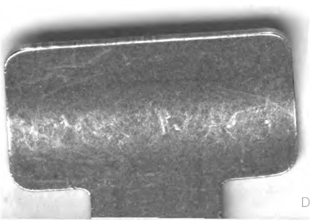


ENGINE LATHE WORK

Published by the

McGraw-Hill Book Company

New York

Successors to the Book Departments of the

McGraw Publishing Company   Hill Publishing Company

Publishers of Books for

Electrical World   The Engineering and Mining Journal

Engineering Record   Power and The Engineer

Electric Railway Journal   American Machinist

Metallurgical and Chemical Engineering

ENGINE LATHE WORK

PRACTICAL SUGGESTIONS WHICH WILL GIVE THE YOUNG MACHINIST OR APPRENTICE THE FOUNDATION PRINCIPLES OF ENGINE LATHE WORK

BY

FRED H. COLVIN

ASSOCIATE EDITOR OF THE AMERICAN MACHINIST, AUTHOR OF "AMERICAN MACHINISTS' HANDBOOK," "THE HILL KINK BOOKS," "MACHINE SHOP CALCULATIONS," ETC., ETC.

FIRST EDITION
SECOND IMPRESSION

McGRAW-HILL BOOK COMPANY
239 WEST 39TH STREET, NEW YORK
6 BOUVERIE STREET, LONDON, E.C.
1909

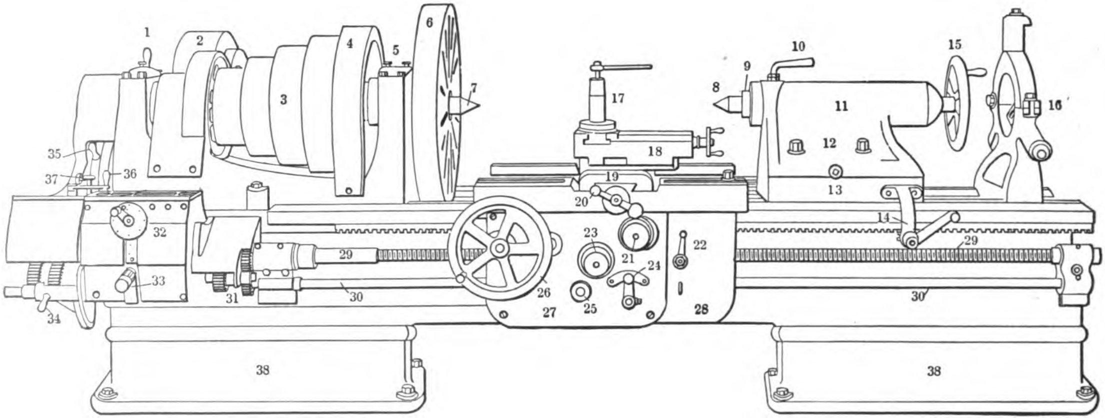
PREFACE

BACK of the proper running of all machine tools are what may be called foundation principles which should be understood in order to get the best results. Every good machinist comes to know these principles and to apply them unconsciously, but this book has been written for the apprentice and younger machinists who may not have had the same opportunities.

There is nothing startlingly new in the suggestions offered as they represent good shop practice for the average machine shop, and it is believed that they are presented in such a way as to be clear and practical. While they have been written especially for those with a limited experience, it is quite probable that many of the ideas and suggestions may be new to some of the older men who have not had a chance to see what other shops were doing.

THE AUTHOR.

CONTENTS



PARTS OF SCHUMACKER & BOYE 32-INCH DOUBLE BACK GEARED ENGINE LATHE

1 Rear bearing.
2 Back gear case.
3 Cone pulley.
4 Face gear guard.
5 Front bearing.
6 Face plate.
7 Live center.
8 Dead center.
9 Tail spindle.
10 Tail spindle lock.
11 Tailstock slide.
12 Locking bolts.
13 Tailstock base.
14 Tailstock pinion.
15 Tailstock handwheel.
16 Steady rest.
17 Tool post.
18 Compound rest.
19 Cross-slide.
20 Cross-feed screw.
21 Cross-power feed.
22 Half-nut handle.
23 Regular power feed.
24 Feed reverse.
25 Gear stud.
26 Hand feed.
27 Front apron.
28 Rear apron
29 Lead screw.
30 Feed rod.
31 Feed gears.
32 Feed box.
33 Change gear handle.
34 Compound gears.
35 Change gear handle.
36 Change gear handle.
37 Change gear handle.
38 Bed.

ENGINE LATHE WORK

CHAPTER I

THE ENGINE LATHE

THE engine lathe is such a well-known machine, the nearest perhaps that we have to a universal tool for shop work, that it is well to become familiar with its parts and their names. For this purpose we show a lathe with its parts numbered and named, as well as a ghost or skeleton view of the lathe carriage, the heart of the lathe and the least understood of all its parts.

In the pages that follow, the work of the engine lathe will be taken up, beginning with centering, chucking and driving, in a way that any young machinist or apprentice can understand and appreciate. The ideas and suggestions are taken from current shop practice and should prove helpful to all who wish to know about the lathe and its work.

The Carriage

The picture shows just what is behind the apron, and instead of photographing it from the back side where it would be left-handed or have everything reversed, the artist has made one just as though the apron was of plate glass so that you can see everything through it just as it is in working order.

While all lathes do not have exactly the same kind of gearing the general principle is much the same, and the Reed has been selected as being a good example of a plain-geared apron with independent feed rod and a power cross-feed.

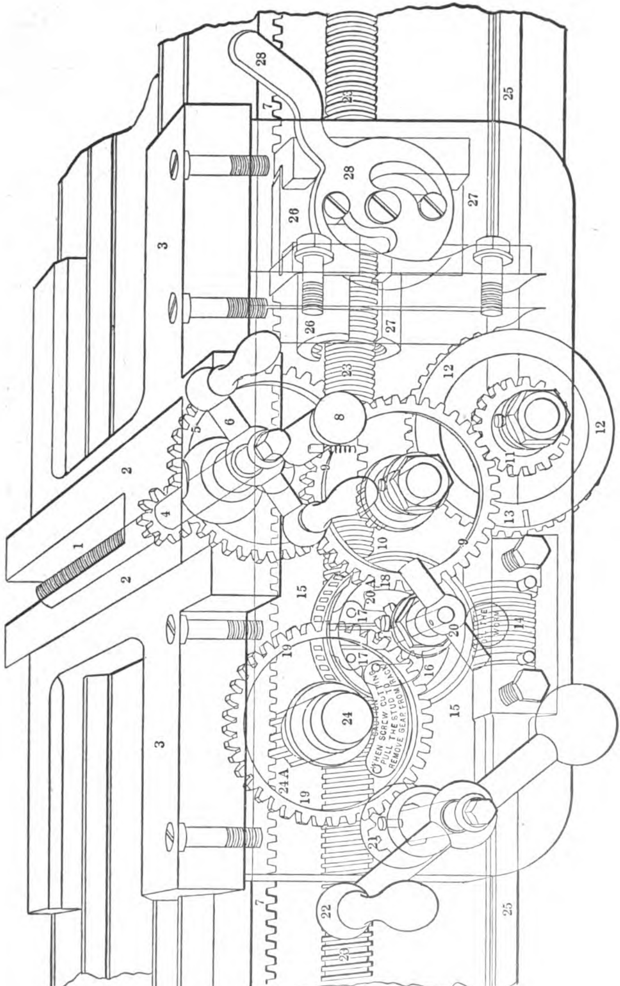
The cross-feed screw is seen at 1, below the surface of the cross-slides 2 which stand above the wings of the carriage saddle 3. The pinion 4 is on the cross-feed screw and is driven by gear 5 when it is slid into mesh by the knob 8. Pulling out this knob disconnects gears 5 and 4 and leaves the cross-feed screw free to be turned by the handles 6. Pushing it in throws the cross-feed in again. This push-rod is held in either position by a small plunger which is forced into the grooves by the spring beneath.

The Cross-feed Gearing

Tracing the driving of the cross-feed down through the gears we find that it starts with the bevel pinion 13 which is connected to the worm 14, and turns with it whenever the feed-rod 25 is driven by the belts from the lathe head. The spline or keyway in the feed-rod allows the worm and gear to be slid along to any point, at the same time driving it positively at all times by the key in the worm.

Starting with pinion 13, this drives the bevel gear 12, and as pinion 11 is fastened to 12 it turns with it, driving the intermediate gear 9. Gear 9 carries pinion 10 fastened solidly to it, and this drives gear 5 which meshes with 4 and turns the cross-feed screw. It will be seen that in each case except the last the motion is reduced by the small gear driving the larger one, but with gear 5 it is necessary to get a large gear to reach up to the cross-feed screw.

These gears run all the time the feed-rod is connected, but by simply pulling knob 8 out the gear 5 is slid out of mesh with 4 but keeps in mesh with pinion 10 which has a wide face.



The Mechanism of a Lathe Carriage

The Regular Rod Feed and the Gear Train Which Drives

To feed the carriage along the bed in taking a cut we start again at the feed-rod 25, driving the worm 14, which should be kept oiled as suggested by the sign on the oil-hole cover. This runs into and drives worm wheel 15 which is bored out to contain a friction clutch. This worm wheel runs all the time the feed-rod is in motion, the same as the main gears of the cross-feed train.

Inside the worm gear 15 is a clutch ring 16, fastened at the bottom but split at the top and connected with a pinion 18. When it is necessary to start a cut we give the handle 20 about a quarter turn in either direction, which turns the button or spreader 20*A* at the other end of the rod, between the clutch levers 17. These are hinged to the loose ends of the clutch ring and bear, at the inside of the short ends, against a pin or block. When the spreader forces the ends of levers 17 apart the friction ring 16 is expanded from both ends and grips the inside of the worm wheel so that the clutch and the pinion 20*A* turns with it.

This pinion drives the gear 19 which turns rack pinion 24*A*. This meshes into the rack 7, and moves the carriage in either direction according to the rotation of the feed-rod. Here as before there is a constant reduction of motion from the feed-rod to the last gear.

Moving Carriage by Hand and Cutting-out Gears for Screw-cutting

The handle 22 and its pinion 21 are revolved whenever the carriage is moved, but it plays no part in the gear train for power feed and is only useful in running the carriage along the bed by hand. This is also geared down considerably to allow the carriage to be moved by hand without too much exertion.

When cutting screws, the knob 24 pulls the rack pinion 24*A* out of mesh so that the regular friction feed cannot be thrown in at the same time and to relieve the screw of the work of turning the idle gears.

How the Lead Screw is Used in Cutting Threads

When it comes to cutting screw threads there are no gears to be considered as the lead screw 23 moves the carriage along the ways without any regard to rack or anything but the half nuts 26 and 27. These slide in the dovetail ways fastened to the back of the lathe apron and are controlled by the curved slots in the cam 28. When this is turned by the handle, it forces the screws which are fastened in the half nuts to follow the curved slots and draws them together, closing the two halves of the nuts around the lead screw. This holds the nut firmly in place, and as the screw turns the carriage is moved along the bed, carrying the thread tool and cutting whatever thread the lathe is geared for. Opening the half nuts by handle 28 leaves the carriage perfectly free to be moved by hand when gear 24*A* has been slid into mesh by pushing knob 24 into place.

In an old lathe built by Bement in the '60s, the nut was solid on the screw and for screw cutting was simply locked against turning. For feeding the carriage by hand the handle was geared to the nut and revolved the nut on the screw, feeding the carriage along the bed just as though the screw was a rack.

Taking the Apron Down for Inspection or Repairs

As will be seen, the apron is held to the saddle by the fillister-head cap-screws, and taking these out separates the two main parts. But that isn't all, because the half nuts are still around the screw and the worm and bevel pinion are on the feed-rod. So before taking out the cap-screws, take out the bolts which hold the worm bracket in place and the two bolts holding the half-nut frame or slide to the web on the apron. Then take down the apron and the gears will come out of mesh and no harm will result. To prevent burring up the slots in the cap-screws on top, be sure the screw-drive is flat on the sides and that it fits the slot fairly well. Sloping sides tend to slide out and to make a bad mess of things.

The names of parts follow:

LATHE APRON, REED — PARTS OF

1. Cross-feed screw.
2. Cross-slides.
3. Wing of saddle.
4. Cross-feed pinion.
5. Cross-feed gear.
6. Cross-feed handle.
7. Rack.
8. Power cross-feed and control.
9. Gear in train.
10. Pinion for cross-feed.
11. Main driving pinion
12. Bevel gear.
13. Bevel pinion.
14. Feed worm.
15. Feed-worm wheel.
16. Clutch ring.
17. Clutch levers.
18. Pinion.
19. Gear in train.
20. Feed-clutch handle.

20A. Clutch spreader.

21. Hand pinion.
22. Carriage handle.
23. Lead screw.
24. Rack pinion knob.

24A. Rack pinion.

25. Feed-rod.
26. Upper-half nut.
27. Lower-half nut.
28. Half-nut cam.

CHAPTER II

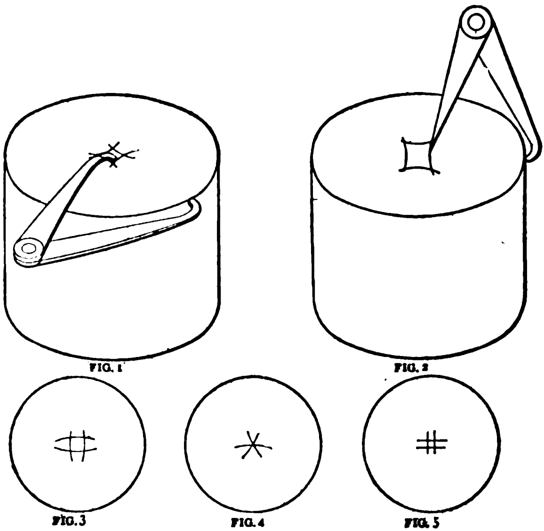
CENTERING LATHE WORK

THE accuracy of finished work depends more on good centering than many realize and is important enough to demand close attention. This includes both the finding of the center and the drilling and countersinking to correctly fit the lathe center.

On plain round work where the bars are practically straight, the ends are all that need attention, and in any case the locating of the center is the first thing to be done, unless the foreman prefers to straighten the bars in a straightening press before starting work on them. The usual way is to locate the center by any of the various methods shown.

The simplest and most common way is to grip the end of the bar in the vise so the end will stand fairly level, flatten or even up the end a little if it has not been cut off in a machine or by a saw, and mark the end with chalk or soapstone to show the marks of calipers, dividers, surface gages or scribers as the case may be.

Fig. 1 shows the end of a round bar marked by taking a pair of calipers set with a little less than half the diameter of the bar, and marked as shown. It is then easy to locate the center with a punch, between these marks. Fig. 2 shows the same thing done with the caliper known as a "hermaphrodite," which has one regular caliper leg, and the other a divider leg. These are much used by some for laying out work of various kinds, but for most purposes the regular caliper, with rather small points, will answer all purposes. Fig. 3 shows the markings when a



Centering the end of a Bar

little more than half the diameter is used, while in Fig. 4 the marking is from three points and with exactly half the diameter. Personally, we prefer the open space in the center as there is apt to be a confusion of lines if the markings meet.

Another method is to lay the bar on the bench with the end on a plate or even on a level spot in the bench, and place a lathe tool or other piece of steel against the end, so as to come a little above or below the center of the bar. Then the scriber is used to draw lines, as in Fig. 5, by turning the bar into four positions.

Along this same line is the surface gage method, as indicated in Fig. 6. The bar rests in a V block, although this is more of a convenience than a necessity, and both bar and surface gage rest on a smooth surface of some kind, such as an iron bench. The marking is done by moving the surface gage scriber across the end, as indicated in the drawing.

The "center square," either as a separate tool or in one of the combination sets now so popular, is another method and one that is quicker than any of those mentioned. Placed across the end of a round bar it allows a line to be drawn directly across the center, as shown in Fig. 7, without regard to the diameter of the bar, as will be seen from the smaller bars, indicated by dotted lines. Turning the bar or the center square part way around the end gives a mark across the first and locates the center where they cross. The illustration shows part of a combination set, the center square as a separate tool not being used much since the introduction of the combinations.

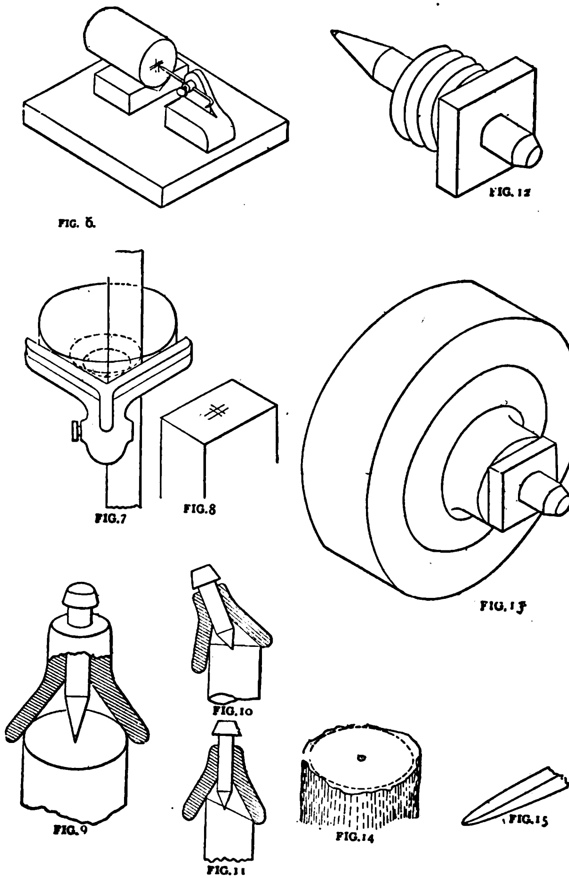
The centers of square or rectangular bars can also be found by calipers in the same way as indicated by Fig. 8. The center square does not work as well as calipers or dividers in this case, and should be kept for round work.

The self-centering punch, shown in Fig. 9, is quite popular in many shops, and is a great time-saver, but it should be used with a little care. In fact, even the most simple operation can always be improved by a little care, and what is commonly called "horse sense." When held squarely over the end of a round bar, a blow on the punch will mark the center with considerable accuracy. If it is not held squarely or the end of the bar is not square, the punch mark may be away off the center, as shown in Figs. 10 and 11.

Another way that used to be more common than it is now that centering machines are used so much more than formerly, is shown in Figs. 12 and 13. A block of cast iron or machine steel is turned up and threaded to fit the screw plate of a universal chuck, generally one with three jaws, though this is not necessary. This was bored or drilled true and a center punch turned up to fit without shake, or a good sliding fit. This was then screwed into the chuck, and the piece to be centered gripped in the chuck, when a blow on the punch would locate the center without any marking whatever, the same as the cone-shaped device shown in Fig. 9.

Forgings are often quite uneven, as shown in perhaps some exaggeration in Fig. 14. In such a case it is well to chalk the entire end, punch a very light mark at what seems to be the center, and try it with dividers until the circle drawn shows it to be about even on each side. If you make too deep a center-punch mark before getting it just right, you can move it a little, either by holding the punch at an angle, and forcing it over in the desired direction, or by using a "drawing chisel" like Fig. 13, which is simply a cold chisel with a small, round point. This will cut a small chip out of the punch mark on the side toward the center, then the center drill will start right.

Fig. 16 is a device used by some shops for centering com-



Centering Work

paratively short pieces, although it might be mounted against a post and used for long work, if desired. A centering machine would be a more profitable investment, however.

The base is heavy, of cast iron, and has a cone-shaped piece set in it, as shown. The upright shaft has a spline or

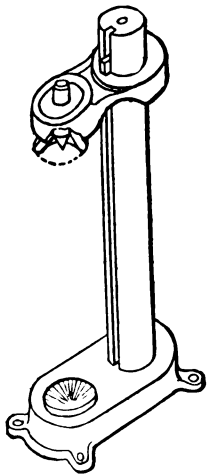


FIG. 16. A Centering Tool

keyway running up its length, by which the upper piece is guided. This is not necessary, but saves time by preventing the piece swinging around out of the way. It would center itself on the upper end of the peice by the cone set in it. The piece is set in the lower cone, the top piece dropped over it, and the punch struck with a hammer, which centers it without any scribing. It can be fastened to the bench or floor, if desired.

Centering in the Lathe

In some cases the bars are gripped in a chuck in the lathe, the outer end supported and held true by a steady rest, and the center drilled by a drill or center drill or cutter carried in the tailstock of the lathe. Where there are many pieces of a kind to center, this is a very good way. The steady-rest guides are set centrally for this size bar, the chuck is set so that the loosening of one jaw in a three-jawed chuck, or of two jaws in a four-jawed chuck, will release the bar, and allow another to be clamped in exactly the same position. Universal chucks are also used for this work in which all jaws move to close or open on a piece of work. For centering they are, perhaps, the handiest, but independent chucks can be used with success in the way indicated.

Some use a hollow-cone center in the live spindle, as shown in Fig. 17, which drives the bar by friction, and also supports that end of it. The other is held by a "cutting" center, sometimes known as a square center, in the tail spindle. The bar is then revolved and the forked or Y centering tool, held in the tool-post, is forced against it until the end runs true, then the cutting center is forced in far enough to mark it plainly, and the work is done for that end. Reversing the bar enables the other end to be done in the same way.

A little practice enables this to be done very rapidly, and the lathe need not be stopped until the whole lot is done, as the bars can be put in and taken out of the cone with it running as fast as necessary. This cone-driving chuck can also be used in combination with the steady rest, but it is not as convenient or as safe to handle with the lathe running as with the forked centering tool in the tool-post.

Still another improvement, as to saving of time, on a lot of bars of the same size, is to put the hollow-cone center on the tail spindle, set the steady-rest guides in the correct position up near the headstock, run the center drill in a drill chuck in the live spindle, and simply lay the bars into

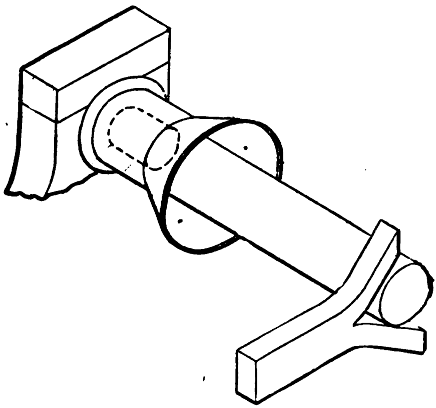


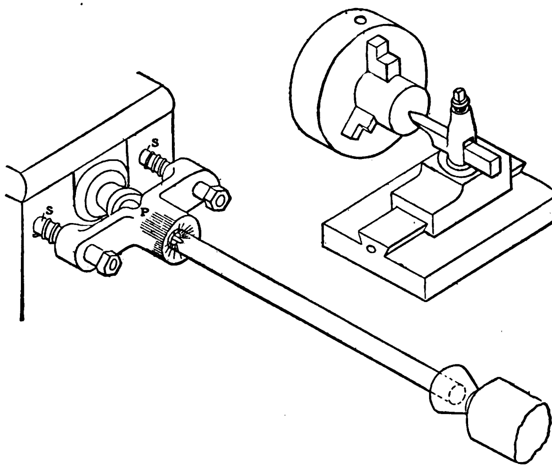
FIG. 17. Using a Cone Center

the steady rest and feed against the drill by the tail spindle.

For centering short work held in a chuck, the lathe tool, shown in Fig. 18, is often used. This has what is practically a flat drill point, moved to the center of the piece and forced against it either by the carriage or by the dead center bearing against the back end of the tool. This can be used in long work with the end supported by a steady rest or with a forked tool, as shown in Fig. 17. Some prefer it to the "cutting" center.

If there is no centering machine in the shop a very good substitute can be made of a small speed lathe, or it can be made removable if desired, so the lathe can be used for other things when not needed for this work, as illustrated in Fig. 19.

Drill and tap into the head for the studs *S S*, in line with



FIGS. 18 AND 19. Centering in a Chuck and a Cheap Centering Lathe

the spindle. Make the studs long enough to take the centering piece *P* and allow the drill chuck and center reamer behind it, and stiff enough to guide well and not be easily bent by accident, say $\frac{5}{8}$ or $\frac{3}{4}$ inch.

The centering piece *P* is a casting drilled to slide freely on the studs, and having a coned or bell-shaped mouth, with a small hole in the center through which the center drill comes into the piece to be centered. By keeping the hole as small as possible, smaller size rods can be centered in it.

With a cup center in the tailstock the bar is put in this center, and forced against the centering piece *P*. This centers it, and then slides back on the studs so the center drill can get to work. The springs return the centering piece on removal of work. Removing the nuts from the studs allows the centering piece to be taken off to change drills, etc.

Drilling and Reaming the Center

If the work comes to the lathe already centered, as it does in most shops of any size, the next move is to get the tool started to work on it as soon as possible. But if centering is part of the lathe man's work, as it always used to be, drill the centers in good shape the first thing.

There is no need of discussion as to the right angle; 60 degrees is the accepted standard, and there is nothing else to use. There is hardly any excuse for not using the regular center drills that can now be had at such a low price. These drill and ream at the same time, and are time-savers. But if the boss will not get them, first drill a small hole, with a $\frac{1}{16}$ drill on small work under an inch, and going up to $\frac{1}{8}$ for work over 3 inches, with $\frac{3}{32}$ for diameters in between. You can use pieces of broken twist drills if you have to, or even make small flat drills; but always drill a hole, anyhow, unless you have the combined drill and reamers. The drilled hole makes it easy to do a good job of reaming, and should always be deep enough to prevent the center from touching the bottom.

A Good Reamer

After drilling the hole, there is nothing better than the half-round reamer, made by turning or grinding the end of a steel rod to a 60-degree angle, and filing or milling one-half of it flat, as shown in Fig. 20. This is easy to make and does a good job. It is sharpened by grinding on the flat side.

The drill, reamer, and countersink combination is shown in the end of a bar in Fig. 21. The object of the countersink is to enable you to face the end without interfering with the center.

Five examples of how not to center a piece of work are shown in Figs. 22 to 26. Their defects are easily seen. Fig. 22 is too blunt an angle, 23 is too sharp, 24 has been made with a punch, and in 25 the reamer has gone in so far that the center does not reach the angle at all.

Square Up the Ends and Straighten

Fig. 26 is reamed all right, but it should have been squared up before centering, by the cutting-off machine, the saw, by grinding, or even by filing. Facing this down with a side tool would cut away most of the center reaming, as shown by the dotted line.

Another point to avoid is the centering of crooked work. Straighten the work first, and then center it. Fig. 27 is an exaggeration of what happens when the bar is centered first and straightened afterward. The centers have a poor bearing here.

Size of Center

The size of the center must depend largely upon the judgment. It should be large enough to hold the work firmly under the heaviest cut that will be taken, and yet small enough not to weaken the end or be out of proportion to the finished diameter at the end. If it is tool steel and to be hardened, make the hole and the center as small as can be done with safety, not small enough to cut the center or prevent the piece being held steadily. A large center is more apt to cause cracking in hardening.

In making arbors and work that is to be run frequently or for long periods on the centers, it is best to use as large centers as possible. For regular work, the following sizes are recommended by J. T. Slocomb, the maker of combination center drills.

TABLE OF LATHE CENTERS IN WORK TO BE TURNED

Diam. of Work	Diam. of Center Drill	Diam. of Countersink	Speed of Drill and Countersink
$\frac{3}{16}$	$\frac{3}{64}$	$\frac{3}{32}$	2000
$\frac{1}{4}$	$\frac{1}{16}$	$\frac{7}{64}$	1700
$\frac{5}{16}$ to $\frac{1}{2}$	$\frac{1}{16}$	$\frac{1}{8}$	1600
$\frac{9}{16}$ to $\frac{3}{4}$	$\frac{5}{64}$	$\frac{9}{64}$	1500
$\frac{13}{16}$ to 1	$\frac{5}{64}$	$\frac{3}{16}$	1300
$1\frac{1}{16}$ to $1\frac{1}{4}$	$\frac{3}{32}$	$\frac{7}{32}$	1000
$1\frac{5}{16}$ to $1\frac{1}{2}$	$\frac{3}{32}$	$\frac{1}{4}$	900
$1\frac{9}{16}$ to $1\frac{3}{4}$	$\frac{3}{32}$ to $\frac{1}{8}$	$\frac{9}{32}$	850
$1\frac{13}{16}$ to 2	$\frac{1}{8}$	$\frac{5}{16}$	800
$2\frac{1}{16}$ to $2\frac{1}{2}$	$\frac{5}{32}$	$\frac{3}{8}$	600
$2\frac{5}{8}$ to 3	$\frac{3}{16}$	$\frac{7}{16}$	500

In drilling the centers on a lot of similar pieces, it pays to be careful that the pieces are all as near the same exact length as possible and are centered to the same depth. This makes it much easier to turn the work to exact distances between shoulders. A little care will save time and work in future operations.

The Lathe Centers

The days of plain round centers, with no means of removing from the spindle, have passed in most shops, and with them the excuse for banging them with the tool-post wrench to jar them loose when it was necessary to take them out. This was apt to spring the centers, and to enlarge the mouth of the spindle taper, neither of which improved the lathe for accurate work.

Then came the center with squares or flats, so that a wrench could be used, and, with the hollow spindle, the use of a rod for driving out the live center. The dead center is usually forced out by the screw in the tailstock, when it is run in all the way. But the square enables it to be taken out in any position.

Some prefer a center made with a threaded nose and a nut, as shown in Fig. 28. This works just like a young screw-jack, the nut bearing against the end of the spindle and pulling the center loose. This should be a fairly fine thread, say 18 to the inch to give a strong pull with little effort, as all that is needed is to start the center.

When a rod is used the end should be turned down and rounded so as to avoid pounding it larger than it should be, just as we head over a rivet. To avoid this many prefer the center with the nut, as shown.

Have Two Sets of Centers

There should be at least two sets of centers, one for heavy work and one that is kept for fine turning. It used to be the style to leave the live center soft, on account of its having no wear; but one real reason was because there

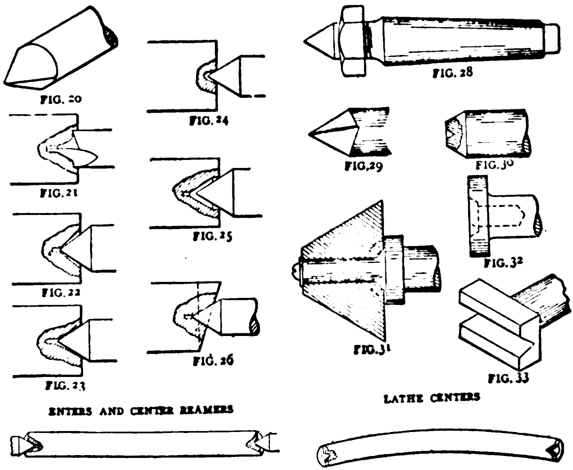


FIG. 27. Effect of Centering Bent Work

were few shops that had any way of grinding the centers after they were hardened.

The dead center was made hard to stand the wear and from the fact that the truth of a dead center is not so important.

Suppose it is an eighth of an inch out of line, and it simply means that the work is held an eighth of an inch too high or too low, or the same amount to one side or the other, but it does not affect the roundness of the piece being turned providing the live center is in line with it. If, however, the live center is an eighth of an inch out of true, the center of the work is carried around in a circle one-quarter inch in diameter, and this end of the work will be turned eccentric to that extent, gradually getting round as it approaches the dead center.

This is the reason that grinding machines have the work run on dead centers at both ends as it avoids all eccentricity due to the live center being out of true.

It is best to keep the live-center spindles for use in that place only, and to be careful that both the hole and the center are wiped clean before putting them in place.

In most cases it is an advantage to mark the centers and the spindle so they can be put back the same way, although modern lathes are generally accurate in this respect. Unless the screw and nut center is wanted, a very handy way is to make them out of a piece of octagon steel and leave the flats of the octagon to be handled by the wrench. This wastes less stock in turning up, and takes less time than using a square bar, as is often done.

Centers for Special Work

A cutting center for centering work, as described elsewhere, is simply a center ground with a square point to act as a blunt reamer, Fig. 29. Needless to say, it cannot be too pointed or too sharp, as it wears out and dulls too quickly.

This kind of a center is also used to drive small work by fitting into a square center, made in one end. This will not stand a very heavy cut, but is useful in some cases, such as turning valve disks having a stem on one end, similar to a gas-engine valve.

Some prefer a three-sided center for cutting, especially on brass or softer metals, on account of the sharper cutting angles it presents.

Some kinds of work, such as small taps, cannot have a center drilled in them, and instead have a point which fits into a female center, as shown in Fig. 30. This requires the work to be centered by pointing, before coming to the machine.

A Pipe Center

For holding such work as cast or wrought-iron pipe, a large center is necessary, unless a cap is placed over the end and a center put in that. So a large cone center is made, with the angle varying according to the ideas of man in charge. For small pipe the regular 60 degrees is all right, but on larger work a 90-degree angle is better.

Instead of having these solid and letting the pipe turn on the center itself, it is customary to have the cone run loosely on the body of the spindle. The bearing is usually against a cone, so as to center it, but sometimes a ball-bearing is used, as indicated in Fig. 31 by the dotted lines. The pipe bears on the cone, which turns with it, and saves wearing a groove where the thin edge of the pipe comes.

Centers for Drilling

When the lathe is used for drilling, as is usually the case at times in a small shop, some drilling center should be provided to hold the work against. A plain drilling pad is shown in Fig. 32 which is simply a flat-faced center, turned square with the spindle and having a recess or hole for the drill to pass into. It is not a good plan to drill into the spindle itself on account of getting chips into it, and for this reason it is better not to have the hole in the drilling center go clear through. The outside diameter can be made to suit the work that is likely to come up.

Fig 33 shows a center that is extremely handy in drilling holes crosswise in round bars. The V should be carefully laid out and cut, so as to be exactly across the center line; otherwise it will not hold the work so as to drill in the center. This is especially useful to the toolmaker in the small shop who has to mortise boring bars by drilling, chipping, and filing.

CHAPTER III

DRIVING THE WORK

CENTERED work is usually driven by a dog of some kind, probably the bent-tail variety is the most common. The only precaution is to be sure that the tail does not bind in the slot of the face plate, at either the sides or top or bottom, so as to draw the work off the live center, or prevent its seating clearly and squarely.

If the piece to be turned happens to be a forging or casting, with a wing or projection of any kind at the end next the face plate, a stud fastened to the face plate is the best way to drive it, as shown in Fig. 34.

Different Kinds of Dogs

Straight-tailed dogs are preferred by many, and with them it is necessary to use a stud on the face plate the same as with the forging. Less trouble is likely to occur from cramping a piece of work when this style dog is used, but the bent-tail is all right if you are careful about putting it in place.

Beside the forged or cast-steel dogs in one piece, there are many uses for the two-piece dog, usually made from two pieces of flat stock, bent up by the blacksmith and drilled as shown in Fig. 35. In some cases one side, and occasionally both sides, will have a bent tail for driving, but they are often made straight and driven with a stud.

Protecting Work from Dogs

On most work roughing cuts can be taken without regard to the screw marking the bar; but if only a little is to be removed it is best to protect the bar by putting in a

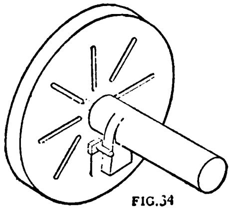


FIG. 34

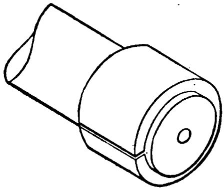


FIG. 37

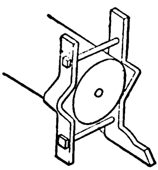


FIG. 35

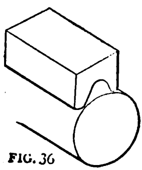


FIG. 36

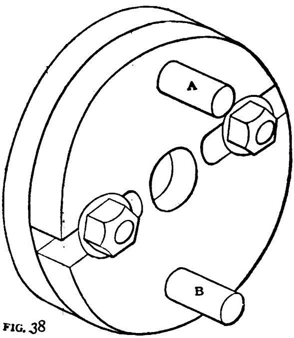


FIG. 38

Methods of Driving Work

piece of iron or brass, as the screw point often makes a deep mark which does not come out with a light cut.

Finished work should always be protected, and a piece of soft iron or hard brass, something like Fig. 36, is always handy to put under the screw. If there are many pieces of the same size to turn it may pay to make a cast-iron collar, like Fig. 37, and make a saw cut in one side of it. Any dog will clamp this on the work in good shape, and drive it without marking the finished part of the bar.

A good machinist has his own ideas as to the best kind of a dog, and most other things about the lathe, but he also

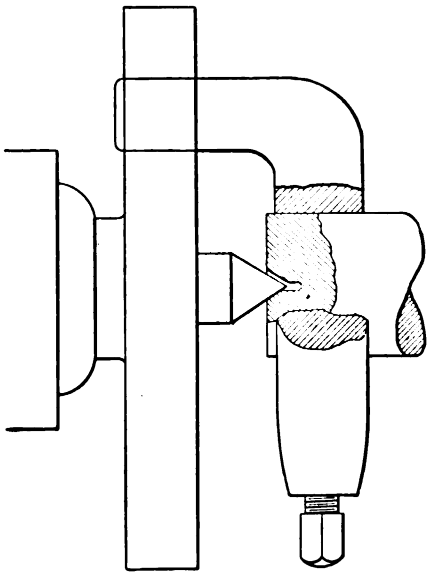


FIG. 39A. Crowding off Center

knows that he can do good work with almost any of them and he uses what he finds, to the best advantage. So if he finds the bent-tail breed, as is most likely, he fastens it on the work, sees that both centers are free from dirt and chips, puts a drop or two of oil in the dead-center hole or on the dead center itself, and puts it in the lathe. It should be tight enough to prevent end play, and yet turn freely, a good running fit. And never have the spindle out of the tailstock more than is necessary.

One of the best methods of insuring equal driving with a double straight-tailed dog is to use what is known as the equalizing face plate as shown in Fig. 38. This consists

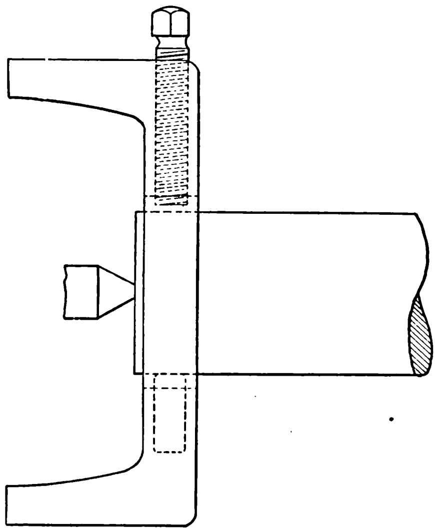


FIG. 39B. Two-tailed Dog

of an additional plate put on the regular face plate as shown and held to it by the two studs working in the slot in the outside plate. The lathe center comes through in the usual way, and the outer plate is left loose enough to adjust itself, so that each of the driving studs *A* and *B* will bear with equal pressure on the two ends of the dogs. This insures an even driving pressure and avoids the tendency to crowd the work to one side, which is sometimes found where the bent-tail dog is used, or even the single straight-tail.

For heavy work it is generally conceded that a two-tailed dog is better as it drives from opposite points and avoids the tendency to cramp that was mentioned before. Here again there is a difference of opinion as to the straight-tail or the bent-tail dog. Prof. John E. Sweet, one of the most practical men I know of, advocates the style of dog shown at *B*, and many are being used. The straight-tail may be a little easier to make, but this avoids all difficulty with locating studs in the right place.

CHAPTER IV

TOOLS AND TURNING

ONE of the disputed points in lathe work is the setting of the tool. But leaving all discussion out of the question, a glance at Fig. 40 will show what happens with the tool point at the center, on the right, and above the center at the left. For ordinary work of fair diameter, over an inch, it is usually best to set the tool point above the center, but for smaller work it is readily seen that it must be at the center to cut at all, at the small diameter. In taper work it *must* be at the center for good work, and in finishing a thread it is also necessary to set it at the center line if a correct shape of thread is desired.

Adjusting Hight of Tool

Most tool-posts have either the curved block, Fig. 41, or the stepped ring, Fig. 42. Some have a hight-regulating device with one sleeve screwing into the other, and some lathes still have the elevating screw at the back, which raises or lowers the whole cross-slide. They are very handy in setting a tool, too, because you can fasten the tool first, and then adjust it for hight afterward. All these tool-posts are handy, but they are also wabbly under a heavy cut, and the four-bolt tool-holder, similar to a planer, is the most substantial though not as handy. Fig. 43 shows one adapted to go in a regular tool-block, in place of the other, when there is heavy work to be done. This has two bolts.

Avoid Overhang for Tools

Whichever style tool-post is used, set the tools with the cutting point as close to it as possible. Do not stick it out so as to overhang the support an eighth of an inch farther than is necessary. Every possible chance for spring should

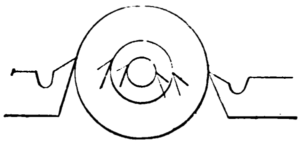


FIG. 40 HIGHT OF CUTTING POINT

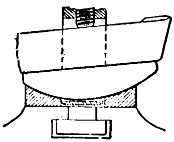


FIG 41

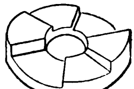


FIG. 42

TWO HIGHT ADJUSTMENTS FOR LATHE TOOLS

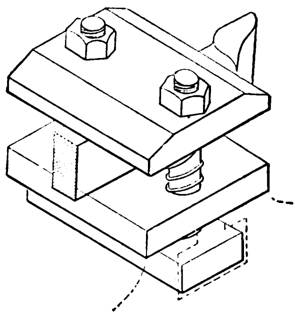


FIG. 43 A SOLID TOOL POST

Setting the Tool

be cut out, so far as it can be done, and many times it is simply a case of overlooking or not thinking about the difference it makes.

The reason these new special lathes can remove so much stock is because they support the work and the tool so as to avoid the springing of either.

Turning the Work

Unless the bar has been faced a little with the countersink, as in Fig. 44, the facing or side tool is the first to be used, so as to get the end square and have an equal bearing on the center. The usual side tool is shown in Fig. 45. Even with the countersinking, it is a good plan to face

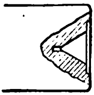


FIG. 44

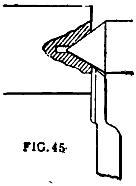


FIG. 45

FACING END OF A BAR

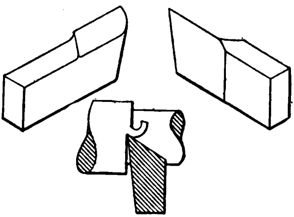


FIG. 46

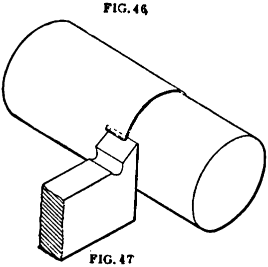


FIG. 47

ROUGHING AND FINISHING TOOLS

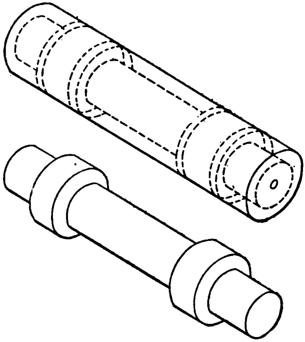


FIG. 48. TURNING A BAR WITH COLLARS

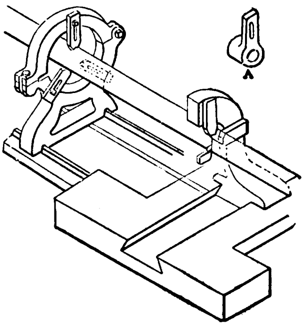


FIG. 49 STEADY AND FOLLOWER RESTS

Facing and Turning

down to meet it, although if the work has come from a cutting-off machine or a cold saw this is probably unnecessary.

The tools to use in turning this bar, the feeds, the depth of cut and the speed, all depend on the hardness of the material to be turned, the kind of tools used, the stiffness of the lathe, and the driving power of its belt. Calling the work machinery steel and the tools of a good grade of carbon steel, tempered in the usual manner, it is a good plan to start work with a tool something like Fig. 46. The lower view shows the cutting action. In the old days the only permissible tool was the diamond point, and a man who used anything else was dubbed a poor mechanic who was not up in the fine points of the trade. But the days of the diamond point have gone for roughing work, and a sort of a side tool with a round nose, plenty of rake on top, and not much clearance at the side, does the best work. It will peel off the chips in a way to make glad the heart of the foreman, and he is the man to please.

The First Cut

Always take the first cut deep enough to get under the scale if it is a possible thing, as this saves the tool. For roughing the work, most mechanics now prefer a deep cut and a fairly fine feed and reverse this in finishing, though in the old diamond-point days this would have been all wrong. Good finishing can be done with a broad-nose tool, with enough rake to cut freely, a rounding corner on the leading point, and a fairly coarse feed.

Fig. 47 shows this at work. The depth of the finishing cut depends on a number of conditions, but if necessary quite a cut can be taken. The only object of the groove back of the cutting point is to allow grinding easily and still retain the top rake. The clearance in front should be slight. If a nice finish is wanted there is nothing better than a stream of water on the work, either plain or flavored with soda to prevent rusting.

One of the Hardest Jobs

Perhaps the hardest job on a lathe is to turn a bar the same diameter from end to end. When you come down to fine points in measurements, it is practically impossible when we have a centered bar, and have to turn the bar end for end to finish it. But even without getting down into the thousandths, it is a hard job to get it even approximately true. By carefully calipering the bar after the first cut, at points as far apart as possible, it is easy to see how nearly right it is, and to move the tailstock a little to correct any slight taper that may be found. But, assuming that the lathe turns straight, when the piece is reversed and we turn up to the meeting point, the slightest variation shows at once, and the meeting point is always noticeable, unless the piece is afterward ground or we resort to the use of emery cloth to produce an even-looking finish.

The nearest approach to work of this kind usually found is turning up an arbor or mandrel, and here there is a short distance at each end smaller than the rest, and the dog can usually be put near enough the end to allow the center cut to run off over the reduced portion.

Work with Shoulders

Most work, however, has one or more shoulders and several varying diameters, such as the bar shown in Fig. 48. A fair allowance for finish on such work is $\frac{1}{8}$ inch in length and same in diameter to be reduced, for the largest part of the bar. As the lengths as well as the diameters are important, one good plan is to lay off these shoulders as soon as the roughing cut has been taken. This can be done by drawing a chalk mark, the length of the bar, laying out the distances on this mark and scoring a line at each point with a sharp-pointed tool held in the tool-post.

Here comes a point which each man must decide for himself. Shall the piece be faced to length, before cutting any shoulders, or shall the facing be left till last, or shall one end be faced and the other left to be finished afterward? If both ends are faced the first thing, there is absolutely no leeway for a slight error in making a shoulder. But if by accident the tool cuts a little too deep at the first shoulder, there is still a chance for saving the piece by facing that end down to meet it. Of course such things should not be done, but they sometimes happen, and if it is possible to leave a loophole it is well to do so. But with four shoulders, as this work has, there are few chances of escape.

Facing the Ends Last

Going on the plan of facing the ends last, lay off the distances from the center, or, if from the ends, allow one-half the extra length of bar for facing. With these marks there is no excuse for not getting the job right, taking care not to undercut the shoulders so that, when you face them up square with a side tool, the collars will be too thin. The facing can best be done by feeding the side tool, or whatever you use, into or out from the center, so as to get it square, rather than to depend on setting the tool square. This is a good plan in facing the ends and in almost all facing work.

Another Method

Another way of doing such a job is to use a cutting-off tool and cut grooves each side of the marks to about the depth needed to bring them down to the required diameter (see dotted lines, Fig. 48). Enough should be left for facing, as the cutting-off tool very seldom leaves the surface smooth enough for a finish.

The accuracy of the work can be measured by a gage of sheet metal, if one is in the place, or by a rule, or with calipers. The odd-legged caliper is especially useful in this work, for all except the thickness of the collars. The regular calipers are best for this.

Springing of Work

On work that is stiff enough to stand a good cut there is little difficulty in making good time. The limit to the material removed is the ability of the tool to keep a fair cutting edge, which means that it must not get hot enough to start the temper, and the ability of the centers to hold the work and the belt to drive the lathe. The belt-driven feed for the carriage is a very good safety valve for the rate of feed. It will slip or come off entirely if the feed gets too much for it. If this happens after the work has been turned a short distance it probably indicates that the tool is dull, and can be remedied by reducing the feed or the depth of cut. Grinding the tool only will repeat the old conditions and not remove the cause. With a geared feed the cutting edge of the tool must be looked after more closely, as the only safety valve is a lot of broken gear teeth, and the foreman never appreciates a present of that kind.

Turning Slender Work

With long, slender work comes the necessity for care and judgment. Cuts must be lighter and the cutting edge keener, so as to spring the work as little as possible. A dull tool often springs the work to the point where it climbs over the tool point, bends out of shape or breaks, and sometimes snaps the points off the lathe centers. It is not an easy job explaining to the foreman just how this happened, because his mind is preoccupied with the thought of delay to the works, the cost of the spoiled work and the centers. Care on slender work will avoid this. Keeping the cutting point a little above the center will help in preventing this kind of an accident, but steadying devices should be supplied wherever possible.

Shapes of Tools

Although there are now thousands of tool-holders in use with inserted tools, the solid tools still have a place, and in Fig. 50 we show what may perhaps be called the standard shapes. The diamond point has given way to the side cutting tool, as in Fig. 46, for roughing, but is still used in many places. Fig. 51 gives the standard shapes and sizes as recommended by William Sellers & Co. and produced by their tool-grinding machine.

Cutting Speeds

Cutting speeds and feeds have changed greatly in the last few years, and this has been made possible, both by the introduction of high-speed steel and the lathes which have been designed to stand as heavy a cut as the tool itself.

LATHE TOOLS

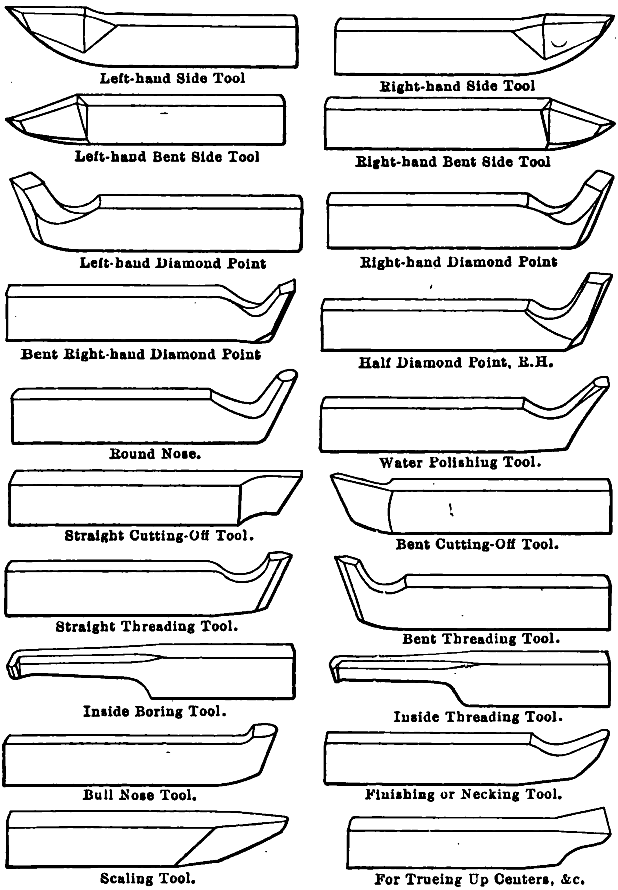
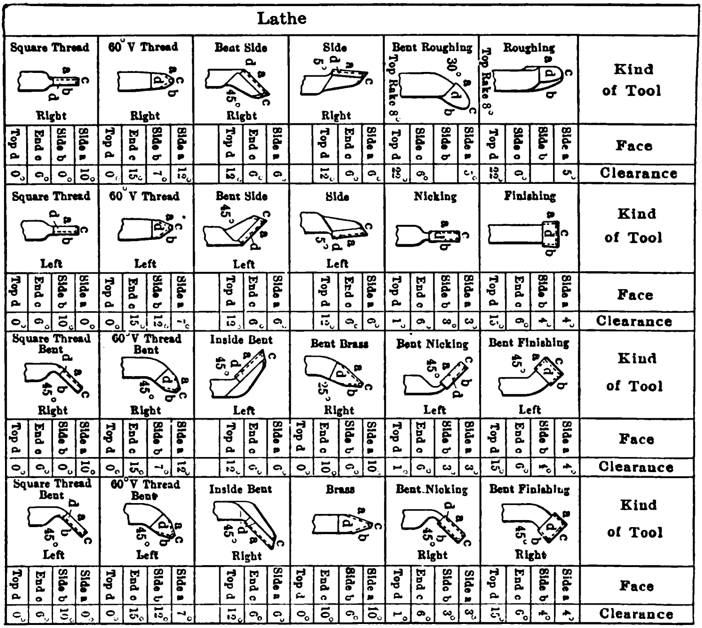


FIG. 50

Whichever steel is being used it is a safe plan to run the lathe as fast as the tool will stand up without too frequent grindings, and then make the cut and feed all the lathe can stand.

LATHE TOOLS — WM. SELLERS & CO.



Lathe — Kind of Tool	Face / Clearance			
Square Thread, Right	Top d 0°	End c 6°	Side b 0°	Side a 10°
60° V Thread, Right	Top d 0°	End c 15°	Side b 7°	Side a 12°
Bent Side, 45°, Right	Top d 12°	End c 6°		Side a 6°
Side, 5°, Right	Top d 12°	End c 6°		Side a 6°
Bent Roughing, 30°, Top Rake 8°	Top d 22°	Side c 6°	Side b	Side a 5°
Roughing, Top Rake 8°	Top d 22°	Side c 6°	Side b	Side a 5°
Square Thread, Left	Top d 0°	End c 6°	Side b 10°	Side a 0°
60° V Thread, Left	Top d 0°	End c 15°	Side b 12°	Side a 7°
Bent Side, 45°, Left	Top d 12°	End c 6°		Side a 6°
Side, 5°, Left	Top d 12°	End c 6°		Side a 6°
Nicking	Top d 1°	End c 6°	Side b 3°	Side a 3°
Finishing	Top d 15°	End c 6°	Side b 4°	Side a 4°
Square Thread Bent, 45°, Right	Top d 0°	End c 6°	Side b 0°	Side a 10°
60° V Thread Bent, 45°, Right	Top d 0°	End c 15°	Side b 7°	Side a 12°
Inside Bent, 45°, Left	Top d 12°	End c 6°		Side a 6°
Bent Brass, 25°, Right	Top d 0°	End c 10°	Side b 6°	Side a 10°
Bent Nicking, 45°, Left	Top d 1°	End c 6°	Side b 3°	Side a 3°
Bent Finishing, 45°, Left	Top d 15°	End c 6°	Side b 4°	Side a 4°
Square Thread Bent, 45°, Left	Top d 0°	End c 6°	Side b 10°	Side a 0°
60° V Thread Bent, 45°, Left	Top d 0°	End c 15°	Side b 12°	Side a 7°
Inside Bent, 45°, Right	Top d 12°	End c 6°		Side a 6°
Brass	Top d 0°	End c 10°	Side b 6°	Side a 10°
Bent Nicking, 45°, Right	Top d 1°	End c 6°	Side b 3°	Side a 3°
Bent Finishing, 45°, Right	Top d 15°	End c 6°	Side b 4°	Side a 4°

FIG. 51

There is a lack of uniformity in shop practice in this respect, probably due to the difference in material and conditions generally. Castings, both iron and steel, affect tools very badly at times, due to sand in the scale and impurities in the iron itself, the highest cutting speeds being obtainable on low-carbon steel.

Cutting speed for cast iron seems to vary from 30 to 60 feet per minute, one shop using a speed of 44 feet per minute, with a cut of $\frac{1}{4}$ inch and a feed $\frac{1}{8}$ of an inch. With a tool $\frac{3}{4} \times 1\frac{1}{2}$ inches this was maintained for one hour without re-grinding and means the removal of 330 pounds of cast iron per hour. This is for high-speed steel.

On low carbon or machine steel this same shop uses a roughing speed of from 80 to 130 feet per minute and from 150 to 300 feet per minute for finishing cuts. Their records show from 4 to 15 pounds of metal removed per minute or from 240 to 900 pounds per hour.

Other shops rarely get above 100 feet per minute, cutting speed, although in one case 140 feet is used in finishing tool steel, but with a very light cut. One good-sized shop uses from 80 to 125 feet per minute for lathe work, with cuts ranging from $\frac{1}{16}$ to $\frac{1}{4}$ inch and feeds from $\frac{1}{64}$ to $\frac{3}{32}$ inch per revolution of the work.

In thread cutting as well as plain turning, the cutting speed has been very materially increased, until we find 90 feet a minute being used on long screws.

These are simply guides that give us something to work by, although the conditions may make it impossible to secure the same results in all cases.

CHAPTER V

STEADY AND FOLLOWER RESTS

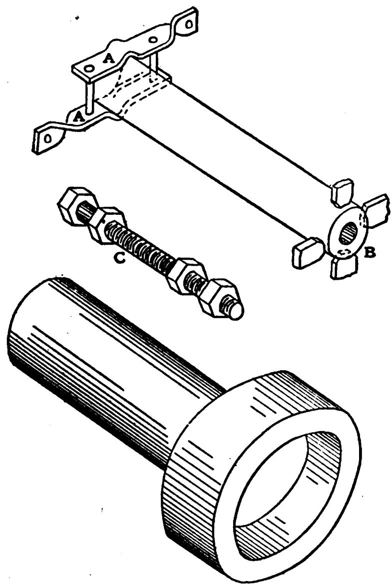
FOR slender work that is liable to spring, the old diamond point still holds its own. Most work of this kind can be helped by the use of a steady rest, when the part to be turned is at one end of the piece. The steady rest comes with every lathe, and has something the appearance shown at the left of Fig. 49. Some classes of work require this kind of a rest, such as long crank-shafts with several cranks, and other pieces.

Occasionally it is necessary to turn a short place in the center or some other point of a long bar as best we can, so as to put the steady rest on a fairly round part of the work.

Boring End of Bar

Another application is shown in Fig. 52. Here is a bar which has been turned on centers, and which is to have a hole bored in the end so as to be true with the outside of the bar. A dog or clamp similar to *A* is put on, and the piece held between the centers, as before. The steady rest is adjusted at the outer end, as shown at *B*, but before the dead center can be removed so as to drill and bore this, we must fasten it to the face plate. Use a stud similar to *C*, for driving and holding the dog *A*. This is put through the face plate from the back and clamped to it by the first nut.

Then the dog is fastened between the second and the third nut, and in this way can be very nicely adjusted, so as to pull the work evenly toward the face plate, and drive it at the same time. Washers should be used under each nut. Then the dead center can be removed, and the hole drilled, bored and threaded, as desired.



FIGS. 52 AND 53. Holding Work to Face Plate

A very common case of this use of the steady rest is in the making of die-holders for large turret-lathe work, such as shown in Fig. 53. This must be turned between centers to insure accuracy, the shank and the enlarged portion being turned on centers by changing end for end. It is then clamped to the face plate, as was done with the bar, in Fig. 52. The steady rest will probably have to go on the shank as near the head as possible, for the jaws of the steady rest seldom open wide enough to take in the large diameter. Taking the tailstock back out of the way leaves the inside of the die-holder ready to be bored and the die ring itself is threaded in this way.

It is better than chucking with the ordinary chuck, though special chucks are sometimes used in connection with the steady rest. This is necessary in all work of this kind.

Follower Rests

Some work is so slender, or the cut must be so deep in proportion to its diameter, that it is necessary to support it right over the point of the cutting tool. This means that it must be fastened to the carriage and move with it, so as to follow the cutting point, which gives it the name of follower rest. All lathes need them if small work is to be done, but they are not always provided. The idea is made plain at the right in Fig. 49.

The value of follower rests can easily be seen by examining any of the new lathes which take heavy cuts. These all have heavy follow rests and in some cases rollers are used in them at the top of bar.

Bushings in Follower Rests

For some classes of work it is advisable to use a cast-iron bushing to follow or lead the tool, as *A*, Fig. 49. This can usually be used for square or flat top threading, as to have it fit after each cut would require reducing the bushing each time, and not be practical. And unless a follower or steady rest really supports the work, it is of little use. The bushings can be bolted to the follower rest in place of one of the regular arms.

A Home-made Follower Rest

If no follower rest is found with the lathe and a job comes along that needs one, such as turning and threading a feed-screw for slide rest, a very fair apology can be rigged

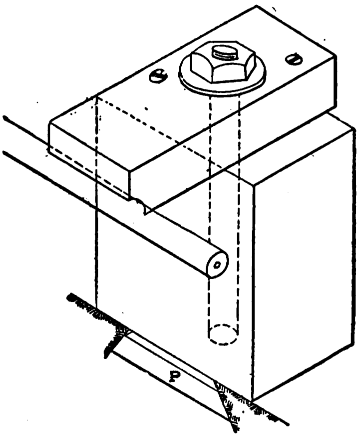


FIG. 54. Wooden Follower Rest

up at short notice, as shown in Fig. 54. The top piece is best made of maple, the material of the large block not being as important. Screw the top piece to the block, drill for a good-sized bolt, say $\frac{5}{8}$; screw the lower end of the bolt into plate *P*, which fits under the cross-slide ways, and allows the whole thing to be clamped solidly. Either bore out the corner of the top piece the size of the work (after the rest is in place, of course), or cut away the corner as nearly as you can, and finish with a sand roll made by wrapping sandpaper around a stick of the right diameter.

This rest, if well fitted and kept oiled, will do good work, but such devices should not be fudged up unless necessary. They take time and often cost as much as one of the regular kind.

CHAPTER VI

FACE-PLATE WORK

FACE-PLATE work requires more care and thought on the part of the lathe man than most other kinds, and is mostly confined to job or repair work. Where a piece that would ordinarily be done on the face plate is manufactured regularly in fair-sized lots, a jig is made and it goes to a drill or a boring mill instead. But the face plate is still necessary and its use is common on a large line of work. The face plate might be called a chuck with removable jaws, for the various clamps used take the place of chuck-jaws.

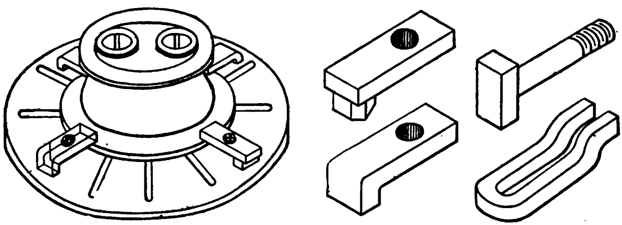
Take a simple job, such as is shown on the face plate in Fig. 55. The problem is to face both sides of this casting square with each other, and then to bore the two holes in their right location, the correct distance apart. The plain side will be done first in this case, as then we can shift the work on the face plate to bore the smaller holes.

Truing Face Plate

If there is any doubt as to the face plate running true, test it with a tool point or a marking gage of any kind. As it will be necessary to use a gage from time to time, it is just as well to use it for this and avoid the danger of marking up the face plate if it is true already. The plate does not need to be particularly smooth; in fact, it holds work better if it is not, but it should be true. Lathe indicators help in this work, as will be seen later.

Clamping the Work

Bent-tail clamps as shown in Fig. 56 are handiest to use, but straight pieces can be used and the outer end clamped down on to a nut or other piece of metal, as shown. It is handy to have the T-head bolts, as they can be put through the face plate, from the front, and a quarter turn gives them a grip. The work can sometimes be clamped best by put-



FIGS. 55 AND 56. Face-Plate Work

ting the face plate on the bench and locating it as nearly as possible, giving the final adjustments after it is put on the lathe. This is to be decided by the man himself and the easiest way adopted. With a portable crane or other hoisting device it can be handled from the bench to the lathe very easily.

The first thing is to lay two or more parallel strips on the plate, so as to raise the work clear of the projecting rings, or "spigots," as they are sometimes called. Then arrange the clamps so as to put the strain on the flange as nearly over the strips as possible, and screw them down tight enough to prevent slipping when put in the lathe. When the piece is swinging in the lathe, true it up by turning the lathe slowly and shifting on the face plate, generally by tapping with a soft hammer, until it runs true. This can be tested very closely with the eye after a little training, but a lathe tool or the marking gage run up near it will show exactly how it is running. When true, clamp solidly to the face plate and face it off with a round pointed tool, taking a cut that will get under the scale. Some prefer facing out from the center, others toward the center. The latter lets you see a little more what you are doing.

Saving a Poor Casting

It sometimes happens that the two faces of a casting like this will not be parallel, and that if all of the inequality is faced from one side the flange will be too thin. But if this is divided up between the two, the casting can be saved. In such a case block up a little under the low side of the casting and in this way divide up the error.

Reversing the casting on the face plate, clamp the side already faced and turned so it will run true. This will be helped greatly by having circles scored on the face plate at regular intervals, such as ½ inch apart, similar to many chuck faces. It can be tested by the gage as before. Turning the outside of the smaller flange, shown uppermost on the face plate in Fig. 55, is an easy matter, and then comes the laying out of the two openings shown.

Laying Out Holes

Drive in, lightly, pieces of wood across the holes and lay off the right distance between them. This gives the centers of the smaller holes. Lay these out with a pair of dividers so as to know how to set the work for turning and boring these. Loosen the clamps so that the work can be moved; turn the face plate so that one opening will be over the other and the weight will help drop the casting to the point where the dead center will come in line with the center mark on the wooden block. Then clamp the work. To test it still further, put the gage in the tool post and see that

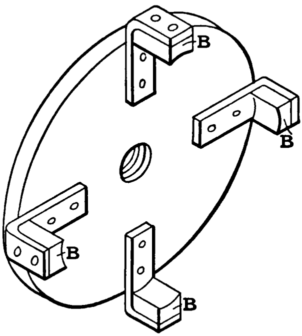


FIG. 57

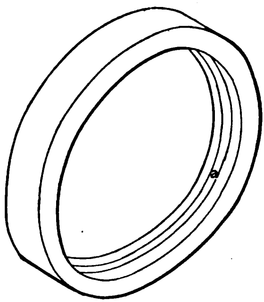


FIG. 58

Holding Large Work

it follows the circle marked by the dividers. If it does, the work is ready to face, bore and turn. Then shift to the other hole and machine that.

Handling Special Jobs

It sometimes happens that a job comes along which cannot be handled by any ordinary methods or by the tools regularly used. Such a piece of work is shown in outline in Fig. 58. It was a lot of large cast-iron caps, 60 inches in diameter, which had to have a dovetail slot cut in them at *a*, in which rubber packing was to be forced to make a tight joint over a pipe. The largest face plate wasn't as big as this, but a bright machinist went to the blacksmith shop and had four pieces of $1\frac{1}{2}$x4-inch iron bent into angles, as shown, drilled holes in them for wood screws at the short end and for bolts at the long end, and screwed on hard-wood blocks *B B B B*. Then he bored these a little tapering, to the size of the outside of the castings, and drove the rims in with a soft hammer till they were square. This

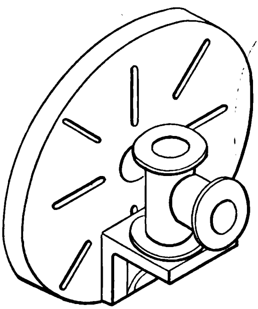


FIG. 59

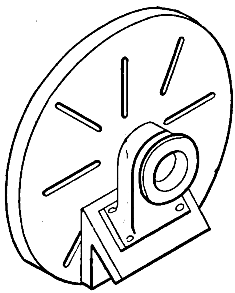


FIG. 60

Use of Angle Plates

held them true without any caution except to drive them in until the face ran true, and held them firmly enough to take any cut that was necessary. Wooden blocks can often be used in similar cases and do good work, as they have more elasticity than iron and grip the work very tightly.

Angle Plates

While many other examples of face-plate work might be given, these show the general plan to be followed, and any work of a similar character can be handled as readily.

Angle plates add to the range of work that can be done on the face plate, as indicated by Figs. 59 and 60, and there is no end to the variety of angle blocks and special blocking-up devices that will be found in shops handling a large variety of jobbing work. The machinist who can tackle the work as it comes in, and handle it with the blocks, straps, and other "fixings" he finds on hand, is the one that is in demand. Angle plates are also made with a plate that can

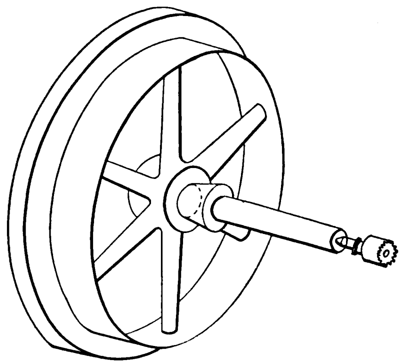


FIG. 61. Centering Pulleys

be shifted to any angle to the face plate and are very handy, although not as apt to be rigid as the solid plates.

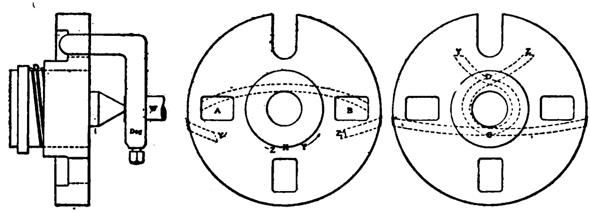
Centering Pulleys

There is a simple plan for centering such face-plate work as pulleys which are to be bored. These are held against the face plate, for boring, by hook bolts which go over the arms; but for centering, the pulleys are held against the face plate by the dead center, as shown in Fig. 61. The dead center holds the bar against the pulley, and it is driven by the friction of the face plate. The back end of a lathe tool, held in the tool posts, "bumps" (no other word quite expresses it) the pulley rim into the center. It is then clamped tight and is ready to bore. The centering takes very little time if done in this way.

These pulley arms could be easily sprung in clamping and thus strain the whole pulley. To prevent this the arms should be clamped against blocks of wood or iron, set between the arm and face plate.

Bridle for Face-Plate Work

Any one who has noticed the amount of time a lathe hand consumes in the different machine shops looking for a



FIGS. 62 TO 64. Using a Belt Lace Bridle

bridle and two bolts of suitable length to bridle a shaft for any work on the head center will agree that more time is often lost in looking up these things than the work costs. Fig. 62 shows how to bridle work on the center without loss of time and no long hunt for bolts, bridle or packing washers. And when at times you forget and place your hand on the steady rest, there is no bolt to come around and take the bark off of your finger as is usual in work of a short length.

Take a piece of strong belt lacing ½ inch wide, about 38 inches long; place one end in right and one end in left hand as in Fig. 63. Place the end *Y* that you have in your left hand through bolt slot in drive plate *A*. Place the other end *Z* in your right hand in slot *B* with the center of lacing on top of work as shown by dotted lines. Now let ends *Y* and *Z* follow arrows *Z* and *Y* and cross beneath the work on center at *X*.

Fig. 64 shows lacing crossed at *X* and *C* and the dotted lines also show lacing crossed at *D*. When you have accomplished this, *i. e.*, placing the lacing according to dotted lines, tie a common knot at *D*. You do not have to pull so hard that it is impossible to reopen the knot; just give it a good pull.

The most important point is the necessity of allowing for the draw. Before starting to lace unscrew drive plate a few turns, until the face plate is about ¾ of an inch from spindle collar. Then apply lacing as directed and draw drive plate up on spindle nose and you are ready for work, as this holds it tight.

When the work is done, loosen the face plate again, which eases up on the knot so that it can be easily untied.

Another plan is shown in Fig. 65, which is more convenient many times than the lacing, although you can sometimes find a substitute if a lacing is not handy. This can be made without difficulty and easily kept in a tool chest until wanted.

Adjustable Face Plate for Accurate Tool Work

The drawings illustrate a handy adjustable chucking plate. Fig. 66 shows the device assembled. This is a tool that is not costly, but in a tool room where subpress tools are made, or on work of a similar character, it is worth its weight in gold. It not only assures perfect alinement of holes, but if used with a micrometer head graduated to read to ten thousandths of an inch, almost any degree of accuracy may be obtained.

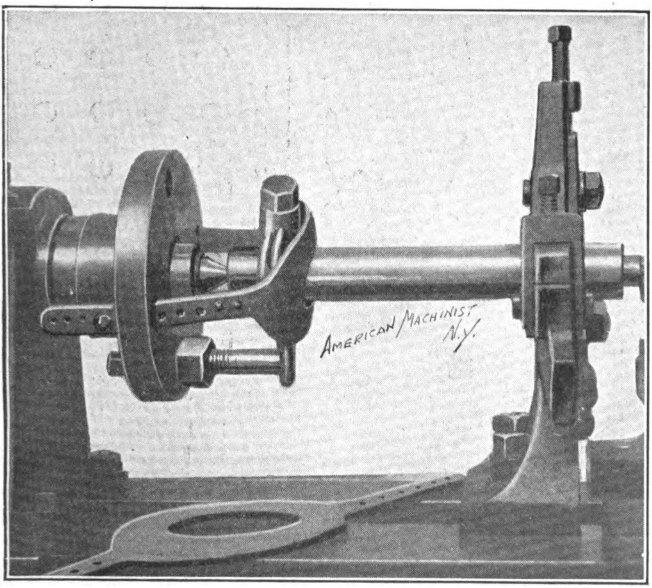


FIG. 65. A Handy Bridle for Lathe Work

Suppose we have a punch pad, stripper and die to produce as shown in Fig. 67. Lay out hole *A*, place the angle plate 0.375 inch below center, clamp the work, and bore the hole at *A*. Then move the micrometer holder to the

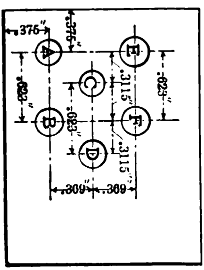


FIG. 67

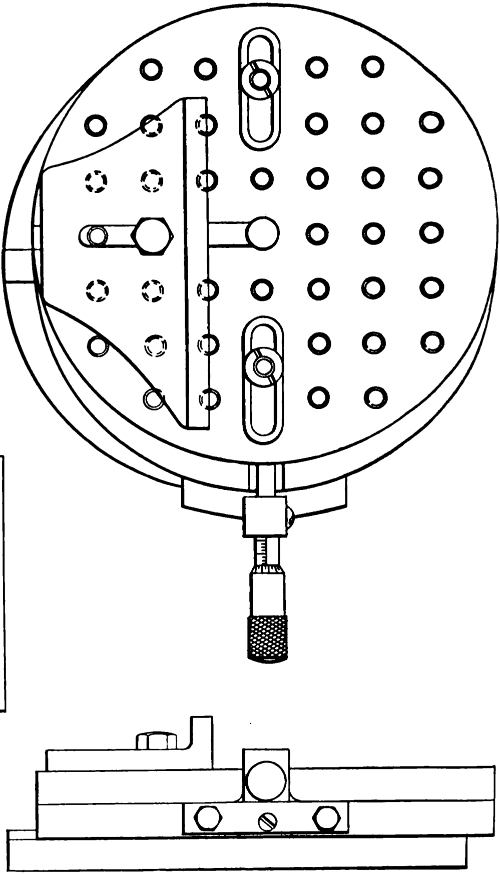


FIG. 66. Adjustable Face Plate for Accurate Tool Work

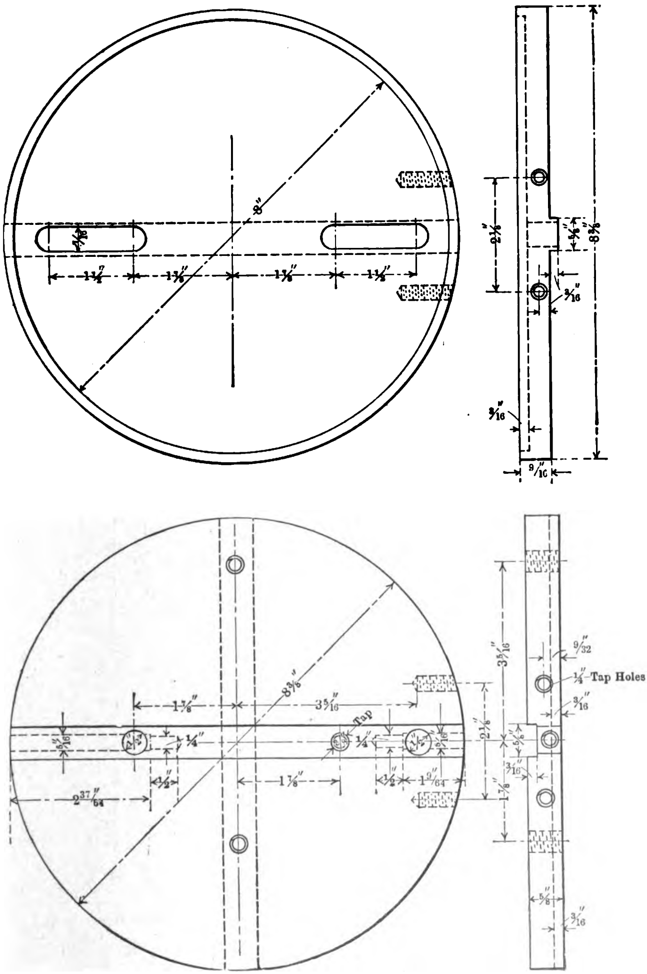


FIG. 68*a*. Details of Adjustable Face Plate

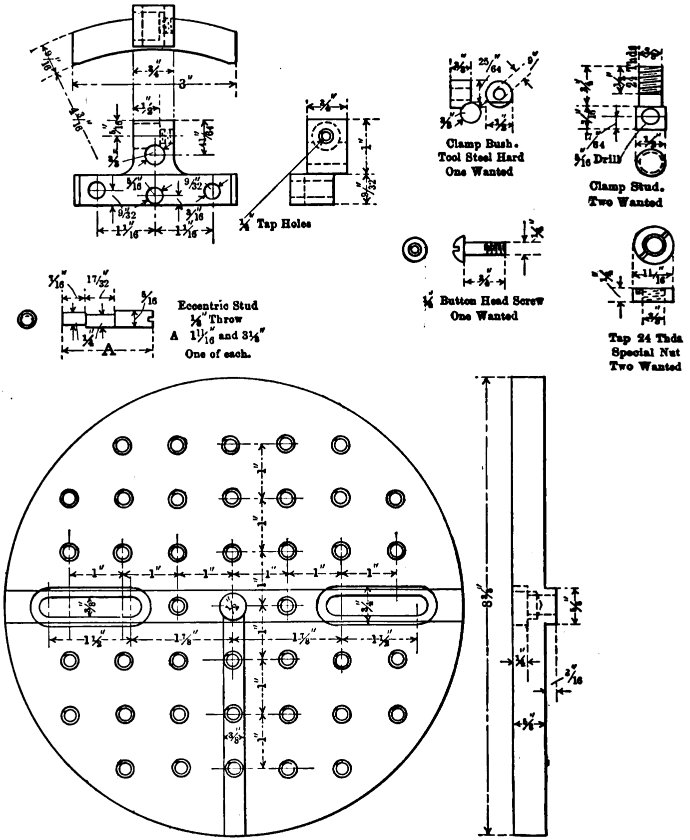


FIG. 68*b*. Details of Adjustable Face Plate

back plate and adjust 0.369 inch to produce holes *C* and *D*, and so on. You can now harden your work and place the chucking plates on the grinder and repeat as before.

Details of the plates are given in Fig. 68. These plates are being used every day in some shops and the men find that they are far superior to the button system for their work.

CHAPTER VII

CHUCKS AND CHUCKING

WHILE the selection of chucks best adapted to the work must depend on conditions in each shop, it is safe to say that for ordinary machine work, especially in a jobbing shop, the independent jawed chuck is best in almost every case. Even with round work, the independent chuck is better if it must be gripped hard for severe work, as the three- or four-jawed universal chuck, with its rack and pinions or other devices, is weak, as compared with the independent chuck. This does not refer to the two-jawed chuck, where the screw runs through from one jaw to the other; these are very strong, but are more useful on turret and chucking lathes than on the engine lathe for general work.

For holding finished work, where the work to be done is light, and the exact truth is not essential, such as polishing with emery cloth, the universal chuck is very handy, but it cannot be depended on for accurate work, as any one can readily prove for himself. And for job work, the independent chuck is indispensable, with odds in favor of the one with four jaws.

By having circles a half inch apart on the face, it is easy to set the jaws very near the center at the first trial, and the work can be gripped as tightly as necessary for any job, as there are no gear teeth to give away.

Handy Jaws for Most Work

Special jaws can be had or false jaws made to go over or on the regulars, but for a large variety of work it is hard to beat the jaws shown in Fig. 69. By setting these with the "steps" out, the long bearing of the jaw is available for the smaller work, and the steps for gripping the inside of larger work.

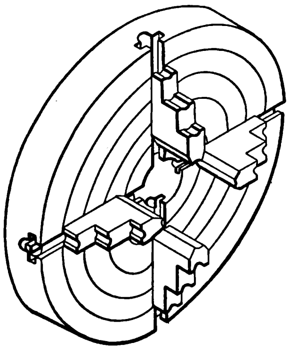


FIG. 69

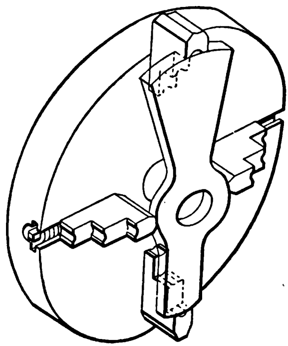


FIG. 70

Chucking Work

Always grip work as near the face of the chuck as possible, simply because it gets back nearer to the spindle. This is especially necessary when the work is not supported at the outer end by the dead center, and is all overhang. Avoid overhang in everything as far as possible, which means keep the tool point near the holder, the dead center in as close as you can, and the work near the end of the spindle.

Using these steps, or even the outside of the end of the jaws, allows chucking pieces considerably larger than the chuck itself. Reversing one or two jaws as in Fig. 70 often gives just the sort of a grip you need for odd-shaped pieces. The faces of the jaws also give a good guide for chucking rings and similar work square on the face, and many applications will come up from time to time. Like everything else in the shop it requires a little thought, if best results are to be secured.

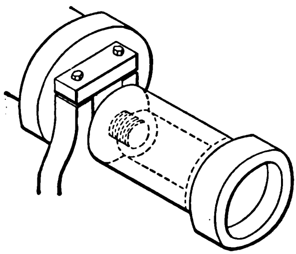


FIG. 71

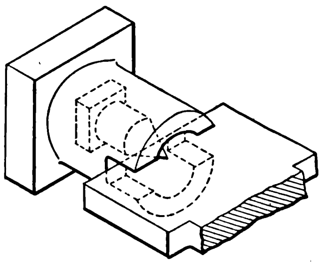


FIG. 72

Special Chucks

Special Chucks

The regular chucks do not end the question, however, as every shop has a variety of special chucks for various uses. These may be simply cast-iron cylinders, screwed on the lathe spindle to hold a standard size bar for finishing, as Fig. 71; or expanding or contracting chucks for a similar purpose, but more accurate work, as Fig. 73; or still another little device for driving a flat cutter or reamer while it is being turned or ground, as Fig. 72. Other variations will be found in many shops, or will come into use as necessity arises.

In Fig. 73 the work is held by springing the jaws through the wedging action of the coned surfaces. The internal chuck at the left is largely used for holding small rods, and the expanding chuck (or mandrel) at the right, for turning work after it has been bored. The jaws are split just the same, but are expanded by forcing the cone in. Sometimes this expanding cone is handled through the lathe spindle and operated while the lathe is in motion. Many devices are also made so as to operate the internal chuck without stopping the lathe, but these are not used on engine lathes.

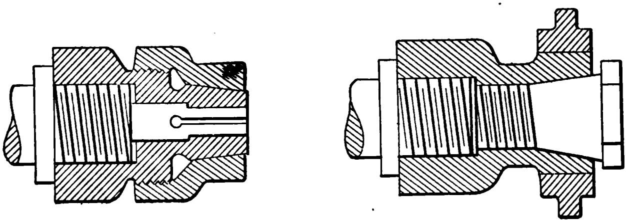


FIG. 73
Split Chucks

Making a Collet for a Lathe

It quite frequently happens in the average toolroom or machine shop that there is a scarcity of collet lathes to meet the requirements. This is perhaps more often the case in the toolroom. In the building of jigs, fixtures, dies, gages, etc., the collet lathe is certainly an indispensable tool, and in the shop where there happens to be only one, it is a case of wait, most of the time. Now any ordinary hollow-spindle lathe can be quickly and cheaply converted into a first-class collet lathe by making a fixture similar to the one shown in Fig. 74.

A represents the lathe spindle, which in this case carried a center about $1\frac{1}{8}$ inches diameter at the small end. To this center is fitted the hardened and ground taper sleeve E. This sleeve was ground to fit the collets used on other lathes. As this rig admits of much larger size collets than

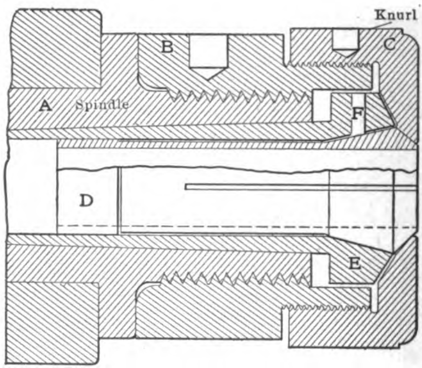


FIG. 74

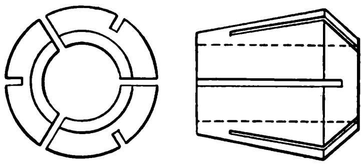


FIG. 75
How a Collet is made

the ordinary draw-in collet lathe some extra sizes can be made. The machinery steel sleeve *B* was next fitted to the spindle, and by means of a spanner wrench screwed up tight to the shoulder. This sleeve has a fine thread on front end to which a cap *C* was fitted having a taper to match the front end of the collet. The working of this rig must be apparent at sight. Both tapers of the collet being used, and as the cap *C* is made with a fine thread (20 per inch), a very powerful grip is obtained. A spanner wrench is fitted to the cap *C* also. Several have used this chuck quite extensively in their shops, where it is usually pre-

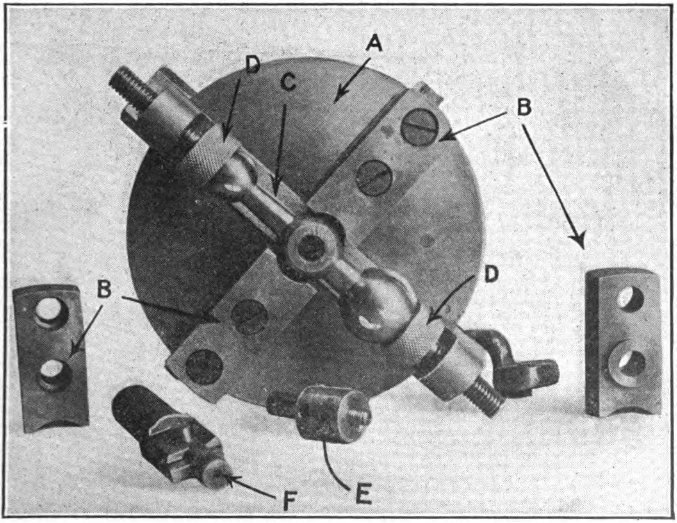


FIG. 76. Details of the Chuck for Ball Cranks

ferred to the ordinary draw-in collet. It is nearly as rapid in manipulation, is very stiff and rigid, and for holding power it is much better than the collet lathe.

Where the use of long bars is not required, an ordinary small center lathe with no hollow spindle can be used by extending the sleeve *B* and fitting it to a short collet as shown in Fig. 75. The sleeves *B* and *C* should be made of machinery steel, case-hardened; a pin *F* forms a key to prevent the collet *D* from turning.

Special Chuck for Boring Ball Cranks

Figs. 76 and 77 show a special chuck made for the Smith

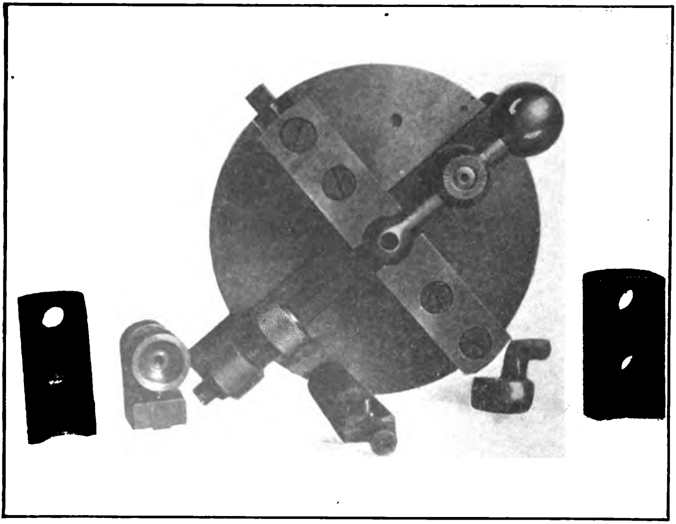


FIG. 77. Boring the End Hole for Handle

& Mills Shaper Company, in Cincinnati, Ohio. They almost explain themselves as the ball cranks are finished complete, bored and faced on both sides, and then the small ball is bored for the handle.

It is a great time saver and, as the work revolves, you can see that the hole will be in the center of the ball, as it is in plain view as the operations proceed.

A is of solid cast iron with two steel jaws with right- and left-hand threads, a dovetail slot running at right angles with the jaws. *B′ B* are two pairs of cast-iron jaws; one bored for the center ball and the other for the small ball, and have a lug on the back to fit in the steel jaw so there is no pressure on the screw that holds them. *C* is a piece of steel dovetailed to slide in the chuck and on which is fastened two lugs to hold cup centers *D D*, which line up the crank handle.

E is a plug to screw in dovetail strip *C* when one of the lugs carrying cup center is removed, and the other end is complete.

Interchanging Lathe Chucks

In the jobbing or other small shop it is quite customary to use the same chucks and face plates on different lathes, and men often say things because the threads on the ends of the spindles are not all the same. When they are different we find adapters or pieces threaded to fit the nose of one spindle and the chuck that goes on another. If the chuck happens to be threaded enough larger than the other thread, we can make a very compact adapter as in Fig. 78, but when the difference is slight, the adapter must be long and this places the chuck at a considerable distance from the spindle bearing of the lathe. This overhang is a bad thing and is permissible only when it cannot be avoided.

But there is another feature about interchanging chucks that is apt to be overlooked in shops where the finest class of work is not done. It is practically impossible to get a chuck to run true on any lathe but its own, and it isn't easy to make it run twice alike even on its own spindle. We do not always realize that a very small piece of dirt, either in the thread or on the shoulder, will throw a chuck out of true more than we might imagine, and this is equally true of face plates.

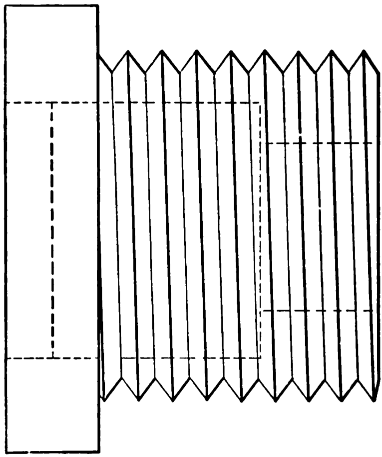


FIG. 78. Adapter for Chucks or Face Plates

If you doubt this, just try putting on a face plate and note that it usually goes on rather snugly. Now clean the thread out and note the difference. You can be sure that it does not go on in exactly the same position in both cases and the condition is changing every time it goes on.

Keeping Face Plates True

The noses of most lathe spindles are not designed so as to make it easy to maintain the truth of face plates.

The small bench or precision lathes are best in this respect as they draw the face plates or chuck onto a taper which comes nearer holding them true than the square shoulder. The long thread is entirely unnecessary as all it does is to keep the face plate or chuck from coming off, but does not hold it true; this must be done by a straight or taper portion.

When a new lathe comes in the shop the face plate should be trued upon the lathe itself before any nice work is attempted on it. In some shops the face plate is trued every time an extra nice job comes along and is not taken off till it is finished. This is a good plan to follow and also to avoid taking it off for another job before the work is done. This is sometimes difficult in a job shop, but the work will very seldom run exactly true when it is put back into the lathe again.

If the face plate is carefully tested it will be found that in some cases the cross-slide will not turn exactly square with the spindle, but will be a trifle concave. Not much, only a trifle, and this is done purposely. A concave face plate is better than one convex because it enables work to be clamped to it more firmly and for another reason.

When the face plate is concaved, it shows that the nose of the spindle is a trifle in front of a center line through the back end of the line spindle and the dead center or that the cross slide is not exactly square. Under a heavy cut there is a tendency to force the spindle nose away from the tool, which may overcome the slight error and make it turn straight. At least that is the theory and as it is next to impossible to have the cross-slide face exactly square, the error is made so as to make the face plate concave instead of convex.

Using Screw Chucks

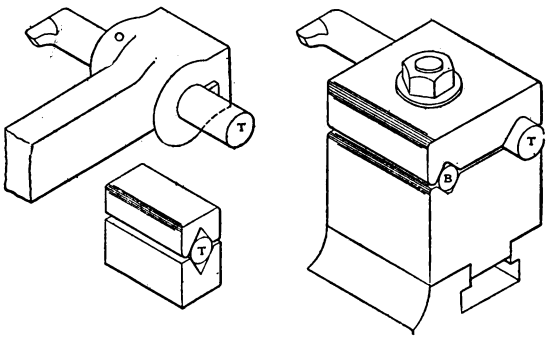
Screw chucks that go on the lathe spindle and are threaded to receive work are a great source of annoyance from this cause. It seems as though they ought to run true, but they never do unless far more than ordinary care is taken. Every joint gives a chance for dirt to get in and that means error in the running of the work.

On very nice work where screw chucks are used, they make a new chuck for every lot that is to be made. Turn it up in the lathe, thread it in place and never take it out till the whole lot is finished. Then the chuck is thrown in the scrap heap and a new one made for the next lot. This is the result of long experience and is much cheaper than any attempt to save money by using the old chuck over again. There is a large variety of work, however, where this precaution may not be necessary, where the other way may be plenty good enough, and it is up to the really practical machinist to decide when to use the ordinary and when the most refined methods.

CHAPTER VIII

BORING TOOLS

Tools for boring are a very different proposition from those for turning. Turning tools can be supported very close to the cutting edge, but this is impossible in boring



FIGS. 79 AND 80. Boring Tools

tools unless the hole is very short. The spring of the boring tool is always with you. The thing to do is to reduce it all you can.

Some of the old forged boring tools were weak in this respect, as they sacrificed strength in the stock to get a long hook on the cutting point. This does not necessarily go with the forged tool, but was done to allow for many grindings without redressing the tool.

Boring Tool Holders

The newer tools, such as shown in Figs. 79 and 80, are better in this respect, as the stock of the tool can be as large as will go in the hole, after allowing for the length of cutting lip. This gives the cutting edge a good support, and allows better work to be done.

The holders in Fig. 79 are both held in the tool-post; neither of them is new but both have many uses. The

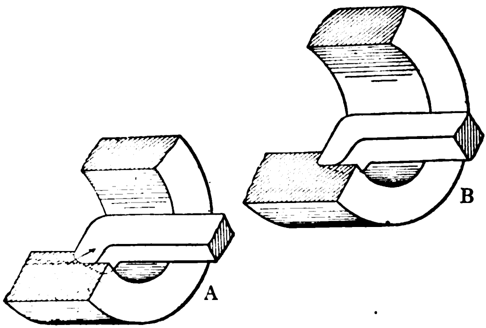


FIG. 81. Tools in the Cut

V-blocks are simple, but the other has the advantage of being able to replace the tool in the same place, as the holder remains fixed in the tool-post.

Fig. 80 is a more substantial block that goes right on the tool-block, and is more solid than the others. By having a large and small *V*, it is easy to hold different-sized boring tools very firmly. In each case the end of the tool is marked *T*, and *B* is simply a piece of steel the size of the small tools to clamp the top piece down against.

Grinding Boring Tools

The tendency of tools ground as shown in *A*, Fig. 81, is to be crowded away from the cut, as shown by the arrow, and as it dulls this is increased. This means that the hole will be smaller at the back than at the front, and that light cuts must be taken to get it straightened out.

With the cutting edge square the spring is all down and not away from the bore, but it is necessary to modify this a little in heavy boring cuts by rounding the corner a little.

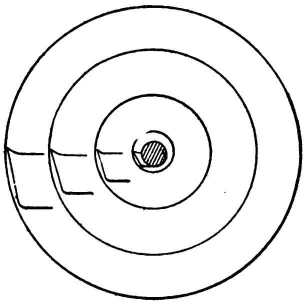


FIG. 82. Setting Boring Tools

The clearance of boring tools must be watched carefully, especially in small holes, as can be seen in Fig. 82, which shows different diameters and the clearances for each.

Boring Bars with Inserted Cutters

On work with a fair diameter it is often possible to use inserted cutters to advantage, as shown in Fig. 83. These have the advantage of costing little to keep up as the cutters can be made of commercial sizes of high-speed steel. These can be removed when dull and sharp ones put in their places without disturbing the setting of the boring bar itself; and where a hole is to be threaded, the threading cutter can be put in place of the boring cutter, and readily set square with the work.

The bar *A* at the left of Fig. 83 is perhaps the most simple, having a round cutter held in place by a screw bearing on a flat place on top of the cutter. Next comes a square cutter, *B*, inserted at an angle so as to reach ahead

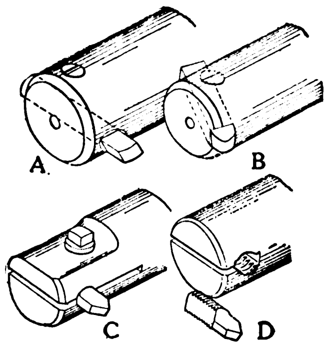


FIG. 83. Inserted Cutter Boring Bars

of the bar, and also held by the screw. Numerous modifications of this can be made to suit individual ideas.

A Threading Tool

The threading tool, shown at *D*, can have either a plain round cutter, clamped by splitting the bar, or a threaded cutter, as shown. The object of the thread is to make it easy always to put the cutter back in the same place after grinding, the threads locating it with the point the same distance from the bar.

There is no need of cutting a full thread on the cutter; in fact a single cut with the threading tool will be sufficient for the threads in the bar to bite into. To make it easily removable, file away the thread at the sides of the hole, and also on the cutter. Then by turning the cutter at right angles, it can be slipped into place, given a quarter turn, and clamped ready to go to work.

Chatter in Boring

Work that overhangs to any great extent will chatter in boring as in turning. This can generally be stopped by supporting the work on the outside with a steady rest of some sort. On very thin work this may not be the case, and here it will be helped by lightly forcing in a piece of wood beyond the point to be bored, taking care not to spring the work. This helps stop the vibration, which is what causes chattering. In turning hollow work on the outside the chatter can often be stopped, by filling the inside with waste, forced in fairly tight. This is simply another case of stopping the vibrations.

Prevent Damage to Chucks

One point in boring, both for chuck and face-plate work, is to be sure to allow a space behind the work for the tool to break through, if the hole goes clear through the work, so as to save digging into the face plate or chuck. This can be done by placing parallel strips behind the work, out of the way of the bore, or by having a ring which can serve the same purpose. But no matter how, be sure it is done, and there is no excuse for ruining a chuck or face plate in this way.

Precision Drilling and Reaming

For some classes of work, the kind of drill shown at *A*, Fig. 84 herewith, would be ideal. Drills like this can be made quickly from drill rod and are commonly known as cannon drills. They will follow the guide of the bushing precisely, and can be used as reamers also. For brass they work as well, and can be made more quickly, as shown at *B*. They are just slanted off exactly to one-half the diameter at the end, and the proper clearance given. For accuracy they are much superior to the twist drill. The hole made will be precisely the size of the hole in the bushing, even if the drill is not a close fit in it.

Nice Boring in the Lathe

One of the essential features in using the boring bar in the lathe is to have the bar supported as close to the cutting edge as possible. *C* shows a method which is used by some toolmakers with considerable success. To begin with, the work is clamped to an angle plate fastened to the face plate, so that it can be clamped perfectly square with the face plate, and not sprung, as is sometimes the case when we attempt to pull the work back against the face plate by two or more bolts.

As shown, the boring tool *A* is clamped in the holder *B* so that the tool-block and carriage travel with the bar when fed into the work by the tail spindle *C*. In some cases the tail spindle is not used for this purpose, and the bar is fed into the work solely by the carriage. Another way which is advocated by many lathe men is to move the holder *B* as close to the work as possible and feed the bar

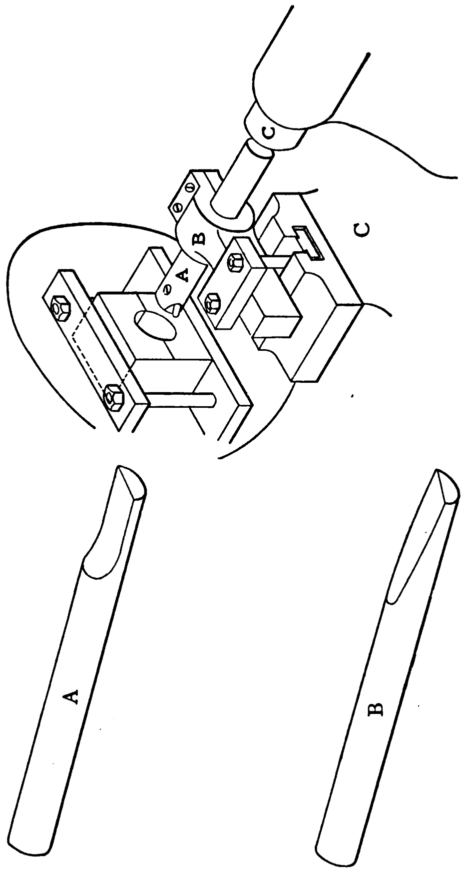


FIG. 84. Boring Tools

A through it by the tailstock *C*, as this keeps the support close to the outside of the work at all times. On the other hand, it allows the tool to spring more as it gets farther into the hole, for the cutting point is constantly going away from the support to the bar, while if it is clamped in the holder, the cutting point is at a constant distance from its support at all times, so that the spring of the bar should be practically constant.

Boring Small Cylinders on a Lathe

The Moline Tool Company had a contract to bore a lot of gasolene-engine cylinders and rigged up one of the lathes for a boring machine in a way, that, while it is not new, is a good rig for a lathe in this work.

It proved itself so efficient that two local concerns building gasolene engines have fitted up lathes in the same way and are getting very good results from them. This is shown in Fig. 85.

The lathe used had rather a long bed, so a bar about 2 feet long and 5 inches in diameter was used for a boring bar. The outer end was threaded to fit the boring heads, which will be described later. The rear end was fitted into the bore of a 3-jaw chuck, although the sketch shows it threaded to fit the nose of the spindle, which is a better plan.

A heavy bearing for the bar was fitted to one V of the lathe bed, allowing the bar to project far enough beyond it for boring the job in hand. This bearing was made very heavy, bolted securely to the lathe bed and babbitted in place.

A suitable jig was made for holding the cylinders on the

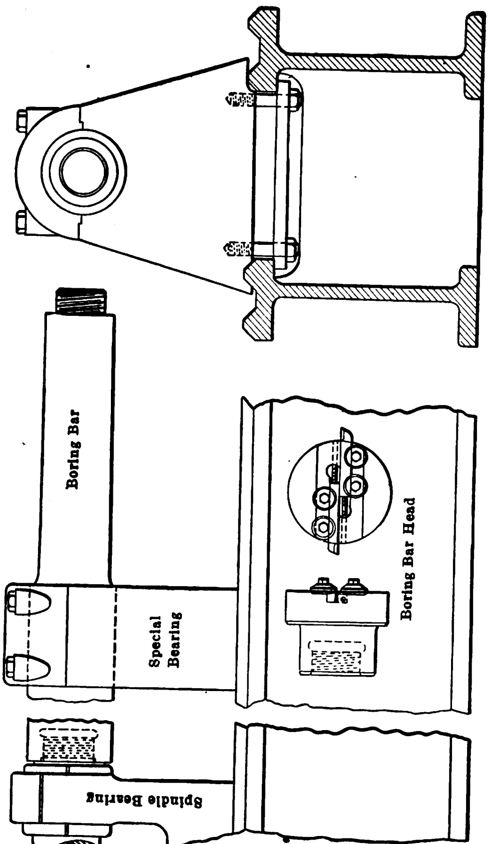


FIG. 85. Rig for Boring Small Cylinders on the Lathe

lathe carriage. The boring heads were threaded to fit the nose of the boring bar and flatted to allow a wrench to be used for readily removing them. Each head had two cutters, which were set in slots so that the cutting edges were radial. Each cutter was held by two cap-screws having case-hardened clamping washers, which bore on top of the cutter and on the face of the cutter head, as shown. The surface of the cutter head was relieved so that these clamping washers would only bear on the back edge. Each cutter was set out to position by a headless set-screw directly back of it.

A roughing and a finishing cutter head was used so as to avoid changing tools. A hole was also drilled in the side of the finishing cutter head into which a small cutter was slipped for beveling the end of the cylinder to allow the piston rings to be entered readily.

After running the second cutter head through, a reamer was screwed onto the end of the bar. Instead of pushing this through, the feed of the carriage was reversed and pulled it through. This seemed to give better results than the customary way, although, of course, it cannot be used in a cylinder having a closed end.

For facing the ends of the cylinders, a sweep tool was first used, which gave fair results when a notched tool was first used to remove the scale.

A facing arm with a traveling head was then made, on which the tool was fed out by a star wheel on the outer end of the feed screw.

By using a comparatively long bar, not only is the rig made very stiff, but the carriage is brought on to a part of the lathe bed which in ordinary lathes is practically not worn at all, so it is an easy matter to keep it gibbed to the bed uniformly.

It was discovered that while boring the cylinders with this rig, that after the cylinders got cold they were not exactly round, due to the fact that the removing of the roughing cut released strains in the castings and allowed them to change their shape. This was overcome by removing the cylinders and allowing them to cool or season after taking the roughing cut.

CHAPTER IX

TAPER TURNING

ANY machinist who has tried to turn a piece of work perfectly straight knows the difficulty of getting the tail center set just right to give exactly the same diameter at each end. But when it comes to setting the lathe for just the right taper, especially without several trials that take a lot of time, it isn't any easier than turning a piece straight.

Next to selecting the change gears for thread cutting, the question of how much to set the tailstock over for a given taper is asked more often than any other. While it is a simpler problem than the other, in some ways it has some points which are not always clear.

Measuring the Taper

The first thing to consider is just what the taper is, as this is sometimes a point for differences of opinion. Some measure the taper on each side, while the usual way is to take the total taper, as with pipe threads. The standard pipe taper is $\frac{3}{4}$ inch to the foot, which is the same as $\frac{1}{16}$ inch to the inch, or 1 inch in 16 inches, whichever way you like best. In many cases it is easier to have the taper per inch, but we have to take these things as they come to us on the blue-prints or drawings.

The amount of taper depends on the diameter at the two ends of the taper part of the bar and on the length of the taper, but the amount of offset for the tailstock depends on the length of the whole bar regardless of how much is turned taper. In Fig. 86, *A* is a section of a taper plug with a taper hole. The difference in diameter is 1 inch in

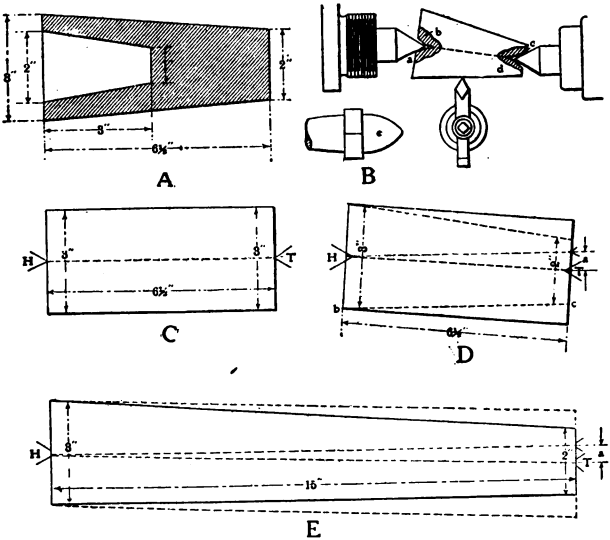


FIG. 86. Turning Tapers

both cases, but the taper of the hole is much sharper, as it is less than half as long as the plug. The outside is 1 inch taper in 6½ inches, while the hole is 1 inch in 3 inches.

The Lathe Centers

There is some question as to what effect the depth which the centers enter the work has on the setting over of the tailstock; but as it is practically out of the question to set it over the exact distance the first time in any case, this need not enter into the question for general work. *B* shows a very short piece being turned at a sharp taper. It can be seen how the outer edge *a* and the point *b* bear on one end; on the other it is opposite, as in *c* and *d*. This is very hard on centers and makes it difficult to keep them from cutting. In fact, if much taper work is to be turned, a point more blunt than the usual 60 degrees may be found better. Some use a somewhat rounded point, as in *e*.

Forgetting about the center and its action and considering that the work is held between the points of the centers as in *C*, *H* being the head and *T* the tail center, how much must we set the tailstock *T* over to cut the outside taper shown in *A*? This is 3 inches at one end and 2 inches at the other, so we must reduce the small diameter 1 inch, which means ½ inch on a side, and we set the tailstock over ½ inch, as at *a* in *D*. The tool moves along the line *b c* and cuts off ½ inch at the small end, running out at *b*. If the tailstock has been set over just ½ inch, the outside will be the correct taper, as shown by dotted lines.

In *E* the tailstock is set over the same amount and the small end reduced to 2 inches as before; but as the piece is 15 inches long instead of 6½ inches, the taper per inch or per foot is very much less.

Count Full Distance Between Centers

In Fig. 87, *A* shows a straight bar in the lathe before setting over the tailstock; *B* with tailstock set over and *C* with the bar turned to the desired taper. Calling this bar 3 inches in diameter and 30 inches long, how much must it

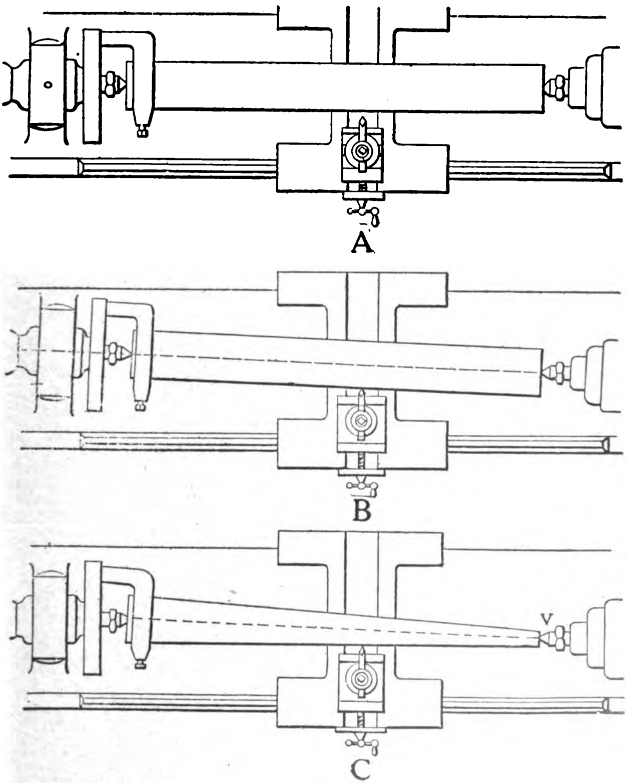


FIG. 87. Setting over the Tailstock

be set over to make a taper of 2 inches in 24 or within 6 inches of the whole length?

As the taper is 2 inches in 24 or 1 in 12 inches or $\frac{1}{12}$ of an inch in 1 inch, in 30 inches it would be $\frac{30}{12}$ or $2\frac{1}{2}$ inches; so the tail center must be set over one-half of this or $1\frac{1}{4}$ inches. An example of this work is in turning the taper on the end of a piston rod, where the taper may be 6 inches long and perhaps an inch to the foot, as in the above case. The rod may be 48 inches long, and the whole length must be considered in setting over the tail center. In 48 inches the taper would be $\frac{48}{12}$ or 4 inches, so that the tailstock would have to be set over 2 inches. The two points to remember are the getting of the right taper and always to consider the total length of the piece regardless of the taper portion. And it makes no difference where the taper portion is, whether at the tail end, the middle or near the headstock, the set-over is the same in any case.

Ways of Figuring Tapers

It is generally easier to figure tapers if we reduce them to the amount per inch in order to get at the offset. If the taper is given per foot, we divide it by 12, as, if it is pipe-thread taper of $\frac{3}{4}$ inch per foot, we have $\frac{1}{12}$ of $\frac{3}{4}$, or $\frac{3}{48}$ or $\frac{1}{16}$ per inch. If the taper is given 1 in 8 or 1 in 15, the taper per inch is of course $\frac{1}{8}$ or $\frac{1}{15}$ inch to the inch.

Tapers are frequently given in degrees, and in such cases they are usually turned with a compound rest divided into degrees; but it is sometimes handy to know what the taper would be in inches, so we give a table which may help if you get a blue-print with the taper marked in degrees. It is given to four places of decimals and you can use as many as circumstances seem to demand.

Tapers in Degrees

This table shows that to cut a taper of 4 degrees on a bar 10 inches long means a total taper of 0.698 inch, and that the tail center must be set over one-half of this or practically 0.35 inch.

TABLE 1. — TAPERS IN DEGREES AND INCHES PER FOOT

Total Taper in Degrees	Equivalent Taper per Foot	Set Over Tailstock or Taper Attachment per Inch of Length
1	.20952	.00873
2	.41904	.01745
3	.62832	.02618
4	.83808	.03490
5	1.04688	.04362
6	1.25664	.05234
7	1.46520	.06105
8	1.67616	.06976
9	1.88496	.07846
10	2.09376	.08716
11	2.31040	.09585
12	2.51328	.10453
13	2.71680	.11320
14	2.93140	.12187
15	3.14064	.13053

TABLE 2. — TAPER AND SET OVER FOR TAILSTOCK

Total Taper in Inches per Foot	Set Over Tailstock or Taper Attachment per Inch of Length
$\frac{3}{32}$ — .09375	.0039
$\frac{1}{8}$ — .125	.0052
$\frac{3}{16}$ — .1875	.0078
$\frac{1}{4}$ — .25	.0104
$\frac{3}{8}$ — .375	.0156
$\frac{1}{2}$ — .5	.0208
$\frac{5}{8}$ — .625	.026
$\frac{3}{4}$ — .75	.0312
$\frac{7}{8}$ — .875	.0364
1 — 1.	.0416
1 $\frac{1}{4}$ — 1.25	.052
1 $\frac{1}{2}$ — 1.50	.0624
1 $\frac{3}{4}$ — 1.75	.0728
2 — 2.	.0832

TAPERS IN DEGREES AND THE EQUIVALENTS IN INCHES

1	0.0175	7½	0.1310	25	0.4434	45	0.8284
2	0.0349	10	0.175	30	0.5360	50	0.9326
3	0.0524	15	0.2633	35	0.6306	55	1.0411
4	0.0708	20	0.3526	40	0.7279	60	1.1547
5	0.0873						

The next table will be a guide in the opposite direction by giving something of an idea as to what angle a given taper is. Taking the pipe taper of $\frac{3}{4}$ inch to the foot or $\frac{1}{16}$ inch to the inch, what angle is it? Consulting a table of decimal equivalents or dividing 1 by 16 gives 0.0625, and this is between 3 and 4, very nearly 3½ degrees.

In the same way a taper of 1 in 12 or $\frac{1}{12}$ inch to the inch is 0.083, which comes very close to being 5 degrees; to cut this, we set the tailstock over 0.042 inch for every inch of length in the bar.

Another Way of Turning Tapers

There has long been an idea that setting the tailstock over was not the best way, although it is probably used more than any other; but many lathes have a taper attachment consisting of a bar at the back *A*, on which a shoe slides and moves the tool-block *T* through the bar *E*, as shown by *D*, Fig. 88. The cross-feed screw is of course disconnected so that the tool-block is free to slide under control of shoe. Blocks *B* and *C* are fastened to the back of the lathe and hold the ends of the bar *A*, as shown, at any desired angle, the angle of the bar giving a similar taper to the work. As shown, the bar would be turned smaller at the head end than the tailstock, just the reverse of the usual way.

These bars are graduated at the ends, sometimes in inches and fractions, sometimes in degrees, sometimes inches at one end and degrees at the other. Some lathes have the bar swing from one end only. This plan allows a much better contact of the centers in the work and has many advantages. Credit for this device belongs to Dwight Slate, of Hartford, Conn., who brought it out in 1867.

Boring Taper Holes

Most taper boring is done with a compound rest, the piece of work being held in a chuck or strapped to the face plate and the outer end supported in a steady rest. But

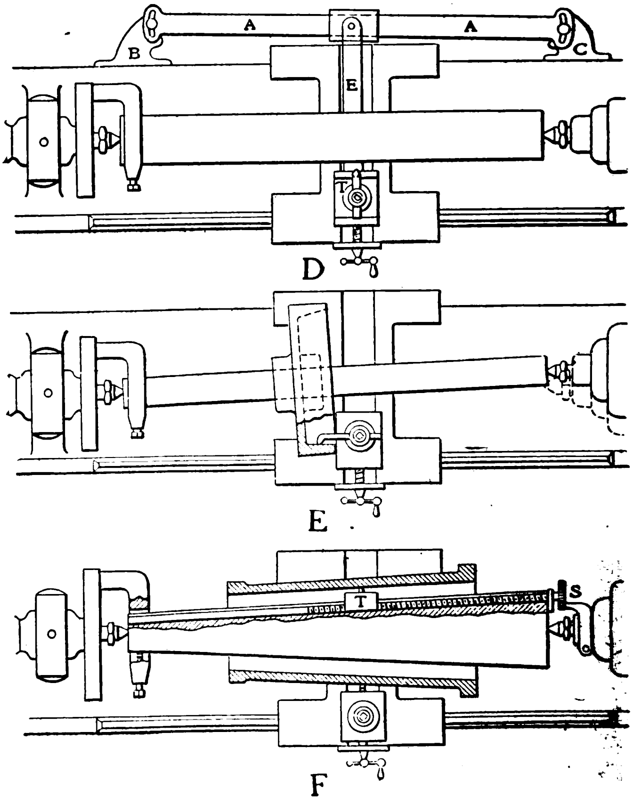


FIG. 88. Other Taper Methods

there are many jobs which can be handled by driving the work on an arbor or mandrel, as in *E*, and setting the arbor over just as though that were the piece to be tapered, except that for boring, the tailstock must be set over the other way, as will be seen in the illustration. This is a pulley for a friction-clutch countershaft, and is bored with a regular lathe tool.

If the taper is $\frac{1}{8}$ inch to the inch and the arbor 24 inches long, the total taper would be 3 inches and the offset one-half of this, or $1\frac{1}{2}$ inches. With a shorter arbor, say 16 inches, the offset would be only 1 inch, and so on.

Use of Boring Bar

Long taper holes are sometimes bored by using a star-feed boring bar. If the regular straight-feed bar is used, as is usually the case, the tailstock is set over in the usual way and the work driven from the face plate, being supported by steady rests or otherwise. The bar does not turn in this case, but is held stationary by the tailstock while the live center turns in the bar. The star feed must also be worked by a pin or pins, which moves around with the work so as to strike the star wheel once or twice during the revolution of the work, according to the feed desired.

For special work it may be better to make a bar of the right taper and drive the bar while the work is held on the lathe carriage, as *F*. The tool *T* travels along the dovetail slide in the bar, driven by the screw, which is turned a portion of a revolution at each turn by one prong of the star wheel striking the stop on the tail spindle at *S*.

This would not pay unless a large number of pieces were to be bored alike, although by driving the work as first suggested and setting the tailstock over as desired, this bar could be used for quite a variety of work.

Another plan of using the star-feed bar in the lathe is shown in Fig. 89. In this case the straight bar is used by locating a center at the proper point to swing the bar to the required taper. The feed is given the star wheel from the tailstock end.

A refinement for the end of any bar which runs at a taper in this way, such as the boring bar shown or the ends of

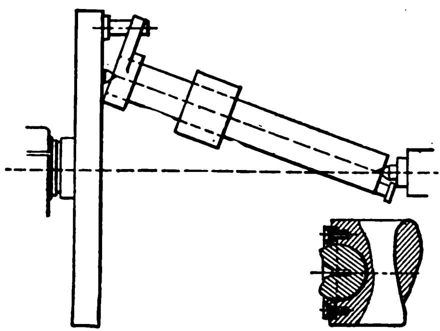


FIG. 89. Taper Boring Bars

an arbor used for taper work, is shown in section in this same figure. It consists of a half-round recess in the end of the bar into which a ball, having a center in the outer end, is held by the retaining plate. Never having used a device of this kind we are not recommending it, although it may be all right for some classes of work.

Some of the old lathes swiveled the head on the bed, which made it very handy for boring, but had objections in other ways. Special lathes, such as those used in turning the shanks of twist drills, swing the head and tailstock, together with the inner set of ways or shears, so that the work is at the right angle with the carriage ways. This is a very good plan when there is work enough to warrant it.

Cutting Taper Threads

Pieces to be threaded are turned up in the regular way to the right taper for the job; but when it comes to cutting the thread, there is a difference of opinion in some quarters, although it is pretty well agreed that there is only one right way to do it.

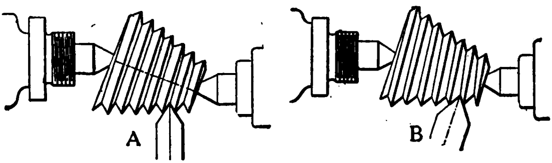


FIG. 90. Cutting Taper Threads

In straight work we set the threading tool square with the surface to be threaded, and some do this with tapered work as well as with straight. It is easier but it isn't right, as will be seen at *A*, Fig. 90. This makes the front side of the thread almost at right angles to the center line, and the back of the thread becomes more nearly parallel and has a very poor hold on the other piece which screws into or on it. The angles would vary with every taper, being less objectionable as the taper decreases.

The proper way is to set the thread tool at right angles with the center line of the piece, as shown at *B*, as this gives an equal angle to both sides of the thread, although the back side is necessarily shorter than the front; but it will hold better than the other.

To do this, hold your thread gage in line with the points of the centers, either by holding a scale between the centers or in any other way that is easy for you. The tool is shown in position in both *A* and *B*, and the setting of the tool is important if you want good threads on the work.

Points to Remember

The main points about taper work are:

Always consider the length of the work or the distance between centers instead of the length of the taper portion.

Make the offset half the amount of the total taper, which would be the difference between the diameters of the large and small ends if the taper were extended the whole length of the piece.

Always set the point of the tool at the hight of the lathe center in taper work. If it is set above or below the center there will be a slight change in the taper between each cut, or as the diameter is reduced. You can see this by imagining the tool set ½ inch below the center. When the work gets down to 1 inch in diameter, the tool will not cut at all at the small end, but will cut more and more as it feeds on to the larger diameter. This is a very important point to remember.

In cutting threads, set the tool square with the center line of the work and not square with its surface.

It is always well not to set the tailstock over the full amount at first, so that the small end will be larger than required, rather than smaller. If the taper is too sharp, from setting the tailstock over too much, there is danger of spoiling the piece by having the taper run up too far on the bar in making a fit into a tapered hole. In making long taper fits it is easier to have the bearing for a short distance on each end with a relief in the center, and this generally answers the purpose equally well.

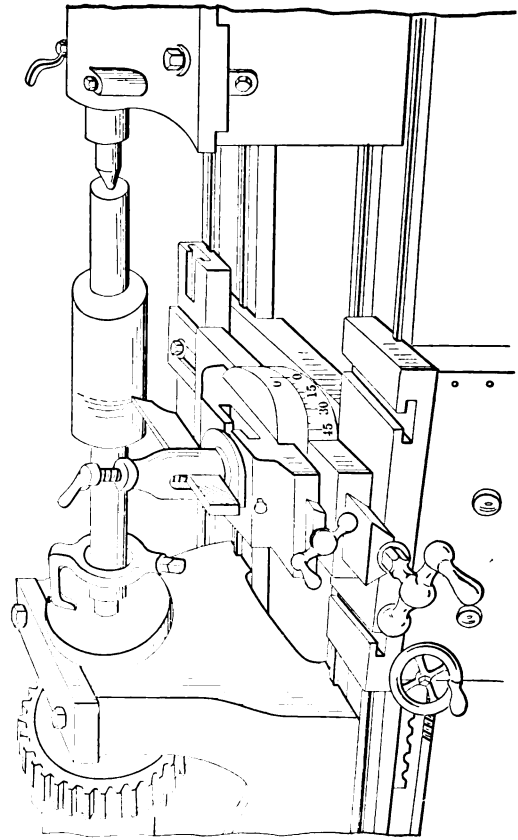
Setting Compound Rests

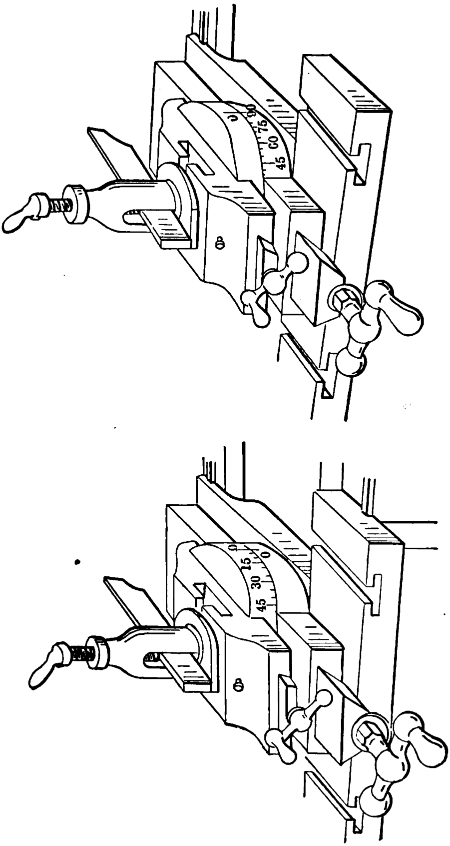
The difficulty experienced in measuring angles and setting dividing heads and compound rests comes mainly from two causes: A confusion of ideas as to whether half of the total angle is meant and the position of the base line.

In the compound rest we have the work measured in angles from a line drawn between the lathe centers while the slide is at right angles to this as in *A*, Fig. 91. If we have to turn the bevel on a valve-seat reamer that is 60 degrees total angle, as in the sketch, we must set the slide rest at 30 degrees from the line of the centers. This very seldom means that we can set it to 30 degrees on the compound rest as they are usually divided to read from the cross-movement at right angles to the center.

Perhaps the easiest way is to swing the compound rest parallel to the lathe centers with the handle toward the headstock, and the two 45-degree marks come together if they are divided in this way. Then move the handle end out until 30 degrees have passed by the 45-degree mark, or until 15 degrees on the upper coincides with the 45 if both are graduated. No matter how it is divided, move the compound rest 30 degrees without regard to the numbers as they appear.

When you are facing work to any desired angle and the work is normally in line with the cross-slide, you can read


A



B C

FIG. 91. Graduations of Compound Rests

the divisions just as they are graduated, bearing in mind that each degree the slide is set off means 2 degrees total angle for the work.

A, *B* and *C*, Fig. 91, show three methods of graduating compound rests on a lathe or swivel head on a planer. In *A* the base is divided into 45 degrees each side of a zero line at the side or at right angles to the cross-slide. With this graduation the scale shows the degrees moved through by the tool-slide with reference to the cross-slide. If we set it to 15 degrees we can face off a piece 15 degrees on each side of the end, but this would leave the end with a total angle of 150 degrees with the center line of the work.

In *B* the graduations are reversed, being on the upper slide, and the zero on the base. The results are the same except that we read on the opposite side of the zero mark of the graduations; swinging the upper slide to the left 15 degrees, we must read the angle on the side of the scale now hidden from view.

C shows a different plan and one which has some things to recommend it: the 90-degree mark in place of the zero, in A, the 75-degree in place of the 15, but, of course, the 45 comes in the same place on account of its being half-way between the two.

This method of graduation shows the exact angle that will be cut each side of the center line, and we get the total angle by doubling the figures of the graduation. If we move it to 75 it will cut 75 degrees each side of the center line. If it is moved to 45 it cuts a 90-degree total angle, and if to 60 it cuts 30 degrees away on each side, leaving 120 degrees included angle. Bearing this in mind, there should be no confusion as to what angle will be cut.

After one becomes familiar with the divisions of the machine he is handling, it is easy to set by the numbers, but as they are apt to be confusing, it is safer to check the reading by counting the degrees from the point when the slide is parallel with the lathe centers.

The following table gives the tapers most commonly used in both degrees and inches per foot. It will save time and trouble in figuring out the desired taper.

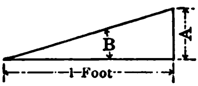


FIG. I

If Taper Foot for included Angle is wanted as C in Fig. 2, Divide the Taper per Ft. by 2 and take the corresponding Angle each Side the Center Line.

FIG. 2

Taper per Ft	Angle B		Taper per Ft	Angle B		Taper per Ft	Angle B		Taper per Ft	Angle B	
A	Deg.	Min.	A	Deg.	Min.	A	Deg.	Min.	A	Deg.	Min.
1/8	0	36	1 7/8	8	53	5 1/4	23	38	8 3/4	36	6
1/4	1	12	2	9	28	5 1/2	24	37	9	36	53
3/8	1	47	2 1/4	10	37	5 3/4	25	36	9 1/4	37	36
1/2	2	23	2 1/2	11	46	6	26	35	9 1/2	38	21
5/8	2	59	2 3/4	12	54	6 1/4	27	30	9 3/4	39	5
3/4	3	35	3	14	2	6 1/2	28	25	10	39	48
7/8	4	10	3 1/4	15	9	6 3/4	29	21	10 1/4	40	30
1	4	46	3 1/2	16	16	7	30	15	10 1/2	41	12
1 1/8	5	21	3 3/4	17	21	7 1/4	31	8	10 3/4	41	52
1 1/4	5	57	4	18	26	7 1/2	32	1	11	42	32
1 3/8	6	32	4 1/4	19	30	7 3/4	32	50	11 1/4	43	9
1 1/2	7	7	4 1/2	20	33	8	33	43	11 1/2	43	47
1 5/8	7	43	4 3/4	21	36	8 1/4	34	30	11 3/4	44	24
1 3/4	8	18	5	22	37	8 1/2	35	18	12	45	0

TABLE 3. A Table of Common Tapers

CHAPTER X

THREAD CUTTING

THERE is nothing mysterious about thread cutting any more than in plain turning if we look into it a little. If we extend the back end of the spindle, cut a 6 thread on it and use a rig like a fox lathe, it is clear that we will cut a 6 thread in the lathe. The engine lathe is just as simple only we put the lead screw down in front of the bed and drive it by gears, part of which are hidden, but it works in just the same way.

If the lead screw is geared "even," that is, if it makes one turn for each turn of the spindle, it must be clear that we will cut the same thread in the lathe as there is on the lead screw. For if the lead screw is 6 threads to the inch and makes one turn to each turn of the spindle, it is plain that the carriage and the tool will be moved $\frac{1}{6}$ of an inch for each turn of the spindle.

Before going ahead with the lathe, put gears having the same number of teeth on both the stud and the lead screw and see if you cut the same thread as the lead screw. If not, take the thread cut as the pitch of the lead screw in all calculations to be made.

Having found what thread will be cut with "even" gears, it is easily seen that to cut a coarser pitch thread the lead screw must be driven faster than the spindle, while to cut a finer pitch the lead screw must run slower than the spindle. This means that for a finer pitch the largest gear must go on the lead screw, and for a coarser pitch the small gear goes on the lead screw.

Still calling the lead screw 6-pitch and wanting to cut a 3-pitch thread we see that the lead screw must turn twice as fast as the spindle, so that the stud gear must have twice as many teeth as the screw gear. This means that a 24 on the lead screw and a 48 on the stud would cut a 3-pitch thread on the work.

If, on the other hand, you wanted to cut a 12 thread, the screw must turn half as fast as the spindle, so that the same gears can be used, but the 48 goes on the lead screw and the 24 on the stud.

Figurings Gears for Thread Cutting

There are many ways of doing this, but they are all based on the proportion between the movement of the thread tool, which depends on how fast the screw is turned and its pitch, and the revolutions of the lathe spindle.

It will pay to take plenty of time to get the principles on which screw cutting is based fixed thoroughly in your mind before going further, as on that depends your success in this line. When the idea is thoroughly grasped you can tackle any job of thread cutting that comes along without a moment's hesitation. Having done this, and it may help to do some actual experimenting on a lathe, you can make rules with your eyes shut, if you want to use them, as they are very simple after you start right.

Perhaps the easiest way is to multiply both the thread to be cut and the pitch of the lead screw by any number (the same number in both cases) that will give gears found in the set. If the lead screw is 4 to the inch and we wish to cut a 10 thread, multiply both 4 and 10 by any number such as 6, giving 24 and 60; or by 5, giving 20 and 50; or by 4½, giving 18 and 45, although the latter are not likely to be found. Put the gear obtained by multiplying the thread to be cut, on the lead screw, and the other on the stud. The reason for this is that the thread to be cut depends directly on the revolutions of the lead screw. When the thread to be cut is finer than the lead screw, it is clear that the lead screw must turn slower than the spindle, so the larger gear goes on the head screw.

This shows us why we make the rules as follows:

1. *To find the gears for cutting any thread.* Multiply the thread to be cut and the lead screw by any number that will give two of the gears in the train. Always multiply both the thread to be cut and the pitch of the lead screw by the same number. Put the gear found by multiplying the thread to be cut on the lead screw and the other on the stud.

2. *To find what thread will be cut by any pair of gears.* Multiply the pitch of lead screw by its gear and divide by the gear on the stud.

3. *To find gears to cut a thread faster than 1 to the inch.* Divide the distance between one complete turn by the distance between threads in the lead screw. This shows how many times the lead screw must turn for each turn of the spindle. Use gears that will do this and you are all right.

Examples: Lead screw 5 to inch to cut a 12 thread? Multiply both by 5. $5 \times 5 = 25$, the gear for the stud. $5 \times 12 = 60$, the gear for the lead screw.

What thread will be cut by a 64 gear on lead screw, 48 on the stud and lead screw 4 to inch?

Multiply 64 by 4 as in Rule 2, and divide by 48. $64 \times 4 = 256$. Dividing by 48 gives $5\frac{1}{3}$ threads per inch. Prove this by Rule 1, using 12 as a multiplier.

What gears will cut a thread $1\frac{3}{4}$ inch pitch, lead screw 4 to the inch?

Lead screw pitch $= \frac{1}{4}$ inch and $1\frac{3}{4} = \frac{7}{4}$ so the thread to be cut is 7 times as fast as the lead screw, and the lead screw must travel 7 times as fast as the spindle. This will be impossible without compounding, as we shall see.

Compounding Gears

The compounding of gears is simply a way of avoiding the use of very large gears and of requiring a much larger number of gears in the set. If the lead screw is 4 to the inch and you want to cut a 36 thread, the screw must turn only $\frac{1}{9}$ as fast as the lathe spindle, so that the spindle will turn 9×4 or 36 times while the screw turns 4 times, as this will move the carriage and the thread tool just one inch.

As the screw moves so much slower than the spindle we must put the small gear on the stud. We seldom use a smaller gear than 24 teeth and $9 \times 24 = 216$ for the gear on the screw. To avoid this we use a compound gear set between the stud and the screw, varying the proportions of the compound gears as we think best.

In this case we can take a 24 and a 72, making a 3 to 1 combination, and reduce the motion between the stud and the screw to one-third, so that we select the gear for a thread $\frac{1}{3}$ of 36 or 12. As 12 is three times the lead of the screw, the gear on the screw must be 3×24 or 72. So we have the

24-tooth gear on the stud mesh into the 72 of the compound, and the 24 of the compound mesh into the 72 on the screw.

Analyzing this we can see that the 24 gear of the stud will turn the 72 gear of the compound just $\frac{1}{3}$ of a turn or 24 teeth. At the same time the 24 gear of the compound is also turned $\frac{1}{3}$ of a revolution or 8 teeth, and moves the 72 on the screw an equal amount, which is $\frac{1}{9}$ of the whole number of teeth or $\frac{1}{9}$ of a turn, so that the thread tool has only moved $\frac{1}{9}$ of $\frac{1}{4}$ or $\frac{1}{36}$ of an inch.

The gear-feed boxes or quick-change devices for threads are simply combinations of compound gears which can be varied by sliding keys or by other means. These give a variety of gear combinations which can all be traced out in this same way with a little care, always remembering that there is nothing mysterious about any of them.

Catching Threads

The time wasted in letting a lathe run the carriage back, even with a fast backing belt, is not good practice, although less objectionable on short threads than on long. Still on threads as short as 2 to 3 inches a good lathe hand can save a lot of time, and while there is a knack to it, a little practice helps a great deal.

If we stop the lathe at any point on a threading job, unlock the carriage and move it along just an inch, it will always catch the thread unless it is faster than one to the inch.

If both the pitch of the thread being cut and the lead screw are even numbers, such as 2, 4, or 6, the thread can be caught at any half inch. But the way to save time is to be able to do this without stopping the lathe or doing any measuring except with your eye. Stopping the lathe with the dog in any particular position takes time and is unnecessary if you are careful and get a little practice.

If the lathe is geared even, or the work being threaded is the same pitch as the true pitch of your lead screw, you can throw in the half nut anywhere and have no fear about catching the thread. This is also true if the thread being cut is any multiple of lead screw, but we must find a way to catch any threads that come along and do it so as to save time and yet not spoil work.

When starting the thread it is a good plan to note the position of the lathe dog, turning it until its tail is at the top. Then bring the thread-cutting tool as near the starting point as you think is safe and lock in the half nut. Now move up the tailstock body till it touches the carriage bridge, or put something on the bed that will give a positive indication, and we are ready.

It is very evident that whenever the lathe dog has its tail at the top and the carriage is back in the same position, the half nut will drop into place and catch the thread every time. So all we have to do after a cut is to open the half nut, run the carriage back to the stop or mark as the case may be, wait until the dog's tail gets to the top and close the half nut. It doesn't require that you wait till it is exactly at the top, as the eye can tell near enough to enable the half nut to catch in at the right place, for the lead screw threads are so coarse that if the nut goes in when the dog is near the top it is all right.

Another way, although not quite as rapid as the one just described, is to run the carriage back until the thread tool is near the beginning of the cut, and run it in until it nearly touches the work. Then with the half nut in hand ready to close, wait till the thread tool points to its proper position in a thread, throw in the nut and the carriage will start in its proper place. Reverse the lathe, run it back a turn or two till the tool clears the work, and then start the cut. But the other way is quicker and better in most cases.

Rapid Work

Thread cutting used to be considered a ticklish job and light cuts were always in order, but the newer methods have reached this as well as turning, and quick thread cutting is now the order of the day.

One of the best examples of this is a lot of special studs used in a gas engine. The threads are 7 pitch, 1⅛ inches in diameter, and the thread is 2½ inches long. These are chased in an engine lathe with a single-point tool in from 4 to 5 cuts by a boy who has learned to catch threads almost with his eyes shut, at the rate of 13 studs an hour, not for one hour but for ten hours, 130 studs a day. And the threads are as smooth as necessary, better than many chased at a slower rate, and accurate as to fitting the nuts.

There is a little trick about this that is worth knowing. The first chip is a reasonably heavy one, perhaps half the depth of the thread because the area cut out is comparatively small. Then on the next cut he deliberately misses the first thread and cuts another the same depth as the first, as shown in Fig. 92. The next time he splits the two cuts and takes out another big chip from the center, getting the thread down nearly to size and not having any of the heavy crowding chips from both sides as when you cut straight down from the center every time. The fourth or fifth chip finishes the stud to size and leaves a good thread.

Some makes of lathes have an indicator for this purpose; although the idea is old, it is not common, probably because there are comparatively few long screws to be cut and the necessity for such a device is not often felt. There are other plans, but those described are all that will be needed.

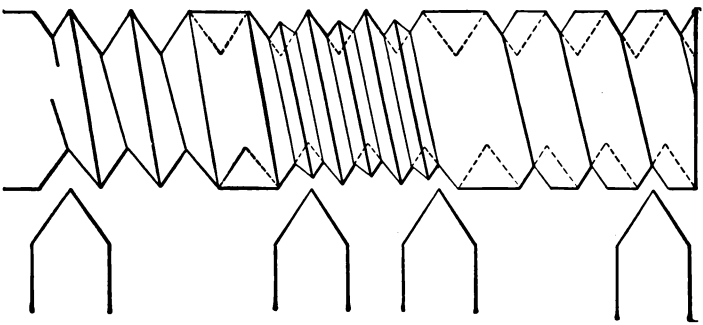


FIG. 92. Splitting Threads for Rapid Work

Measuring the Thread

The old notion of measuring the thread with a pair of broad-nosed calipers to go over two or three threads has given way to the better plan of measuring half-way down the sides of the thread. Measuring the top gives the outside diameter, but the threads fit on the sides and not the top or bottom, and if they should fit at these points when new they would not after being put together a few times.

The bottom of the thread depends entirely on the point of the thread tool, while the top depends on whether the cut is brought up so as to be just sharp in a V-thread or to just the right width of flat for the United States Standard. But measuring the sides of the threads tells the story.

The most refined and accurate thread measurements are made by the use of wires laid in the threads, then the outside of the wires measured by micrometer or else a special thread micrometer, but for lathe work grind the points on a pair of calipers so as not to touch the bottom of the thread, and then measure a bolt that fits the nut properly. Using this as the standard, you may be surprised to know how accurately you can cut threads by measuring in this way.

Of course fixed calipers or snap gages can be made in this way if the number of pieces to be threaded warrant it, but the calipers made and used as shown will give good results.

Cutting the Thread

Having found the proper gears for the thread wanted, either by figuring out the change gears or by moving the handle in the gear box if the lathe has one, the next step is to decide on the tools to use. In some cases men seem to forget all about the question of clean cutting when it comes to thread tools, and grind up a tool to fit the thread gage without regard to top rake or easy cutting. In cutting brass the top rake can be omitted, but for iron and steel it will help in roughing out a job quickly and getting ready for the finishing cuts in short order.

Pitch and Lead

Before taking steps to cut any threads it is best to fix in the mind what the pitch of a screw is. As usually measured, we say 10 pitch, meaning 10 threads to the inch, or 20 or 9 or any other number. On the other hand we sometimes run across a drawing marked ¾ pitch, which should mean three-quarters of a turn to the inch or one turn in 1⅓ inches. If it says ¾ inch pitch it means ¾ inch from one thread to the next.

The next point to watch is "pitch" and "lead." The pitch of a thread is the distance from the center of one tooth to the center of the next. The lead of a screw is the distance a nut will advance in one revolution of the screw. If

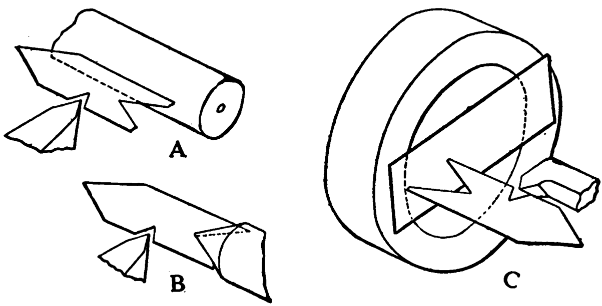


FIG. 93. Setting Thread Tool

it is a single-thread screw, the pitch and lead are always the same; but for double, triple or any multiple thread, the lead is just as many times the pitch as there are multiple threads. A double thread has a lead twice the pitch, a triple screw three times, and so on. The pitch might measure 12 with a screw thread gage, but be a quadruple thread of 3 to the inch. The angle the thread makes with the bar it is cut on tells the story here.

Grind your tool to fit the thread gage and set it square with the work, as shown in Fig. 93. This insures each side of the thread being 30 degrees. The tool can be set on straight work by resting the back side of the gage against the work and using the small V on the side to see that it is right as at *A*, Fig. 93. Some place the large V in the end over the center, as shown in *B*, but the first is usually the better way. For inside thread cutting the tool can be set as in *C* if the work is small, or by reversing the process of *A* if the hole is large enough.

Roughing Out the Thread

The old standby for a roughing thread tool is shown in *A*, Fig. 94. This is practically a diamond lathe tool, with

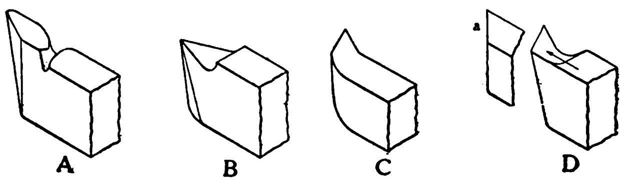


FIG. 94. Thread Tools

the sides 60 degrees, plenty of top rake, and the point rounded so as to stand a heavy chip. For soft steel or wrought iron this will get out most of the metal in a hurry and leave the finishing for a tool like *B*, which, from the amount ground down on top, has seen service.

Some prefer a bent thread tool, and they are necessary at times to get up close to a shoulder. They are usually bent around as at *C* or even more than this, but an easier way is to grind the end of a straight bar as shown at *a*, in tool *D*, which is an easy tool to make and gets up to a shoulder very nicely. When ground with plenty of top rake at the front edge, it looks like *D* and is an excellent tool for what might be called angle cutting. It can also be ground with a rake from the point back toward the body, for regular or front cutting.

Most shops use one of the many tools with removable points, and these have the advantage of being able to reset the tool in the same place after grinding, without wondering if it is just the same as before.

Take Up All Lost Motion

One of the most frequent causes of unsatisfactory thread cutting is to have "slack" in the cross-feed screws or lost motion in the slides of the tool-block or the compound rest. Any of these is apt to make or to allow the tool to dig into the work; there is nothing more exasperating to a good machinist than this, especially if it is near the finishing cut. So it is well to have all the slides gibbed down quite tight to be sure the slack is taken out of the screws, and to have the tool clamped tight and with as little overhang as possible, before starting to cut a thread.

Sometimes, too, a slight difference in the hight of the tool will make it cut much better, and for finishing a thread it is a good plan to have the tool just about at the center of the lathe. For this purpose the rising and falling tool rest on a lathe carriage is very handy, as the tool can be raised or lowered after it is set in position. With this rest the roughing cuts can be taken with the tool above the center and the tool lowered to the center for the finishing touches.

Chasing threads is a job that takes a fair amount of horse sense, but any boy who is careful can soon get the hang of it and get along without much trouble if he doesn't try to take out too much metal at one cut and will go slowly in the finishing. Another point is to be sure that the centers fit the work and are snug so that the work cannot jump or ride the tool and make it dig into the work. With a tool like *A*, Fig. 94, a large chip can be cut out and finishing left for a few light cuts.

When the thread tool is fed straight into the work, it cuts on both sides of the *V* and acts something as a wedge. It is possible to slope the whole top back, as in *A*, and get a kind of top rake; but to make it cut as nicely as a side tool, it would be necessary to hollow it out in the center something like a wood-worker's gouge or a wood-carver's parting tool.

Angular Feeding of Tool

To get away from this difficulty and to get the advantages of the side-cutting tool, some one thought out what can be called the angular thread-cutting method. It is rather hard to say where this originated; but all indications point to Prof. John E. Sweet as the first to bring it into use, and it adds one more to the practical ideas he has fathered.

The object of this is to get the advantage of a side-cutting tool by feeding the tool into the cut so as to cut only on one side of the thread until the roughing is all done, and then finish with a regulation tool on both sides. It is possible, however, to cut the entire thread from one side if the tool is in good shape and fed in at exactly the right angle, or the last cut can be taken by feeding the cross-slide in straight.

The swivel or compound rest is necessary for this kind of work, and it should be set at 30 degrees, as shown in *A*, Fig. 95. This has the tool shown in *D*, Fig. 93, with the top ground away so as to give a side-tool rake from the cutting edge *a*. The tool is fed up to the work by the regular cross-slide, which should then be locked by tightening the gibs or in any other way, and the tool fed into the work by the swiveled slide. This makes the front or shearing edge *a* do all the cutting, and it will be seen that the other edge of the tool simply follows the angle of the thread. The notches *a*, *b*, *c*, *d* and *e* show different cuts and how the front edge does all the work, but still leaves a perfect thread at the bottom every time.

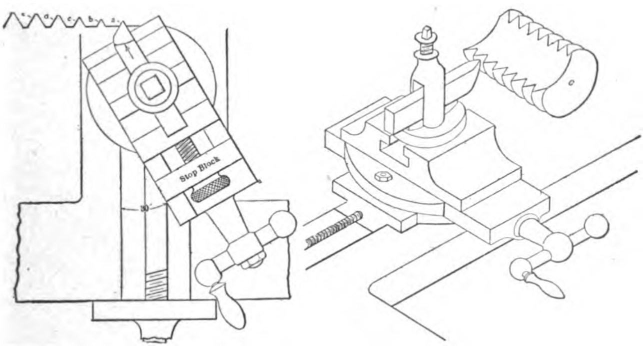


FIG. 95. Using Compound Rest in Thread Cutting

As a matter of fact it is very hard to cut a *V*-thread that is perfectly sharp at the bottom, and the roughing cuts should be taken with the point of the tool either rounded or flattened, saving the tool with a perfectly sharp point for the very last cuts. Even then they should be fed into the work very gingerly, as it is the easiest thing in the world either to break the extreme point or dull it a trifle before the thread is finished. In fact the V-thread has now been discarded by tap and die makers, who now use the United States Standard for all regular work and make the V only as a special.

Another way of using the compound rest in thread cutting is to set it at right angles to the cross-slide, as shown at *B*, and move the thread tool first to one side of the thread and then the other, cutting on only one side at a time until the finishing cut. This method is said to be quite common in English shops.

The United States Standard Thread

The United States Standard or Franklin Institute or Sellers thread is so called because it was designed by William Sellers, recommended by the Franklin Institute, and has been adopted by the United States Government. This thread has the same angles as the V, 60 degrees, but has one-eighth of the depth taken off the top and bottom,

The depth of a V-thread of this pitch is 0.86 inch and of the United States Standard 0.65 inch, while the flat at top and bottom is 0.125 inch. To find the depth for any other thread, divide these figures by the number of threads to the inch. To help in allowing for the thread when boring a die or other piece with internal thread. Table 4 will be found useful. This also gives the width of the flat for the point of the thread tool, but it is fully as easy to measure this with a standard thread gage and there is much less chance of error. Simply grind the tool to fit the gage for whatever thread is to be cut, being sure it is a United States Standard thread gage and not a *V*.

Using the Stop Block

In feeding the tool into the work it is a great help to have a screw in the stop block with divisions on the head like a micrometer. Then you can see just how far the tool is fed in each time. This is not absolutely necessary, as you soon get to know how much of a turn to give the screw each time to allow for feeding it into the work. The stop screw and block are usually made as in *A*, Fig. 95, but for inside work it is necessary to take out the screw, move the block out and put the screw head inside the block so the head will stop against it when drawn back. This can be done away with by having a collar on the screw inside the block with space enough to allow for the deepest thread you are likely to cut. A half-inch allowance will be ample to allow the tool to be withdrawn from the cut when running back, on either outside or inside work.

Double or Triple Thread

Having decided on the double thread to be cut, the first thread is cut to one-half the depth for that pitch and the second thread cut half-way between the grooves of the first thread. This space can be divided by measurement, by turning the work half-way round or by turning the *stud gear* just half-way round. The last is probably the easiest except when an odd-tooth gear, as when the change gears jump by 5, is on the stud. In that case it is probably easier to measure and reset the tool.

If there is much double-thread cutting to be done, the best way is to have an indexing face plate made like Fig. 96.

This shows a face-plate fixture used on various numbers of threads. On an ordinary driving plate is fitted a plate having, as shown, twelve holes enabling one to get two, three, four, or six leads if required. This ring carries the

TABLE 4. — PROPORTIONS OF SCREW THREADS

Number of Threads Per Inch	Lead of Thread	Sharp V-Thread		U. S. Standard Thread		
		Single Depth	Double Depth	Single Depth	Double Depth	Width of Flat
1	1.00000	0.8665	1.7330	0.6495	1.2990	0.1250
1⅛	0.88888	0.7702	1.5404	0.5773	1.1546	0.1111
1¼	0.80000	0.6932	1.3864	0.5196	1.0392	0.1000
1⅜	0.72727	0.6302	1.2603	0.4723	0.9447	0.0909
1½	0.66667	0.5776	1.1553	0.4330	0.8660	0.0833
1⅝	0.61538	0.5332	1.0664	0.3997	0.7994	0.0769
1¾	0.57142	0.4951	0.9902	0.3711	0.7422	0.0714
1⅞	0.53333	0.4621	0.9242	0.3464	0.6928	0.0666
2	0.50000	0.4332	0.8665	0.3247	0.6495	0.0625
2⅛	0.47058	0.4077	0.8155	0.3056	0.6113	0.0588
2¼	0.44444	0.3851	0.7702	0.2888	0.5773	0.0555
2⅜	0.42105	0.3648	0.7296	0.2734	0.5469	0.0526
2½	0.40000	0.3466	0.6932	0.2598	0.5196	0.0500
2⅝	0.38095	0.3301	0.6602	0.2474	0.4948	0.0476
2¾	0.36363	0.3151	0.6302	0.2361	0.4723	0.0454
2⅞	0.34782	0.3014	0.6028	0.2259	0.4518	0.0435
3	0.33333	0.2888	0.5776	0.2165	0.4330	0.0416
3¼	0.30769	0.2666	0.5332	0.1998	0.3997	0.0384
3½	0.28571	0.2475	0.4951	0.1855	0.3711	0.0357
3¾	0.26667	0.2311	0.4621	0.1732	0.3464	0.0333
4	0.25000	0.2166	0.4332	0.1623	0.3247	0.0312
4½	0.22222	0.1925	0.3851	0.1443	0.2886	0.0277
5	0.20000	0.1733	0.3466	0.1299	0.2598	0.0250
5½	0.181818	0.1575	0.3151	0.1180	0.2361	0.0227
6	0.166666	0.1444	0.2888	0.1082	0.2165	0.0208
7	0.142857	.1237	0.2475	0.0927	0.1855	0.0182
8	0.125000	0.1083	0.2166	0.0812	0.1624	0.0156
9	0.111111	0.0962	0.1925	0.0721	0.1443	0.0138
10	0.100000	0.0866	0.1733	0.0649	0.1299	0.0125
11	0.090909	0.0787	0.1575	0.0590	0.1180	0.0113
11½	0.086956	0.0753	0.1507	0.0564	0.1129	0.0108

12	0.083333	0.0722	0.1444	0.0541	0.1082	0.0104
13	0.076923	0.0666	0.1333	0.0499	0.0999	0.0096
14	0.071428	0.0619	0.1237	0.0464	0.0927	0.0089
16	0.062500	0.0541	0.1083	0.0406	0.0812	0.0078
18	0.055555	0.0481	0.0962	0.0361	0.0721	0.0069
20	0.050000	0.0433	0.0866	0.0324	0.0649	0.0063
22	0.045454	0.0394	0.0787	0.0295	0.0590	0.0057
24	0.041666	0.0361	0.0721	0.0271	0.0541	0.0052
26	0.038461	0.0333	0.0666	0.0250	0.0499	0.0048
27	0.037037	0.0321	0.0642	0.0240	0.0481	0.0046
28	0.035714	0.0309	0.0618	0.0232	0.0463	0.0045
30	0.033333	0.0289	0.0577	0.0216	0.0433	0.0043
32	0.031250	0.0271	0.0541	0.0203	0.0405	0.0039
34	0.029411	0.0255	0.0510	0.0191	0.0382	0.0037
36	0.027777	0.0241	0.0481	0.0180	0.0360	0.0035
38	0.026315	0.0228	0.0456	0.0171	0.0342	0.0033
40	0.025000	0.0216	0.0433	0.0162	0.0324	0.0031
42	0.023809	0.0206	0.0412	0.0154	0.0309	0.0029
44	0.022727	0.0196	0.0393	0.0147	0.0295	0.0028
46	0.021739	0.0188	0.0376	0.0141	0.0282	0.0027
48	0.020833	0.0180	0.0360	0.0135	0.0270	0.0026
50	0.020000	0.0173	0.0346	0.0129	0.0259	0.0025
52	0.019231	0.0166	0.0333	0.0125	0.0249	0.0024
54	0.018518	0.0160	0.0320	0.0120	0.0240	0.0023
56	0.017857	0.0154	0.0309	0.0116	0.0232	0.0022
58	0.017241	0.0149	0.0298	0.0112	0.0224	0.0021
60	0.016666	0.0144	0.0288	0.0108	0.0216	0.0020
64	0.015625	0.0135	0.0271	0.0101	0.0202	0.0019
68	0.014705	0.0127	0.0254	0.0095	0.0191	0.0018
72	0.013888	0.0120	0.0241	0.0090	0.0180	0.00172
76	0.013157	0.0114	0.0228	0.0085	0.0170	0.00184
80	0.012500	0.0108	0.0216	0.0081	0.0162	0.00156
84	0.011904	0.0103	0.0206	0.0077	0.0154	0.00148
88	0.011363	0.0098	0.0196	0.0073	0.0147	0.00142
92	0.010869	0.0094	0.0188	0.0070	0.0141	0.00135
96	0.010416	0.0090	0.0180	0.0067	0.0135	0.00130
100	0.010000	0.0086	0.0173	0.0065	0.0129	0.00125

driving stud, and is clamped at the back of the plate by two bolts as an extra safeguard. All that is necessary in operation is to slack off the bolts, withdraw the index pin, move the plate the number of holes required, and re-tighten the bolts. It is used on different lathes, as occasion requires, by making the driving plates alike and drilling a hole for the index pin. It is found that the index pin works best when made taper, and a light tap is sufficient to loosen or fix it.

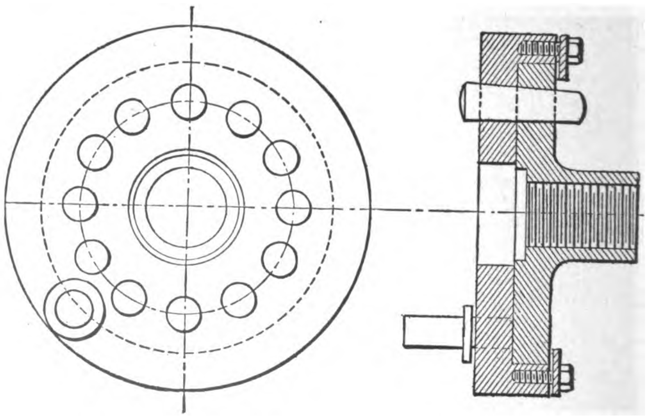


FIG. 96. Indexing Face Plate for Multiple Thread

With this, you cut one thread to its full depth, or a partial depth if you prefer to leave a finishing cut to be taken after they are all roughed out, then pull the index pin and turn to the right point. There are fewer chances of springing a bar if the threads are partially cut in each place and then light finishing cuts are taken in all the threads.

Square Threads

Square threads are sometimes puzzling in two ways: in grinding the tools and in cutting the thread. The width of the tool is half the pitch because the land and the space must both be the same, although some make the space a little wider to allow for play and for any variation in the pitch. In *A*, Fig. 97, a pitch of 1 to the inch is shown.

TABLE 5. — SQUARE-THREAD TOOLS — WIDTH AND DEPTH OF THREADS ARE THE SAME

WIDTH OF THREAD TOOL			
Pitch or Threads per Inch	Single Thread	Double Thread	Triple Thread
1	0.5	0.25	0.166
2	0.25	0.125	0.083
3	0.166	0.083	0.055
4	0.125	0.062	0.042
5	0.10	0.05	0.033
6	0.083	0.042	0.028
7	0.071	0.035	0.023
8	0.062	0.031	0.021

This means either the distance from center to center of threads or from the face of one thread to the corresponding face in the next thread.

Dimensions of Screw Threads

While the table almost explains itself, an illustration may help a little. Let it be required to make a special nut, 1 inch diameter, 12 threads per inch. From Table 4 we find the double depth of thread to be 0.1082 inch, which, subtracted from 1 inch, leaves 0.8918 inch, the required diameter of bore. This for a U. S. S. thread.

Or, if we wish to know how far a nut will advance for one turn of a screw having 1⅝ threads per inch, we find this from the column marked lead, opposite the number of threads per inch, to be 0.61538.

If it is a double or triple thread, we must be careful and not confuse pitch and lead. The depth of the square

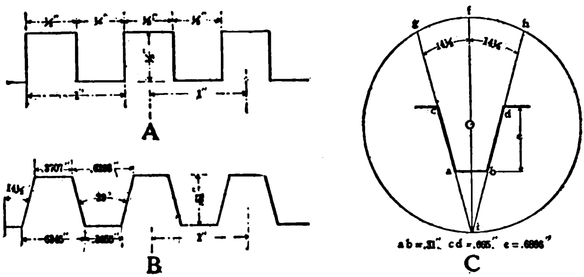


FIG. 97. Square, Acme and Worm Thread

thread is usually the same as the width of the land or the space, although here again there is a difference of opinion, some allowing clearance at the bottom. The square thread is rather difficult to cut on account of giving clearance on the sides to avoid rubbing from the angularity of the thread.

In cutting double threads of square section, the same precautions must be observed as with V or other threads. Table 5 gives the width of square-thread tools for use in cutting single, double, and triple threads.

Top rake on square-thread tool gives a good cutting edge, and the chips can often be rolled out in a hurry if the stock is good and clear.

Acme Threads

Square threads were not always easy to cut, and so it often happened that feed screws, lead screws, etc., were

TABLE 6. — PROPORTIONS OF ACME THREADS

No. Threads per In. Linear	Depth of Thread	Width at Top of Thread	Space at Bottom of Thread	Space at Top of Thread	Thickness at Root of Thread
1	0.5100	0.3707	0.3655	0.6293	0.6345
2	0.2600	0.1853	0.1801	0.3147	0.3199
3	0.1767	0.1235	0.1183	0.2098	0.2150
4	0.1350	0.0927	0.0875	0.1573	0.1625
5	0.1100	0.0741	0.0689	0.1259	0.1311
6	0.0933	0.0618	0.0566	0.1049	0.1101
7	0.0814	0.0529	0.0478	0.0899	0.0951
8	0.0725	0.0463	0.0411	0.0787	0.0839
9	0.0655	0.0413	0.0361	0.0699	0.0751
10	0.0600	0.0371	0.0319	0.0629	0.0681

made flat top and bottom, but with slanting sides of any angle that pleased the eye of the man who ground the thread tool. As these were neither square nor V, they soon had a name of their own and were called bastard. In some parts of the country this term is applied only to odd pitches, but any old hand will recall bastard threads of a great variety of shapes and sizes.

Of course no two of these were alike, and the natural course of events brought about a standard which is now known as the Acme thread. The proportions for a pitch of one to the inch are shown in *B*, and Table 6 gives full details for other sizes. Thread gages can be had for the Acme thread if desired.

TALBE 7. — BROWN & SHARPE WORM THREAD PROPORTIONS

Worm Threads						
Pitch in Threads per Inch	Lead per Revolution	Depth of Thread	Width of Tool Point or Bottom of Thread	Width of Top of Thread	Width of Space at Top of Thread	Width of Root of Thread
1	1	0.6866	0.31	0.335	0.665	0.69
2	0.5	0.3433	0.155	0.167	0.332	0.345
3	0.333	0.2288	0.103	0.111	0.222	0.23
4	0.25	0.1716	0.077	0.084	0.166	0.17
5	0.20	0.1373	0.06	0.067	0.133	0.14
6	0.166	0.1144	0.05	0.056	0.111	0.115
7	0.141	0.0981	0.044	0.048	0.095	0.098
8	0.125	0.0858	0.039	0.042	0.085	0.086

Worm Threads

The Acme thread is so near the worm thread that care must be taken to avoid using one for the other or getting the proportions mixed. The angle is the same, 29 degrees; but the depth is greater, as can be seen in *C*. This also shows an easy way to lay out the angle of 29 degrees if you want to make a gage for yourself.

Take a piece of sheet iron, draw a circle say 2 inches in diameter; draw a line through the center, as *f i*. Take one-quarter the diameter, or ½ inch, in the dividers and mark off *g* and *h* from *f*. Connect *g i*, and *h i* and the enclosed angle is 29 degrees.

The point of the tool *a b* is 0.31 × the pitch: the space *c d* is 0.665 × the pitch; and the hight *e* is 0.6866 × the pitch, according to the Brown & Sharpe standard, or practically one-third deeper than the Acme thread. The details are given in Table 7.

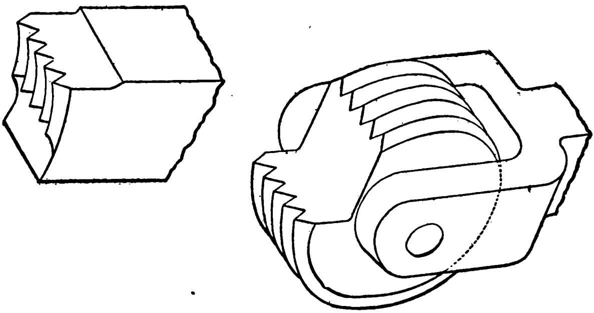


FIG. 98. Thread Chasers

Chasers and Special Tools

A chaser, as known to the engine-lathe man, is a thread tool with several points, as in Fig. 98; when used, it is generally kept for finishing threads and not for roughing. The old chaser was like the new at the left, being a straight tool with threads cut in the end like a die chaser, this being cut with a hob tap. The other is a later development and though usually confined to screw machines is sometimes used in the engine lathe. It is made by cutting a thread of the desired pitch on a steel roll, milling out a flute and clamping the roll so as to bring the cutting edge at the right position on the work. It can be sharpened by grinding the face and will last until the whole roll has been ground away. Chasers are well thought of by many for finishing long threads, and you may find them useful at times. One holder will handle all the rolls you make for different pitches of threads.

Another form of threading tool might be called a turret thread cutter, in which the cutter resembles a milling cutter with different lengths to the teeth; it is known as the Rivett-Dock threading tool. The tool is brought up to the work and the shortest tooth takes a cut. Then the next longer tooth takes the next cut, and so on until the last tooth finishes the thread. There is no feeding of the tool into the work, as the cutter takes care of that and each tooth cuts only its own portion of the thread.

Thread milling has been done for more years than most people dream of, the first we know of being done in tap-making by old Ben Bee on Cape Cod in 1866. Then Horton began milling the threads of his chuck screws in 1873, and the thread miller is now a regular institution.

There is a great difference of opinion as to the best lubricant for use in chasing threads. Nearly every machinist has his pet screw-cutting oil, which includes fish oils, heavy crude oils, emulsions of soap, oil, and water, soda water with variations, etc., but perhaps more swear by pure lard oil than anything else. It is expensive, but there is no economy in saving a little money and turning out rough threads on work where it should be nice. The time saved by having everything go right and not having to stop and try to coax the tool to behave itself will pay for more oil than you will use on a dozen threads. There may be other oils just as good for most work; but if in doubt, try lard oil and ten chances to one you'll get along all right.

An Attachment for Turning and Threading the Ends of Long Well Tubes in the Lathe

Lathes have been built that would do this work without the necessity of cutting off the end of the piece and welding it in again. They would be built with a very large hollow spindle. To do the largest sizes of these stems requires that the hole through the spindle be something like 7 inches diameter, and anything under 6 inches would not be of much use. There are at least two objections to such a lathe. In the first place it has to be built special, which makes it very much more expensive than a regular one of the same swing, and in the second place it is an inconvenient tool for ordinary work. The re-threading of the stem takes but a small part of the time of the lathe and very few shops can afford to have them idle during the balance of the time.

At one shop where they have all the facilities for welding they have still found it cheaper to avoid doing it and have made a very cheap attachment that allows them to do the re-threading on an ordinary lathe. The idea might be used on other long work.

This attachment consists of a sleeve that has a hole through it large enough to take the largest stem. It has a bearing at each end. These bearings are made to fasten to the ways in the same way as a steady rest and bring the sleeve so that its center coincides with the lathe centers.

On the end of the sleeve nearest to the headstock is fitted a chuck and a gear is mounted on the outside. A small shaft with a pinion on each end of it furnishes the drive. One of these pinions meshes into the large cone gear of the lathe and the other one with the gear on the sleeve. There is a bearing for this shaft as part of the bearing for the sleeve and a bearing made to clamp onto the headstock holds the other end. An adjustable roller support holds the outer end of the stem being operated on.

Fig. 99 shows how this would look from above the lathe. The castings *GG* carry the hollow sleeve *SS* and forms the

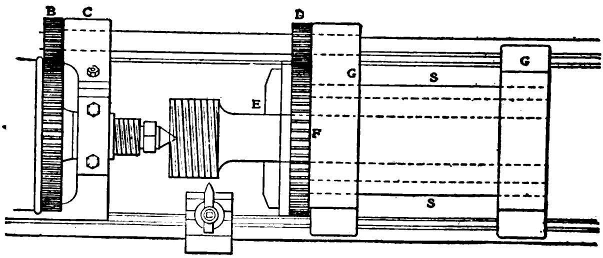


FIG. 99. Threading Long Rods

bearing for one end of the driving shaft as shown. The chuck is at *E*, the geared portion *F* meshes with pinion *D*, which in turn is driven by pinion *B* from head gear. Casting *C* bolts to headstock.

In doing a job the lathe is worked in the usual way except that the work is at the right of the workman instead of at the left. I can imagine that an old latheman would have some peculiar sensations the first few jobs he did with this device.

That it is thoroughly practical is shown by the statement of the owner that it could be put on and a pin re-threaded in an hour and a quarter. As the usual charge there is six dollars for cutting off a pin, re-threading it, and welding it back on, it can be seen that it would not take long for it to pay for itself. We understand that the cost of making it was less than one hundred dollars.

The pins threaded in this way are sure to be in line with the stem and there is no fear of a bad weld.

It will be understood that the tailstock is removed to allow for this attachment being used, and that the carriage is between it and the headstock. Where large pipe has to be threaded and a pipe machine large enough to handle it is not at hand, it is much more convenient to use than it is to catch the end of the pipe in the chuck while the body of it runs in the steady rest. Besides saving something in the length of the pipe it avoids the annoyance of having the pipe crawl endwise, which it is very liable to do if caught in the chuck jaws. Unless a piece is fitted into the end of a pipe that is caught by the end in a chuck it is very liable to be crushed out of shape; and unless a considerable piece is cut off, the thread will not be round when finished. In using this attachment the pipe is caught so far from the end that it is much more rigid than when caught at the end, and it is also farther removed from the end so that there is not much danger of the end being affected.

It is a job-shop rig and there is no doubt that the job-shop man will see a number of applications on jobs where long pieces need to have work done on the end.

Two Steep-pitch Thread Jobs

Figs. 100 to 102 shows details of a job of thread cutting

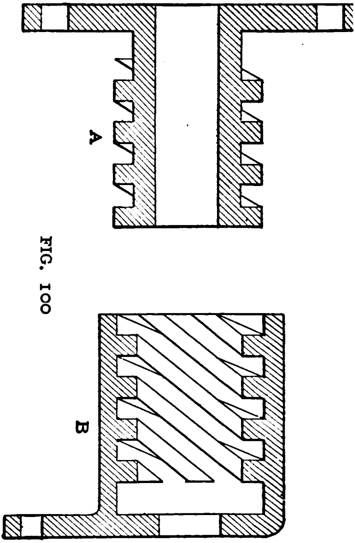


FIG. 100

The Work to be Threaded.

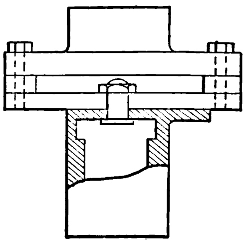


FIG. 101

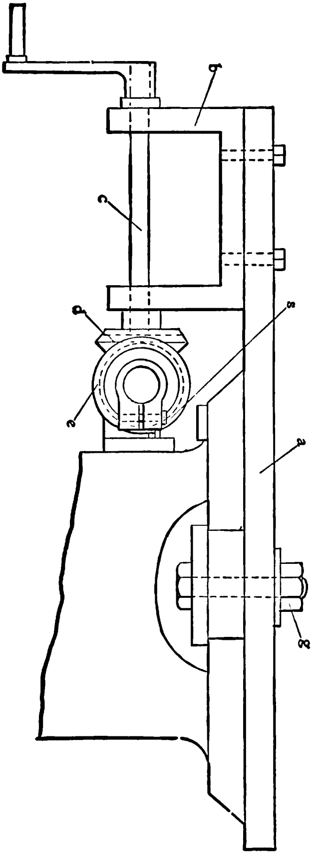


FIG. 102. Driving the Lathe from the Lead Screw

and illustrates the manner in which it was done on a very old English lathe, the screw being two threads per inch.

In Fig. 100, *A* and *B* represent the screw and nut respectively. The thread to be cut was a quadruple one, 2-inch pitch, right hand. After considering the large pitch it was thought preferable to drive from the lead screw. As shown in the sketch, both screw and nut had shoulders standing in the way of the threading tool when it is running out of the cut; so the power drive was abandoned in favor of the handle, which when all the castings were ready for threading were fixed to the end of the lead screw.

After doing a couple of screws it was found awkward to manipulate the driving and cross-feed handles, on account of their being too far apart. To bring the driving mechanism nearer to the carriage, another rig was necessary. The scrap heap supplied the few necessaries consisting of a bar of flat steel *A*, Fig. 102, to which was secured bracket *B* carrying shaft *c* and bevel gear *d*. Bevel gear *e* was slid on the lead screw. Bar *A*, as mounted, was then placed across the lathe bed as close to the carriage as the travel of the latter would allow, the two bevel gears put in mesh and nut *g* and screw *s* tightened. To effect the division into four, the split nut was opened, the carriage moved along over one thread of the lead screw and closed the nut down. Fig. 101 shows how the nut was held for the screwing operation.

The job required a little elbow grease, but the time was very satisfactory.

Cutting Steep Pitch Worms on an Engine Lathe

Having occasion for cutting some steep pitch worms of various leads and having a heavy 18-inch lathe idle, it was decided to equip this lathe for the work. The most feasible scheme seemed to be to increase the speed of the change gears by driving them directly from the small gear on the cone pulley of the lathe in place of the usual small gear

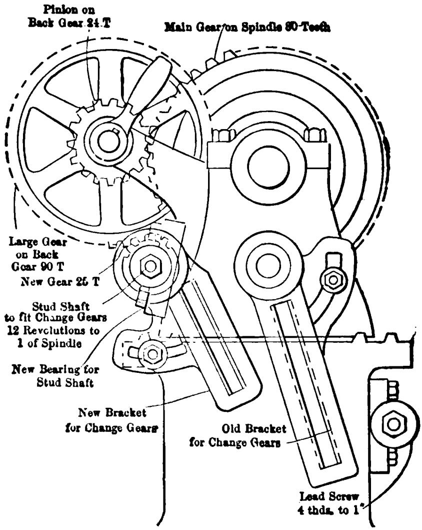


FIG. 103. Cutting Steep Pitch Worms

on the spindle, thus giving ample driving power and utilizing the regular change gears. This was accomplished by the device shown in Fig. 103.

It consists essentially of a new stud shaft mounted in a bearing secured to the rear of the headstock in such a manner that a new gear, made with the proper number of teeth and secured to the inner end of the stud shaft, would, when the back gear was thrown in, mesh properly with the large gear on it. The other end of the stud shaft was made to fit the regular change gears of the lathe. The main gear on the latter spindle was found to have 80 teeth, while the pinion on the back gear had 24 teeth and the large gear on the back gear 90 teeth, so that the new stud-shaft gear was made with 25 teeth, causing it to make 12 revolutions to one of the spindle.

It was necessary to use three intermediate gears to reach over to the new stud shaft when a right-hand thread was to be cut, or two for a left-hand. The stud shaft was made long enough to use a compound gear if necessary.

In laying out the work it was found advisable to add an additional bracket to accommodate the extra intermediate gears. On the whole the device proved very satisfactory and accomplished all that was hoped for it.

Making Accurate Screws in the Lathe

This plan for producing practically perfect screws such as are necessary in good micrometers is due to J. P. Lavigne.

The simple ingenuity of the scheme will be of interest to all mechanics, for it is doubtful if many would know how to proceed to cut a precision screw without a lead screw, and in this case a well-known maker of precision screws had failed to furnish a screw which was considered satisfactory.

An ordinary make of 14-inch lathe was secured, and stripped of all its feed mechanism, both turning and screw-cutting, and a substantial cast-iron frame *a* bolted to the back end of the headstock, as shown in the accompanying freehand sketches. The back or inside of frame *a*, Fig. 104,

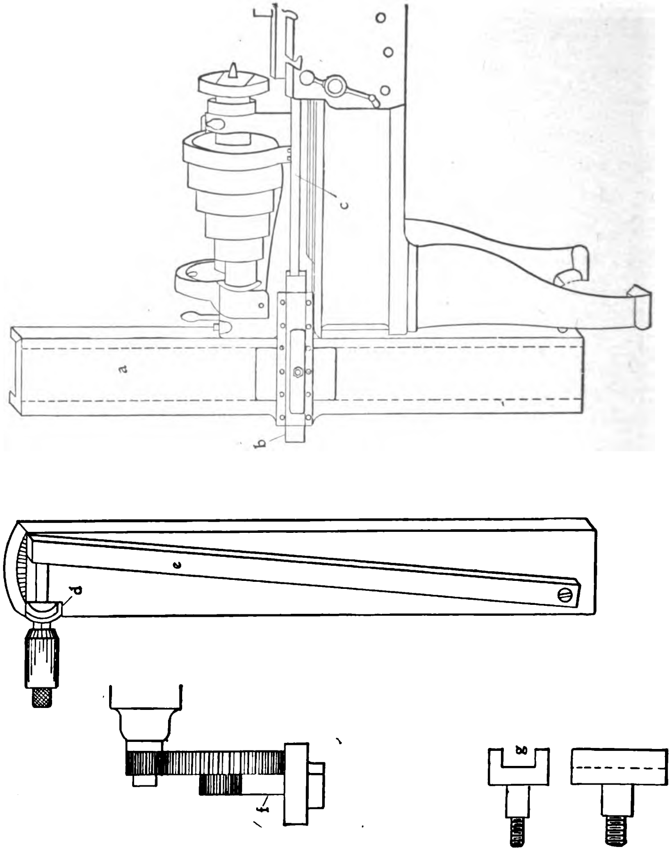


FIG. 104. Cutting Accurate Screws

was planed straight through lengthwise, and a slide *d* carefully fitted; on the face of this slide was fitted the lead bar which was attached at one end with a dowel bolt, and made adjustable at the other with an adjusting screw and clamping bolt, and set at an angle as shown.

On the front of frame *a*, and at right angles to its axis, was planed a seat for the slide *b*, which was carefully fitted in place, secured with a plate gib, and attached to the carriage by the rod *c*. Attached to the slide *b* by a dowel stud and nut washer was the yoke shown in detail at *g*, the fork of which fitted accurately on the lead bar *e*.

The lead bar was made as accurately as possible, three being made, and all scraped and lapped, until any two would show straight when placed together, which proved them all to be straight.

Firmly attached to the back of the slide *d* was the rack *f*, and connecting this with the lathe spindle was a train of gears as shown in detail sketch. These gears and rack were made by a firm of gear specialists, the specification calling for a tooth which represented complete turns of the worm in the dividing head, which assured an accurate gear, the same applying to the rack.

Now it is clear that with this rig in position when the spindle was run forward or back the slide *d* moved up or down, the lead bar *e* moved the slide *b* laterally in or out, and the latter the lathe carriage, so that the relation of the spindle and carriage was the same as when cutting threads with a lead screw; and it is also obvious that if the lead bar was adjusted, to correct a screw say one thousandth in its total length, this thousandth would be accurately subdivided into as many parts as there were threads on the screw.

Mr. Lavigne states that he submitted one of the screws cut with this rig to a firm which has a world-wide reputation for accurate screws and screw gages, and their expert pronounced it a perfect screw within his ability to measure it.

It may never be necessary for this to be done in any ordinary shop, but it is useful as showing that a thread can be cut without a lead screw.

Allowing for Contraction of Threads in Hardening

Another refinement in screw cutting is shown in this method of allowing for contraction in hardening.

It is well known that a tap shrinks when being hardened so that the pitch of the thread will not be the same as when it was cut. To correct the error due to shrinkage it is customary when making long taps to increase the pitch by allowing about 0.001 inch per inch of length of the tap. Thus, if it is desired that the finished tap have an accurate pitch of 12 threads per inch, it would be necessary to cut 12 threads on a length of 1.001 inch. The gear train for cutting this thread can be figured by the use of the general rule for lathe gearing, that is, divide the number of threads per inch on the lead screw by those on the work, and select gears having the ratio of this fraction.

The number of threads per inch to be cut on a tap of 12 pitch is $\frac{12}{1.001}$, which equals $\frac{12,000}{1,001}$. When the denominator of the fraction is reduced to a whole number the result is obtained by multiplying both terms of the fraction by 1000. According to the rule the ratio of the gears where the lead screw has five threads per inch is

$$5 \div \frac{12,000}{1,001} = \frac{5005}{12,000}.$$

This ratio can be separated into two factors, thus,

$$\frac{5005}{12{,}000} = \frac{65 \times 77}{120 \times 100}.$$

The separation of the ratio into two factors must be done by inspection, or by factoring, and the process is easy, or

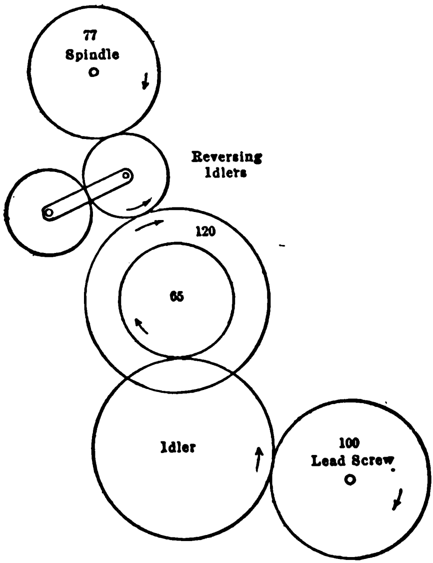


FIG. 105. Gears to Correct for Hardening

difficult, depending upon the knowledge of arithmetic possessed by the one making the calculations.

The result shows that if two pairs of gears are connected up as in Fig. 105, so that the ones having 65 and 77 teeth will be the drivers and the ones having 120 and 100 teeth will be the followers, the combination will cut the thread on the tap and allow 0.001 inch per inch shrinkage. Another combination of gears that will give the required ratio is $\frac{91}{100}$ and $\frac{55}{100}$. The combination to select in any case will depend upon the gears at hand and the cost of the ones that must be made or purchased. The method of arranging the gears on the lathe would be like that shown in the illustration.

A Tool for Threading Slender Work

This tool was used in a very difficult job of threading and may help out in a tight place. The thread to be cut was 14 inches long, 5 per inch and left-handed. This was

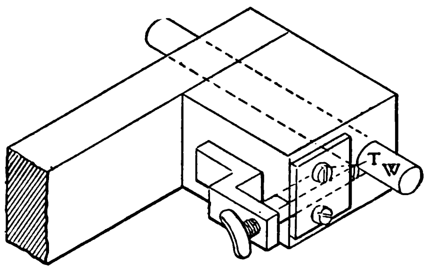


FIG. 106. Thread Tool for Slender Work

for an arbor for winding springs of No. 20 spring brass wire.

The outside diameter of arbor was 0.345, the bottom of thread 0.310 inch, and it had to be perfectly straight. The whole length of arbor was 18 inches. So a guide was made out of 2-inch square steel with a hole drilled through it the size of arbor ends, fastened on a strip to fit holder for tool post, planed out a place for a thread tool worked in and out by a screw, as shown in Fig. 106. The rest was quite easy, as this held the work straight while being threaded.

Testing the Lead Screw of a Lathe

The lead screw is very seldom considered in buying a lathe unless it is bought especially for screw-cutting purposes. This screw can be easily tested on its own lathe to see what results it will produce. Having a large face plate on the lathe, we can take a surface gage, set it on a parallel across the ways and scribe a line about to the center. Now the carriage should be run back as far as possible, and the half-nuts engaged, the gearing being set for any pitch desirable. Suppose we set the gears for 4 pitch, then one revolution of the spindle will make the carriage travel ¼ inch. On the ways behind the carriage we will clamp a small block of cast iron *A*, in which is inserted a micrometer barrel, as shown in the sketch. On the carriage is a block *B* with a pin in the end the same diameter as the micrometer spindle. Also we will use a 6-inch distance rod. This is shown in Fig. 107.

To begin with, turn the face plate *D* one revolution until the line is even with the needle of the surface gage *C*. Now set the micrometer barrel at zero against the carriage. It is probably best to test the screw every 6 inches, as this will be near enough for ordinary screw cutting. Turn the face plate 24 revolutions, try the 6-inch distance rod and see whether the distance is short, long, or correct. Move the micrometer-barrel block up to the carriage and again set at zero; turn the lathe 24 revolutions more and test again. This will show the exact operation of the lathe regardless of bad gearing, defective lead screw, incompetent fitting of keys, etc.

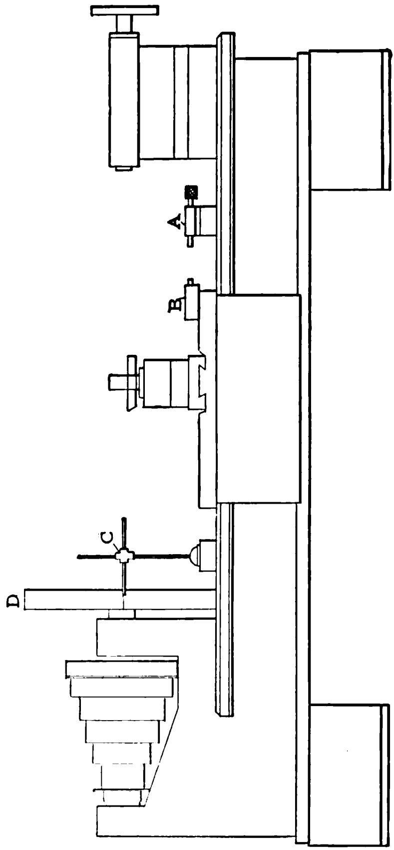


FIG. 107. Testing the Lead Screw of a Lathe

CHAPTER XI

TEST INDICATORS AND THEIR USE

THERE are many kinds of test indicators for lathe work, some of which are quite elaborate and some quite simple. the indicator shown in Fig. 108 is extremely simple and yet will show any variation in the truth of the work.

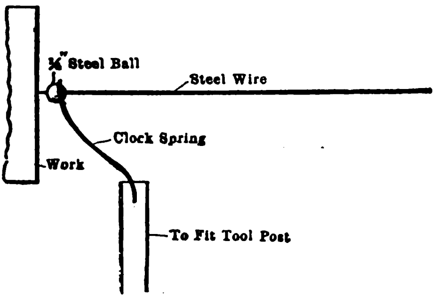


FIG. 108. A Simple Indicator

The only material required is a ¼-inch steel ball, a piece of wire 5 inches long, a piece of clock spring and a piece of bar steel to put in the tool-post. Anneal the steel ball and drill a hole through the center, a drive fit for the wire; then drill a $\frac{3}{16}$-inch hole in the end of the clock spring; slot one end of the steel bar and solder the end of the spring in the slot and the indicator is ready for use.

Use of the Lathe Indicator

Perhaps the simplest example of using an indicator in a lathe is for locating a piece of work on the face plate or in a chuck. For ordinary work, where the limit of accuracy may not be closer than a thirty-second of an inch, the indicator is hardly necessary as you can locate the prick-punch mark or the hole, either by the eye or with the point of a lathe tool. But for really nice work the indicator is a fine tool, even in its cruder forms as shown in Fig. 109, which is one of many types in general use.

This is simply a piece of steel, *A*, pointed at both ends and having the steel wire *B* instead in a crosswise hole, with the long end bent down so as to be in line with the centers of *A*. It will be readily seen that as the arm *B* is much longer than the piece *A* the end of the pointer will multiply the distance the point *C* is out of center when the work is revolved.

When the point *C* is exactly central, the point *E* will remain stationary; but if the point *C* is out of center one-hundredth of an inch the point *E* will travel in a circle having a radius as many times this as the distance from *D* to *E* is longer than from *C* to *D*.

To use this, a punch mark is made on the side of a lathe-tool shank and this fastened in the tool-post so that the punch mark comes as near the center of the lathe as possible. Then the point *D* is placed in the punch mark, the point *C* on the work and the work revolved, usually by hand. Watching the point *E*, tells the story whether it is sighted by the dead-lathe center or not. When the end of the pointer stands perfectly still while the work is being turned,

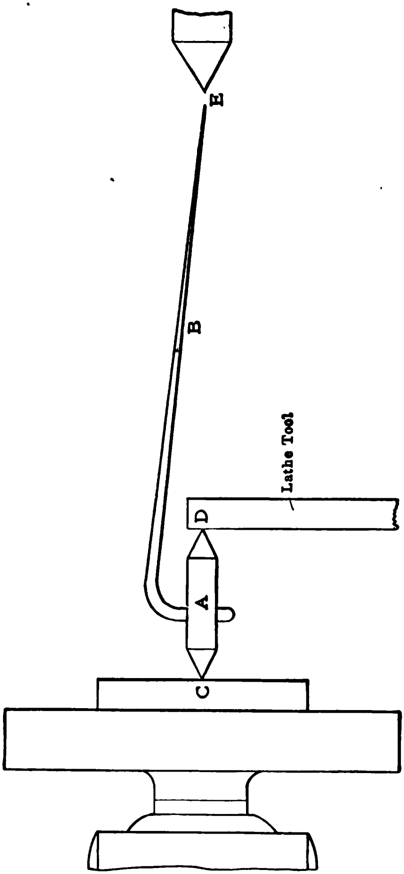


FIG. 109. Another Simple Lathe Indicator

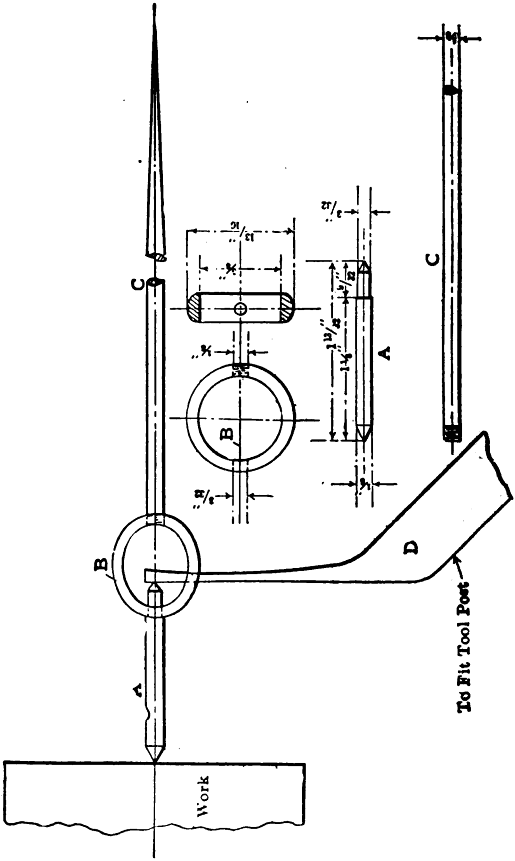


FIG. 110. A more Complete Indicator

the center of the work is located at the exact center of the lathe.

Another kind is shown at Fig. 110 which is also easily made. It consists of piece *A* which is stubs' steel, hardened and made a tight fit in steel ring *B*, and piece *C* which is threaded on one end to fit *B*. *A* and *B* are dimensioned clearly in the sketch. Pointer *C* can be made any length desired, the one shown being 12 inches long. The sketch shows the tool in use, the large end of *A* being held in posi-

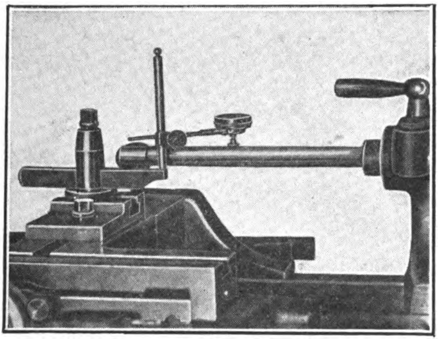


FIG. 111. Testing Alinement of Tail Spindle

tion by shank *D* which has a flexible end hardened, spring-tempered, and centered to receive the small end of *A*. One side of ring *B* rests on *D* to keep it from turning.

More Sensitive Indicators

The refinements of modern tool making have brought out many very sensitive indicators which show variations of a thousandth of an inch or less. Some of these are made with multiplying levers which increase the movement a

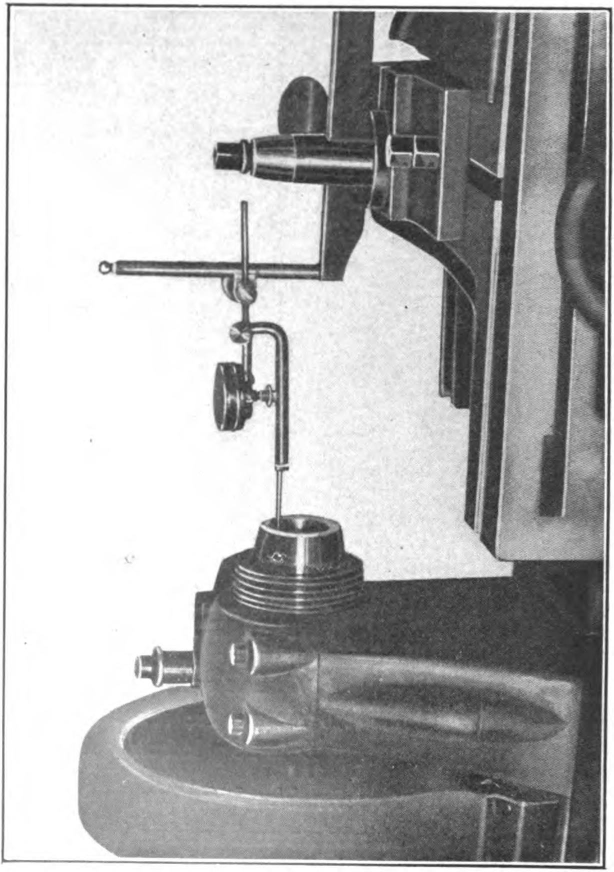


FIG. 112. Testing the Nose of the Spindle

hundred times. Others add dials to them and read the variations directly in that way.

Three applications of such an indicator are shown in Figs. 111 to 113 which are made from photographs furnished by the L. S. Starrett Company.

Fig. 111 shows the instrument fastened in the tool-post to test the accuracy of the tail spindle of a lathe by means

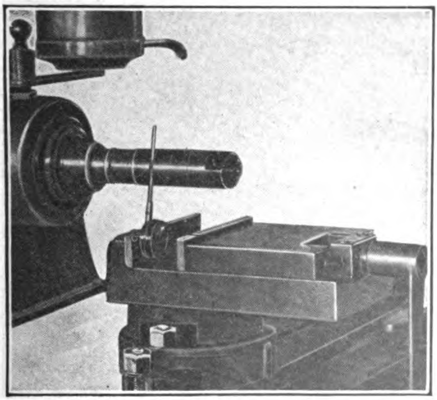


FIG. 113. Squaring Work with the Spindle

of the test bar shown. This bar is carefully turned up to fit the taper of the tail spindle and then the whole bar turned true with the taper. Putting this in the tail spindle and testing it at different points shows how the tail spindle lines up with the ways of the lathe.

Fig. 112 shows the testing of the truth of a lathe-spindle nose in much the same way, except that a multiplying arm has been put on in addition to the indicator itself. This also shows a form of spindle nose that is better for getting chucks and face plates on true than the usual design. Here the thread draws it on the taper in front; in some, the taper is at the back and the threaded portion is smaller in diameter than the taper.

Fig. 113 shows the indicator in use on a milling machine,

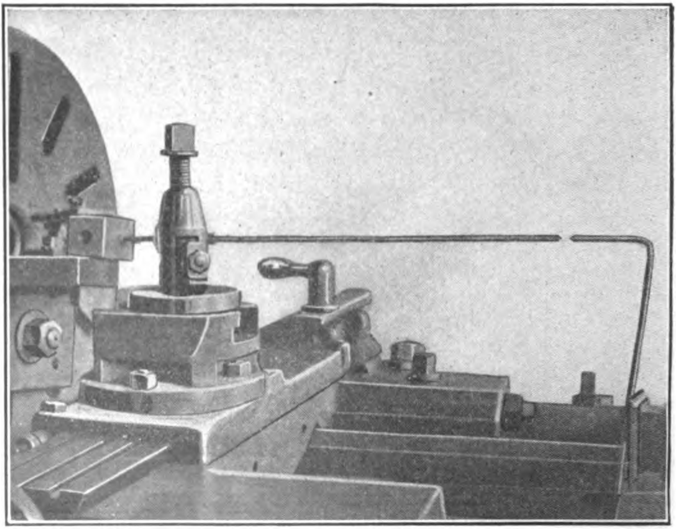


FIG. 114. A

although a similar use on a lathe would be to square a bar or casting across the lathe or to test whether the cross-slide was at exact right angles to the lathe spindle. If it is not it will not face exactly square. The piece to be tested, the vise jaw in this case, is adjusted till the indicator points to zero when set to one end of the jaw. Then the spindle is turned until the indicator comes to the other end of the jaw, and the difference, if any, will be noted on the dial. By carefully adjusting the vise or whatever the work may be, until the reading is the same at both ends, we know the work is square with the spindle.

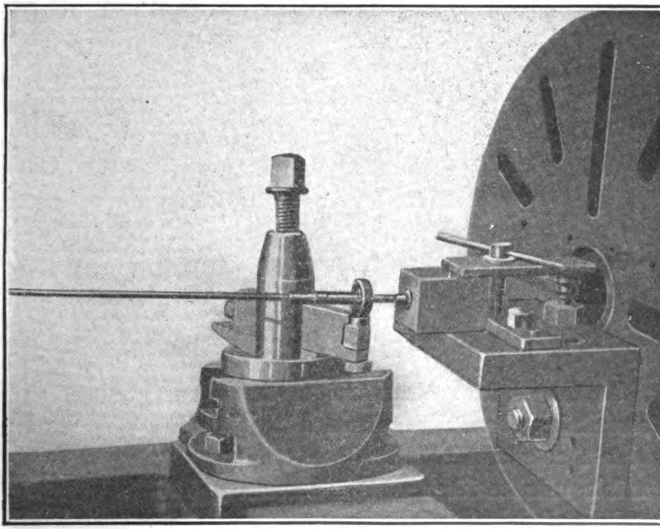


FIG. 114. B

The Indicator on a Nice Face Plate Job

The boring of two holes that cross each other at right angles and exactly central with each other is a nice job. The first thing is to square up the block on all sides and then use a stiff angle plate clamped to face plate as shown.

Having done this it is clamped to the angle plate, as shown in *A*, Fig. 114, and centrally adjusted. After this has been done the angle plate must be bolted to the face plate in such a manner as to insure it from not moving during the operation of boring both holes. Holding in this way also makes it free from all clamping strains.

In turning the block to position shown for boring second hole, care must be taken not to turn the block over; that is to say, the same side must be placed on the angle plate as was used in boring the first hole.

At *B* is shown how the two points should line up when the block has been centrally adjusted. The use of a surface gage in this way gives about the closest alinement of the lathe indicator that can be had. It will be noted that a point is fastened to the lathe as a guide for the indicator point as this makes any movement of the pointer readily visible.

Finishing Up Work

In the various operations of machining, fitting, and assembling of machinery and tools, too often little details in method and finish are overlooked; resulting in a finished article which is defective in operation, and unsightly to the truly mechanical trained eye. Had there been any care, forethought, or possibly knowledge of proper methods on the part of the workman, a neater and superior machine or tool at less cost would have been the result.

Were the little omissions referred to confined to the novice, it would be hardly worth while to mention them, but past experience has proved that the middle-aged journeyman is often as great a sinner in this respect as the apprentice. It sometimes happens that those placed in charge, through lack of shop training, are not able to see small defects or judge their importance.

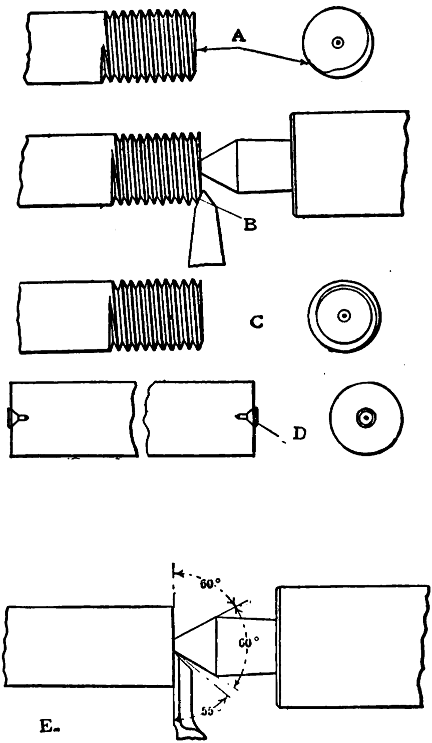


FIG. 115. Finishing the Ends of Work

Notice the first thread on the screw shown at *A*, Fig. 115. This is the way many so-called lathe hands turn the job over to the bench hand to finish with a file.

After the thread has been cut to size, before removing the tool or work from the lathe, open the lead-screw nut, move the carriage by hand and chamfer as shown at *B*, leaving it finished as shown at *C*. This not only looks better but is better, and takes less time. The same method applies to the threading of a nut on the lathe. The above does not apply to harvesting machinery, but to tool work where accuracy usually is economy.

To have the lathe boss bring you a shaft like this, with the centers as shown at *D*, is exasperating. And on protesting at his not having them faced off, to have him tell the bench hand to file "em" off, because "Bill hasn't any cutaway centers for his lathe and can't face them out!" This can be done by backing the tailstock center a little, and they can then be faced off the end clear into the countersink without a cutaway center.

Obviously great-grandfather was aware of those little kinks, and no doubt some who have actually learned their trade will smile when reading things so elementary; but the fact remains that there are plenty of men running lathes that have not learned them yet.

By looking at Fig. *E* it must be apparent to most any one that 55 degrees is about the proper angle to grind a facing tool. This gives ample strength and permits facing close to the center without interfering. But take a look around the shop and see how many facing tools you will find on the tool-boards, ground this way.

A Drilling Kink

Any one who has ever drilled a hole in a lathe by holding the drill with a dog, and feeding it in to the work with the tail spindle, has probably had the drill catch just as it broke through the work, draw off the center, and raise more of a disturbance than is pleasant for any of the parties concerned.

To prevent this, especially in brass work, it is customary to grind the lips of the drill so that they have no top rake, or what corresponds to that in a lathe tool, but there is a

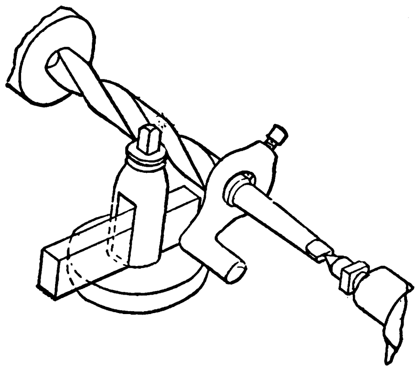


FIG. 116. A Good Drilling Kink

better way, even without the use of a drill chuck of any kind.

Just put a piece of steel or a lathe tool into the tail post, backward, as is often done, but, instead of letting the dog rest on the top of the tool have it rest on the tool-holder and back up against the tool, as shown in Fig. 116. This pulls the carriage along as the drill is fed into the work, and it effectually prevents any dragging of the drill into the work with the damage that often goes along with it.

Grinding Lathe Centers

If there is no center grinder in the shop, a man can, with very little skill, grind a lathe center perfectly true on a common emery wheel in less time than he can set up the ordinary grinding rig on his lathe. Fig. 117 shows the

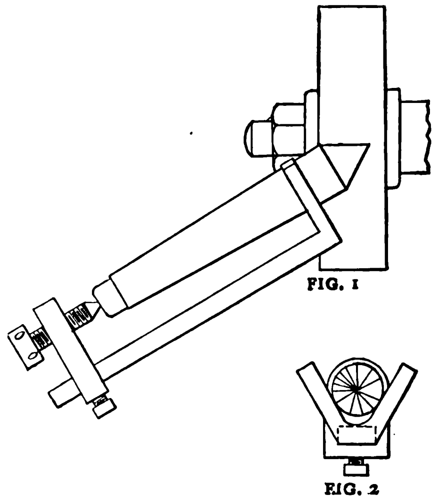


FIG. 117. Grinding Lathe Centers

position of holding the center to the wheel when giving the center the last finish. If the center is in bad shape, by lowering the left hand the operator can make the wheel cut very fast. It does not make any difference how much the center is out of true, the high speed at which it will run generates a true center and makes a good enough job for any except the nicest work.

CHAPTER XII

THREE TYPES OF CENTERING ARBOR

ARBOR work, or the use of arbors in order to have the last turning operations come true with the hole previously bored, has changed somewhat in the last few years. Where a piece cannot be finished at one setting, as a blank for a change gear or similar work, it is customary to first bore the hole and face the hub. Then, in the old days, it would have been forced onto an arbor which was made a very slight taper, from 0.001 to 0.002 inch per inch of length; the arbor driven between the lathe centers and the work turned off. If the arbor is true the piece will then be finished square and true with the hole.

All who have done this know of the occasional slipping of the piece through the cut being too heavy or the arbor a trifle too small. They also know of the old way of enlarging the arbor by prick-punching it to raise burs all over it. If the work is not particular, it can be driven on over the burs and will hold, though it will not be quite true. Or it can be ground after prick punching and trued up again.

The solid arbors took tons of steel and iron as each was good for only one size, and the expansion arbor came to be common on account of the saving in stock and the fact that you could vary them within reasonable limits to fit any size.

When the hole is small in comparison with the outside diameter it is often difficult to drive work in this way and the work slipping or the arbor bending are common occur-

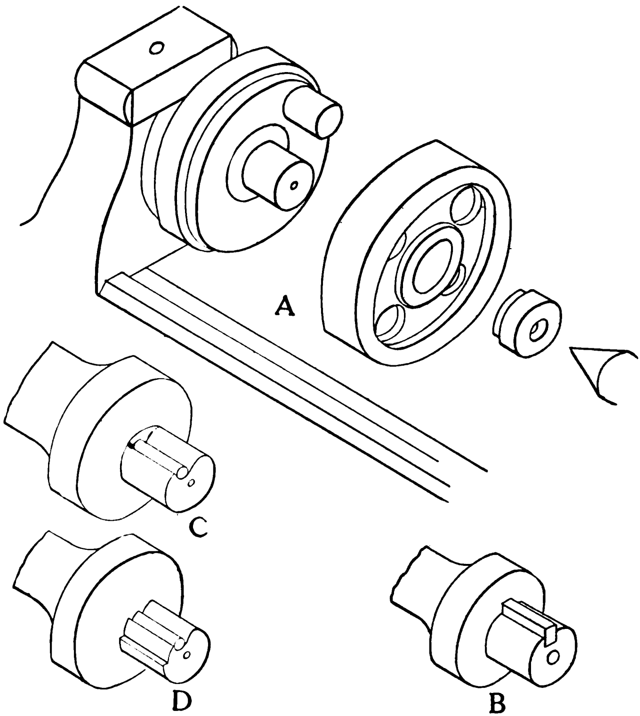


FIG. 118. Centering Arbors

rences. For this reason, as well as to facilitate the handling of work, the centering instead of the driving arbor has come into use. Forcing an arbor into the work takes time and this is avoided in the newer method. This means that the arbor merely centers the work but does not drive it as shown in *A*, Fig. 118.

In such cases the arbor might be called a stud or pin and is carried in the lathe spindle, projecting just far enough to carry the work, but usually not projecting through the hub. The piece is usually first chucked and bored and the face turned off at the same setting. Then, after the pieces are all bored, the centering arbor is put in place, the face plate and stud screwed on and the lathe is ready.

If the arbor runs true and the face plate is square, it is very evident that the work will run true with the hole. The blank is slipped on the arbor so that the stud will come between the spokes or in the holes and drive the gear. Then a cap or center plug, as shown, is slipped into the outer end of the bore to hold the work square and keep it in place, this being removed when the outer hub is squared. Or the hub is squared first and then the plug put into place. This is not necessary in all cases as the pressure of the facing tool will tend to keep the work against the face plate on cast-iron work. On steel castings or metal that rolls up a chip there is more danger of the work being pulled off the arbor and something must be done to prevent it.

In some shops the centering arbor is fitted with a key, *B*, and the work is driven in this way. This necessitates key-seating or splining the work after boring, but in a shop with a key-seater this is easily and quickly done. It is, of course, not advisable to use this method with work that is large as compared with the hole, it being always better to drive the work as far from the center as possible. This plan is used whether the work needs to be key-seated or not, for, as a rule, the key-seat does no harm whether it is needed or not.

In work where the key-seat is objectionable or where there is no key-seater handy, the device shown in *C* is sometimes used. The arbor fits and centers the work as before, but as the tool tries to turn the work on the arbor, the roller tends to roll up the incline, as in a roller ratchet, and holds the work from turning. There is always the tendency to crowd the work off center with this device, much more so than where the key is used, and may be objectionable on this account in very accurate work.

This is modified in some cases by making the work bear only on the lower half of the arbor as in *D*, and cutting away the upper half, allowing the roller to bite as before, so that it draws the bore of the work up against the arbor. In this way the bearing part of the arbor can be exactly to size instead of having a slight allowance for the work being slipped on.

In all of these the object is to hold the work accurately and securely, to drive it steadily and allow it to be put on and taken off easily and quickly.

With the arbor best adapted for the work in hand, and face plates that allow the work to be held and driven, yet faced down on the back side, a large variety of duplicate work can be done on the engine lathe at a low cost, much lower than is usually considered possible. A fork tool can go down each side of the rim and face it to size, while a regular tool turns the outside diameter. Suitable stops allow the tools to be moved to the same place every time and in this way do away with much measuring, as they will all be alike except for the wear of tools. The centering arbor has many advantages along this line of work.

CHAPTER XIII

GOOD EXAMPLES OF ENGINE LATHE WORK

In the development of the automatic and special machines, both hand-operated and automatic, we have, in too many cases, largely lost sight of the older machines, such as the engine lathe, and it has received scant attention in the way of fixtures or appliances that might improve its output. A number of shops are taking the matter up, however, and the engine lathes are making a splendid showing in many cases, and are turning out more work than was deemed possible a few years ago.

The examples shown here are from the shop of the Whitcomb-Blaisdell Company, Worcester, Mass., the fixtures being due to Mr. Wiggin, their superintendent. They give an idea of some of the simple methods and fixtures by which the output of a lathe can be largely increased under good conditions.

Starting with the shaft-turning attachment shown in Figs. 119 and 120, which is used for turning feed-rods and lead screws, we have a holder which spans the lathe and fastens to both wings of the carriage. This carries a cutter head with four circular cutters and one flat cutter between two of the others. The circular cutters are set so that each takes its share of the cut, the last being the finish size. The fifth cutter is also set to the finishing size, but does not cut unless the other finishing cutter slips, or loses its edge, then the fifth or emergency cutter takes up the work to prevent the rod from being left large enough to bind or seize in the bushing. Where a water finish is desired, as on the feed-rods, a lathe tool which is ground for this work goes in the tool-post behind the holder.

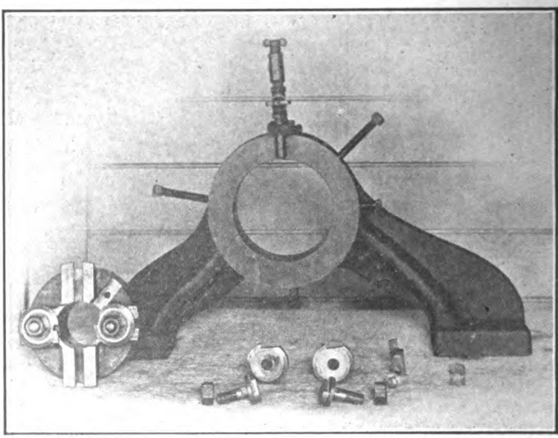


FIG. 119. Details of Fixture Used

Lead screws vary from $1\frac{1}{4}$ to $1\frac{3}{4}$ inches, and feed-rods from $1\frac{1}{8}$ to $1\frac{1}{2}$ inches in diameter, the average reduction in diameter being about $\frac{3}{16}$ of an inch. On the lead screws which do not require the water finish, owing to having the thread cut afterward, the turning from the rough is at the rate of 13 inches per minute, while with the feed-rods, finished ready for use except the spline, the rate is 7 inches per minute.

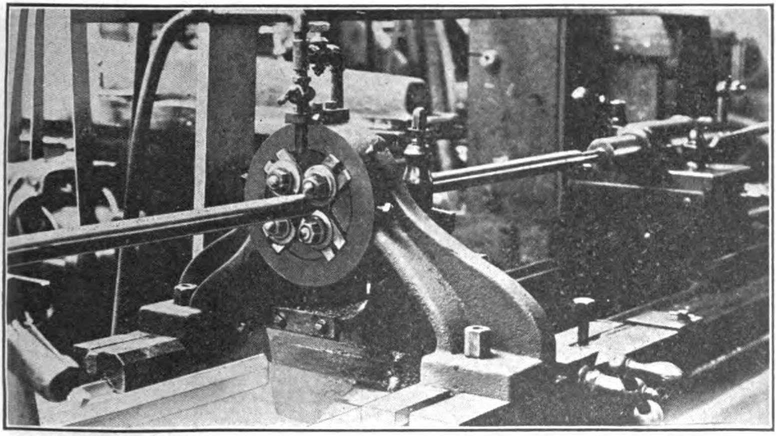


FIG. 120. The Fixture in Use Turning Shafting

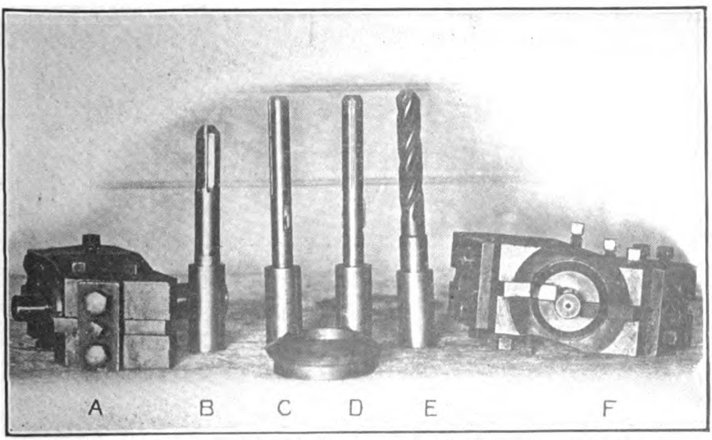


FIG. 121. Tools Used for Bevel-Gear Blanks

Bevel Gear Blanks

The bevel gear blanks shown are 5½ inches in diameter and show the work of the engine lathe equipped with a turret in place of the usual tailstock. This, of course, takes it out of the plain engine-lathe class, but it is a modification that can be made at a comparatively low cost and is evidently more than justified by the results obtained.

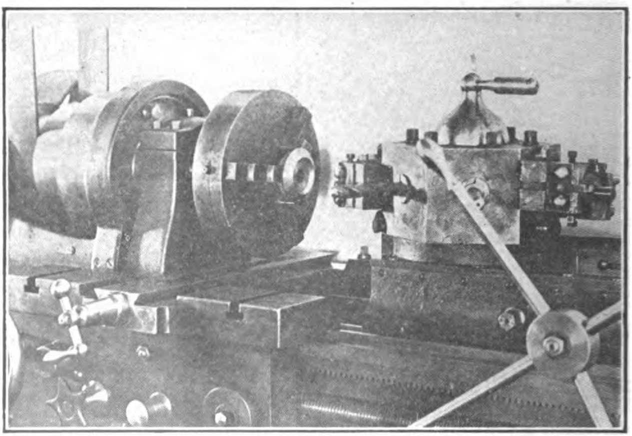


FIG. 122. Turning the Blanks

Figs. 121 and 122 show the tools used on two sizes of gear blanks. The gear blank is first chucked, then rough bored with a flat cutter held in bar *D*, the front end of the bar being guided in a hardened steel bushing in the lathe spindle, which gives a hole that is not far from true. Then the three-lip core drill *E* is used, followed by the sizing flat cutter *C* and the reamer *B*.

The roughing and finishing forming tools *A* and *F* are next used, being guided by the hardened-steel plug in the holder. This has taken about three minutes so far. Then the blanks are taken to a key-seater, splined and slipped on a splined arbor in the lathe spindle with the beveled face against a true face plate. Two forming tools, similar to *A* and *F*, rough and finish the hub of the gear blank and the job is done with a total time of eight minutes each for all operations, including splining, in fair-sized lots. The allowance for error is 0.001 inch.

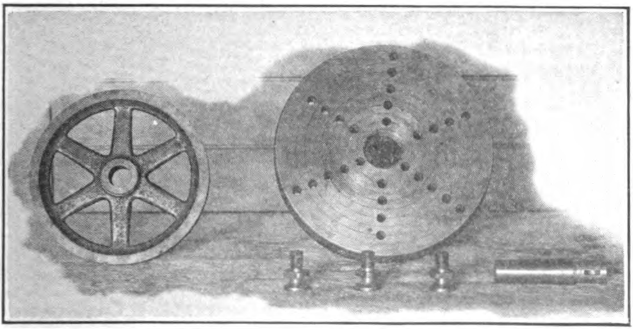


FIG. 123. The Work and the Face Plate

Spur Gear Blanks

These and the next job are plain engine-lathe work with only a few slight expenses for fixtures. The spur gears are intermediates for the change-gear tram and are 9½ inches in diameter by 1⅛-inch face. Figs. 123 and 124 show this operation and the fixtures required. First the face plate, with six series of holes, tapped in radial lines for locating the driving studs shown beneath. These are made with a cone head, the cone being at the right distance from the face plate to catch the arms of the gear blank and not only drive them, but hold them against the face plate. Then the centering arbor, with a keyway for locating the gear blank, completes the outfit.

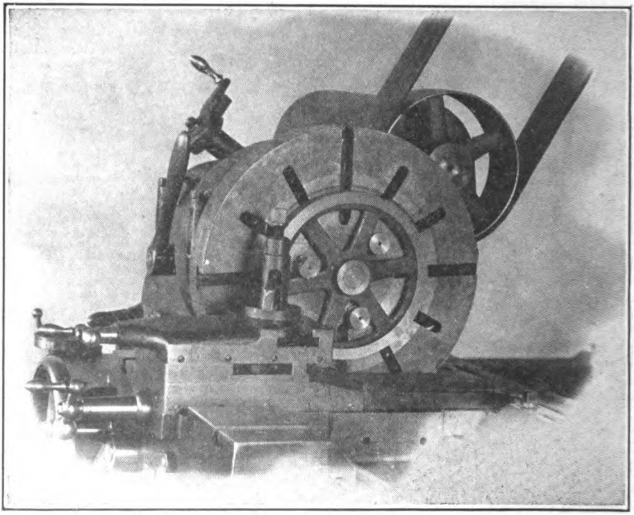


FIG. 124. Turning the Gear Blank

The first operation is to chuck, bore, and ream the same as with the other gear blanks, also square the hub. They are then slipped over the spindle arbor and driven by three cone-head bolts, as shown in Fig. 124. The face plate is recessed so the turning tool can turn over the whole rim, the rim is faced and then turned over to be faced on the other side. The total time in these gears was 14 minutes each, and the limit of accuracy 0.003 inch.

Facing Feed Plates

After the boring operation, which is always done first and in the manner described for the bevel-gear blanks, the only fixture needed is the expanding arbor shown in Fig. 125. This fits the taper of the lathe spindle and the long portion is made a nice sliding fit to the hub of the feed-gear plate. The front end is turned smaller for the expanding bushing shown at *B*. The stud *C* slips through the bushing with the square head this way and the pin *E* holds the collar *D* in place so that the expanding bushing is moved in and out on the tapered end of the arbor and in this way centers the outer end of the work perfectly so that the tail center can be used to insure steadiness under a heavy cut.

In finishing the feed plate as in Fig. 126 the driving is done by a face-plate stud near the outer end of the handle, and in this way the outer diameter of the long hub is turned and the back of the plate faced off with the power feed. Both the roughing and finishing cuts are four to the inch. The hub is $4\frac{1}{2}$ inches long by $2\frac{1}{4}$ inches outside diameter, while the hole is $1\frac{1}{4}$ inches in diameter. The plate is $\frac{7}{8}$ inch thick, making the total length of hole $5\frac{3}{8}$ inches. These feed plates are finished all over, the total time for all operations being 50 minutes each.

Gages are used for everything, a set being shown on the tool slide, held together by the brass ring. These gages aid in getting out the work quickly, as, if the piece fits the gage, it is right, and no measuring has to be done.

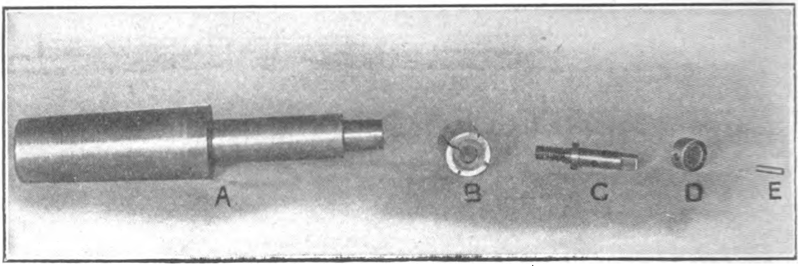


FIG. 125. The Expanding Arbor in Detail

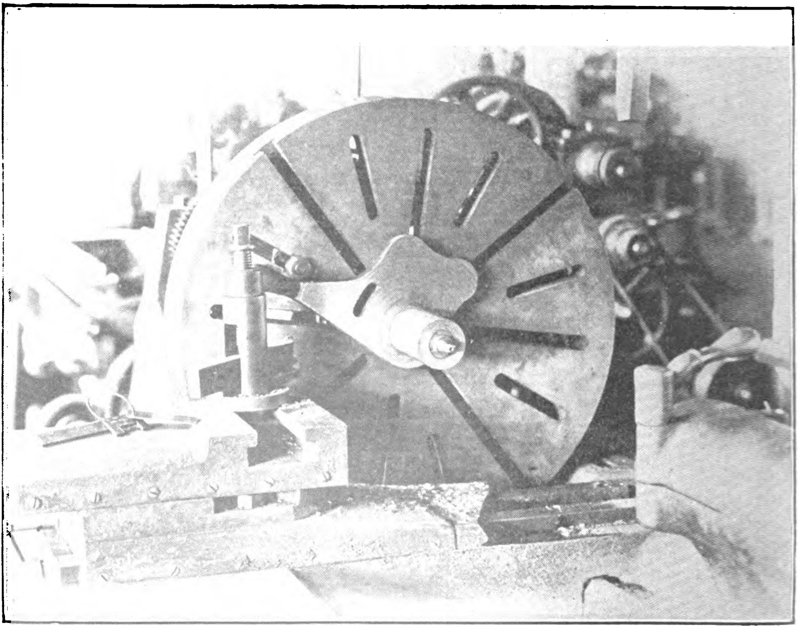


FIG. 126. Turning the Feed-Plates Gear

CHAPTER XIV

CARE OF THE LATHE

THE care of lathes and other shop tools deserves more attention than it usually receives. While in some shops they will be found clean and bright, in others they are just the reverse, and both the quantity and the quality of work suffer. This does not mean that it is necessary to polish every part of the lathe, and scour it all bright with emery cloth or other abrasive. In fact these should be used sparingly. If the lathe has been allowed to get "gummy" from the oil not being wiped off, use a little kerosene or benzine in preference to emery, and in very few cases should it be used on the ways; never if it can be avoided.

When you see a lathe with the ways dented and scored and other parts in a similar condition, it is safe to say the man in charge is not a high-grade machinist. Even if he is not responsible for the condition, he will endeavor to remedy it as quickly as possible, even in his own time, if no other is allowed him. Character shows up just as strongly in the shop as elsewhere, and unless a man or boy thinks enough of the tools he uses to take decent care of them, and has enough interest to do this, it is pretty safe to say he will never be a first-class machinist. If it is the right kind of a shop the care of the tools will be noted by the man higher up, and it counts when a better job comes along.

Oiling the Lathe

A little oil and often, same as the homeopathic doctor gives medicine, is the best with lathes, and other machinery. The spindle bearings should be tight enough to prevent lifting or jumping, and require oil to prevent heating and cutting. Lathes with cast-iron bearings must be watched closely. Cast iron makes a fine bearing after it gets glazed but it must not be allowed to run dry. Some careful machinists fill the top of the oil pipe (or bottom of the oil cup if there is one) with clean waste, to keep dust and chips from being washed down to the bearing with the oil.

Keeping the ways lubricated is not easy, as the angle lets the oil run down, and it also attracts dust and dirt for the carriage to run over. The plan adopted by some of the best shops is to wipe the ways clean each morning, put on a few drops of oil, and run the carriage from one end to the other, so as to work the oil under it. Any surplus is wiped off to prevent its running down and gumming.

The various bearings on the apron of the carriage should all have their little drop of oil, through the hole provided in the front of the apron. The back gears, the bearings of the feed rod, and the lead screw all deserve a little attention, as well as the screws and bearings of the cross-feeds and the tailstock. The dead center also needs a drop of oil occasionally, especially on heavy work. To assist in this, heavy centers are often provided with an oil groove on top for feeding the oil to the bearing. Some prefer to cut an oil groove in the center of the work itself, so that it will carry the oil around with it, but this is not usually considered advisable.

Keep the Lathe-bed Clear

The piling of work, of tools and other things on the lathe-bed and on the carriage is not a good plan. It marks up the bed, gets in the way of the carriage at times, and is usually

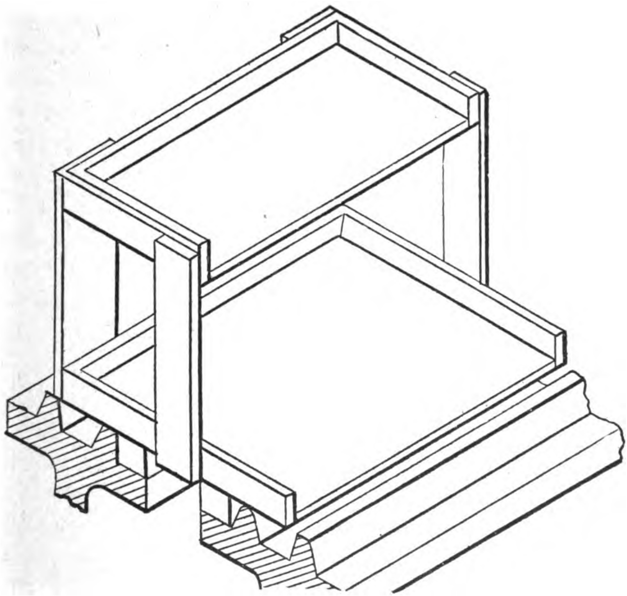


FIG. 127. A Double-Decked Tool Board

more of a habit than a necessity. While it is not a criminal offense, the habit should not be encouraged.

When it is necessary to have work or special tools around they can usually be kept on the tool board at the foot of the bed. This tool board should have the sides raised enough to prevent the tools from falling or being pushed off, and it can be double-decked like an apartment house, if more room is needed. A common design is shown in Fig. 127 and can be modified to suit conditions and needs. Two strips underneath, to fit the inside of the ways, prevent its being accidentally pushed off the end.

Care of the Spindle

The center holes in the lathe spindles require especially good care. It does not take a diagram to show any one how the center will be thrown out of true if anything, even a thread of waste, is left in the hole or on the center. Even the "fuzz" from waste affects it on fine work, and for this reason some of the best mechanics use a small brush instead of waste, for this purpose. When it is not being used, it is a good plan to insert a cork or a blank center to prevent dirt from entering.

The threads on the nose of the spindle should also be well cared for. One trouble that arises is the "crossing" of threads in putting on chucks and face plates. This can be avoided by being sure that the chuck or face plate is held square with the spindle when putting it on. When no chuck or face plate is on the spindle, some protect the nose by screwing on a cap or slipping one of sheet metal over it.

The centers themselves also demand good treatment if first-class work is to be done. They must not be dropped or used for hammers or center punches. In some shops even this self-evident fact may be a revelation.

Overhang of any part is always bad, though it cannot always be avoided. The lathe tool, the tail spindle, and the compound rest are all to be watched in this respect.

A Common Failing

Perhaps the most common abuse of a lathe or its parts is the practice of using the tool-post wrench for a hammer to force the lathe tool into position. Even men who are careful of the working parts of a lathe will do this at times, and it may be a trifle finicky to object to it. But it is just as well to avoid it unless the habit is too firmly fixed.

Some machinists, especially the older men who had a similar feeling of ownership in their lathe that an engineer did in his locomotive, were honestly disturbed at the beginning of higher speeds and deeper cuts owing to the new steels. To them it seemed like abuse of their favorite machine, and it really hurt them to do it. But abuse of this character is up to the foreman or the owner, and the duty of the machinist is to run the lathe at the speeds desired, as the wear or breakage coming from this is beyond his control. Until the lathe refuses to run or the tool fails, there is nothing to do but follow instructions to the letter.

The weekly cleaning of lathes, usually the last half or quarter-hour on Saturday, is a good thing in many ways for most shops. In some cases it may pay to have special men to clean them after hours, but the mechanic is better fitted for this than a laborer unless the latter is especially trained for the work. And after you train him he is apt to get another job, because his first wife's cousin works in a place where they serve tea in the afternoon.

The cleaning process, however, should not be turned into a scouring match, as sometimes happens. A clean lathe doesn't necessarily dazzle your eyes, but has the gum and dirt carefully wiped off.

INDEX