears at one time. The eight feed changes obtained through the tumbler gear and the quick change gear are multiplied three times by means of the three positions of the compound lever (41), which operates a sliding gear and gives three different gear ratios to the feed shaft. In the feed train there are two ratios through the slip gear on the quadrant, each of which is multiplied by the eight different ratios through the tumbler and change gears, making sixteen changes. These sixteen changes are multiplied by the 3 additional ratios obtained by the use of the compound lever, giving a total of 48 changes.

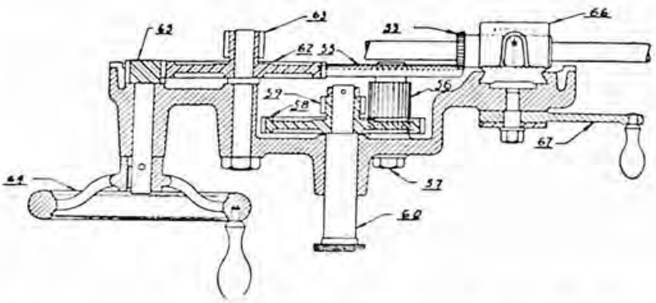
The index plate is direct reading, mounted on the quick change box directly over the tumbler sleeve. The numbers on the plate refer to the threads or cuts per inch that the lead-screw and the gear combinations will cut when the tumbler is engaged directly under the number on the index plate. "In" and "Out" refer to the location of the slip gear on the quadrant, and the location of the lever refers to position of compound lever (41). For example, if you wish to cut 8 threads per inch, engage the tumbler under 8 on the index plate, push in the slip gear, and move the compound lever to the right-hand position (C). You will also note on the top line of the index plate, feeds=so many times the threads, as on the 10″ plate shown; Feed=3½ times thread—which means that if you have the quick change box set to cut 8 threads per inch, the feed would be 28 cuts per inch (3½ times 8) or 1/28 of an inch per revolution. This expressed in thousandths of an inch would be one inch divided by 28, or .036 of an inch per revolution of the spindle.



Apron

10-inch

The apron (52), page 18, comprises the necessary mechanism to give the longitudinal feed to the carriage (68), page 28, the cross feed to the compound rest (74), page 29, and is provided with the means for engaging and disengaging the feeds. The apron also carries the half-nuts that engage and disengage the lead-screw when cutting threads, and an interlocking device that prevents the engagement of the lead-screw and feed rod at the same time. The apron operates in the following manner: The splined feed rod passes through the feed bevel pinion (53), which has a key to engage the



spline (keyway) in the feed rod. The bevel gear (55) has its bearing in the apron. The feed trip lever (54) engages and disengages the feed bevel pinion (53) with apron bevel gear (55). The feed trip lever (54) serves to engage and disengage the feed for both the longitudinal and cross feeds. Forming the hub of the apron bevel gear is a wide-faced pinion (56); both apron bevel gear (55) and pinion (56) revolve on a stationary stud (57) in the apron wall. The apron sliding gear (58) and sliding pinion (59) are both mounted on a shaft revolving on pull bush (60), which slides in a long bearing in the apron wall. There are three positions for this bush. When pulled all the way out, the sliding gear (58) engages the cross feed pinion (61), page 28, on the cross feed screw and gives the cross slide its movement in and out; when pushed all the way in, the sliding pinion (59) engages the rack wheel (62) attached to the rack pinion (63), which in turn engages the rack fastened to the bed and gives the carriage its longitudinal movement. A neutral position, where neither gear (58) nor pinion (59) is engaged, is used for screw cutting. When the gears are in this neutral position the apron gears are not driven by the rack wheel and pinion.

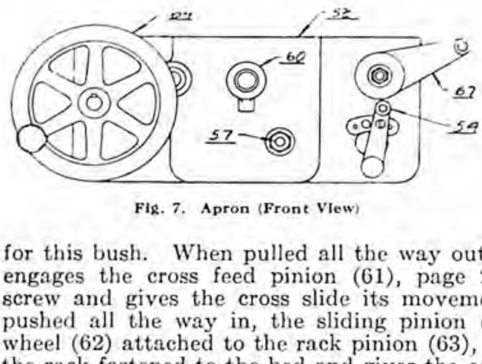


Fig. 7. Apron (Front View)

To move the carriage by hand, the apron hand wheel (64) is provided. The hand wheel shaft has a pinion (65) cut on it to engage the rack wheel (62) and provide hand feed movement to the carriage. The half-nut (66) is carried in a dovetail bearing and it is moved in and out of engagement by the half-nut lever and cam (67). The lead-screw is supported directly under the half-nut by a lead-screw support bearing. A simple interlocking device prevents the engagement of the half-nut with the lead-screw when the cross or longitudinal feed is in engagement; that is, the feed bevel pinion (53) must be out of engagement with bevel gear (55) before the half-nut can be engaged with the lead-screw.

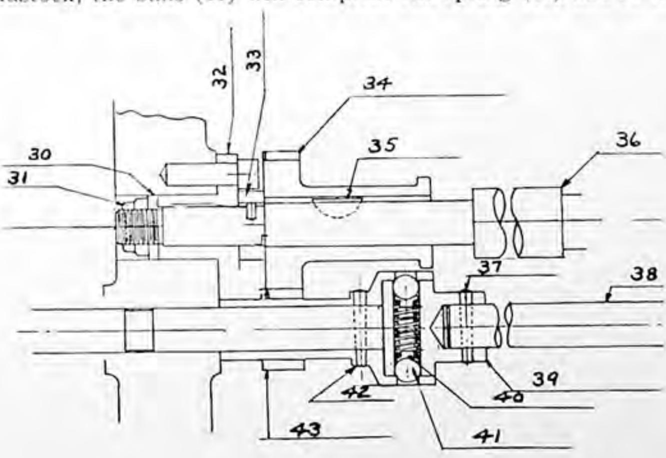
Feed Rod

The feed rod (44), page 16 or (38) below transmits the power from the quick change box to the apron. Most lathes of this type are not provided with separate feed rod, but use a splined lead-screw for both turning and chasing, thus the lead-screw is always in use. On Regal Lathes a separate feed rod is provided to transmit the power for turning and facing. The feed rod is connected to the final drive shaft through a safety device. Should the carriage meet with any obstruction on the bed or run into the chuck or face plate, the safety device will release and save the feed mechanism of the lathe.

Safety Devices

All Regal Lathes, because of their extensive use in schools where they are operated by comparatively inexperienced persons, are equipped with a safety device, which releases when the load on the feed rod becomes too great for the machine.

At a predetermined factor of safety point the spring-ball clutch releases the feed rod, and automatically engages it again when the strain is released. Thus, if the carriage runs into the headstock, the balls (41) will compress the spring (40) and release



the shaft (38) and save the feed mechanism from breakage, but as soon as the feed is disengaged at the apron, the safety device engages again and resumes turning the feed rod.

Lead-screw

The lead-screw (45), page 16, is used for thread-cutting, and it is driven by the lead-screw gear (46) and the feed rod gear (47). The lead-screw slip gear (46) has a sliding fit on the feed box end of the screw and can be engaged with, or disengaged from, the feed rod gear by a short sliding movement on the lead-screw. When not chasing threads, disengage the sliding gear so that the lead-screw does not revolve. On other lathes of this type, where splined lead-screws are used to drive the feed gears, the lead-screws are subjected to torsional strains at all times and soon become inaccurate. The key engaging the spline (keyway) in the lead-screw also burrs up the edges of the threads and the lead-screw acts as a tap, constantly wearing the half-nuts. The lead-screw on a Regal Lathe remains accurate for the life of the machine, as it is not subject to these conditions.

The headstock end of the lead-screw runs in the lead-screw bush (48), which is held in a bearing in the quick change box by two screws. The lead-screw is held endwise between a shoulder and the adjusting nut (50), with hardened thrust washers on each side of the bush. End play is eliminated with the adjusting nut (50). Care must be taken to keep the lead-screw free from end play or the threads will be spoiled when the lathe is reversed without backing the tool away from the work. To adjust the nut, take out the screws holding the lead-screw bush (48) to quick change box; engage the half-nut to the screw by raising the half-nut lever (67), below; move the carriage toward the tailstock sufficiently to allow the withdrawal of the bush (48) and draw up the nut (50) until there is no end play in the screw.

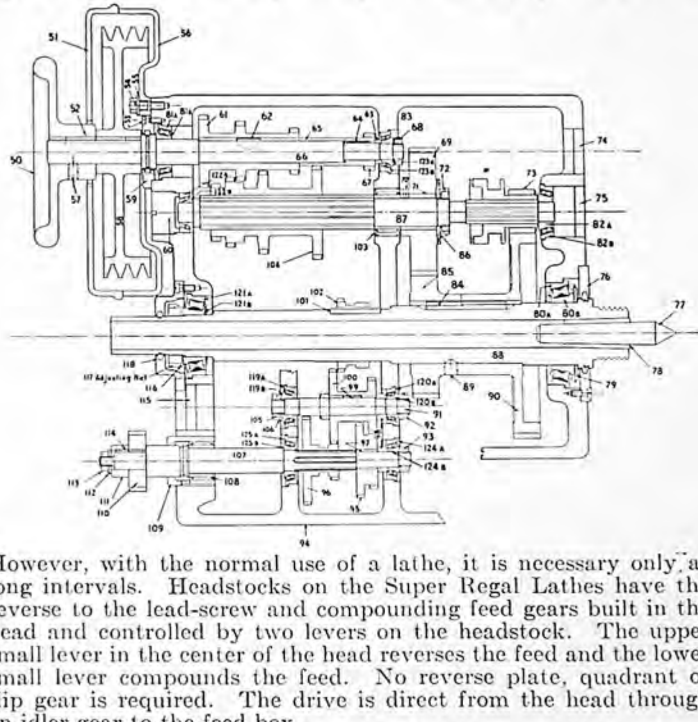
All right-hand threads, and the majority of turning cuts on a lathe, are cut toward the headstock. For this reason the Regal Lathes are equipped with left-hand threaded lead-screws. This also reduces the number of gears in mesh between the spindle and the lead-screw. The thrust of the lead-screw is taken at the feed box end of the screw, and as most threads cut are right-hand, the lead-screw is in tension under this condition. The lead-screw takes a bearing in back box (51), but it takes no thrust at this point. The back box supports both the lead-screw and the feed rod on the tailstock end of the lathe.

When cutting threads, it is good practice to put a few drops of oil on the lead-screw. Also, put a few drops of oil on your hand and run your hand over the feed rod. This will not only lubricate the parts but keep them from rusting. Oil both bearings in the back box daily.

Headstock

13 to 24-inch

The headstock spindle and shafts on the 13″ to 24″ lathes are all on anti-friction bearings, splash oiled, and require very little attention. The spindle bearings 80 A & B and 121 A & B can be adjusted by removing cover (118) and turning nut (117) to the right as tight as it can be pulled by hand with a spanner wrench.



However, with the normal use of a lathe, it is necessary only at long intervals. Headstocks on the Super Regal Lathes have the reverse to the lead-screw and compounding feed gears built in the head and controlled by two levers on the headstock. The upper small lever in the center of the head reverses the feed and the lower small lever compounds the feed. No reverse plate, quadrant or slip gear is required. The drive is direct from the head through an idler gear to the feed box.

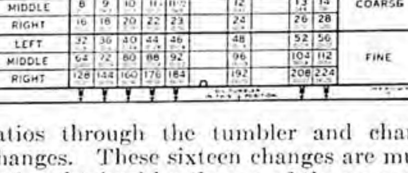
The 13″ to 19″ sizes also have the speed selector on head. A simple device that indicates the cut speed and the proper r.p.m. for all commonly machined materials. The lever positions are indicated on the dial below the r.p.m. Just set the work diameter dimension on the dial at the material being machined and the arrow will indicate the lever setting for most effective speed for the job.

See page 27 for direction operation of feed reverse lever.

Quick Change Mechanism

13-15-inch

The construction of the 13″ and 15″ feed box is similar to the 10″ size: The quick change box gear (27), page 16, is driven by an idler gear from the feed gears in the head. It (27) is mounted on the end of the sleeve gear (35). This sleeve gear runs inside of the quick change box sleeve (36). The quick change box tumbler (37) is mounted on the sleeve (36). The back of the sleeve is cut out to permit the tumbler gear (38) to engage the sleeve gear. By means of the tumbler gear handle (39) the tumbler gear can be rocked into engagement with any of the change gears (40) in cone formation. This is done by pulling out the handle plunger and bearing down on the handle to disengage the tumbler gear, then sliding the tumbler along the sleeve to the proper location and lifting the handle up to bring the gears into engagement. When the gears are in engagement the plunger in the handle locks the tumbler in place so that it cannot kick out when cutting left-hand threads. A series of slots milled in the quick change box sleeve and a pin engaging these slots prevent the tumbler from engaging two change gears at one time. The eight feed changes obtained through the tumbler gear and the quick change gear are multiplied three times by means of the three positions of the compound lever (41), which operates a sliding gear and gives three different gear ratios to the feed shaft. In the feed train two ratios are obtained by compounding feed gears in the head. They are controlled by the lower small handle near the center of the head and are multiplied by the eight different ratios through the tumbler and change gears, making sixteen changes. These sixteen changes are multiplied by the 3 additional ratios obtained by the use of the compound lever, giving a total of 48 changes.



The direct reading index plate is mounted on the quick change box directly over the tumbler sleeve. The numbers on the plate refer to the threads or cuts per inch that the lead-screw and the gear combinations will cut when the tumbler is engaged directly under the number on the index plate. "Coarse" and "Fine" refer to the location of the lever refers to position of compound lever (41).

Quick Change Mechanism

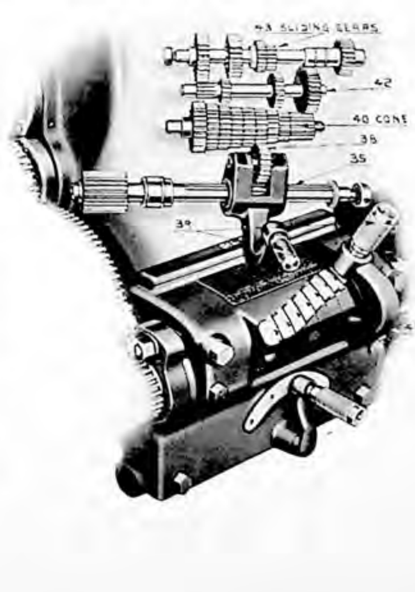
17-19-21-24-inch

All of the shafts in the improved quick change feed box are on anti-friction needle bearings with only two levers to make all changes except compounding, which is accomplished by a single lever on the head. The mechanism is simple, consisting of a cone of gears (40) an intermediate shaft with four gears (42) and a set of sliding gears (43). The shifter, operated by handle (39), slides easily on a splined shaft (35) and carries the tumbler gear (38) which can be dropped into engagement with any one of the gears of the cone (40), and positively held by means of a plunger in the shifter handle. A simple direct reading plate indicates the 56 feed and thread changes.

THREADS PER INCH

A	2 7/8	[illegible]	2 3/4	[illegible]	2 1/2	[illegible]	2 1/4	[illegible]	2	[illegible]	1 3/4	[illegible]	1 5/8	[illegible]	1 1/2	[illegible]	A
B	5 3/4	[illegible]	5 1/2	[illegible]	5	[illegible]	4 1/2	[illegible]	4	[illegible]	3 1/2	[illegible]	3 1/4	[illegible]	3	[illegible]	A
C	11 1/2	[illegible]	11	[illegible]	10	[illegible]	9	[illegible]	8	[illegible]	7	[illegible]	6 1/2	[illegible]	6	[illegible]	A
A	23	[illegible]	22	[illegible]	20	[illegible]	18	[illegible]	16	[illegible]	14	[illegible]	13	[illegible]	12	[illegible]	B
B	46	[illegible]	44	[illegible]	40	[illegible]	36	[illegible]	32	[illegible]	28	[illegible]	26	[illegible]	24	[illegible]	B
C	92	[illegible]	88	[illegible]	80	[illegible]	72	[illegible]	64	[illegible]	56	[illegible]	52	[illegible]	48	[illegible]	B
D	184	[illegible]	176	[illegible]	160	[illegible]	144	[illegible]	128	[illegible]	112	[illegible]	104	[illegible]	96	[illegible]	B

The numbers on the plate refer to threads per inch in one column and feeds in thousandths of an inch in the adjoining column that can be cut with the gear combinations in conjunction with either the lead-screw or feed rod when the tumbler gear is engaged directly below the arrow. The letters A, B, C and D on the left side of the plate indicate the setting of the compound lever (41) on the feed box. The letters A and B on the right side of the plate indicate the position of the feed compound lever on the head. For example, if you want to cut 11½ threads per inch, engage the tumbler (39) under the first arrow on the left side of the plate, set the lever (41) on the front of the feed box in the C position and set the lever on the head in the A position. The same setting engaging the feed rod instead of the lead-screw will give a feed of .016 inch per revolution.



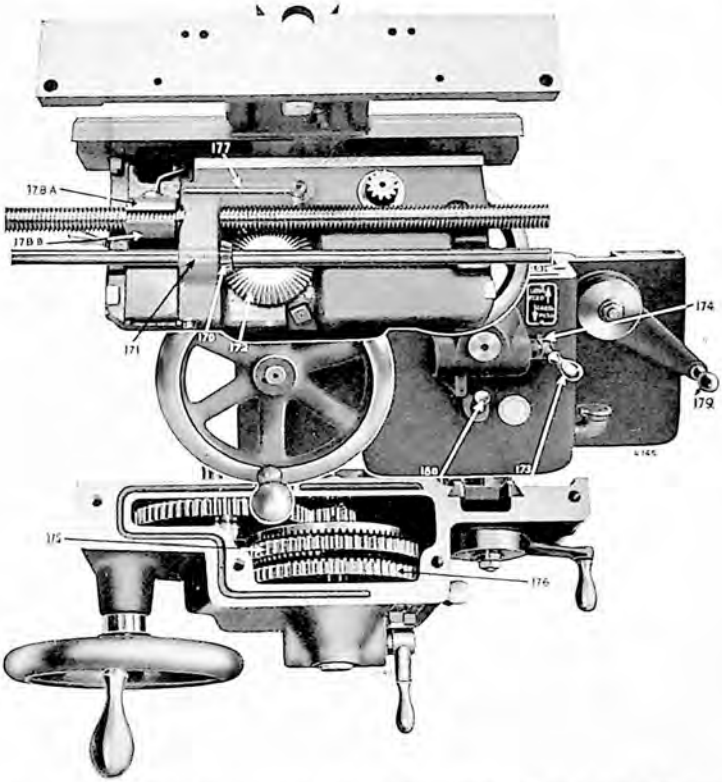
Feed changes can be made while the machine is running, and although the changes can be made under cut, we recommend that the feed trip lever on the apron be disengaged and the feed thrown out. The gears are of alloy steel and will stand any load within the capacity of the lathe, but changing feeds under cut is bad practice because it damages the cutting edge of the tool.

Ample provision has been made for oiling all of the bearings of the quick change box from a large reservoir on the top of the box. Oil holes lead from the reservoir to the various bearings carrying the shafts in the box. The tumbler gear bearings are oiled from the reservoir by placing the tumbler in the central position of the mark "Oil Here". The quick change box reservoir should be filled each morning before starting the operation of the machine.

Apron
13-inch and 15-inch

The apron on the 13-inch and 15-inch lathes is a double walled one-piece casting in which all shafts and gears are supported on both ends. The splined feed rod passes through the feed bevel pinion (170). A key (171) in the bevel pinion engages the spline (keyway) on the feed rod. The bevel gear (172) is always in engagement with the bevel pinion (170), which slides on the feed rod. The feed trip (173) controls both the cross and longitudinal movements, but it is interlocked to prevent accidental shifting from cross to length feed or vice versa. When the shifter handle (173) is moved to the right to clear the safety lug (174) and pressed down, it slides the gear (175) into engagement with gear (176) which is always in mesh with the cross feed pinion in the carriage and the cross slide moves forward or back depending on the direction of the lead-screw as explained on page 20. When the feed reverse lever on the head is in the left-hand position, that is, in the position farthest from the operator, the cross slide moves to the front, toward the operator. When the shifter is moved to the left, past the safety lug and pulled up, with the head reverse lever to the left, the carriage moves toward the tailstock. The direction is toward the head if the head reverse lever is moved to the right, that is, toward the operator.

When the shift lever (173) is in the neutral position, the safety rod (177) is in the slot of the half-nut (11) and allows the half-nuts (178 A & B) to be closed on the lead-screw by lever (179) to chase threads. When the feed shifter (173) is moved up or down,



the safety rod (177) moves to the right or left and locks the upper half-nut so it cannot be moved. When the half-nut is closed, the safety rod is in the slot of the shifter shaft and prevents the movement of the feed lever (173).

A one-shot lubricating system forces oil to all bearings by means of plunger (47).

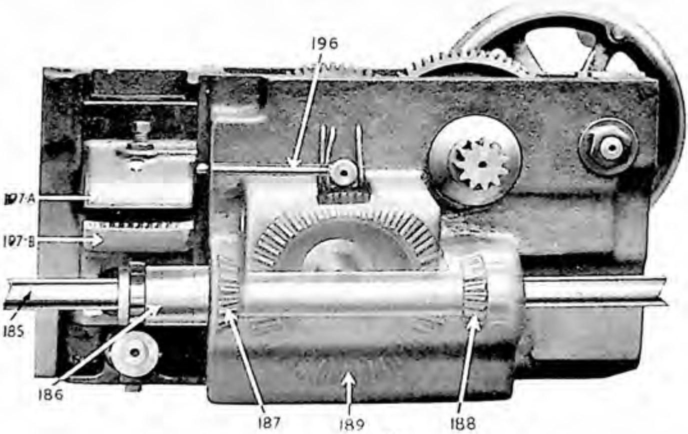
Apron

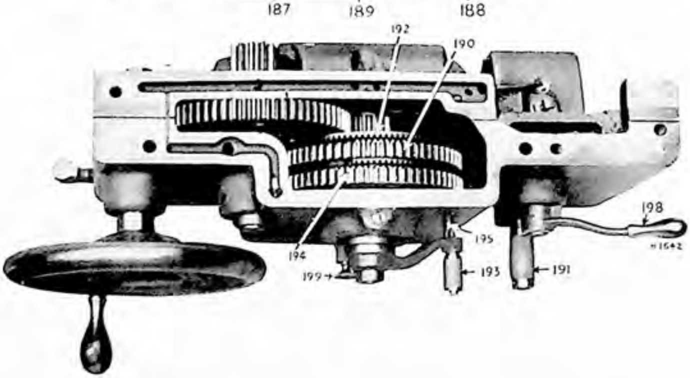
17-inch, 19-inch, 21-inch and 24-inch

The splined feed rod (185) passes through the bevel pinion sleeve (186); the two bevel pinions (187) and (188) are mounted on this sleeve. When the sleeve is in the position so that the pinion

(187) is driving, the feed movement is in one direction, and when the pinion (188) is driving, it is in the opposite direction.

The drive is transmitted from these pinions to the bevel drive gear (189). This gear is pinned to the gear that engages clutch gear (190). When either bevel pinion (187) or (188) is engaged with bevel gear (189) by reverse shifter (191) the feed clutch gear (190) is in motion, and when gear (192) is engaged with it by feed control lever (193), motion is imparted to the carriage. When gear (194) is meshed with the clutch gear the cross slide moves.





In outlining instructions for the operation of the apron, where the feed reverse mechanism is in the head, it is necessary to establish the position of both the head reverse lever and the apron reverse lever. Let us assume that both the feed reverse lever on the head and the apron reverse lever (191) are in right-hand positions, then when the apron feed control handle is pulled up, the carriage moves toward the headstock; when pushed down past the neutral stop (29-S) the cross slide moves toward the operator. If the position of both the head and apron levers are reversed, the movements of the carriage and cross slide will be the same as outlined above. However, if the position of either lever is changed, then the direction of movement will be reversed for both carriage and cross slide, that is, the carriage will move toward the tailstock and the cross slide to the back of the lathe. The use of the lead-screw reverse is particularly valuable in chasing odd threads that cannot be picked up with the thread dial on short threading jobs.

When the handle (191) which actuates the bevel pinion sleeve is in the "mid" position, the bevel pinions (187 and 188) are both out of mesh, the interlock (196) is in a neutral position and the half-nut can be closed.

The half-nuts (197 A & B) are actuated by the handle (198).

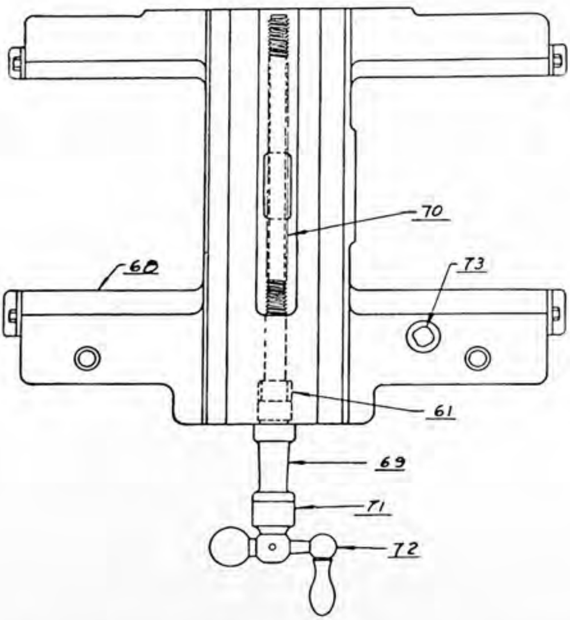
The carriage may be locked in position on the bed by the stud on the right-hand front wing, which tightens the clamps.

A plunger pump on the apron forces oil to all bearings in the apron. Work the plunger (199) slowly in and out three or four times. This should be done daily. Fill the reservoir through the intake on the left side of the apron. The reservoir is full when the oil can be seen in the bottom of the intake. A good grade of machine oil SAE 20 is recommended.

Note—*When the handle* (193) *is not in the "mid" position, it is impossible to move handle* (198) *to close the half-nut. A camming action on pin* (196) *locks the half-nut.*

Carriage

The carriage (68) travels along the bed and is guided by an inverted V way or shear in front and a flat way in back. The movement of the carriage is by means of the gear train in the apron to which it is attached. The bearings (bed ways) are protected by shear wipers to prevent chips and dirt getting between the carriage and the bed. The carriage is gibbed to the bed both in front and back. The cross feed bush (69) forms the bearing for the cross feed screw (70), on the front end of which is a micrometer dial



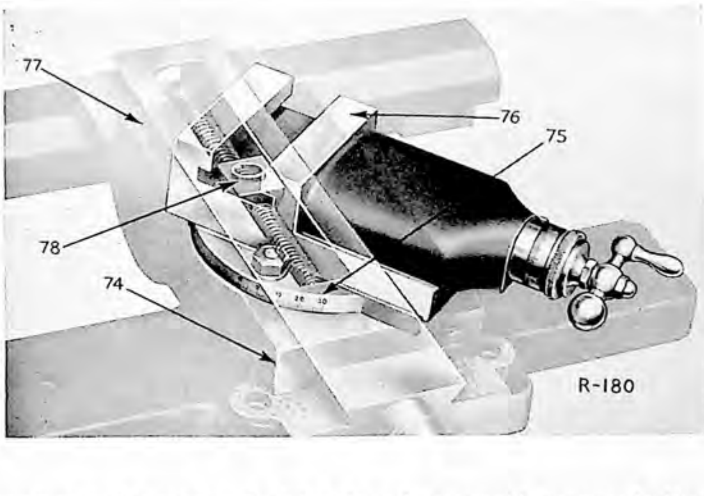
(71) and cross feed handle (72). The carriage clamp screw (73) is used to clamp the carriage to the bed for facing and cutting off operations. Before engaging the longitudinal feed, *be certain that the clamp screw is loose and that the carriage can be moved by hand.*

The carriage is clamped to the bed when the lathe leaves the factory to prevent movement during transit.

Remove the compound rest dirt guard (77) and oil the screw and the cross feed bush. Also see that the dovetailed cross slide is cleaned and oiled occasionally. Oil the felt in the wipers. When working with cast iron, remove the wipers occasionally and clean them in gasoline or kerosene.

Compound Rest

The compound rest unit consists of compound rest bottom slide (74), compound rest swivel slide (75), compound rest top slide (76), cross feed dirt guard (77), and cross feed nut (78). The bottom slide (74) is fitted to the dovetailed cross slide of the



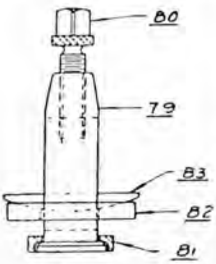
carriage and is equipped with a flat gib to provide means of adjustment for wear. The gib adjusting screws in the side of the bottom slide can be tightened or loosened to obtain the proper adjustment. The bottom slide should be adjusted so that it will move freely on the dovetail and still be a snug fit. The cross feed nut (78) is attached to the bottom slide and runs on the cross feed screw (70), page 28. The compound rest gets the movement on the carriage through the cross feed screw and nut.

The compound rest swivel slide (75) is fitted on top of the bottom slide and swings around to the angles selected. It is clamped in position by two T-slot bolts whose heads are in a circular T-slot in the bottom slide. The swivel slide (75) is graduated in degrees so that the compound rest can be accurately set at the desired angle. This feature is used when turning angles on bevel gears, boring holes having short steep tapers, turning and grinding centers, etc., where the angle is too steep to use the taper attachment.

On the swivel slide a dovetail is planed, to which is fitted the compound rest top slide (76). The top slide is also fitted with a gib for adjustment. A screw with a micrometer dial which engages the nut mounted in the swivel slide provides hand feed to the compound rest top slide.

Tool Post

The tool post unit comprises the tool post (79) itself, with component parts as follows: Tool post screw (80), tool post washer (81), tool post collar (82) and tool post wedge (83). The washer (81) fits the T-slot in the compound rest top slide. The collar and wedge elevate and lower the point of the tool, and the screw is used for clamping. When placing a tool in the tool post, be sure there are no chips or turnings between the collar and the compound rest, or between the wedge and the collar to prevent the tool securing a firm foundation. Also see that the tool does not extend out of the tool post more than is necessary. The compound rest top slide should not extend over the bottom slide when taking heavy cuts, and the tool post should be located as near the center of the top slide as possible. Failure to observe the above precautions will often cause chatter. Do not tighten the tool post screw with a long wrench, but use the wrench provided for that purpose. Clean and lubricate the compound rest slides occasionally. Also put a few drops of oil on the compound rest screw.

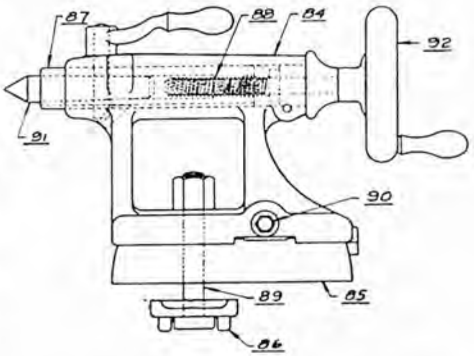


Bed

The bed is the foundation of the lathe. On it the different parts described in the foregoing paragraphs are mounted. The bed has been polished and accurately scraped at the factory, and the length of time that it stays in this condition depends entirely on the operator. *Do not use the bed as an anvil for driving arbors in and out, or as a bench for hammers, wrenches and chucks.* If you have no place to lay your tools, arrange a neat little board at the tailstock end of your lathe, on which you can place them without damaging the bed. Do not lay chuck wrenches across the bed or wings of the carriage or leave tool post wrenches lying on the bed. Many lathes have been wrecked by allowing the carriage to feed against a chuck wrench or a tool post wrench lying on the ways of the bed, between the carriage and the headstock. Also see that the tops of the girths in the bed are free from heavy turnings or chips, as there is only a small clearance between the carriage bridge and the bed girths. Keep the shears clean. Wipe them off occasionally with a rag or waste, following up with a little oil on a piece of cloth. See that the bed rack is kept tight on the bed. Remember, the condition of the bed usually tells what kind of a mechanic has been running the lathe.

Tailstock

The tailstock unit comprises the tailstock top (84), tailstock bottom (85), tailstock clamp (86), tailstock spindle (87),



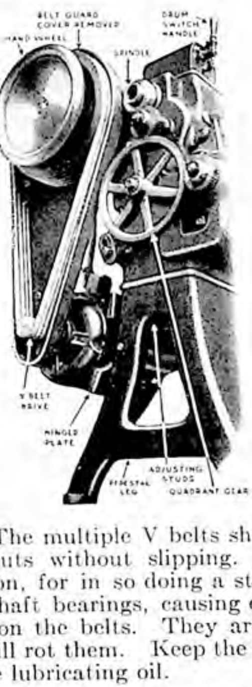
and tailstock screw (88). The entire unit is movable on the ways along the length of the bed to accommodate pieces of varying lengths between centers within the capacity of the machine. The tailstock is kept in alignment with the headstock by a V on the rear shear (way) of the bed and can be clamped in position with the tailstock clamping bolt (89). Before moving the tailstock along the bed, wipe the ways carefully to clean off any chips. Turnings on the ways will throw the tailstock out of alignment.

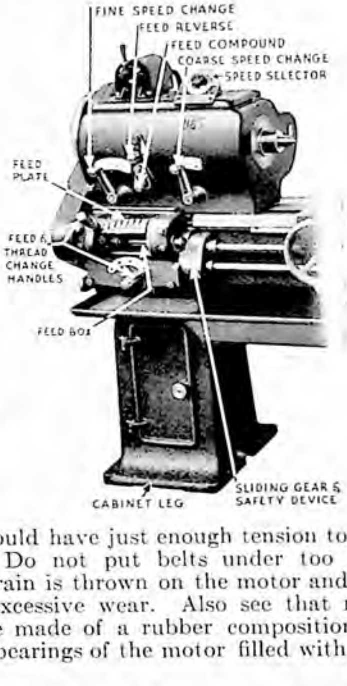
The tailstock top (84) sets on the bottom (85) and is held in alignment by a cross tongue. For turning tapers a setover is provided for the tailstock top. A setover adjusting screw (90) on each side of the tailstock top provides means for setting, and a raised boss in the rear is graduated to show the amount of setover. The tailstock spindle (87) is moved in and out of the tailstock barrel by means of the screw (88), which fits a tapped hole in the spindle. The front end of the spindle is bored and reamed to a Morse taper to hold the tailstock center (91), drills, drill chucks and reamers. To remove the tailstock center, run the spindle back as far as it will go until the center hits the end of the screw, which will force it out of the tapered hole in the spindle. Before replacing the center, carefully wipe out the hole; clean the tapered part of the center; move the spindle forward by a few turns of the handwheel (92), and push center in. When using drills, drill chucks and reamers, be sure they are tight in the taper hole. If they are not tight they will revolve and cut the tapered hole, destroying its accuracy. Should the hole become scored, carefully ream out the burrs or score marks in the taper with a Morse taper reamer.

The design of the tailstock allows the spindle to be clamped in any position by means of a binder screw. The spindle (87) should be removed occasionally, in order to oil the spindle nut and the outside of the spindle barrel.

Motor

The motor is mounted on a hinged plate on the back of the leg below the headstock. The hinged plate is equipped with adjusting screws to regulate the tension on the multiple V belts which drive the lathe. For greater safety, the belts are enclosed in a driving belt guard, shown below with cover removed. Two screws hold the belt guard cover.





The multiple V belts should have just enough tension to take the cuts without slipping. Do not put belts under too much tension, for in so doing a strain is thrown on the motor and driving shaft bearings, causing excessive wear. Also see that no oil gets on the belts. They are made of a rubber composition and oil will rot them. Keep the bearings of the motor filled with high-grade lubricating oil.

Motor Control

The drum reversing switch, above, is mounted on the headstock cover and the wiring to the motor is protected by a flexible conduit. The switch starts, stops and reverses the lathe. When the handle of the switch is vertical, it is in the "off" position. Turning the handle to the right starts the lathe running forward—turning the handle to the left starts the lathe running backward or in reverse. The 21″ and 24″ lathes are also built with a multiple disc clutch and brake. The clutch is operated by convenient handles, on the control rod, on either side of the carriage. An instruction plate with instructions for adjustment of the clutch is on the front of the headstock.

Changing spindle speeds. When you have selected the required speed (see article on selection of proper speeds, pages 103 and 113), consult the speed plate on 10″, 21″, 24″, or speed selector on 13″, 15″, 17″, 19″, for the proper position of levers to obtain that speed. Stop the lathe. By turning the speed change handwheel, the gears are slowly turned so they will mesh with one another. The proper spindle speed will thus be obtained. We do not recommend changing speeds while the machine is running or under cut. The gears are heat-treated, and no damage would result if the operator accidentally shifted gears while the machine was running, but doing this flakes and burrs the edges of the teeth, and it becomes difficult to mesh the gears. Learn to shift the gears in the proper way and you can very quickly obtain any desired spindle speed. These instructions apply to all Regals.

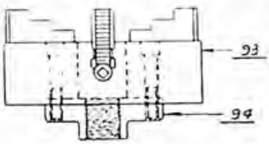
> When ordering repair parts always specify size of lathe and serial number. The serial number is stamped on the flat surface of the front way at the tailstock end of the bed. Prior to 1940 it was stamped on the tailstock end cross girth of the bed.

ACCESSORIES REQUIRED FOR LATHE WORK

For the full and complete operation of a lathe certain tools and accessories are necessary, such as chucks and dogs for holding and driving the work; lathe tools for the actual turning or boring operations, and measuring tools such as calipers, micrometers, scales and gauges. These accessories are illustrated and described below.

Four-Jaw Independent Chuck

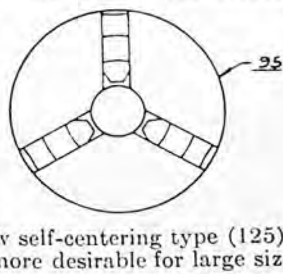
The four-jaw independent chuck (93), as the name implies, has four jaws, each jaw being independently adjusted with a chuck wrench. The jaws are reversible so that the chuck can be used for inside or outside chucking. It is attached to the spindle of the lathe by means of a chuck plate (94), which is threaded to fit the spindle nose. The flange of the chuck plate is fitted to the recess in the back of the chuck and the chuck is bolted to the flange. See page 56.



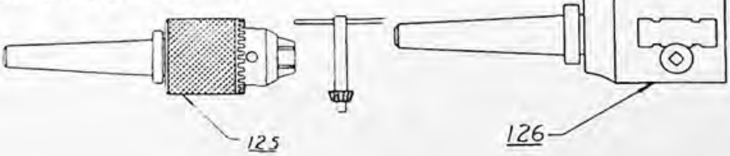
The four-jaw chuck is used to hold rough pieces that are not perfectly round, and other irregular shaped pieces; for example, a cast-iron gear blank.

Three-Jaw Universal Chuck

The three-jaw universal chuck (95) is used to hold pieces that are semi-machined, and parts made of cold-rolled steel or drill rod which is ground to close limits. When the chuck wrench is used, all jaws move to or from the center in unison, gripping the pieces quickly. The chuck is attached to the spindle in the same manner as the independent chuck, described above.

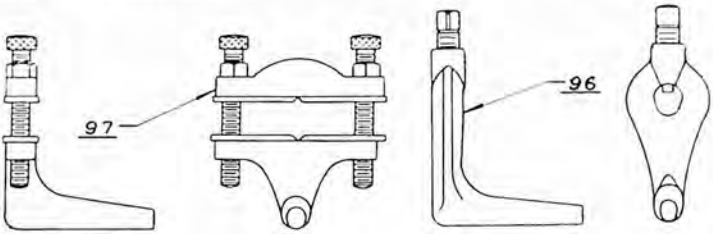


Drill Chuck and Shank. For holding center drills and straight shank drills, a drill chuck with a shank fitting the Morse taper hole in the tailstock spindle is almost a necessity. These chucks are made in the three-jaw self-centering type (125), and the two-jaw type (126) which is more desirable for large size drills, taps, and reamers.



Lathe Dogs

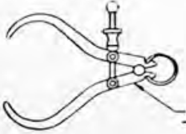
Lathe dogs (96) are used to grip and to drive pieces between centers. They are used with the small face or driving plate (page 47). Dogs of different sizes can be obtained for use with corresponding diameter of work. The usual form of lathe dog has a bent tail which engages a slot in the driving plate. Another form of a dog is known as the clamp dog (97) and it is used to drive square, hexagon or octagon shaped pieces between centers.



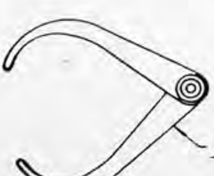
We have described the most essential tools used with lathes. Many other tools are used and some of their uses will be explained in connection with the different lathe operations described later.

Outside Calipers

Outside calipers are used to measure the diameters of the work being turned. There are three kinds: Spring calipers (98), firm joint calipers (99), and micrometer calipers or "mikes" (102). Spring calipers are provided with a screw and adjusting nut for quick setting to size, whereas the firm joint calipers are set by tapping one leg against a solid object. Both types have their advantages. The spring type is much preferred on small work, while on large diameter work, the solid joint type (because of stiffer legs) is better.

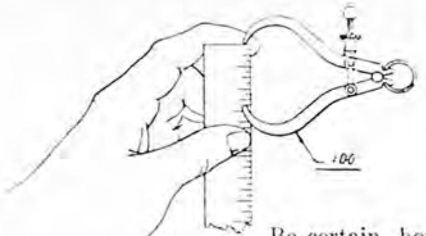


98

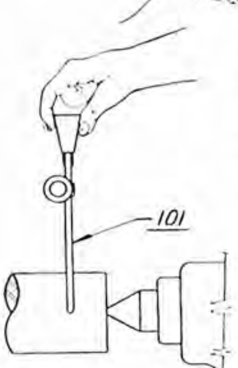


99

To set calipers to a diameter with the use of a scale, hold the scale in left hand and the caliper in the right hand, using the forefinger to keep the one leg from slipping off the end of the scale and adjust the caliper (100) to the dimension



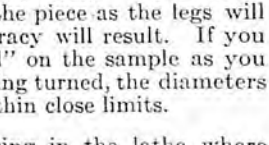
required. Some mechanics become quite expert at setting calipers, acquiring a "feel" that enables them to set calipers to .005 of an inch. In many cases your work will be to reproduce broken parts, and the calipers can be set from the broken part. Be certain, however, that the part is not worn where you set the calipers. Try the calipers at different points to see if the piece is round and that you are not calipering on the smallest diameter. When calipering a piece of work it is best to hold the caliper in a vertical position (101) with the legs at right angles to the axis of the piece, and adjust to a point where the weight of the calipers will just allow the points to pass over the diameter of the piece. This slight resistance is known as the "feel." Never force the calipers over the piece as the legs will spring, and inaccuracy will result. If you get the same "feel" on the sample as you get on the piece being turned, the diameters will correspond within close limits.



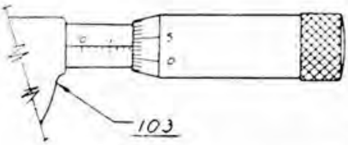
Never try to caliper a piece revolving in the lathe where accuracy is required. For obtaining the approximate diameter this is permissible, but for accurate dimensions, the lathe should be stopped.

Outside Micrometer Caliper

The micrometer caliper (102) is used for measuring to very close dimensions, its graduations reading to one thousandth of an inch. It consists of five principal parts, the frame "A", the anvil "B", the spindle "C", the sleeve "D", and the thimble "E". The spindle "C" has a thread cut on it which fits a tapped hole in the sleeve "D" that is not exposed. The threads are cut 40 to the inch so that exactly one revolution of the thimble "E" advances the spindle "C" one-

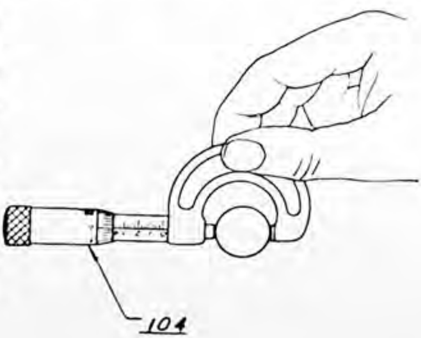


fortieth of an inch, or twenty-five thousandths (.025) which is the gap between the anvil "B" and the spindle "C", the measuring point. The sleeve "D" is graduated with 40 divisions to the inch and numbered 0-1-2-etc. to 10, a number at every fourth division, so that figure one, or four divisions, represent 4 x .025 of an inch (1/40″) or one-tenth of an inch (.100). The number last showing on the sleeve "D" when the "mike" is set on a diameter is the first numeral after the decimal point in your reading. On the bevel edge of the thimble "E" are twenty-five graduations, each of these representing one thousandth of an inch. In illustration (103) you have showing on the sleeve seven graduations representing .025 each, equaling 7 x .025 or .175, and three graduations on the thimble beyond the zero (0) mark, each representing .001, so that the caliper is set at .175″ plus .003″, equaling .178″, or for greater convenience read the highest numeral showing on the sleeve, 1 or one-tenth (.1); beyond this read the number of graduations showing (3), which equals 3 x .025″ or .075″, then add to this the graduations on the thimble, three making .178″. For convenience in using micrometers, tables of fraction and decimal equivalents, also English and metric equivalents, will be found on page 109.



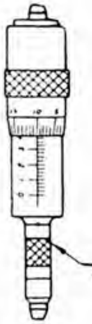
103

When calipering with micrometers the same "feel" is necessary as with other calipers, and they should not be forced over the work. Hold the "mikes" between the forefinger and the thumb (104) and let the weight carry them over the diameter. Do not caliper with the piece revolving as this will damage the anvil and the end of the spindle. Check up the "mikes" occasionally with a reference disc (standard gauge) to see that they register correctly.



104

Inside Micrometer Caliper

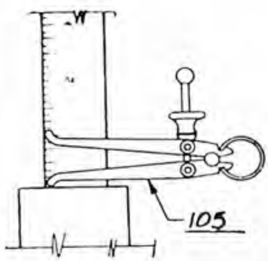


107

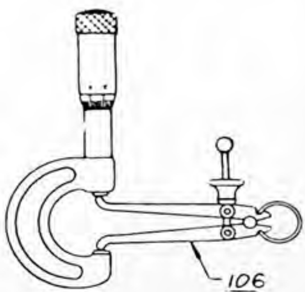
Inside micrometers (107) are manufactured for diameters of one-half inch and up. They are read the same as outside micrometers.

Inside Calipers

Inside calipers are used for internal work such as bored and reamed holes and are made both in the spring and firm joint type. In setting the calipers to a scale, hold the scale against a flat surface, placing one leg against the flat surface and adjusting the other leg to the required dimension (105). If you are boring to accurate dimensions, set a micrometer to the dimension required and transfer this to your inside caliper. This can be done best by holding the micrometer in the left hand and the inside caliper in the right hand (106) and adjusting it until the proper "feel" is obtained. The



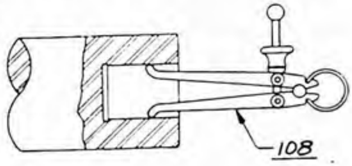
105



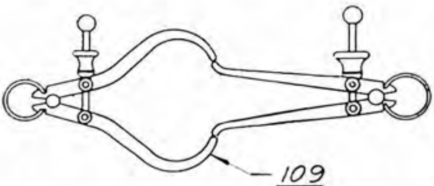
106

adjustment is obtained by rocking the caliper in a vertical plane between the axis of the anvil and spindle of the micrometer.

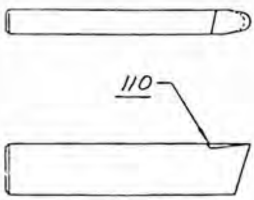
When calipering a hole (108) set one leg of the inside micrometer caliper in the hole, pivoting the caliper in a vertical plane and adjusting until the other leg enters the hole. By rocking and adjusting the caliper, the proper "feel" across the largest diameter can be obtained. If the calipers are forced into the hole, the legs will spring and an inaccurate measurement will be obtained.

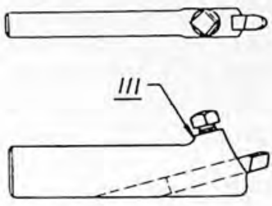


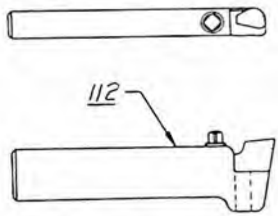
When boring a hole to fit a shaft or turning a shaft to fit a hole, it is necessary to transfer measurement from outside to inside calipers and vice versa. This can best be done by holding the outside caliper in the left hand, supporting it between the thumb and forefinger (109) with the second finger of the left hand supporting the lower legs of the outside and inside calipers to be set. The inside calipers are held in the right hand with the adjusting nut between the thumb and forefinger. By rocking the inside caliper in a vertical plane and adjusting it until the proper "feel" is obtained, accurate transfer from one to the other can be made.



LATHE TOOLS

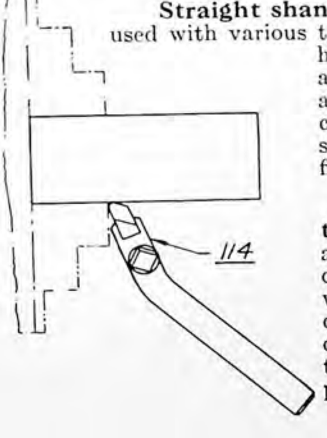




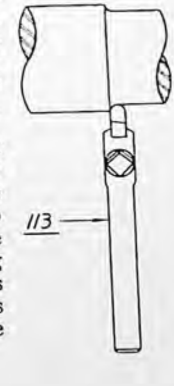


Each lathe should be equipped with a complete set of lathe tools for turning, facing, threading and boring. They can be either of the forged type (110), or of the tool holder type using high-speed steel bits, such as are made by Williams, Armstrong (111) or O. K. (112). The tool holder type is the most commonly used because it is more convenient. No forging or dressing is necessary with this type.

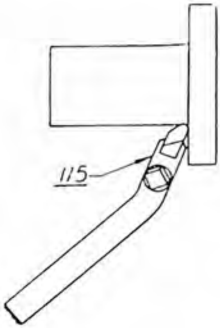
A complete set of tools consists of the following:



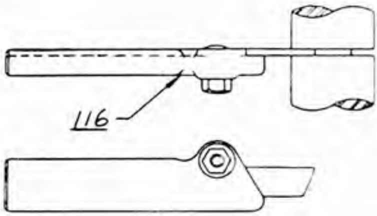
Straight shank turning tool (113), which can be used with various types of tool bits for right and left-hand turning, right and left-hand facing, and threading. Bits can also be ground to special shapes for fillets in corners, etc.



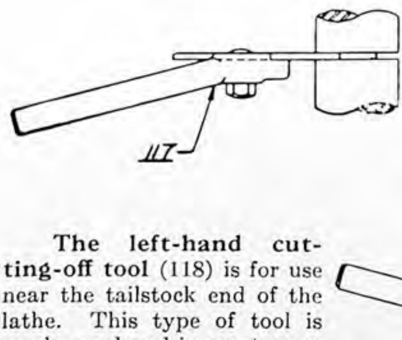
Left-hand offset turning tools (114) are desirable when the operator wishes to work close to the chuck or the driving dog. By using this type of tool it is possible to turn the piece up to the chuck and at the same time have the chuck clear the side of the carriage and compound rest.



Right-hand offset turning tools (115) answer the same purpose when working at the tailstock end. Various shaped bits can be used in either right or left-hand turning tool holders to suit the work being done.

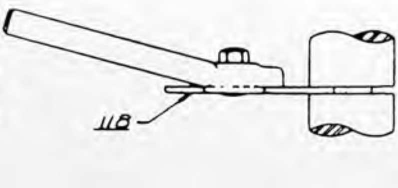


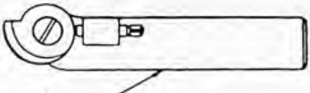
Straight cutting-off tools (116) are used for cutting off in the lathe. The blade used in this tool is ground with the proper side clearance. Then, when properly set in the holder, the tool does not drag.



The right-hand cutting-off tool (117) is used for cutting off work close to the chuck. To prevent chatter, when using a tool of this type, excessive overhang of the work should be avoided.

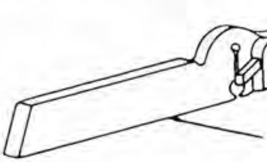
The left-hand cutting-off tool (118) is for use near the tailstock end of the lathe. This type of tool is rarely used and in most cases the right-hand cutting-off tool serves all purposes.





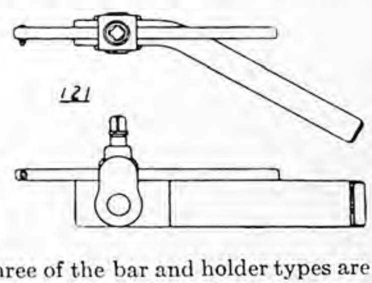
119

Threading or chasing tools (119). There are many patented types of threading tools. The tool shown is provided with a formed cutter, and is ground on the top to maintain the correct thread form. On some classes of material where it is not possible to cut a smooth thread with a rigid chasing tool, a spring threading tool (120) will produce better results.



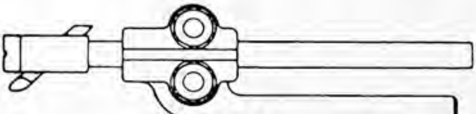
120

The spring threading tool is built with a nut for the lockable spring head which provides rigid backing for coarse threads and heavy cuts, and when loosened, the holder becomes a spring tool for finishing work. Neither of these tools is absolutely essential as most threads can be cut by grinding a tool bit to the proper shape, with the use of a thread gauge. The bit is held in a turning tool holder.



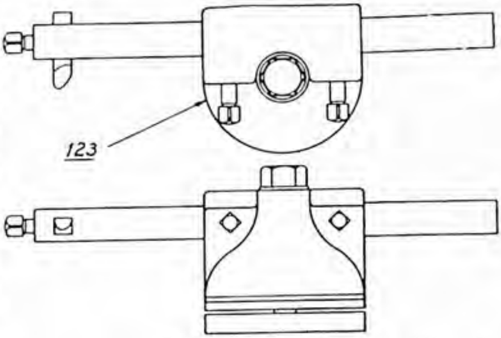
121

Boring bars and holders. Tools of the type shown are used for boring holes and chasing internal threads. Two basic types of boring bars are used, namely, the forged type and the bar and holder type. Three of the bar and holder types are shown in the accompanying illustrations. No. 121 is used for small holes, which in addition to holding bars with bits as shown, can be used to hold boring tools made of drill rod for very small holes, and also for holding drills for drilling. The boring bar shown in illustration (122) is used for medium size holes and is adjustable for different depths of holes. It is held in the regular tool post by a shank, the same as the turning tool.

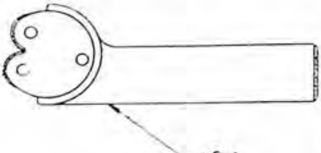


122

Figure (123) shows a boring bar for heavy work that is held in the compound rest T-slot. This makes the bar more rigid than when held in the tool post. Various shaped tool bits for boring, facing, counterboring and threading holes can be used in the different bars.



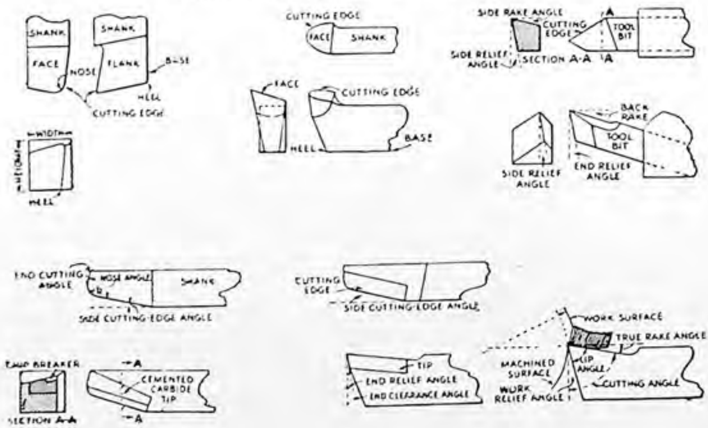
123



124

Knurling tool (124). On some classes of work such as thumb screws, it is necessary to roughen the diameter to give a better grip and prevent oily fingers from slipping. This is called knurling and is used extensively on optical, radio and other electrical work. A tool holder carrying two knurled wheels is fed into the work while it is revolving and impresses a pattern on the work. Wheels of different patterns can be obtained for the knurling tool holder.

Nomenclature of Lathe Tools



REGAL EQUIPMENT AND ATTACHMENTS

The compound rest was developed to give a rigid yet adjustable mounting to the tool.

In facing, for instance, where the carriage is locked and the cross feed is used to traverse the tool across the work, the compound rest may be set with the screw parallel to the axis of the lathe. In this position the micrometer dial on the screw can be read directly to determine the necessary movement of the compound rest to secure the proper depth of cut in the face.

The compound rest is made to swivel a full circle of 360° to allow convenience of set-up and also to enable facing or turning to be done at any angle with axis of work.

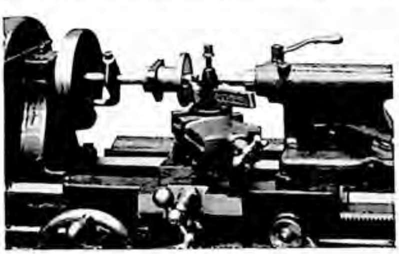
The compound rest is graduated and the bottom slide marked with the zero line at the point where the compound rest would face at 90° to axis of lathe centers.

When turning or facing a taper, it is necessary to note the reference axis of the angle on drawing, or whether included angle of taper is given.

If for instance the angle is given as 60° from axis of work the compound rest should be set at 90° minus 60° or 30° from the zero position which is at right angles to line of center of lathe.

If a tapered piece is marked to be turned 32° included angle of taper, the taper from axis to side is one-half the included taper or 16° and compound rest should be set at 90° minus 16° or 74° from zero position.

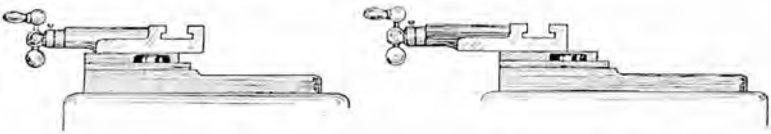
When taking finish cuts to secure a given size in facing, the compound rest, as illustrated, may be set with the axis of the compound rest screw 60 degrees from the axis of the lathe (at 30 degrees) on the compound rest graduations. In this position one-thousandth of an inch motion of the compound rest along the screw will move the tool exactly one-half thousandth of an inch in the direction of the axis of the lathe. This rule may be relied upon implicitly as it is a trigonometric fact and not subject to variation.



To Duplicate a Tapered Surface Using the Compound Rest

(a) Mount sample between centers. (b) Mount test indicator in tool post. (c) Swivel compound rest to approximate angle and set indicator to contact on center line of sample. (d) Run compound rest along work and note error. (e) Reset compound rest to correct error and repeat (d); continue until error is within limit of accuracy desired. (f) Remove sample and insert work piece. (g) Set cutting tool of proper shape so edge is exactly on center line of work. (h) Turn, using compound rest screw to obtain feed travel, until correct size plus finish allowance. (i) Finish by grinding or filing and polishing as in other work.

Various attachments may be bolted to the compound rest, thus widening the scope of operations possible with the lathe.



Caution. Note position of compound rest slide in the two extremes. In the forward position the slide overhangs the rest to which it is clamped. In this position it is not advisable in any lathe to take roughing cuts as top slide may break in the middle of the tee slot. When taking heavy cuts, always have top slide flush with bottom slide so the metal is all in compression.

The steady rest is used to give support to long round pieces, or bars of small diameter, rotating between centers, and to provide a fixed support between the headstock and tailstock ends of the work while it is being turned. The steady rest makes it possible to support a piece of round work of small diameter (see page 67), so that it can be accurately turned with a faster feed. Otherwise it would spring away from the tool and chatter. Many jobs are impracticable without the use of a steady rest.

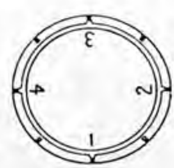
The steady rest is also used when it is necessary to drill or bore the end of a piece of work which is too long to be held entirely by a chuck on the headstock spindle. The steady rest is clamped on the bed and is aligned on the lathe by a bearing which fits the same inverted vee and aligns the tailstock with the headstock.

The follow rest is used to support the bar or round piece of work being turned between centers, against the forces of the cut being taken. It is bolted to the carriage and consequently moves with it. (See page 68). To set the follow rest, the cut is started and turned slightly longer than the width of the follow rest jaws. The jaws of the follow rest are then set to the turned diameter, after which the cut can be taken across the entire length of a piece of work of small diameter.

Face Plates. The Regal Lathe is furnished with one small and one large face plate. The small face plate is used to drive pieces of work such as shafts, mandrels, etc., by means of a lathe dog clamped to the work. The lathe dog is made with a bent tail which engages a slot cast in the face plate. The face plate also has other cored slots radial to the center, which makes it convenient to mount special drivers or fixtures to the face of the plate.

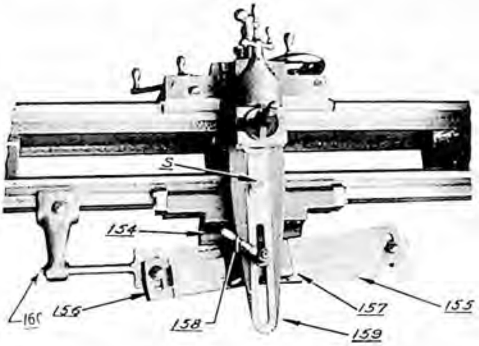
The large face plate performs the same functions as the small one, except that it provides more room to mount fixtures such as angle plates, vee blocks, etc. The large face plate can also be drilled for the planer type hold-down stops to hold large irregular pieces that cannot be accommodated in chucks of the conventional type. See page 67 for details.

Thread Indicator. The chasing dial thread pick up is supplied as an extra to the lathe. This unit comprises a worm wheel which meshes with the lead-screw, and a shaft connecting the worm with the indicator dial. The dial is marked with numbered lines as illustrated. When chasing an even number of threads, the half-nut may be engaged at any line on the dial; for odd threads, at any numbered line; and at any odd numbered line for half-threads.



Using the chasing dial in this manner the operator can take a cut, back the tool out of cut and return the carriage to the starting position, set the tool for the next depth of cut, and re-engage the half-nut without stopping or reversing the lathe spindle.

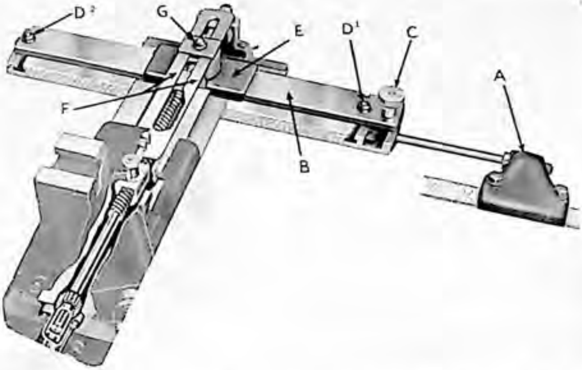
The plain taper attachment on the Regal Lathe is simple but exceptionally rugged and accurate. The principal parts are the carriage bracket (154) (a substantial casting bolted to the rear of the carriage), in which is machined a dovetail slot to take the dovetail on the lower side of the slide (155), the adjustable taper or swivel bar (156), graduated on one end in degrees and the other in inches, and the shoe (157), which slides over the bar and follows the taper at which the bar is set. The clamp handle (158) clamps the shoe to the extension on the cross slide bottom slide (159). The cross slide with the extension is a special casting comprising the bottom slide and extension in one piece, thus eliminating any inaccuracy due to loose joints, etc., that are inherent in a two-piece construction. This special casting is supplied with the taper attachment and is included in the published taper attachment price.



When preparing to use the taper attachment, first set the turning tool on line with the centers of the lathe, advancing the tool to the approximate position of the cut to be turned, then remove the screw "S", next set the swivel bar at the desired angle or taper and tighten the clamping screws. Bolt the bracket (160) to the bed and tighten handle (158). The final adjustment of the tool for size, after screw "S" is removed, must be made by the movement of the compound rest.

The telescopic taper attachment is ruggedly constructed and simple to operate. When the carriage is brought into position for the taper operation, the bed bracket (A) is tightened on the flat bed way. The swivel guide bar (B) is adjusted to the selected taper, which is marked in inches on one end of the lower bar and in degrees on the other end of the bar. Adjustment, for the selected taper, is made by means of the set-over knurled knob (C) that engages the

Telescopic Taper Attachment



rack in the end of the taper bar to move the swivel bar to the desired taper. It is held secure by the screws (D^1-D^2) on the ends of the bar. Now with the taper bar clamp screws loosened and assuming that the shoe bracket gib and the carriage bracket gib are properly adjusted, we are ready to chase taper threads or to finish taper turning, or finish boring. When carriage feed is engaged, bed bracket and its connecting rod hold lower taper bar and adjustable swivel bar in a fixed position with relation to bed and work. Movement of carriage slides gibbed shoe (E) along taper bar.

With light cuts the pull is on the cross feed screw and nut and the end of the screw telescopes in the cross feed telescopic bush, allowing the slide to move in or out.

On heavy cuts, none of the pull is on the cross feed screw, but on the flat draw bar (F) shown over the cross feed screw, which is connected to the bottom slide of the tool rest. On heavy cuts the clamping nut (G) over the sliding shoe is tightened and the action of the sliding shoe is the same as on light cuts, except that the pull is then on the draw bar. This relieves the cross feed screw of all strain and pull and wear, and insures longer life and also retains the original accuracy of the screw. Adjustment of the cut is made by loosening clamping nut (G) and setting tool to proper diameter and, of course, tightening nut again after adjustment is made. Most adjustments of the tool are made by the compound rest. There is no necessity of adjusting clamping nut after taper is set.

Grinding Attachment. The grinding attachment is a complete unit with its own motor, and it is ready for use when it is bolted on the compound rest and attached to a light socket.

There are many grinding jobs that can be done advantageously in a lathe with a suitable attachment. Hard centers can be repointed, cutters and reamers sharpened, and all kinds of straight and taper cylindrical and internal grinding can be done in a lathe.

There should never be more than a few thousandths left on the work for grinding.

Grinding a Mandrel

The illustration shows a mandrel being ground in a lathe. The procedure for this job is to set the compound rest at "0" or with the axis of its screw perpendicular to the axis of the lathe, and clamp the grinding attachment with the axis of its spindle parallel to the surface to be ground. Since mandrels are ground with a slight taper, it is necessary to set the tailstock over or else use the taper attachment. When the taper is set and all is ready, start the grinder and take a very light cut across the work to make sure the taper is correct. Then run the wheel back and forth across the work by power, feeding it in toward the center about .0005" each time until the mandrel is ground to size. When a diameter has been ground to size, run the wheel over the work a few times, without feeding in to allow it to "spark out." This produces a better finish.

For internal grinding, small wheels are used on special quills or extensions screwed on the regular spindle. For wheels less than 2" diameter the pulleys should be reversed to give the spindle greater speed.

When grinding attachments are used on a lathe, utmost care must be exercised that no grinding grit is allowed to work into the slide bearing surfaces.

Safe Speeds

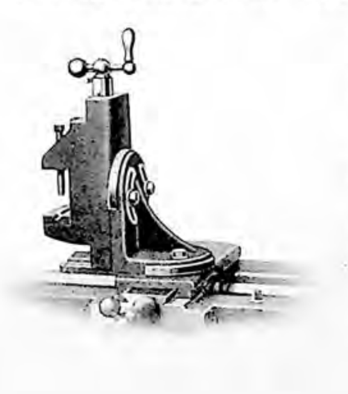
Size of Wheel	2"	3"	4"	5"	6"
Spindle R. P. M.	9550	6380	4775	3825	3190

The micrometer stop is useful when the operator is obliged to bring the carriage to the same definite position a number of times, or where a series of cuts are desired at accurate decimal dimensions from one another, as cutting threads, etc.

The micrometer stop is clamped to the front vee or way of the lathe bed. The design of the clamping surfaces is such that there is no danger of scoring the vee of the bed while using this attachment. A micrometer screw with hardened ends is turned by a large diameter collar. The collar is knurled to provide a finger grip for turning and is graduated to show the travel of the screw in thousandths of an inch. The carriage is brought up against the end of the stop by hand in actual use.

Draw-in Attachment. For turning small parts from round finished stock there is no more accurate, rapid, and economical method than the use of a draw-in collet chuck. This attachment in the spindle allows stock bar to pass through the collet tube and to be gripped in the draw-in collet ready to be machined. When applying the draw-in attachment to the lathe, be sure that the ends of the spindle hole and the outside of the collet closer are clean. Any chips or particles of dirt on these surfaces will destroy the accuracy of the work.

Milling Attachment. The milling attachment is used to cut slots, keyways, or flats on shafts or small pieces of work. Screw heads can be slotted, short length racks cut, etc.



The attachment consists of a graduated (swinging 90 degrees) angle plate support mounted on the cross slide in place of the compound rest. A vise is mounted on a slide which is in turn bolted to the angle plate. Vertical motion is imparted to the vise slide by a screw mounted in the vertical slide which engages a nut in the vise slide. The vertical slide screw is equipped with a graduated collar reading in thousandths of an inch travel.

The vertical slide swivels 45 degrees each side of the vertical position, and the angle plate assembly swivels 360 degrees on the cross feed slide, giving universal setting possibilities.

Horizontal feed parallel to the axis of the lathe is imparted to the carriage and consequently to the milling attachment vise in the regular manner, while the motion in and out, with respect to the center line of the lathe, is derived from the cross feed screw.

With the use of this attachment, the field of work possible on the lathe is extended very much, since for light work many milling operations can be accomplished in the lathe instead of a heavier, more expensive milling machine.

The Garrett Millerette Converter is recommended where accurate indexing is required. The "Millerette" gives a lathe the wide range of a complete milling machine with a dividing head. It is a lathe attachment that takes the place of a milling machine for all ordinary purposes. The "Millerette" on a lathe makes it possible to cut gears of all kinds—spur, bevel and angular—to graduate diameters, to cut various types of keyways, to spline, to slot, in fact to do any milling machine and dividing head work.

The principle used in the dividing head construction is that regularly used on gear cutting machines, the interchange of gears.

For division from 2 to 360 the index plate shows the proper gears to use and the number of turns required of the index lever. It is easy to set up and does not require a skilled operator. The lever or handle makes one or more complete revolutions for each divisional space.

METHODS OF HOLDING WORK IN THE LATHE

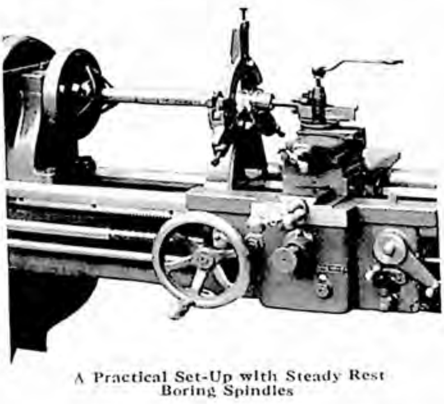
For practical purposes there are five general methods of holding work in a lathe: between centers, on mandrels, in a chuck, clamped on the face plate, and clamped to the carriage.

There are many combinations of the above methods such as chucking one end of a piece and using the steady rest to support the other end while facing, or boring the free end of the work. Another is to hold one end of the work in a chuck and the other end on the tailstock center, etc.

Work on Centers

By far the greatest percentage of work machined on a lathe is held between centers. With this method the work revolves on conical holes drilled in the ends to fit the lathe centers and is driven by a lathe dog (page 36) clamped on the work. The bent tail of the dog fits the cored slot of the driving plate, illustration (page 87). The tool is fed along the work by the movement of the carriage.

A variation of the above method of holding work is to use a chuck to clamp and drive one end of the work while the other end is supported by the tailstock center; the tool is fed by the carriage as before.



A Practical Set-Up with Steady Rest Boring Spindles

Another variation of center holding is to support the work on the headstock center and drive it by a lathe dog, using a steady rest to support the piece at the tailstock end. With this method, however, it is necessary to provide straps or some similar method to hold the work up tight on the headstock center, otherwise the piece will work off center and destroy the accuracy of the cut or spoil the piece of work.

Work on Mandrel

Many parts such as bushings, gears, collars, etc., require all the finished external surfaces to run true with the hole which extends through them. That is, the outside diameter must be true with the inside diameter or bore.

General practice is to finish the hole to a standard size, that is, exactly right within the limit of the accuracy desired. Thus a ¾″ standard hole would ordinarily be held from .7495″ to .7505″ or a tolerance of ½ thousandth of an inch above or below the true standard size of exactly .750″.

The usual practice in machining work of this kind is to drill or bore the hole to within a few thousandths of an inch of the finished size, then remove the remainder of the material with a machine reamer, following with a hand reamer if the limits are extremely close. The piece is then pressed on a mandrel. A dog is clamped on the mandrel and is mounted between centers. Since the mandrel surface runs true with respect to the lathe axis, the turned surfaces of the work on the mandrel will be true with respect to the standard hole in the piece.

A mandrel is simply a round piece of steel of convenient length which has been centered and turned true with the centers. On mandrels of about ⅝″ and smaller, the whole length is usually tapered. The common practice is to make the small end ¼ to ½ thousandth of an inch under the standard size of the mandrel, while the large end is about ½ to 1 thousandth of an inch over standard size. This taper allows the standard hole in the work to vary according to the usual shop practice, and still provides a drive to the work when the mandrel is pressed into the hole. On mandrels over ⅝″ diameter about two-thirds of the length is turned straight, about ¼ thousandth of an inch undersize, and the other third tapered up to about two thousandths of an inch oversize for drive. Some are made with a very gradual taper on two-thirds the length from ½ thousandth of an inch undersize on the small end to standard at the end of the two-thirds length, with the remaining third tapered about two thousandths of an inch for drive.

Where the hole in the work piece is not of standard size or if no standard mandrel is available, a soft mandrel may be made to fit the particular piece.

When pressing a mandrel into work, it is well to remember that clean metallic surfaces when pressed together sometimes gall or stick. A few drops of oil on the mandrel before pressing it into the work will prevent sticking.

Commercial mandrels are made of tool steel, hardened, drawn, and the working surface ground, with the centers lapped for accuracy. Each end is turned smaller than the body of the mandrel and provided with a flat which gives a driving surface for the lathe dog. The size of the mandrel is always marked on the large end to avoid error, and for convenience when placing work on it.

Testing a Mandrel

It is necessary, of course, to have the centers true in both the head and tailstock spindles and to have the tailstock set to turn straight, otherwise the finish turned surface will not be true.

When finish turning accurate work it is well to test the mandrel between centers before placing any work on it. The best test for run-out is made with an indicator.

When taking roughing cuts on a piece of work mounted on a mandrel, it is necessary to have a tighter press fit than for finishing. Therefore, on pieces with a thin wall or section of metal, it is advisable to remove the work from the mandrel after the roughing cut and reload lightly on the mandrel before taking the finish cut.

Where close limits are to be held, it is also advisable to see that the work is not hot when the finish cut is taken, as the cooling of the piece will leave it undersized if it has been turned to the exact size.

Types of Mandrels

In addition to the solid mandrel just described, there are expansion mandrels of various types, and the straight mandrel as shown in the illustration.

In the expansion type shown a hardened taper mandrel spreads the split cast-iron sleeve, expanding it to fit the hole and to drive the piece under the cut. With the straight mandrel there is no friction drive on the mandrel itself, it being necessary to have a shoulder on one end with a nut and collar on the other end to clamp and drive the work.

The expansion plug is another type of mandrel. The part holding the work is similar to the expanding end of an expansion mandrel and it is supported on a taper plug which fits in the headstock spindle tapered hole. The work holding portion of the expansion plug is bored out and split and a taper headed screw expands the sections which grip the hole of the part to be machined.

Proper Fit of Fixtures Mounted on Spindle Nose

The small face plate is usually used only as a drive plate for work held on centers. As a general practice work is not mounted on the small face plate. For this reason the fit of the small face plate on the threads of the spindle nose is not important and may vary from "snug" to "loose" without harm.

With the use of the large face plate and chuck plates, however, the work is held and positioned by the plate, and any shake in the plate from a loose thread fit on the spindle nose would show up as an inaccuracy on the finished work.

Therefore, large face plates and chuck plates are finish tapped and fitted to the lathe spindle as tight as the plate can be screwed on the clean well lubricated spindle nose.

On a well fitted thread the bearing will be distributed evenly on the sides, and the position of the plate will be the same each time it is mounted.

Fitting Chuck Plates to Chucks

The accuracy of a three or four-jaw chuck in holding work in the proper position is largely a question of the care used in fitting the chuck plate to the chuck.

It is imperative that the chuck plate should have a good fit on the spindle nose threads. The chuck plate should be screwed on the spindle nose without undue force and yet not be loose enough to allow shake on the threads. The chuck plate on all Le Blond Lathes is held true to the lathe spindle axis by the fit of the spindle nose threads in the threaded bore of the chuck plate. The chuck plate is held square with the lathe spindle axis by the fit of the chuck plate shoulder against the square face of the spindle shoulder.

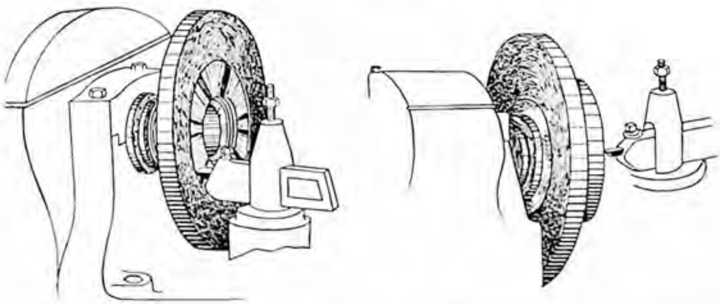
Chuck plates are supplied by Le Blond in both semi-fitted and full-fitted styles.

A semi-fitted chuck plate is machine bored, machine tapped and counterbored, and is then "finish hand" tapped, to a plug screw-gauge fit, before being shipped.

The full-fitted chuck plate, as its name implies, is completely machined and assembled to the chuck so that no further work is necessary.

So much of the accuracy depends on the thread fit (accurately obtained only by use of a precision ground finish tap such as we use in our plant) that we strongly recommend the use of genuine Le Blond semi-finished chuck plates in fitting up chucks to Le Blond Lathes.

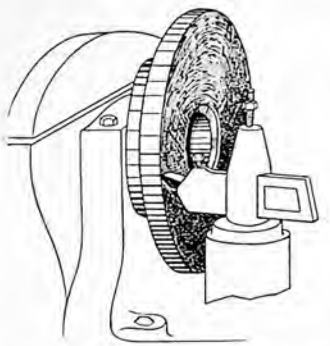
The procedure to fit plates is as follows:



First, screw chuck plate on spindle in regular position about two-thirds of way up to shoulder so that bearing of chuck plate is entirely on threads. Take rough and finish cuts on face of chuck plate so that this finished surface will be square with thread and large enough to clear diameter of collar used in Step No. 2, above.

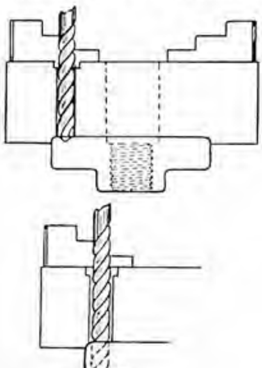
Second, reverse chuck plate on spindle and insert a collar with true parallel faces between spindle shoulder and part of chuck plate just machined.

This collar should be wide enough to extend from spindle collar to the first full depth thread, since the face of chuck plate is not counterbored as is the back. Next take a skim cut off end of hub. Clean up counterbore of hub at 45 degrees and true up diameter of hub and take cut off back of chuck plate.



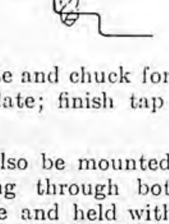
Next take chuck plate off; remove collar; clean spindle nose and chuck plate thread and screw chuck plate on spindle in proper position. Rough face front face and rough turn outside diameter $\frac{1}{32}$-inch above diameter of counterbore in chuck. Also counterbore tapped hole at 45 degrees to ⅛-inch across flat; next take finish cut off face of chuck plate.

Caliper counterbore of chuck and transfer size to outside mikes. Finish turn OD to fit counterbore allowing for a slight tap fit.

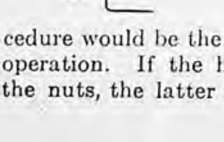


Next transfer the holes in the chuck to the chuck plate, using the chuck as a drill jig.

Lightly tap the chuck plate into the chuck counterbore and spot drill through the chuck body to the plate with body size drill. Without removing chuck plate, drill in spotted holes with the proper tap drill for the screws furnished with the chuck.



Start taps into tap holes, tapping from front of chuck to insure proper tap alignment.



Center punch plate and chuck for location; remove chuck plate; finish tap holes and file off burrs.

The plate can also be mounted with the chuck bolts going through both the chuck body and plate and held with nuts on the face of the chuck plate. The procedure would be the same as the foregoing, omitting the tapping operation. If the hub on the plate is shorter than the width of the nuts, the latter method cannot be used.

When body holes are not drilled through chuck, chalk chuck plate face thoroughly, wipe off mating face of chuck back, then tap chuck plate in position in chuck counterbore. Outline of bolt holes will show up on chalked surface when chuck body is removed. Center punch plate and chuck for location and remove. Mark off holes, center punch circle and center for drilling. Drill with body size drill to allow clearance for chuck screws.

Clean chuck counterbore and chuck plate thoroughly. Reassemble in proper position and insert and draw screws up tight.

When chuck plate assembly is put on lathe spindle the body of chuck should run true.

A true piece of short shaft when clamped in a universal chuck should run true within .002″ if work has been properly done.

Work in Chuck

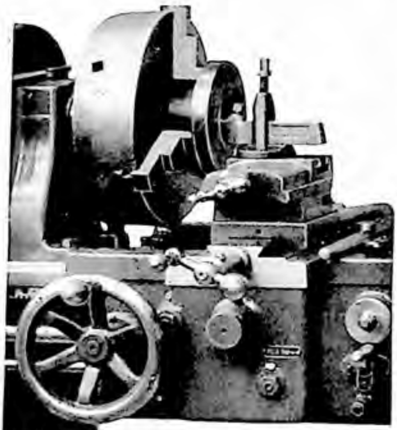
A chuck is usually employed to hold work which may be large in diameter in proportion to its length, irregular in contour, extremely long, which makes it necessary for the piece to be passed through the spindle, or for pieces to be drilled or bored on the axis of the lathe. Two types of chucks are commonly used, four-jaw independent chucks and three-jaw universal chucks.

When putting on chucks or face plates, care should be taken to see that the threaded hole in the chuck plate is clean and free from dirt. Also wipe off any dirt or grit from the spindle nose and put a few drops of oil on the threads.

Do not try to run the chuck on the spindle nose when the machine is running. This may damage the thread on the spindle, or even more disastrous, the operator's sleeve may be caught in the revolving jaws and cause him serious injury.

The safe and better procedure is to hold the chuck and revolve the spindle by hand until the internal thread in the chuck plate engages the first thread on the spindle nose, then run the chuck on by hand. For a heavy chuck, a board should be laid across the wings of the carriage to support it, moving the chuck toward the spindle by means of the carriage handwheel. Keep a board of the proper thickness for this purpose.

When removing the chuck, the foregoing operation should be reversed. When the chuck sticks, which occurs occasionally, run the lathe at the slowest spindle speed and jar the chuck loose with a piece of babbitt metal or wood block set endwise on the lathe bed in a position tending to stop the motion of the slowly revolving jaws. This binding or sticking of the thread can be prevented by cutting a collar from blotting paper and putting it between the spindle shoulder and the chuck plate.



Work held in a 3-Jaw Universal Chuck

The three-jaw universal or scroll chuck is made so that all jaws move together or apart in unison. The combination scroll chuck allows each jaw to be moved independently in addition to the universal movement.

We do not recommend the use of the combination chuck.

Mounting and Adjusting Work in Chucks Three-Jaw Universal*

(a) Adjust jaws roughly to size required. Place wrench in any pinion socket and turn to right to advance jaws toward the center.

(b) Place work in chuck, seating face of work against vertical faces of jaws.

(c) Tighten jaws, as indicated above.

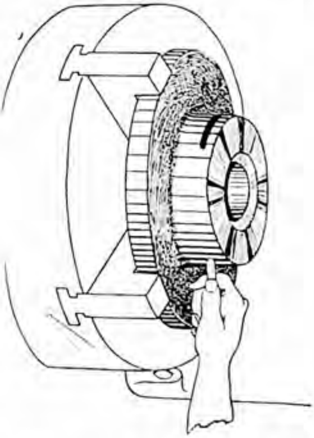
(d) Revolve lathe at speed set for the operation to be performed.

*A "Universal Chuck" is called universal because the jaws are all operated at once, universally, being moved toward or away from the common center together when the wrench is applied to any one of the pinions.

(e) Run carriage up to work, rest hand on carriage and hold chalk to just touch revolving work as indicated below. Chalk will touch high spot, indicating high side.

(f) Loosen chuck jaws, revolve work one-third or one-quarter turn, tighten and retest. Repeat if necessary.

(g) If work still runs out, mount a tool in the tool post, back-end to, and while revolving lathe very slowly by hand or power, adjust cross slide until butt of tool holder just clears high spot on work. Revolve work one-half revolution and note amount of error.



(h) Select shims one-half as thick as the observed run-out. Loosen jaws and insert shims between work and the jaw or two jaws nearest the chalk mark on the side.

(i) If work must be chucked very accurately it is easier and much more accurate to use an indicator and secure an exact reading of the run-out in thousandths of an inch.

When work is finally centered, be sure that all jaws bear on work hard enough to drive work without slipping while under cut.

When work is to be roughed, and then finished while held in a chuck and the section of metal is not solid, it is usually advisable to release pressure on jaws and then reclamp lightly for finish turning. Otherwise the work may be turned "round" while in a sprung condition. When pressure is finally released, the work assumes its normal position and the turned or bored position is no longer round.

The four-jaw chuck has, as its name implies, four jaws, each individually adjustable with an appropriate wrench. Where considerable adjusting is necessary, it is much simpler and quicker to use a four-jaw independent chuck. This type of chuck is the most adaptable of all for regular machine shop practice as the equalization of jaw position may be made so much more quickly, easily and accurately with the individual jaw adjustment.

Work held in a 4-Jaw Independent Chuck

Using the four adjustments, any shaped piece of work may be positioned to bring the desired point on the work over the axis or center of lathe.

When chucking an irregular piece in the four-jaw chuck so that a round boss will run true, the following procedure should be followed: The piece should be inserted in the chuck and the jaws brought down to an approximate clamping position. The piece should then be held flat against the back face of the jaw steps and clamped. Next, the lathe spindle should be rotated either by hand or slowly by power, and a piece of chalk held to touch the high spot as the piece revolves. The screw or two screws directly opposite the chalk mark should then be loosened slightly, and the opposite screw or screws tightened and the chalk test repeated. A few trials should be sufficient to locate the work in the desired position.

The same procedure is followed in clamping semi- or fully finished pieces in the chuck, except that the position is necessarily held to a closer limit before chucking is considered completed. An indicator of the dial type may be used to ascertain the run-out if the limit is extremely close.

A universal chuck will center almost exactly at the first clamping, but after a period of use it is not uncommon to find inaccuracies of from 2 to 10 thousandths of an inch in centering the work and consequently the run-out of the work must be corrected.

Sometimes this may be done by tightening the screw over the high spot, but at other times it is necessary to pack a piece of paper or thin shim stock between the jaw and the work on the high side.

After the positioning has been accomplished in a chuck, be sure to tighten all the screws evenly, so that each jaw is tight against the piece to prevent it slipping under cut.

When chucking thin sections be careful not to clamp the work too tightly, as then the diameter of the piece will be turned round when it is in a distorted position. When the pressure of the jaws is released after the cut, there will be as many high spots as there are jaws, and the turned surface will not be true.

Care of Chucks and Face Plates When Not in Use

Face plates, chucks, etc., when not in use, should be hung on pegs so that the threads are kept free from dirt and chips.

Caution

Some operators, through a mistaken zeal in attempting to have all parts working very freely, will lap out the threads in the chuck or face plate, by placing grinding grit and oil on spindle nose and then screwing face plate back and forth till fit is loose enough to suit them.

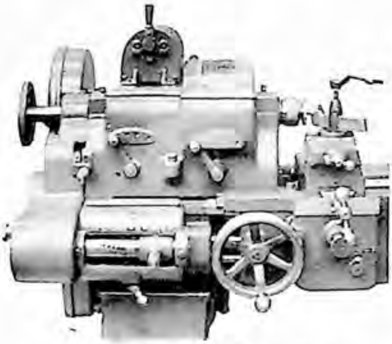
This is not to be recommended.

In the first place, the spindle nose will be worn below standard size, thus ruining the fit over all other fixtures previously fitting on spindle correctly.

In the second place, large face plates and chuck plates should fit tight.

In the third place, with the ordinary grits employed, the cast iron becomes charged with grit and continues to cut even after a loose fit is obtained.

The recommended procedure is to write our factory stating how far plate will screw on spindle nose, both in regular and reversed positions, and send plate in to be tapped larger, or request loan of spindle tap which will be forwarded to you through agent from whom you bought the lathe. Your only charge will be parcel post both ways for the tap.



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 81.

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.

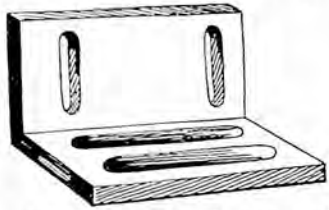
Work on Face Plate

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 47) 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.

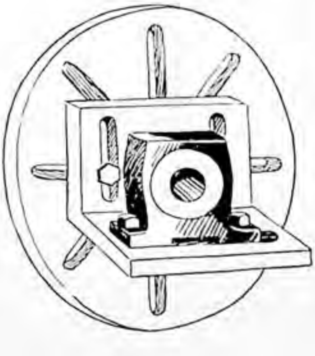
The large plate is usually used in a different manner.

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.

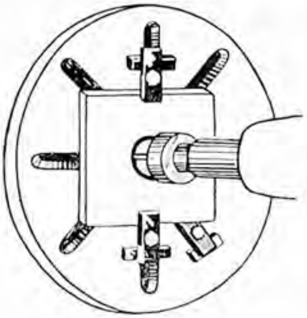
For the purpose of fastening work to the face plate, radial slots are provided to engage bolts used to clamp the work.



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.



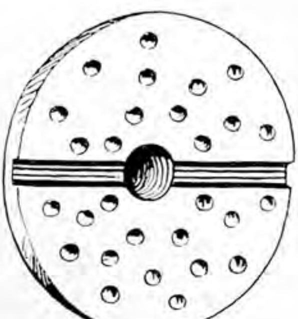
3. Trepanning may be easily accomplished by bolting material to face plate, as illustrated.

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.

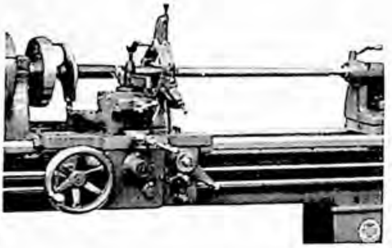
For finishing, use right-hand finishing tool as before, except that tool should be fed into work at center so as to leave no point at exact center of work. The tool is then fed outward as before.

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 53, 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 5/8" to 3/4" 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.

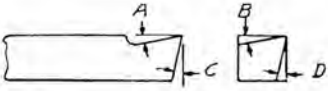
HOW TO GRIND LATHE TOOLS

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 tungsten carbide. The stellite and tungsten carbide tools are used only on high production work in large manufacturing plants because their cost is too great for universal use. There are still some carbon steel tools used, but the general practice is to use high-speed steel tool bits in holders. Be sure, however, to determine the kind of a tool you are grinding, as carbon and high-speed steel require 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.

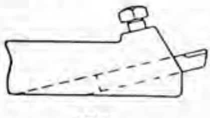
Carbon steel tools should be ground on a grindstone or wet emery wheel or dipped into water often to keep the tool cool while grinding to prevent the drawing of the temper. On the other hand, high-speed steel tools should not be dipped into water when hot as it will crack the tool and crumble the cutting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

Four angles are important in grinding tools and these vary with the material being machined. The top rake (see illustration 127) and page 44 "A", the side rake "B", the front clearance "C" and the side clearance "D".



127

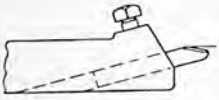
The top rake is usually provided for in the tool holder by the tool being set on an angle (128), which is correct for the machining of steel and cast-iron. On solid steel tools it is necessary to grind the top rake in the tool. By adjusting the tool in the tool post through wedge or rocker, this top 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.



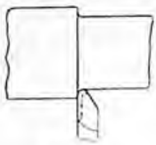
128

129

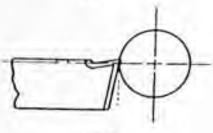
For turning brass, there should be no top rake (129) and the cutting edge of the



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.

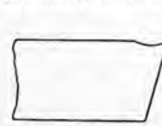
The tool is ground with the side clearance "D" (127) to take care of the feed advance and to prevent the dragging of the tool on the shoulder formed by the cut (131). This angle is usually about 6 degrees from the vertical and is constant.

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.

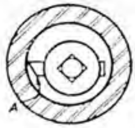
After a tool has been ground on the emery wheel, it will produce better work and last longer if the cutting edge is stoned with an oil stone. This takes out the wheel marks and gives a smooth cutting edge. Care must be taken in grinding cut-off tools to see that both sides of the tool have the necessary 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

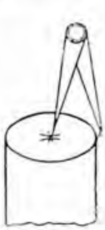
In grinding boring tools, see that the front clearance is sufficient to prevent the tool from rubbing in the hole and dragging at the point "A" (135).



135

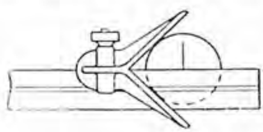
Centering Work

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.

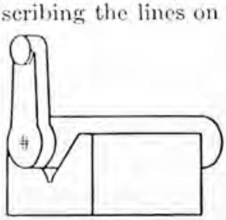

136

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

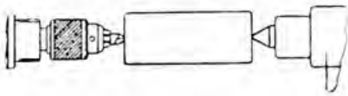
To center irregularly shaped pieces such as a drop forged brake lever, lay the piece in a V-block (138) on a plane surface and use a surface gauge for scribing the lines on each end. First set the gauge to the approximate height of the center and scribe a line. Turning the piece, scribe three more lines,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

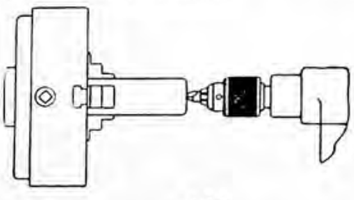
139

Center Drills (139) are usually used for centering. The drill is held in a drill chuck (as shown in Fig. 125 or 126, Page 35), 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 32). 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

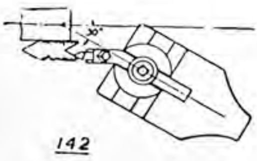
To center finish ground stock such as drill rod or cold-rolled steel, where the ends are to be turned and must be concentric with the unturned body, other methods must be used.



141

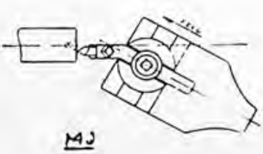
If the piece is small enough in diameter to pass through the spindle, it can be held in a universal chuck on the spindle nose (141), or a draw-in collet (Page 64). Hold the center drill in a drill chuck in the tailstock spindle and center drill one end; reverse the piece, and proceed as before.

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).

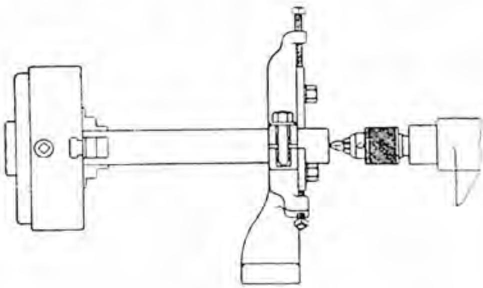


143

144

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

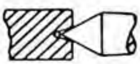
To center long pieces that will not pass through the spindle, a steady rest is used to support the outside end of the piece as near the end as possible (145). The drill and chuck are held in the tailstock spindle and the drill fed to depth by advancing the tailstock spindle. To correct any run-out of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



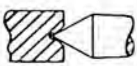
145

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.

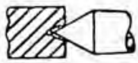
146



147



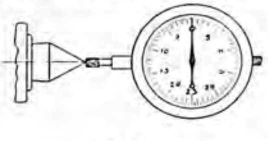
148



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 32.

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.

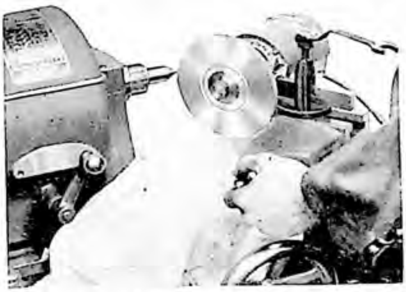
The third cause of center run-out is inaccuracy of the center point itself. Since both hard and soft centers may be used in the headstock spindle, two methods of correction are outlined on next page.

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 76.

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:

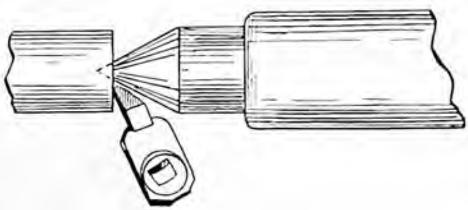
1. Measure piece to determine how much stock is to be taken off.

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

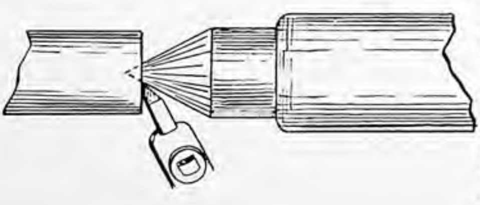
Put red or white lead or other center lubricant in center on tailstock end.

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.

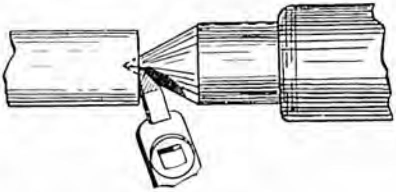
There should be no burr on the tool point such as would be caused by running point of tool into tailstock center when removing ridge around center hole.

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.

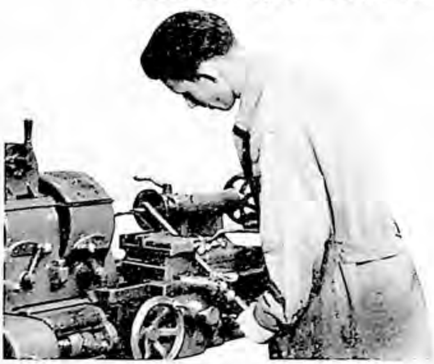
Chatter marks on finish cut are caused by one of the following:

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 may 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 as illustrated. 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 53.

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.

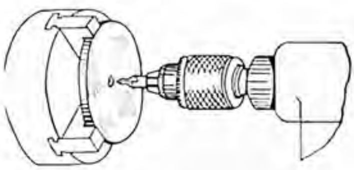
Boring tools may be either of the solid forged type or of the inserted cutting bit type as shown on pages 41 and 43. The use of either depends upon the operator's choice or preference with reference to the respective costs.

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.

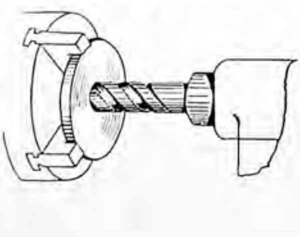
The three above operations, while distinct items of individual operations, are often performed in sequence on a single piece of work at the same work setting, that they will be considered together.

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, pages 59 and 65.)

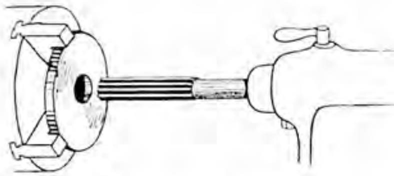
Spot center with center drill to provide true point of entry for small drill.



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.



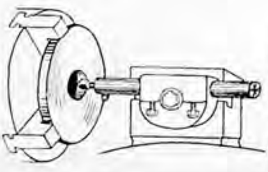
If hole is to be just a rough job both as to size and straightness, use large drill $\frac{1}{64}''$ or $\frac{1}{32}''$ smaller than finish size, and run through piece. Drill is held in taper hole in tailstock spindle if taper shank drill is used, or in drill chuck mounted in tailstock spindle if straight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

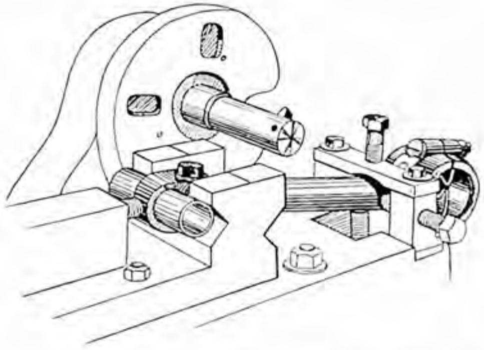
Boring is very accurate with work mounted on a face plate and offers possibilities in boring a number of holes at accurate center dimensions by use of buttons (see page 64), disks or some similar method of location.

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 80, 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.

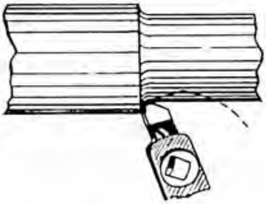
Straight Turning on Centers

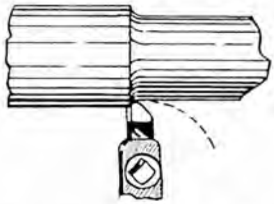
Be sure lathe centers are in alignment and that live center runs true.

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 $1\frac{1}{2}$ 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 46.

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 103 and 113) 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 103) and set feed for finish cut.

Set tool for a light cut to true up all around; turn about 1/4" 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 1/4". 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.

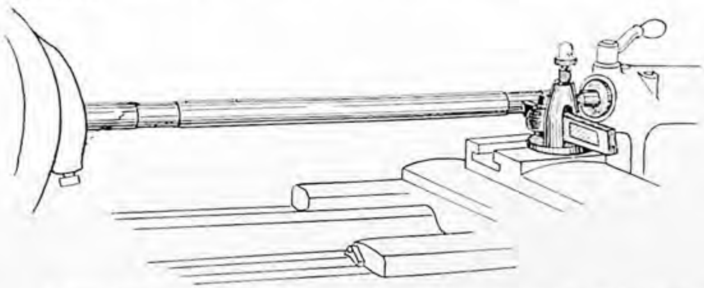
The finish turned surface of steel is usually considerably smoother if cutting oil or soapy water is dropped slowly over the work and tool point on this cut. The liquid may be dropped from an oil can, saturated sponge or waste squeezed in the hand, or by a regular drip spout connected to a can of solution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

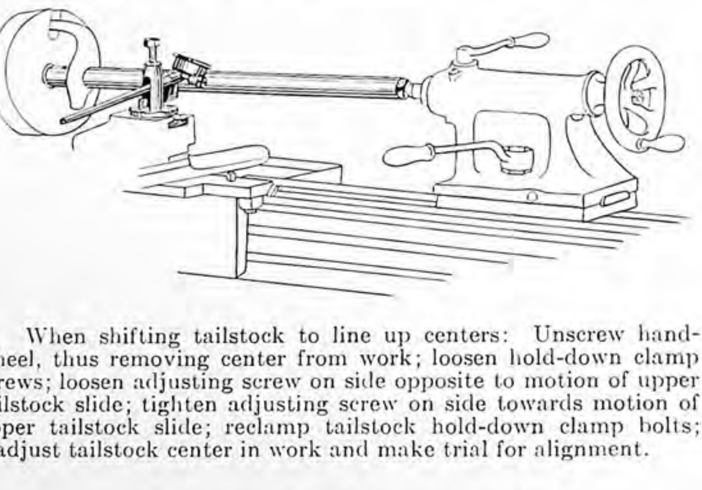


A test bar as shown in illustration is easy to make and use and gives positive results.

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 hand-wheel, 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.

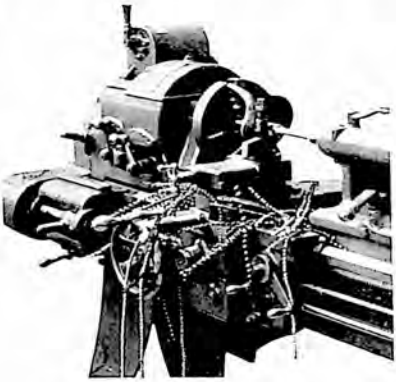
Taper Turning and Boring

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:

$$S = \frac{T}{2} \times \frac{L}{12} \quad \text{where} \quad \begin{array}{l} S = \text{setover in inches.} \\ T = \text{taper per foot in inches.} \\ L = \text{length of piece in inches.} \end{array}$$

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 48.

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 48), 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 $\frac{3}{4}''$.

J. Put in finishing tool and take cut 8 thousandths deep to leave taper stick out from gauge $\frac{3}{8}''$ after cut. This $\frac{3}{8}''$ 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 $\frac{1}{8}''$ away from final position before turning is completed. The last $\frac{1}{16}''$ or $\frac{1}{8}''$ 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 $\frac{3}{8}''$ 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 Set-Over 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 48).

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.

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

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 69 to 71.

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 82.

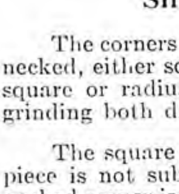
A third variation is possible. The work may be mounted on the carriage or cross slide and the hole bored out by a tool bit mounted in a boring bar held between centers of the lathe. This method is very useful when boring long holes or where the piece is mounted easier by bolting down flat than by holding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

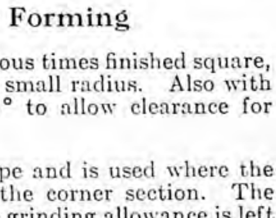
The tool bit may be held in any practicable manner, such as clamping by set screws, by screw nuts or by taper clamping plugs. Bars of this kind are easily made and present no difficulty, since all that is necessary is to take a bar as large as can be passed through rough hole with 1/8" clearance—long 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.

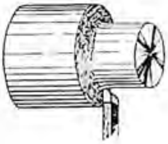
Shoulder and Radius Forming

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.

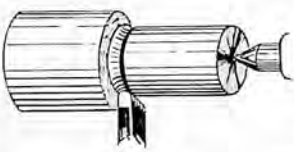
The square corner is the simplest type and is used where the piece is not subject to excess stress at the corner section. The necked corner is used as above, also where grinding allowance is left in turning. The undercut neck prevents undue wear on the corner of the grinding wheel. This design is weaker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







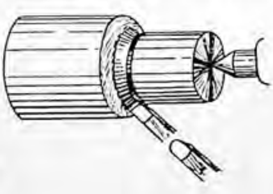
From the design standpoint, the rounded fillet corner is the best due to its strength. It is more costly in production, however, and so is used only in certain classes of work (where vibration is present) to prevent crystallization of steel at sharp corners.



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 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 sleeve of shirt 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.

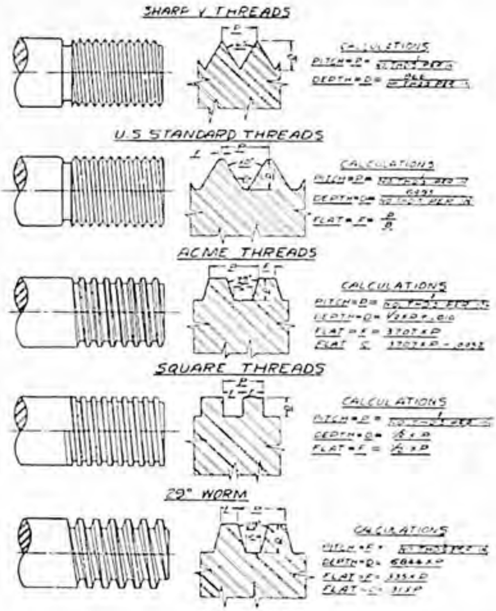
The lead-screw unit on a lathe is rotated by means of a gear train connection between the lead-screw and the spindle. A nut mounted in the apron engages the lead-screw to move the carriage. If, for instance, the lead-screw is 6 threads per inch, then for each revolution of the lead-screw the carriage 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.

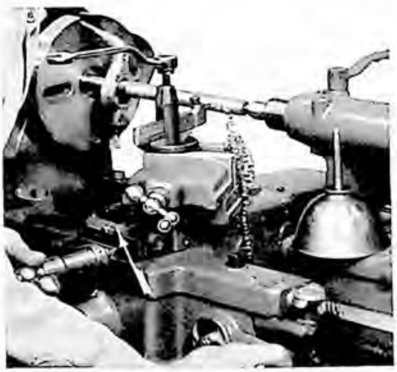
When cutting threads, it is necessary to set the tools at a right angle to the piece to be turned; that is, the axis of the thread tool should be exactly 90 degrees from the axis of the work. This is easily done by use of a thread-setting gauge shown below and on page 72, No. 142. Edges of the gauge 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 lead-screw 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, and consists of a worm wheel meshing with the lead-screw, 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 46. 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 chasing stop indicated by the arrow in the illustration may be used in several ways to aid the operator when cutting threads. When the lathe is set for the first chasing cut, set the stop and clamp it to the carriage dovetail. Bring the head of the screw up to the chasing stop 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

*On the 13" to 24" lathes the more accurate way is to reverse the lead-screw by means of the feed reverse lever on the head.

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:

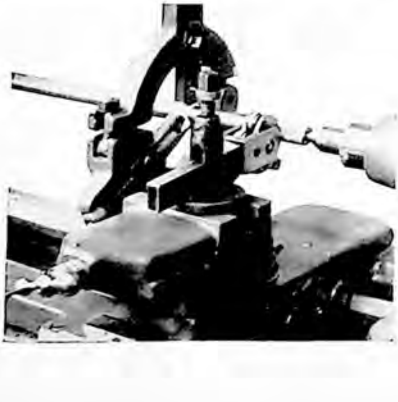
Thread-Cutting Data

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 a 4″ diameter rotating at 60 R. P. M., will have $\frac{4 \times 3.1416}{12} \times 60$, or 60.3′ per minute. The formula for surface speed is diameter

*For Cemented Carbide Tools, see page 113.

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. In this connection it may be mentioned that 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, take a case 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 with the second setup.

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, the work should be run as fast as the tool will properly stand to secure the maximum speed in this operation. A sharp tool should be used when finish turning or the tool whetted 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.

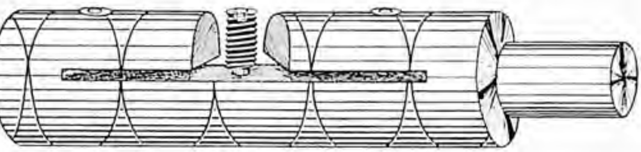
Laps are made in both the solid and the expanding types. The expanding type is preferred to the solid type because it can be expanded, trued up, and recharged when the cutting surface of the lap is worn down.

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 $\frac{1}{4}''$ to $\frac{3}{8}''$ 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″ 21″—24″	127T—120T	$\frac{3}{16}$ M. M. Pitch to 16 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 large 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.

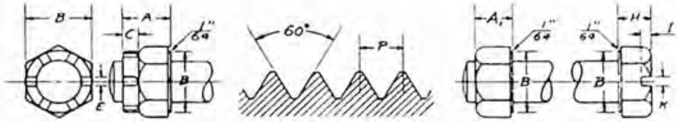
Allowances for Fits

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 1.5 + 1/4" = Length of thread.
$\frac{P}{8}$ = Width of flat.

D	1/4	5/16	3/8	7/16	1/2	9/16	5/8	11/16	3/4	7/8	1	1 1/8	1 1/4	1 3/8	1 1/2
N	28	24	24	20	20	18	18	16	16	14	14	12	12	12	12
A	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	1	1 [illegible]	1 1/4	1 [illegible]	1 1/2
A1	[illegible]	[illegible]	[illegible]	3/8	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	7/8	[illegible]	1 [illegible]	1 [illegible]	1 [illegible]
B	7/16	1/2	9/16	5/8	3/4	7/8	15/16	1	1 1/16	1 1/4	1 7/16	1 5/8	1 13/16	2	2 3/16
C	[illegible]	[illegible]	1/8	1/8	[illegible]	[illegible]	1/4	1/4	1/4	1/4	1/4	[illegible]	[illegible]	3/8	3/8
E	[illegible]	[illegible]	1/8	1/8	1/8	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	3/4	3/4
H	[illegible]	[illegible]	[illegible]	[illegible]	3/8	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	3/4	[illegible]	[illegible]	1 [illegible]	1 5/8
I	[illegible]	[illegible]	1/8	1/8	1/8	1/8	1/8	1/8	1/8	1/8	1/8	[illegible]	[illegible]	1/4	1/4
K	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]
d	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	1/8	1/8	1/8	1/8	1/8	1/8	[illegible]	[illegible]	[illegible]	[illegible]
S	7/32	17/64	21/64	3/8	7/16	1/2	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	1 [illegible]	1 [illegible]	1 [illegible]	1 [illegible]

U. S. Standard Threads, Bolts and Nuts

The tap drill diameters in the table provide for a slight clearance at the root of the thread in order to facilitate tapping and reduce tap breakage. If full threads are required, use the diameters at the root of the threads for the tap drill diameters.

Diameter	No. of Threads per Inch	Diameter at Root of Thread	Diameter of Top Drill	AREA IN SQ. IN. Of Bolt	At Root of Thread	Tensile Strength at Stress of 6,000 Pounds per Square Inch	DIMENSIONS OF NUTS AND BOLT HEADS				
1/4	20	0.185	13/64	0.049	0.026	160	1/2	0.578	0.707	1/4	1/4
5/16	18	0.240	1/4	0.076	0.045	270	19/32	0.686	0.840	5/16	19/64
3/8	16	0.294	5/16	0.110	0.068	410	11/16	0.794	0.972	3/8	11/32
7/16	14	0.345	23/64	0.150	0.093	560	25/32	0.902	1.105	7/16	25/64
1/2	13	0.400	27/64	0.196	0.126	760	7/8	1.011	1.237	1/2	7/16
9/16	12	0.454	31/64	0.248	0.162	1,000	31/32	1.119	1.370	9/16	31/64
5/8	11	0.507	17/32	0.307	0.202	1,210	1 1/16	1.227	1.502	5/8	17/32
3/4	10	0.620	41/64	0.442	0.302	1,810	1 1/4	1.444	1.768	3/4	5/8
7/8	9	0.731	3/4	0.601	0.419	2,520	1 7/16	1.660	2.033	7/8	23/32
1	8	0.838	55/64	0.785	0.551	3,300	1 5/8	1.877	2.298	1	13/16
1 1/8	7	0.939	31/32	0.994	0.694	4,160	1 13/16	2.093	2.563	1 1/8	29/32
1 1/4	7	1.064	1 3/32	1.227	0.893	5,350	2	2.310	2.828	1 1/4	1
1 3/8	6	1.158	1 7/32	1.485	1.057	6,340	2 3/16	2.527	3.093	1 3/8	1 3/32
1 1/2	6	1.283	1 11/32	1.767	1.295	7,770	2 3/8	2.743	3.358	1 1/2	1 3/16

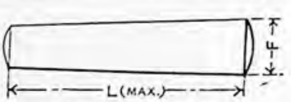
Standard Taper Pin Reamers*

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

No. or Taper Pin	Diameter F at Large End of Pin	Approx. Fractional Size F	Maximum Length L
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	....	.....

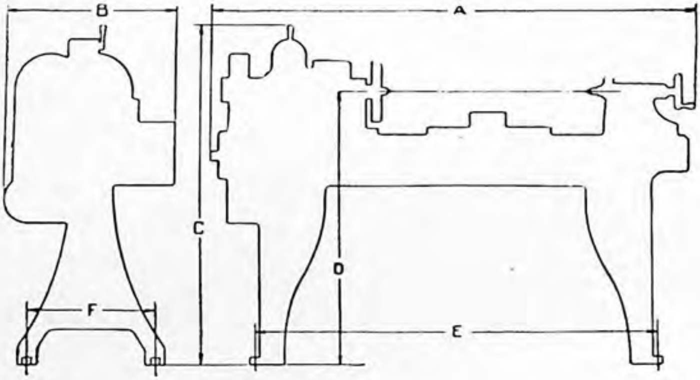
(1) Interrupted cut.

(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	B		Total Height C, Inches	Height to Center D, Inches	APPROX. DISTANCE BETWEEN BOLT HOLES	
			Inches	With Taper Att., Inches	Without Taper Att., Inches			E, Inches	F, Inches
			FLOOR TYPE						
10	18	3	44 3/16	24	21	48 3/8	39	35 3/4	18
13	18	4	58 1/2	33	28	51	42	45 7/8	21
15	18	4	61 1/2	36 1/4	30 1/4	51 1/2	42 1/2	47 1/2	21 1/2
17	30	6	78	40	35	54	42 3/4	63 1/2	19 3/4
19	30	6	84 1/4	42	35	54	42 3/4	64	20 1/2
21	36	7	86 3/4°	59	46 1/2	56 3/8	42 1/2	78 1/4	21
24	36	7	86 3/4°	59	46 1/2	57 1/8	43 3/4	78 1/4	21
			BENCH TYPE						
10	18	3	44 3/16	24	21	22 7/8	13 3/8	43	7 3/4
13	18	4	58 1/2	33	28	27	17	45 3/4	9 1/2

*Base Lengths—For longer beds, add additional length.
°With disc clutch and brake add 7 3/4".

CHAPTER INDEX

ALPHABETICAL INDEX

REGAL

A Line of

Modern Geared Head Precision Lathes at Popular Prices

Built in Sizes

10-13-15-17-19-21-24-inch

Sold on the Le Blond
Deferred Payment Plan

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

Printed in United States of America

Printed by The Bohnett Co.