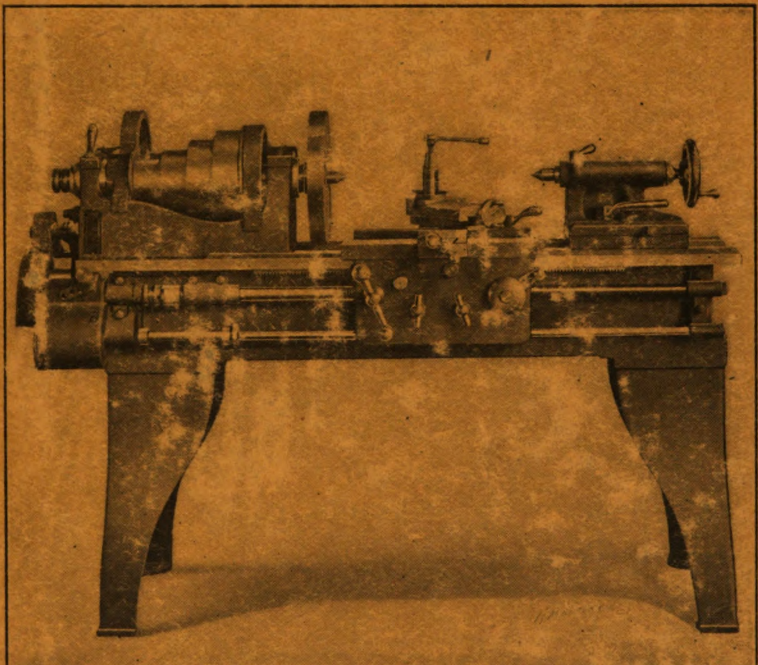
25 CENTS

OPERATION OF MACHINE TOOLS

BY FRANKLIN D. JONES

THE LATHE- PART II

SECOND EDITION



MACHINERY'S REFERENCE BOOK NO. 92

PUBLISHED BY MACHINERY, NEW YORK

MACHINERY'S REFERENCE SERIES

EACH NUMBER IS A UNIT IN A SERIES ON ELECTRICAL AND STEAM ENGINEERING DRAWING AND MACHINE DESIGN AND SHOP PRACTICE

NUMBER 92

OPERATION OF MACHINE TOOLS

By FRANKLIN D. JONES

SECOND EDITION

THE LATHE

PART II

CONTENTS

CHAPTER I

TAPER TURNING

It is often necessary, in connection with lathe work, to turn parts tapering instead of straight or cylindrical. If the work is mounted between the centers, one method of turning a taper is to set the tailstock center out of alignment with the headstock center. When both of these centers are in line, the movement of the tool is parallel to the axis of the work and, consequently, a cylindrical surface is produced; but if the tailstock h_1 is set out of alignment as shown in

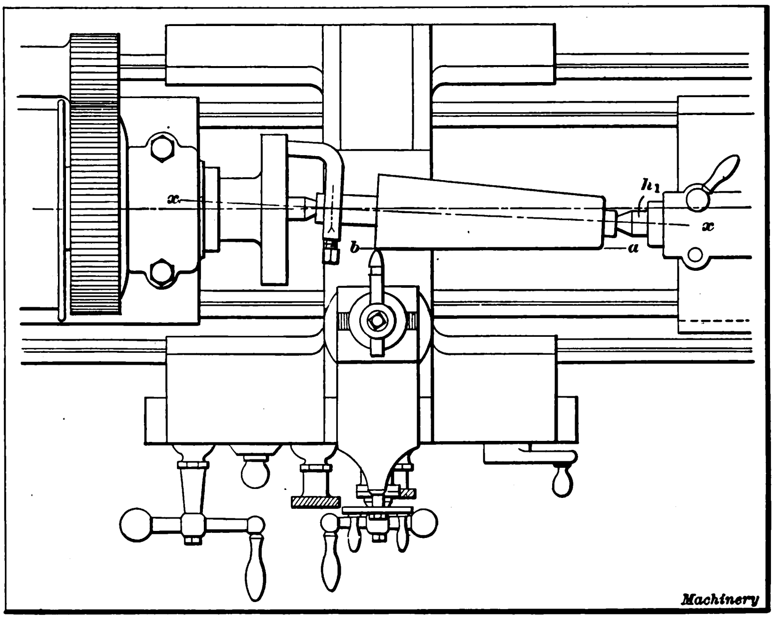


Fig. 1. Taper Turning by the Offset-center Method

Fig. 1, the work will then be turned tapering as the tool is traversed from a to b, because the axis x—x is at an angle with the movement of the tool. Furthermore the amount of taper or the difference between the diameters at the ends for a given length, will depend on how much center h_1 is set over from the central position.

The amount of taper is usually given on drawings in inches per foot, or the difference in the diameter at points twelve inches apart. For example, the taper of the piece shown at A, Fig. 2, is 1 inch per foot, as the length of the tapering surface is just twelve inches and the difference between the diameters at the ends is 1 inch. The conical roller shown at B has a total length of 9 inches and a tapering surface 6 inches long, and in this case the taper per foot is also 1 inch, there being a difference of ½ inch in a length of 6 inches or 1 inch in twice that length. When the taper per foot is known, the amount that the tailstock center should be set over for turning that taper, can easily be estimated, but it should be remembered that the setting obtained in this way is not absolutely correct, and is only intended to locate the center approximately. When a taper needs to be at all accurate, it is tested with a gage, or by other means, after taking a trial cut, as will be explained later, and the tailstock

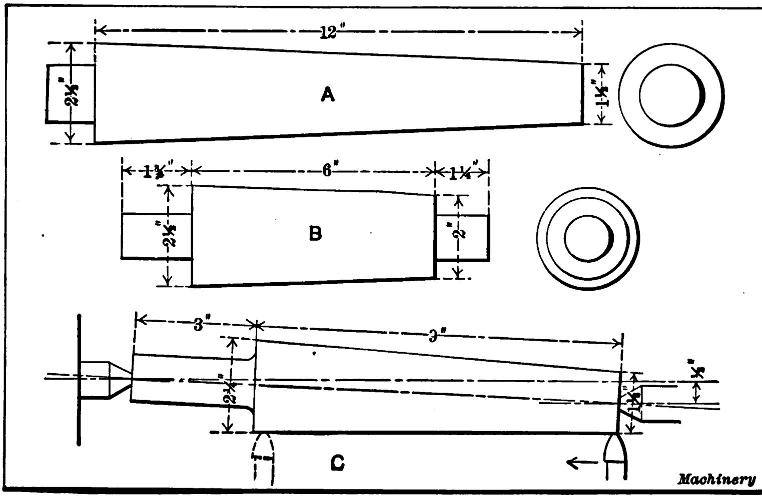


Fig. 2. Examples of Taper Work

center is re-adjusted accordingly. There are also more accurate methods of setting the center, than by figuring the amount of offset, but as the latter is often convenient, this will be referred to first.

Setting Tailstock Center for Taper Turning

Suppose the tailstock center is to be set for turning part *C*, Fig. 2, to a taper of approximately 1 inch per foot. In this case the center would simply be moved toward the front of the machine ½ inch, or one-half the required taper per foot, because the total length of the work happens to be just 12 inches. This setting, however, would not be correct for all work requiring a taper of 1 inch per foot, as the adjustment depends not only on the *amount* of the taper but on the *total length* of the piece.

For example, the taper roller *B* has a taper of 1 inch per foot, but the center, in this case, would be offset less than one-half the taper per foot, because the total length is only 9 inches. For lengths longer or shorter than twelve inches, the taper per inch should be found first; this is then multiplied by the *total* length of the work (not the length of the taper) which gives the taper for that length, and one-half this taper is the amount to set over the center. For example, the taper per inch of part *B* equals 1 inch divided by 12 = 1/12 inch. The total length of 9 inches multipled by 1/12 inch = 3/4 inch, and ½ of ¾ = ⅜, which is the distance that the tailstock center should be offset. In this example if the taper per foot were not known, and only the diameters of the large and small ends of the tapered part were given, the difference between these diameters should first be found (2½ — 2 = ½); this difference should then be divided by the length of the taper (½ ÷ 6 = 1/12 inch) to obtain the taper per inch. The taper per inch times the *total* length represents what the taper would be if it extended throughout the entire length, and one-half of this equals the offset, which is ⅜ inch.*

Example of Taper Turning

As a practical example of taper turning let us assume that the piece *A*, Fig. 4, which has been centered and rough turned as shown, is to be made into a taper plug, as indicated at *B*, to fit a ring gage as at *C*. If the required taper is 1½ inch per foot and the total length is 8 inches, the tailstock center would be offset ½ inch.

To adjust the tailstock, the nuts *N* (Fig. 3) are first loosened and then the upper part *A* is shifted sidewise by turning screw *S*. Scales are provided on some tailstocks for measuring the amount of this adjustment; if there is no scale, draw a line across the movable and stationary parts *A* and *B*, when the tailstock is set for straight turning. The movement of the upper line in relation to the lower will then show the offset, which can be measured with a scale.

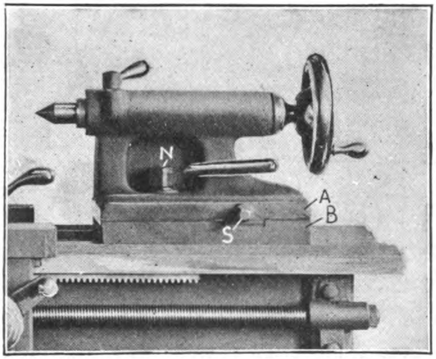


Fig. 3. Detail View of Lathe Tailstock

When the adjustment has been made, nuts *N* are tightened and the work, with a dog attached, is placed between the centers the same as for straight turning. The taper end is then reduced by turning, but before it is near the finished size, the work is removed and the taper tested by inserting it in the gage. If it is much out, this can be felt, as the end that is too small can be shaken in the hole. Suppose the plug did not taper enough and only the small end came in contact with the gage, as shown somewhat exaggerated at *D;* in that case the center would be shifted a little more towards the front, whereas if the taper were too steep, the adjustment would, of course, be in the opposite direction. A light cut would then be taken, to be followed by another test. If the plug should fit the gage so well that there was no perceptible shake, it could be tested more closely as follows: Draw three or four chalk lines along

*See also MACHINERY'S Reference Book No. 18, "Shop Arithmetic for the Machinist."

the tapering surface, place the work in the gage and turn it a few times. The chalk marks will then show whether the taper of the plug corresponds to that of the gage; for example, if the taper is too great, the marks will be rubbed out on the large end, but if the taper is correct, the lines throughout their length will be partially erased.

Another and more accurate method of testing tapers, is to apply a thin coat of Prussian-blue to one-half of the tapering surface, in a lengthwise direction. The work is then inserted in the hole or gage and turned to mark the bearing. If the taper is correct, the bearing marks will be evenly distributed, whereas if the taper is incorrect, they will appear at one end. Tapering pieces that have to be driven tightly into a hole, as a piston-rod, can be tested by the location of the bearing marks produced by actual contact.

After the taper is found to be correct, the plug is reduced in size until it just enters the gage as at *C*. The final cut should leave it slightly above the required size, so that a smooth surface can be

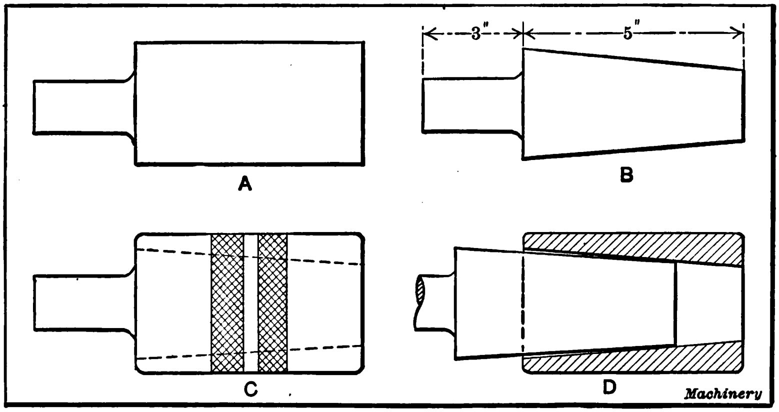


Fig. 4. Taper Plug and Gage

obtained by filing. It should be mentioned that on work of this kind the final finish is very often obtained by grinding in a regular grinding machine instead of by filing. When this method is employed, a lathe is used merely to rough turn the part close to size.

When the amount that the tailstock center should be offset is determined by calculating, as in the foregoing example, it is usually necessary to make slight changes afterward, and the work should be tested before it is too near the finished size so that in case one or more trial cuts are necessary, there will be material enough to permit this. When there are a number of tapered pieces to be turned to the same taper, the adjustment of the tailstock center will have to be changed unless the total length of each piece and the depth of the center holes are the same in each case.

Setting the Tailstock Center with a Caliper Tool

Another method of setting the tailstock center for taper turning is illustrated in Fig. 5. The end of a rod is to be made tapering as at *A* and to dimensions *a*, *b*, *c* and *d*. It is first turned with the centers in line as at *B*. The end *d* is reduced to diameter *b* up to the beginning of the taper and it is then turned to diameter *a* as far as the taper part *c* extends. The tailstock center is next set over by guess and a caliper tool is clamped in the toolpost. This tool,

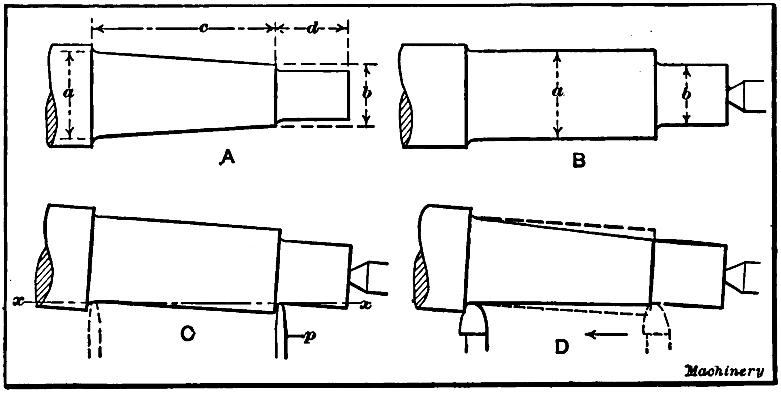


Fig. 5. Setting Work for Taper Turning by use of Caliper Gage

a side view of which is shown in Fig. 6, has a pointer *p* that is free to swing about pivot *r*, which should be set to about the same height as the center of the work. The tailstock center is adjusted until this pointer just touches the work when in the positions shown by the full and dotted lines at *C*, Fig. 5; that is, until the pointer makes

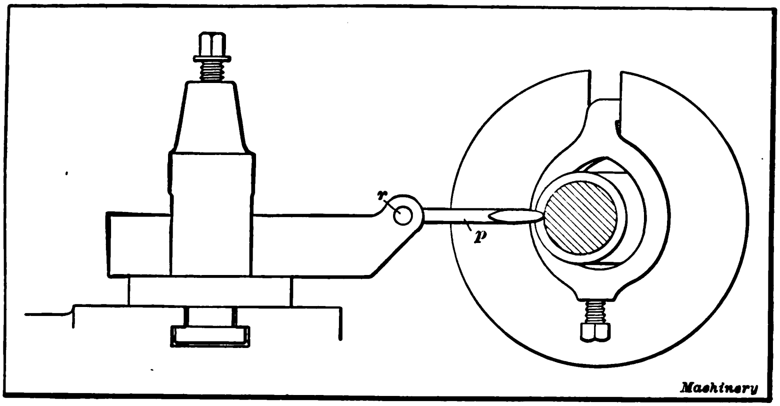


Fig. 6. Side View showing Relative Positions of Gage and Work

contact at the beginning and end of the taper part. The travel of the carriage will then be parallel to a line *x—x*, representing the taper; consequently, if a tool is started at the small end, as shown by the dotted lines at *D*, with the nose just grazing the work, it will also just graze it when fed to the extreme left as shown. Of course, if the taper were at all steep, more than one cut would be taken. If these various operations are carefully performed, a fairly accurate taper can be produced. The straight end *d* is reduced to size after the tail-center is set back to the central position. Some mechanics turn notches or grooves at the beginning and end of the tapering part, having diameters equal to the largest and smallest part of the taper; the work is then set by these grooves with a caliper tool. The advantage of the first method is that most of the metal is removed while the centers are in alignment.

Setting the Tailstock Center with a Square

Still another method of adjusting the tailstock for taper turning, which is very simple and eliminates all figuring, is as follows: The part to be made tapering is first turned cylindrical or straight

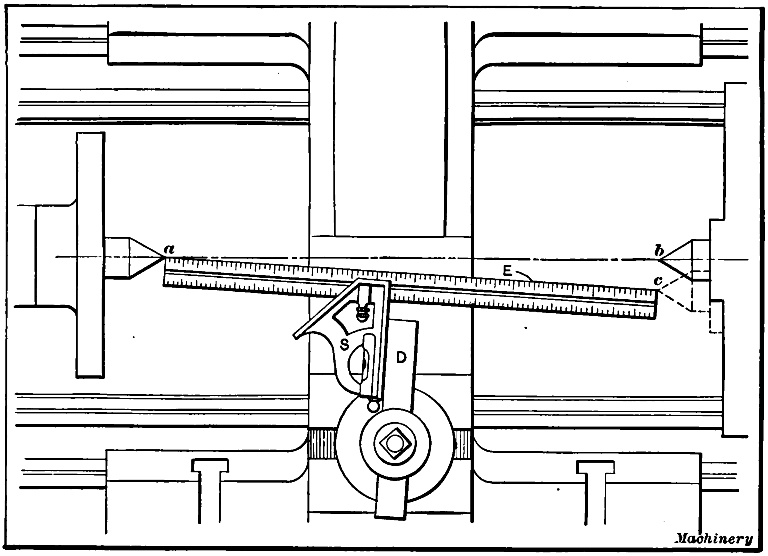


Fig. 7. Obtaining Tailstock Center Adjustment by use of Square

for 3 or 4 inches of its length, after the ends have been properly centered and faced square. The work is then removed and the tailstock is shifted along the bed until the distance *a—b* between the extreme points of the centers, is exactly 1 foot. The center is next offset a distance *b—c* equal to one-half the required taper per foot, after which a parallel strip *D*, having true sides, is clamped in the toolpost. Part *D* is then set at right angles to a line passing from one center point to the other. This can be done conveniently by holding a 1-foot square (preferably with a sliding head) against one side of *D* and adjusting the latter in the toolpost until edge *E* of the square blade is exactly in line with both center points. After part *D* is set, it should be clamped carefully to prevent changing the position. The angle between the side of *D* and an imaginary line which is perpendicular to axis *a—b*, is now equal to one-half the angle of the required taper. The axis of the part to be turned should be set parallel with line *E*, which can be done by setting the cylindrical surface which was previously finished, at right angles to the side of *D*. In order to do this the work is first placed between centers, the tailstock being shifted along the bed if necessary; the tail-center is then adjusted laterally until the finished cylindrical surface is square with the side of *D*. A small try-square can be used for testing the position of the work, as indicated in Fig. 8. If the length of the work is less than 1 foot, it will be necessary to move the center toward the rear of the machine, and if the length is greater than 1 foot, the adjustment is, of course, in the opposite direction. (This method has been described in MACHINERY by Mr. H. C. Lord.)

The Taper Attachment

Turning tapers by setting over the tailstock center has some objectionable features. When the lathe centers are not in alignment,

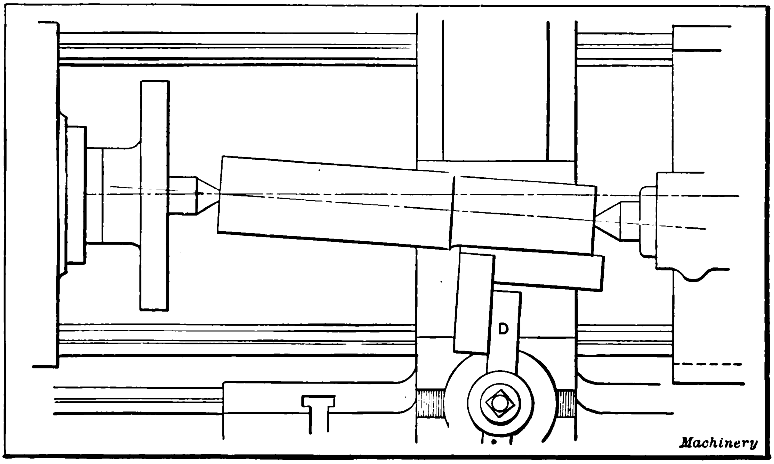


Fig. 8. Second Step in Adjusting Tailstock Center by use of Square

as when set for taper turning, they bear unevenly in the work centers because the axis of the work is at an angle with them; this causes the work centers to wear unevenly and results in inaccuracy. Furthermore, the adjustment of the tailstock center must be changed when turning duplicate tapers, unless the length of each piece and the depth of the center holes are the same. To overcome these objections, many modern lathes are equipped with a special device for turning tapers, known as a taper attachment, which permits the lathe centers to be kept in alignment, and enables more accurate work to be done.

Taper attachments, like lathes, vary some in their construction, but all operate on the same principle. An improved form of taper attachment is illustrated in Figs. 9 and 10, which show a plan view of a lathe carriage with an attachment fitted to it, and also a sectional view. This attachment has an arm *A* on which is mounted a slide *S* that can be turned about a central pivot by adjusting screw *D*. The arm *A* is supported by, and is free to slide on a bracket *B* (see also sectional view) that is fastened to the carriage, and on one end of the arm there is a clamp *C* that is attached to the lathe bed when turning tapers. On the slide *S* there is a shoe *F* that is connected to bar *E* which passes beneath the tool slide. The rear end of the cross-feed screw is connected to this bar, and the latter is clamped to the tool-slide when the attachment is in use.

When a taper is to be turned, the carriage is moved opposite the taper part and clamp *C* is fastened to the bed; this holds arm *A* and slide *S* stationary so that the carriage, with bracket *B* and shoe *F* can be moved with relation to the slide. If this slide *S* is set at an angle, as shown, the shoe as it moves along, causes the tool-slide and tool to move in or out, but if the slide is set parallel to the carriage travel, the tool-slide remains stationary. Now if the tool, as it feeds lengthwise of the work, is also gradually moved crosswise, it will turn a taper, and as this crosswise movement is caused by the angularity of slide *S*, different tapers are obtained by setting the slide to different positions. By means of a graduated scale at *G*, just what taper would be obtained for any angular position of the slide, is shown. On some attachments there are two sets of graduations, one giving the taper in inches per foot and the other in degrees. While tapers are ordinarily given in inches per foot on drawings, sometimes the taper is given in degrees instead. Fig. 11 shows an enlarged view of the scale with the slide set for turning a taper of 1 inch per foot. The attachment is set for turning tapers by adjusting slide *S* until pointer *p* is opposite the division or fractional part of a division representing the taper. The whole divisions on the scale represent taper in inches per foot, and by means of the subdivisions, the slide can be set for turning frac-

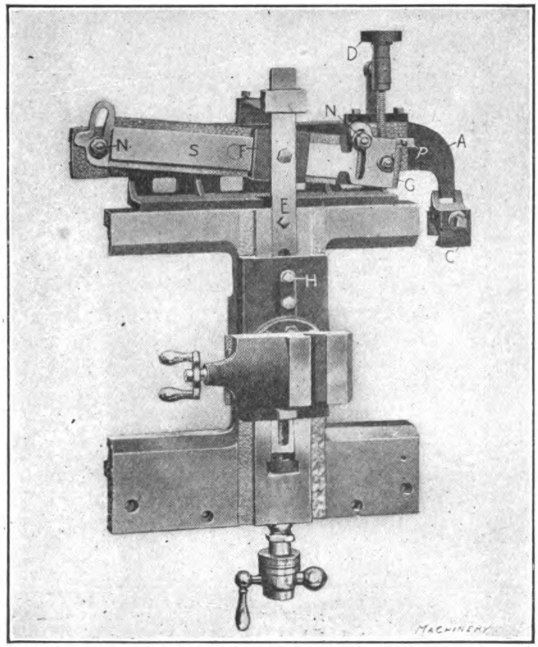


Fig. 9. A Lathe Taper Attachment

tional parts of an inch per foot. When slide S is properly set, it is clamped to arm A by the nuts N. Bar E is also clamped to the tool-slide by bolt H, as previously stated. The attachment is disconnected

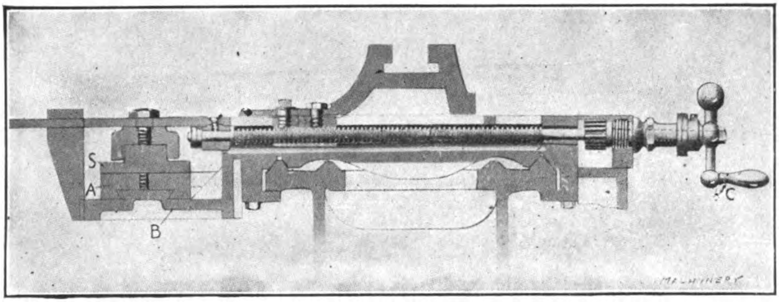


Fig. 10. Sectional View of Taper Attachment

for straight turning by simply loosening clamp C and the bolt H. An example of taper turning with the attachment is given in Chapter II.

Height of Tool when Turning Tapers

The cutting edge of the tool, when turning tapers, should be at the same height as the center or axis of the work, whether an attachment is used or not. The importance of this will be apparent by referring to Fig. 12. To turn the taper shown, the tool T would be moved back a distance x (assuming that an attachment is used) while traversing the length l. If the tool could be placed as high as point a, for the sake of illustration, the setting of the attachment remaining as before, it would again move back a distance x, while moving a distance l, but the large end would be undersized (as shown exaggerated by the dotted line) if the diameters of the small ends were the same in each case. Of course, if the tool point were only slightly above or below the center, the resulting error would also be small. The tool can easily be set central by comparing the height of the cutting edge at the point of the tool with one of the lathe centers before placing the work in the lathe.

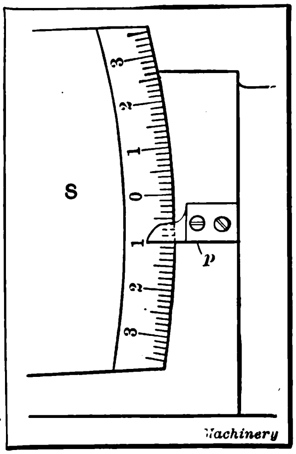


Fig. 11. Scale of Taper Attachment

Taper Turning with the Compound Rest

The amount of taper that can be turned by setting over the tailstock center and by the taper attachment, is limited, as the centers can only be offset a certain distance, and the slide S of the attachment cannot be swiveled beyond a certain position. For steep tapers, the compound rest E is swiveled to the required angle and used as indicated in Fig. 14, which shows a plan view of a rest set for turning the valve *V*. This compound rest is an upper slide mounted on the lower or main cross-slide *D*, and it can be turned to any angular position so that the tool, which ordinarily is moved either lengthwise

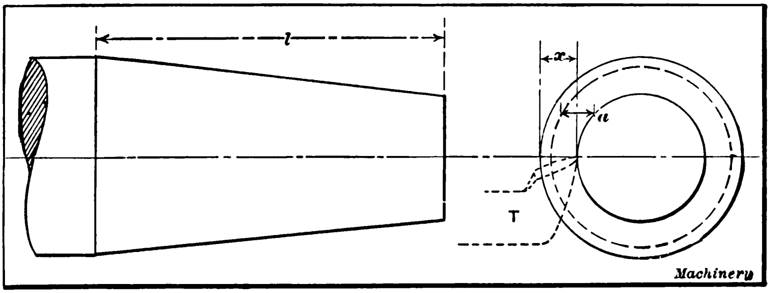


Fig. 12. Tool Point should be in same Horizontal Plane as Axis of Work for Taper Turning

or crosswise of the bed, can be fed at an angle. The base of the compound rest is graduated in degrees and the position of these graduations shows to what angle the upper slide is set. Suppose the seat of valve *V* is to be turned to an angle of 45 degrees with the axis or center, as shown on the drawing at *A*, Fig. 13. To set the compound rest, nuts *n* on either side, which hold it rigidly to the lower slide, are first loosened and the slide is then turned until

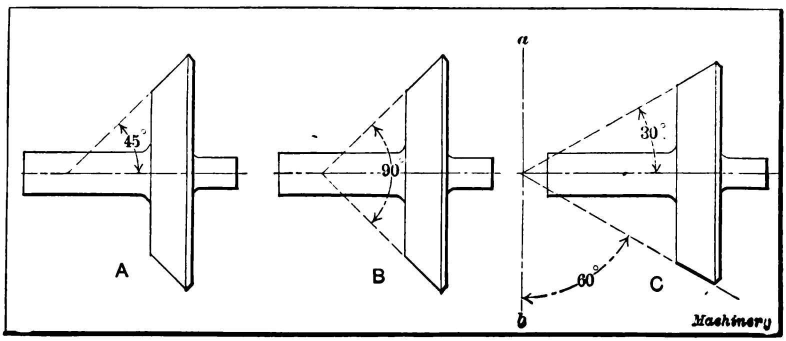


Fig. 13. Example of Taper Work Turned by using Compound Rest

the 45 degree graduation is exactly opposite the zero line; the slide is then tightened in this position. A cut is next taken across the valve by operating handle *w* and feeding the tool in the direction of the arrow.

In this particular instance the compound rest is set to the same angle given on the drawing, but this is not always the case. If the draftsman had given the included angle of 90 degrees, as shown at *B*, which would be another way of expressing it, the setting of the compound rest would, of course, be the same as before, or to 45 degrees, but the number of degrees marked on the drawing does not correspond with the angle to which the rest must be set. As another illustration, suppose the valve were to be turned to an angle of 30 degrees with the axis as shown at *C*. In this case the compound rest would not be set to 30 degrees but to 60 degrees, because in order to turn the work to an angle of 30 degrees, the rest must be 60 degrees from its zero position, as shown. From this it will be seen that the number of degrees marked on the drawing does not necessarily correspond to the angle to which the rest must be set, as the gradu-

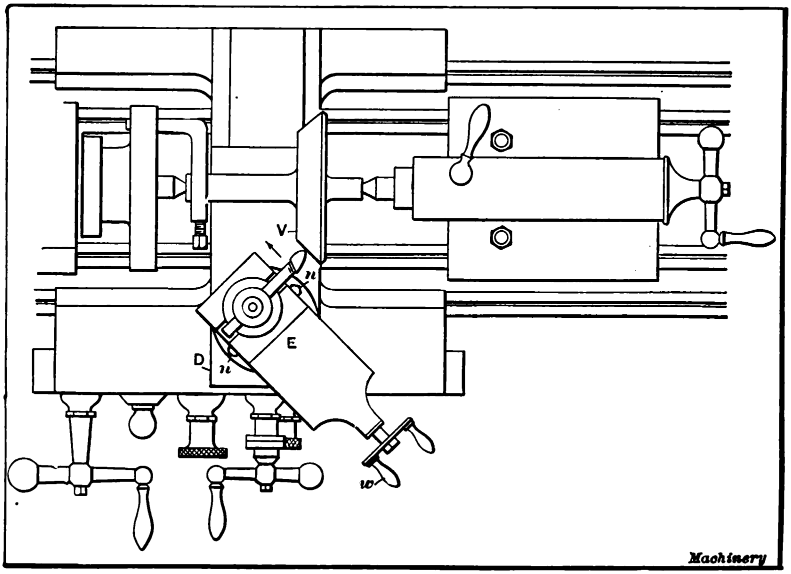


Fig. 14. Plan View showing method of Turning a Taper with the Compound Rest

ations on the rest show the number of degrees that it is moved from its zero position, which corresponds to the line *a*—*b*. The angle to which the compound rest should be set can be found, when the drawing is marked as at *A* or *C*, by subtracting the angle given from 90 degrees. When the included angle is given, as at *B*, subtract one-half the included angle from 90 degrees to obtain the required setting. Of course, when using a compound rest, the lathe centers are set in line as for straight turning, as otherwise the angle will be incorrect. The compound rest can also be used for boring taper holes by setting it to the angle that would give the right taper and then feeding the boring tool by hand, as when turning.

CHAPTER II

EXAMPLES OF CYLINDRICAL AND TAPER TURNING, BORING AND THREAD CUTTING

A practical example of lathe work which requires both straight and and taper turning, thread cutting, taper boring, reaming, and turning by the use of a mandrel, is illustrated in Fig. 15, which represents the drawing of an engine piston and rod. The various steps connected with turning these two parts in an ordinary engine lathe will be explained.

The piston is usually bored and reamed before the rod is turned so that the latter can afterward be fitted to it. The first turning opera-

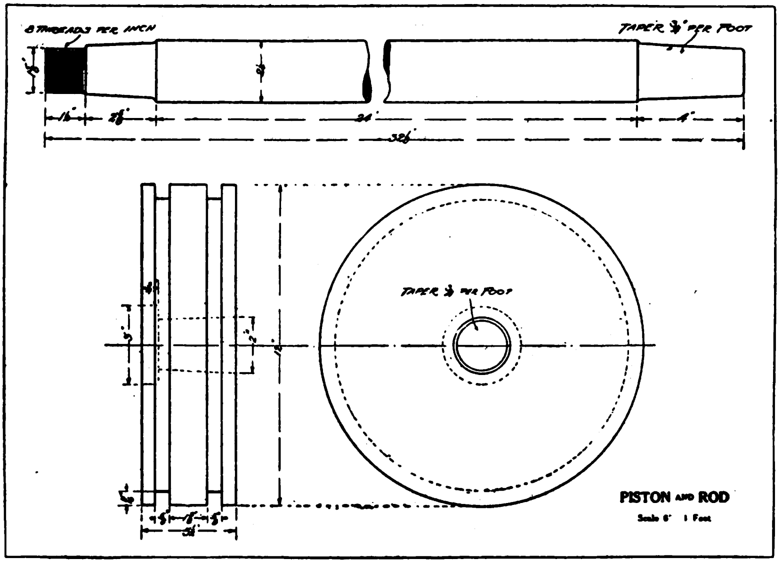


Fig. 15. Drawing of Engine Piston and Rod

tion consists in boring the hole into which the rod is to be fitted; therefore, the casting must be held either in a chuck *C*, as in Fig. 16, or on a faceplate if too large for the chuck. The side of the casting (after it has been "chucked") should run true and also the circumference, unless the cored hole for the rod is considerably out of center, in which case the work should be shifted to divide the error. The side of the casting for a short space around the hole is faced true with a round nose turning tool, after which the rough cored hole is bored with an ordinary boring tool *t*, and then it is finished with a reamer to exactly the right size and taper. If the lathe has a taper attachment, the hole can be bored to the right taper, by setting the attachment to the taper given on the drawing, which, in this example, is ¾ inch per foot. This is done, as will be recalled, by loosening nuts *N* and turning slide *S* until pointer *P* is opposite the ¾-inch division on the scale; the attachment is then ready, after bolt *H* and nuts *N* are tightened, and clamp *C* is fastened to the lathe bed. The hole is bored just as though it were straight, and as the carriage advances, the tool is gradually moved inward by the attachment.

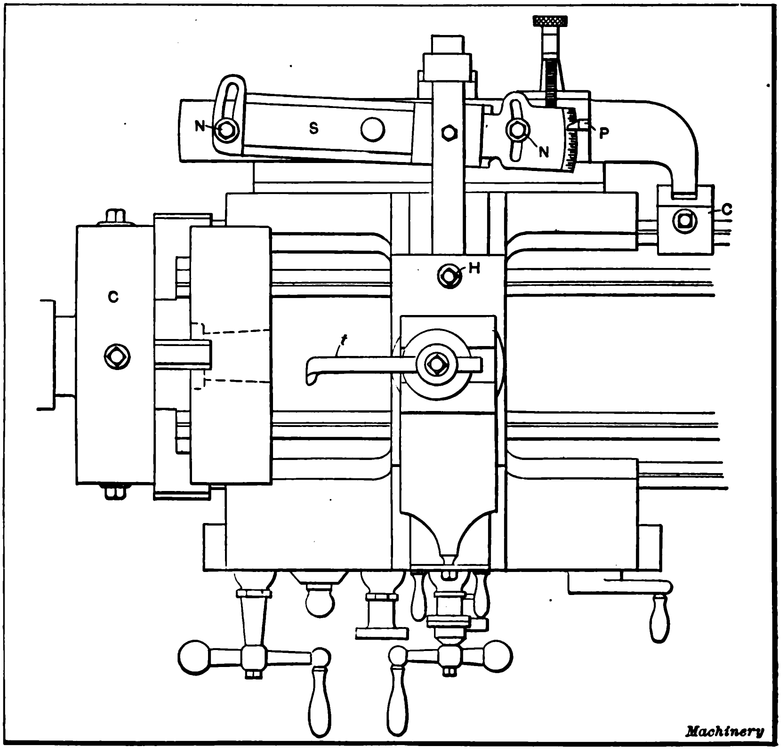


Fig. 16. Lathe with Taper Attachment arranged for Boring Taper Hole

If the lathe is without an attachment, the hole must be bored by using the compound rest. A convenient way to set the compound rest to the required angle is illustrated in Fig. 17. A bevel protractor *P* is first set to the taper of the reamer; this protractor is then placed against the finished spot on the casting as shown in Fig. 17, or against the faceplate, if the casting has not been chucked, and the compound rest is adjusted to the same angle as the protractor blade. The tool is set for boring by adjusting the carriage and cross-slide *D*, and it is fed by hand through the hole by compound slide *E*.

The hole is bored slightly under the finished size, and then a reamer is placed in the hole. The outer end of the reamer, which should have a deep center hole, is supported by the tailstock center. The lathe is run very slow for reaming and the reamer is fed into the hole by feeding out the tailstock spindle. The reamer can be kept from revolving with the work, either by attaching a heavy dog to the end or, if the end is squared, by the use of a wrench. A common method is to clamp a dog to the reamer shank, and then place the tool-rest beneath it to prevent rotation. If the shank of a tool is clamped to the toolpost so that the dog rests against it, the reamer will be prevented from slipping off the center as it tends to do; with this arrangement, the carriage is gradually moved along as the tailstock spindle is fed outward. A reamer of the type illustrated at *B*, Fig. 18, is fed in until the stop collar *S* comes against the finished side of the casting. By having this stop, the holes in any number of

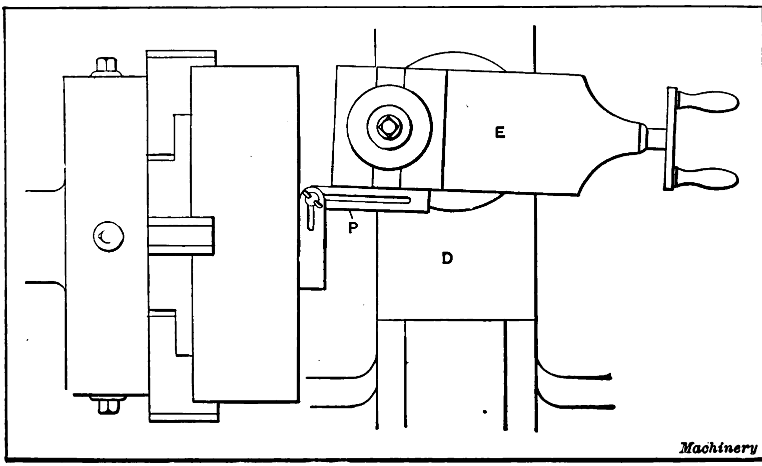


Fig. 17. Setting Compound Rest to Required Angle by using Bevel

pistons can be reamed to the same size. If a plain reamer *A* were used, the hole would probably be tested by inserting a plug gage.

After the reaming operation, the casting is removed from the chuck and a taper mandrel is driven into the hole for turning the outside of the piston. This mandrel should run true on its centers, as otherwise the outside surface of the piston will not be true with the bored hole. The mandrel *M* and the casting are next mounted between the lathe centers as shown in Fig. 19, after the chuck has been replaced with a faceplate. The driving dog *D*, especially for large work of this kind, should be heavy and stiff, because light flexible clamps or dogs vibrate and frequently cause chattering. For such heavy work it is also preferable to drive at two points on opposite sides of the faceplate, but the driving pins must be carefully adjusted to secure a uniform bearing on both sides. The outside of the piston might be turned either to the diameter given on the drawing, or be fitted to the cylinder of the engine for which the piston is intended. When turning work of this diameter, it must revolve quite slowly as otherwise the turning tool will be quickly dulled, and it is for such large work that the slow speeds obtained by driving through the back-gears are used. Ordinarily a piston casting could be reduced to finished diameter by taking one roughing and one light finishing cut, though

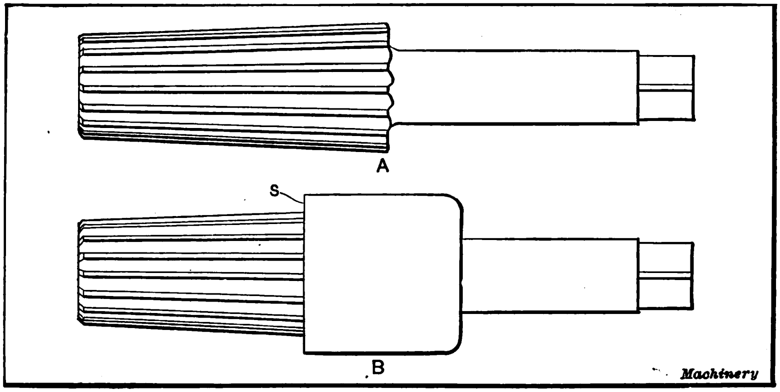


Fig. 18. Plain Reamer—Reamer with Stop Sleeve

this would depend, of course, on the diameter of the rough casting.

After turning the outside, grooves for the packing rings are laid out, as shown at *A*, Fig. 20, by scribing arcs from a central point *a*, that are the same distance apart as the grooves. The dimensions are

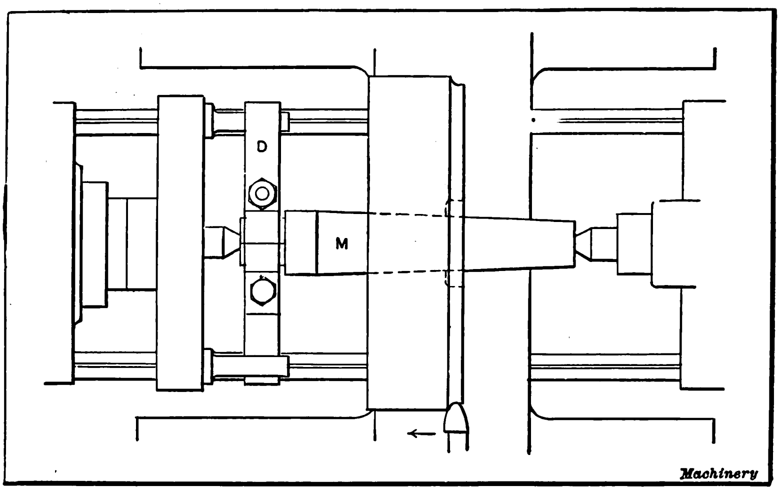


Fig. 19. Piston Mounted on Mandrel, in Position for Turning

obtained from the drawing, and the lines should be marked by light punch marks as shown. One method of cutting these grooves would be to use a square-nosed tool *t* (similar in shape to a parting tool) for turning them to depth, and side tools for finishing the sides. Grooves that are quite wide would be formed by first taking a cut on each side and then turning away the central part, as shown at *B*. The grooves should then be finished to the required width either by using right- and left-hand side tools, or a "square nose" ground to the right size. The width of the grooves should be exactly the same, and ordinarily they are fitted to some form of gage *g*. This particular style is double-ended, the upper end being used to measure the packing rings that fit into the grooves. When the grooves are finished, the outside of the piston is filed to make it smooth.

The final operation is to finish the pocket for the rod nut, which can be done by using a bent square-nosed tool t_1. It may be necessary to grind part of the under side of this tool away to provide

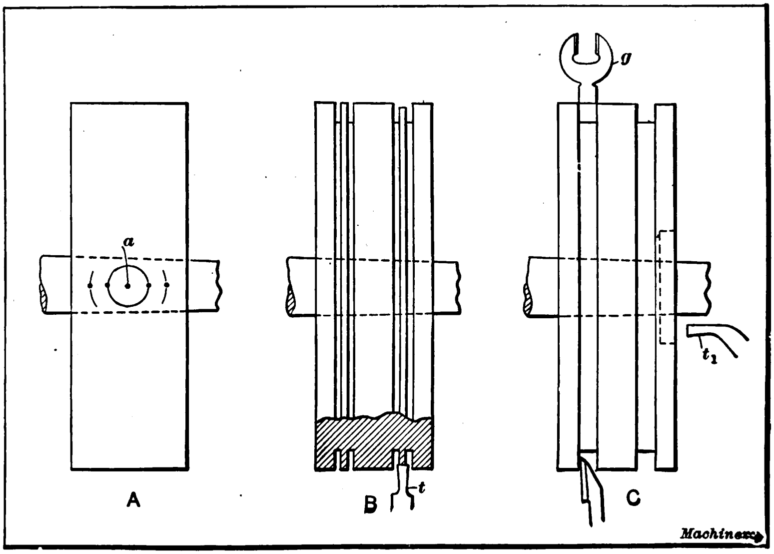


Fig. 20. Successive Steps in Turning Packing-ring Grooves

clearance, or in other words, to make a kind of special tool that would be kept for this particular job.

The foregoing method of machining a piston is one that would ordinarily be followed when using a standard engine lathe, and it would, perhaps, be as economical as any if only one piston were being made; but where such work is done in large quantities, time could be saved by proceeding in a different way. For example, the boring and reaming operation could be performed much faster in a turret lathe, which is a type designed for just such work, but a turret lathe cannot be used for as great a variety of work as a lathe of the regular type. There are also many other classes of work that can be turned more quickly in special types of machines, but as more or less time is required for arranging these special machines and often special tools have to be made, the ordinary lathe is frequently indispensable when only a few parts are needed; in addition, it is better adapted to some turning operations than any other machine.

Turning the Piston-rod

The stock for the piston-rod is cut off to the right length (probably in a hacksaw machine), and the ends are centered. The work is then placed between the lathe centers with a driving dog *D* (Fig. 21) attached to the faceplate end, and the tailstock center, after being oiled, is adjusted rather snugly but not tight enough to prevent a free rotary movement of the work. The body of the rod is first rough turned say 1/16 inch above the finished size, the cut being continued until the tool is near the driving dog. Light punch marks *a* and *b* are then made on the rod to mark the location of the shoulders or the length of the rod body which, in this case, is 24 inches. The

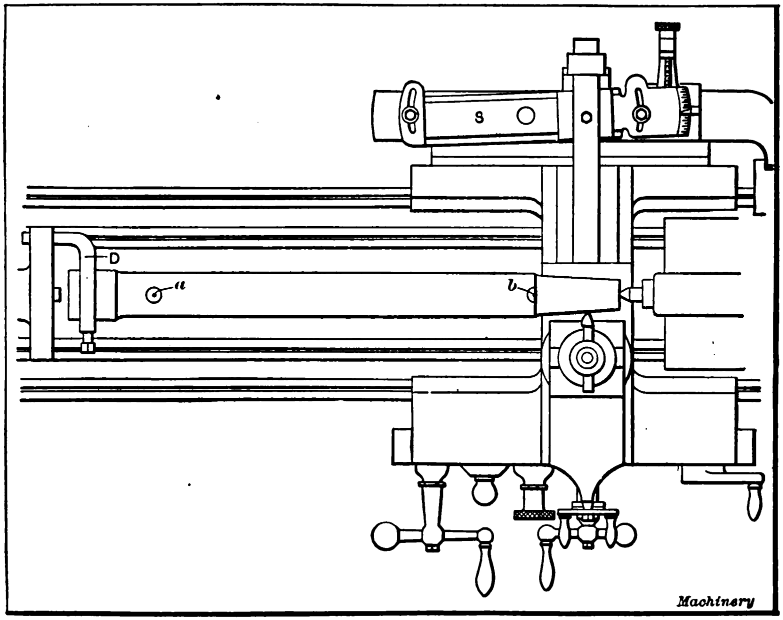


Fig. 21. Taper Attachment Set for Turning Taper End of Rod

marks should also be the right distance from the ends. The right-hand mark is laid out for the crosshead end which is to be fitted first. The taper attachment is next set to turn a taper of ¾ inch per foot, as marked on the drawing. While this taper corresponds to the taper of the hole in the piston, slide *S* will have to be re-set to the ¾-inch division on the opposite side of the central zero mark (see Fig. 11, Chapter I) because the taper of the hole decreased in size during the boring operation whereas the rod is smallest at the beginning of the cut, so that the tool must move outward rather than inward as it advances. The taper part is turned practically the same as a cylindrical part; that is, the power feed is used and, as the carriage moves along the bed, the tool is gradually moved outward by the taper attachment. If the rod is being fitted directly to the crosshead, as is usually the case, the approximate size of the taper end could be determined by calipering, the calipers being set to the size of the hole at a point 4 inches (in this case) from the shoulder or face side. If the crosshead was bored originally to fit a standard plug gage, the taper on the rod could be turned with reference to this gage, but, whatever the method, the taper should be tested before turning too close to the finished size. The test is made by removing the rod from the lathe and driving it tightly into the crosshead. This shows how near the taper is to size, and when the rod is driven out, the bearing marks show whether the taper is exactly right or not. If the rod could be driven in until the shoulder is say 1/8 inch from the crosshead face, it would then be near enough to finish to size by filing. When filing, the lathe is run much faster than for turning, and the most filing should be done where the bearing marks are the heaviest, to distribute the bearing throughout the length of the taper. Care should be taken when driving the rod in or out, to protect the center-holes in the ends by using a "soft" hammer or by holding a piece of soft metal against the driving end.

After the crosshead end is finished, the rod is reversed in the lathe for turning the piston end. The dog *D* is clamped to the finished

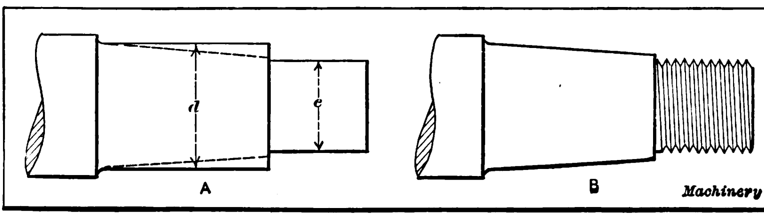


Fig. 22. Final Operations on Piston Rod

end, preferably over a piece of sheet copper to prevent the surface from being marred, and the end is then rough turned as at *A*, Fig. 22, diameter *d* being made slightly greater than the largest diameter of the taper, and *e* equal to the diameter of the thread. The attachment is then engaged and the taper part turned to the same taper as the opposite end, as called for on the drawing. When turning this end, either the piston reamer or the finished hole in the piston can be calipered. The size and angle of the taper are tested by driving the rod into the piston, and the end should be fitted so that by driving tightly, the shoulder will just come up against the finished face of the piston. When the taper is finished, the attachment is disengaged and a finishing cut is taken over the body of the rod with a sharp tool and rather fine feed to obtain a smooth surface.

The next and final turning operation is that of threading the end as at *B*. As there are eight threads per inch (see drawing, Fig. 15) the lathe is geared for cutting that number, and the thread is cut as explained in Part I of this treatise. (See Chapter VIII, MACHINERY'S Reference Book, No. 91.) The final operation consists in filing and polishing the body of the rod, the file being used first to take off the ridges left by the tool, and then emery cloth to polish the surface.

CHAPTER III

CUTTING SPEEDS AND FEEDS

In all turning operations there are two very important questions that must be considered: One has to do with the cutting speed that is used, and the other relates to the feed of the tool and depth of the cut. The cutting speed is the number of feet per minute that the tool point passes over, or practically speaking, it is equivalent to the length of a chip which would be turned in one minute. The term cutting speed should not be confused with revolutions per minute, because the cutting speed depends not only on the speed of the work but also on its diameter. The feed of a tool is the amount it moves across the surface of the work for each revolution; that is, when turning a cylindrical piece, the feed is the amount that the tool moves sidewise for each revolution of the work. Evidently the time required for turning is governed largely by the cutting speed, the feed, and the depth of the cut; therefore, these elements should be carefully considered. It is impossible to give any definite rule for determining either the speed, feed, or depth of cut, because these must be varied to suit existing conditions. We shall, however, point out some of the underlying principles which must be considered in determining the proper speed and feed.

TABLE OF CUTTING SPEEDS FOR TURNING STEEL

Depth of Cut in Inches	Feed in Inches	Cutting Speed in Feet per Minute for a Tool which is to last 1 Hour and 30 Minutes before Regrinding		
		Soft Steel	Medium Steel	Hard Steel
	$\frac{1}{64}$	476	238.0	108.0
$\frac{3}{32}$	$\frac{1}{32}$	325	162.0	73.8
	$\frac{1}{16}$	222	111.0	50.4
	$\frac{3}{32}$	177	88.4	40.2
	$\frac{1}{64}$	352	176.0	80.0
	$\frac{1}{32}$	240	120.0	54.5
$\frac{3}{16}$	$\frac{1}{16}$	164	82.0	37.3
	$\frac{3}{32}$	131	65.5	29.8
	$\frac{1}{8}$	112	56.0	25.5
	$\frac{1}{64}$	264	132.0	60.0
$\frac{3}{8}$	$\frac{1}{32}$	180	90.2	41.0
	$\frac{1}{16}$	122	61.1	27.8

The cutting speed is governed principally by the hardness of the metal to be turned; the kind of steel of which the turning tool is made; the shape of the tool and its heat treatment; the feed and depth of cut; the power of the lathe and also its construction. It is the durability of the turning tool or the length of time that it will turn effectively without grinding, that limits the cutting speed; and the hardness of the metal being turned combined with the quality of the tool, are the two factors which largely govern the time that a tool can be used before grinding is necessary. The cutting speed for very soft steel or cast iron can be three or four times faster than the speed for hard steel or hard castings, but whether the material is hard or soft, the kind of tool to use must also be considered as the speed for a tool made of ordinary carbon steel will have to be much slower than for a tool made of modern "high-speed" steel.

When the cutting speed is too high, even though high-speed steel is used, the point of the tool is softened to such an extent by the heat resulting from the pressure and friction of the chip, that the cutting edge is ruined in too short a time. On the other hand, when the speed is too slow, the heat generated is so slight as to have little

TABLE OF CUTTING SPEEDS FOR TURNING CAST IRON

Depth of Cut in Inches	Feed in Inches	Cutting Speed in Feet per Minute for a Tool which is to last 1 Hour and 30 Minutes before Regrinding		
		Soft Cast Iron	Medium Cast Iron	Hard Cast Iron
$\frac{3}{32}$	$\frac{1}{32}$	169.0	84.6	49.4
	$\frac{1}{16}$	122.0	61.2	35.7
	$\frac{1}{8}$	86.4	43.2	25.2
	$\frac{3}{16}$	70.1	35.1	20.5
$\frac{3}{16}$	$\frac{1}{32}$	137.0	63.6	40.1
	$\frac{1}{16}$	99.4	49.7	29.0
	$\frac{1}{8}$	70.1	35.0	20.5
	$\frac{3}{16}$	56.8	28.4	16.6
$\frac{3}{8}$	$\frac{1}{32}$	111.0	55.4	32.3
	$\frac{1}{16}$	80.0	40.0	23.4
	$\frac{1}{8}$	56.4	28.2	16.5
	$\frac{3}{16}$	45.8	22.9	13.4

effect and the tool point is dulled by being slowly worn or ground away by the action of the chip. A tool operating at such a low speed can, of course, be used a comparatively long time without re-sharpening, but this is more than offset by the fact that too much time is required for removing a given amount of metal when the work is revolving so slowly. Generally speaking, the speed should be such that a fair amount of work can be done before the tool requires regrinding. Evidently it would not pay to grind a tool every few minutes in order to maintain a high cutting speed; neither would it be economical to use a very slow speed and waste considerable time in turning, just to save the few minutes required for grinding. For example, if a number of roughing cuts had to be taken over a heavy rod or shaft, time might be saved by running at such a speed that the tool would have to be sharpened (or be replaced by a tool previously sharpened) when it had traversed half-way across the work; that is, the time required for sharpening or changing the tool would be short as compared with the gain effected by the high work speed.

On the other hand, it might be more economical to run a little slower and take a continuous cut across the work with one tool.

Sometimes the work speed cannot be as high as the tool will permit, because of the chattering that often results when the lathe is old and not massive enough to absorb the vibrations, or when there is unnecessary play in the working parts. The shape of the tool used also effects the work speed, and as there are so many things to be considered, the proper cutting speed is best determined by experiment. The two accompanying tables, giving cutting speeds for hard, medium, and soft steel and cast iron will be found useful, in a general way, in determining the most economical speed. These tables represent a few of the experiments conducted by Mr. Fred W. Taylor, and the figures given are based on the use of a tool correctly ground and made of a good grade of high-speed steel, properly heat treated.

It will be noted that the cutting speed is much slower for cast iron than for steel, and also that the feed and depth of cut have a very decided effect on the speed. Cast iron is cut with less pressure or resistance than soft steel, but the slower speed for cast iron is probably because the pressure of the chip is concentrated closer to the cutting edge, combined with the fact that cast iron wears the tool faster than steel, the wear occurring close to the cutting edge.

The number of revolutions required to give any desired cutting speed can be found by multiplying the cutting speed, in feet per minute, by 12 and dividing the product by the circumference of the work in inches. Expressing this as a formula we have

$$R = \frac{C \times 12}{\pi d}$$

in which

R = revolutions per minute;
C = the cutting speed in feet per minute;
π = 3.1416; and
d = the diameter in inches.

For example if a cutting speed of 60 feet per minute is wanted and the diameter of the work is 5 inches, the required speed for the work would be found as follows:

$$R = \frac{60 \times 12}{3.1416 \times 5} = 46 \text{ revolutions per minute.}$$

If the diameter is simply multiplied by 3 and the fractional part is cmitted, the calculation can easily be made, and the result will be close enough for practical purposes. In case the cutting speed, for a given number of revolutions and diameter, is wanted, the following formula can be used:

$$C = \frac{R \pi d}{12}$$

Of course, machinists that operate lathes do not know, ordinarily, what cutting speeds in feet per minute are used for different classes of work, but are guided entirely by past experience.

The amount of feed and depth of cut also vary, like the cutting speed, with different conditions. Ordinarily coarser feeds and a greater depth of cut can be used for cast iron than for soft steel because cast iron offers less resistance to turning, but in any case, with a given depth of cut, metal can be removed more quickly by using a coarse feed and the necessary slower speed, than by using a fine feed and the accompanying higher speed. When the turning operation is simply to remove metal, coarse feeds and deep cuts are taken, but sometimes the cut must be comparatively light, either because the work is too fragile and springy to withstand the strain of a heavy cut, or the lathe has not sufficient pulling power. The difficulty with light slender work is that a heavy cut may cause the part being turned to bend under the strain, thus causing the tool to gouge in which would probably result in spoiling the work. Steady-

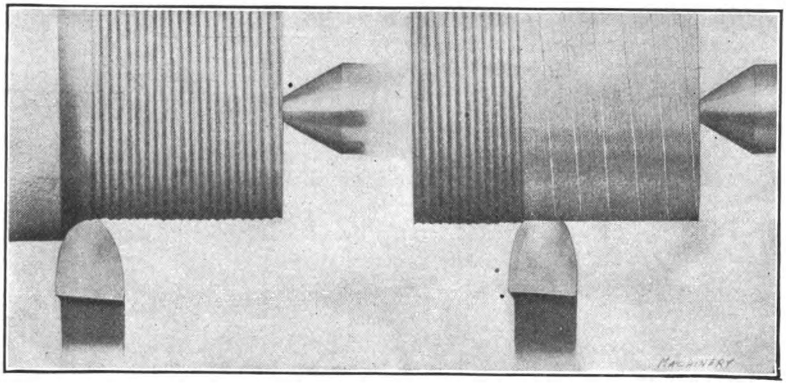


Fig. 23. Roughing Cut—Light Finishing Cut and Coarse Feed

rests can often be used to prevent flexible parts from springing, but there are many kinds of light work to which the steadyrest cannot be applied to advantage.

The amount of feed to use for a finishing cut might, properly, be either fine or coarse. Ordinarily fine feeds are used for finishing steel, especially if the work is at all flexible, but a finishing cut in cast iron is often accompanied by a coarse feed. Fig. 23 illustrates the feeds that are often used when turning cast iron. The view to the left shows a deep roughing cut and the one to the right, a finishing cut. By using a broad flat cutting edge set parallel to the tool's travel, and a coarse feed for finishing, a smooth cut can be taken in a comparatively short time. Some castings which are close to the finished size in the rough, can be finished by taking one cut with a broad tool, provided the work is sufficiently rigid. It is not always practicable to use these broad tools and coarse feeds, as they sometimes cause chattering, and when used on steel, a broad tool tends to gouge or "dig in" unless the part being turned is rigid. Heavy steel parts, however, are sometimes finished in this way. Much of the work that is turned, at the present time, is afterwards finished in a grinding machine so that often it is not necessary to take a finishing cut to secure a smooth surface.

CHAPTER IV

THREADS OF DIFFERENT FORM AND METHODS OF CUTTING

Three forms of threads which are very common in this country are shown in Fig. 24; these are the V-thread *A*, the U. S. standard *B*, and the square thread *C*. The shapes of these threads are shown by the sectioned parts. The V-thread has straight sides which make an angle of 60 degrees with each other and a like angle with the axis of the screw. The U. S. standard thread is similar to the V-thread except that the top of the thread and bottom of the groove is left flat, as shown, and the width of these flats is made equal to 1/8 of the pitch. The square thread is square in section, the width *a*, depth *b* and space *c* being all equal. All of these threads are right-hand, which means that the grooves wind around to the right so that a nut will have to be turned toward the right to enter it on the thread. A left-hand thread winds in the other direction, as shown at *D*, and a nut is screwed on by turning it to the left. Threads, in addition to being right- and left-handed, are single, as at *A*, *B*, *C*, and *D*, double, as at *E*, and triple, as at *F*, and for certain purposes quadruple threads are employed. A double thread is different from a single thread in that it has two grooves, starting diametrically opposite, whereas a triple thread has three grooves cut as shown at *F*. The object in having these multiple threads is to obtain an increase in lead without weakening the screw. For example, the threads shown at *C* and *E*, have the same pitch, but the lead of the double-threaded screw is twice that of the one with a single thread so that a nut would advance twice as far in one revolution, which is often a very desirable feature. To obtain the same lead with a single thread, the pitch would have to be double, thus giving a much coarser thread, which would weaken the screw, unless its diameter were increased. (The lead is the distance *l* that one thread advances in a single turn, or the distance that a nut would advance in one turn, and it should not be confused with the pitch *p*, which is the distance between the centers of adjacent threads. The lead and pitch of a single thread are the same.)*

Cutting a U. S. Standard Thread

A U. S. standard thread is cut in the same way described for a V-thread, in Chapter VIII, MACHINERY's Reference Series No. 91, but as it has a different form, a tool of corresponding shape is used. This tool is first ground to an angle of 60 degrees, as it would be for cutting a V-thread, and then the point is made flat as shown in Fig. 25. As the width of this flat is equal to 1/8 of the pitch, it varies, of course, for different pitches. By using a gage like the one shown at *G*, the tool can easily be ground for any pitch, as the notches around the

*See also MACHINERY'S Reference Book No. 31: "Screw Thread Tools and Gages."

periphery of the gage are marked for different pitches and the tool-point is fitted into the notch corresponding to the pitch wanted.

When the cutting the thread, the tool is set square with the blank, and a number of successive cuts are taken, the tool being fed in until the width *w* of the flat at the top of the thread is equal to the width at the bottom. The thread will then be the right size provided the outside diameter *D* is correct. As it would be difficult to measure

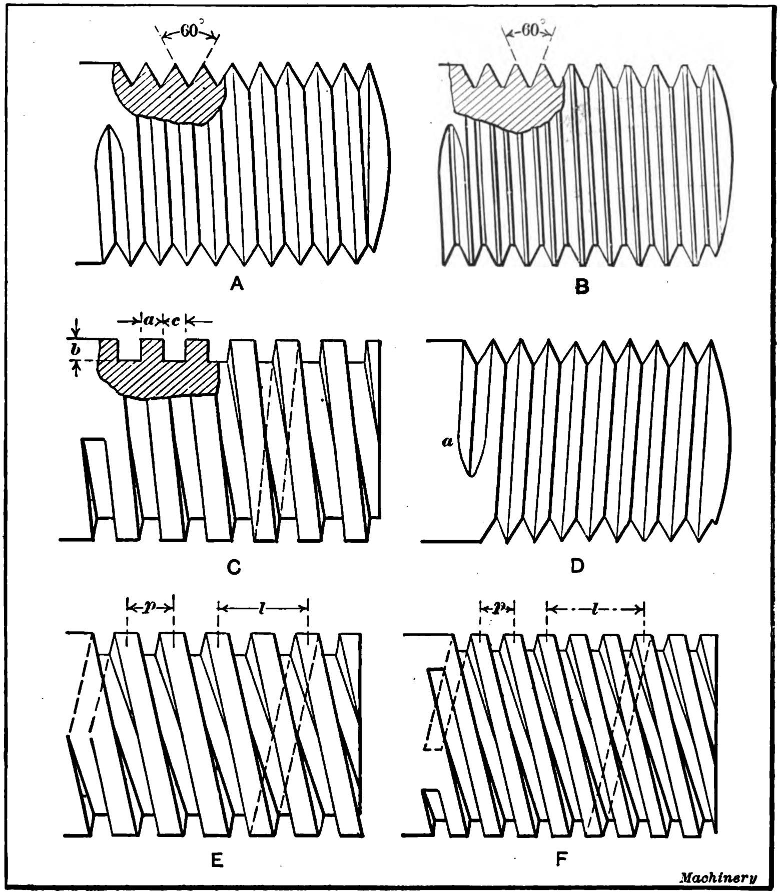


Fig. 24. (A) V-thread. (B) U. S. Standard Thread. (C) Square Thread. (D) Left-hand Thread. (E) Double Square Thread. (F) Triple Square Thread

the width of this flat accurately, the thread can be tested by screwing a standard nut over it if a standard thread is being cut. If it is being fitted to a tapped hole, the tap itself is a very convenient gage to use, the method being to caliper the tap and then compare its size with the work. Calipers or micrometers, such as illustrated in Fig. 60 (Part I), can be used.

A good method of cutting a U. S. standard thread to a given size is as follows: First turn the outside of the blank accurately to diameter D, and then turn a small part on the end to diameter r of the thread at the root. The finishing cut for the thread is then taken with the tool point set to just graze diameter r. If ordinary calipers were set to diameter r and measurements taken in the thread groove, the size might be incorrect owing to the angularity of the groove, which makes it necessary to hold the calipers at an angle when meas-

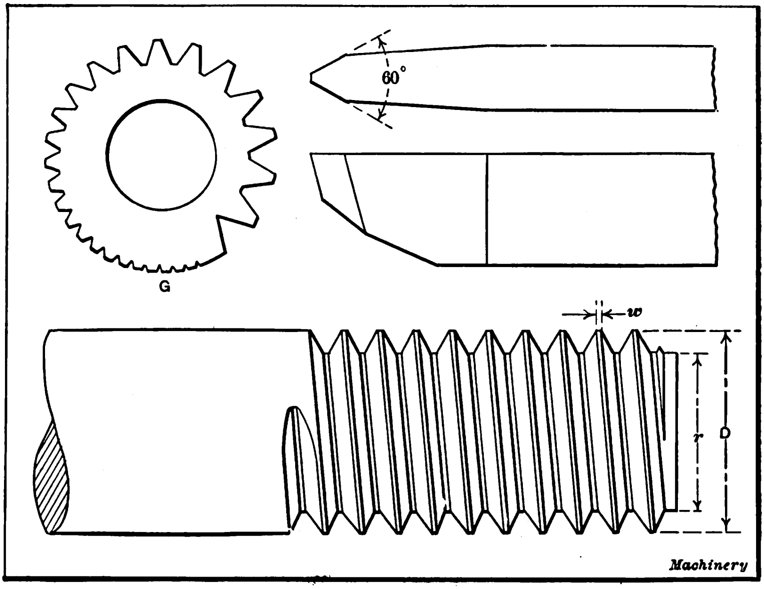


Fig. 25. U. S. Standard Thread, Thread Tool, and Gage

uring. A table, giving root diameters for various pitches, is convenient to have, but this diameter can be found by the following formula:

$$D - \left(\frac{1.299}{N}\right) = r$$

in which D equals outside diameter, N the number of threads per inch, and r the root diameter. The number 1.299 is a constant that is always used.

Cutting a Left-hand Thread

The only difference between cutting left-handed and right-handed threads in the lathe, is in the movement of the tool with relation to the work. When cutting a right-hand thread, the tool moves from right to left, but this movement is reversed for left-hand threads because the thread winds around in the opposite direction. To make the carriage travel from left to right, the lead-screw is rotated backwards by means of reversing gears a and b (Fig. 26) located in the headstock. Either of these gears can be engaged with the spindle gear by changing the position of lever R. When gear a is in engagement, as shown, the drive from the spindle to gear c is through gears a and b, but when lever R is raised thus shifting b into mesh, the drive is direct and the direction of rotation is reversed. The thread is cut by starting the tool at *a*, Fig. 24, instead of at the end.

Cutting a Square Thread

The form of tool used for cutting a square thread is shown in Fig. 27. The width *w* is made equal to one-half the pitch of the thread to be cut and the end *E* is at an angle with the shank, which corresponds to the inclination *x–y* of the threads. This angle *A* depends on the diameter of the screw and the lead of the thread; it can be determined graphically by laying off a line *a–b* equal to the circumference of the screw to be cut, and a line *b–c*, at right angles, equal to the lead of the thread. The angle α between lines *a–b* and

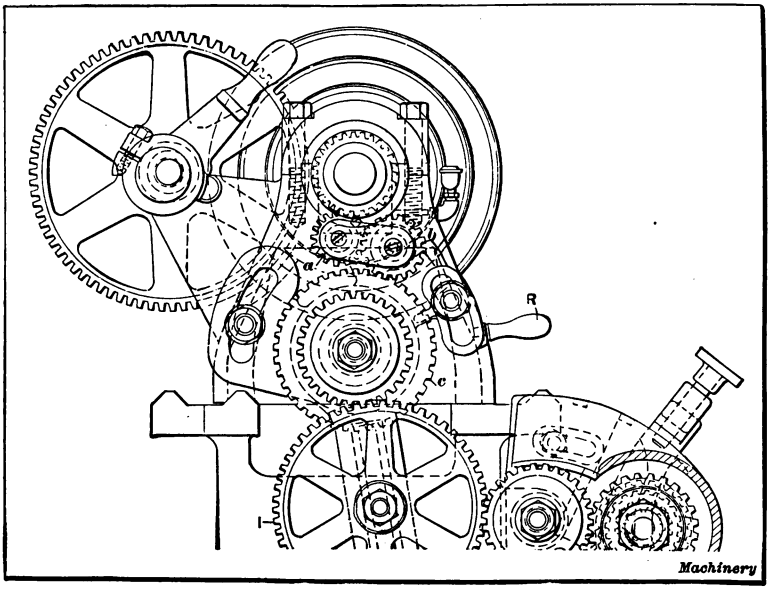


Fig. 26. End View of Lathe Headstock

a–c will be the required angle *A*. It is not necessary to have this angle accurate, ordinarily, as it is simply to prevent the tool from binding against the sides of the thread. The end of a square thread tool is shown in section to the right to illustrate its position with relation to the threads. The sides *e* and e_1 are ground to slope inward, as shown, to provide additional clearance.

When cutting multiple threads, which, owing to their increased lead, incline considerably with the axis of the screw, the angles for each side of the tool can be determined independently as follows: Lay off *a–b* equal to the circumference of the thread, as before, to obtain the required angle *f* of the rear or following side e_1; the angle *l* of the opposite or leading side is found by making *a–b* equal to the circumference at the root of the thread. The tool illustrated is for cutting right-hand threads; if it were intended for a left-hand thread, the end, of course, would incline in the opposite direction. The square thread is cut so that the depth *d* is equal to the width.

Cutting Multiple Threads

When a multiple thread is to be cut, as a double or triple thread, the lathe is geared with reference to the number of single threads to the inch. For example, the lead of the double thread, shown at *B*, Fig. 28, is one-half inch, or twice the pitch, and the number of single threads to the inch equals $1 \div \frac{1}{2} = 2$.

Therefore, the lathe is geared for cutting two threads per inch. The first cut is taken just as though a single thread were being cut, leaving the work as shown at *A*. When this cut is finished the work is turned one-half a revolution (for a double thread) without disturbing the position of the lead-screw or carriage, which brings the tool midway between the grooves of the single thread as indicated by

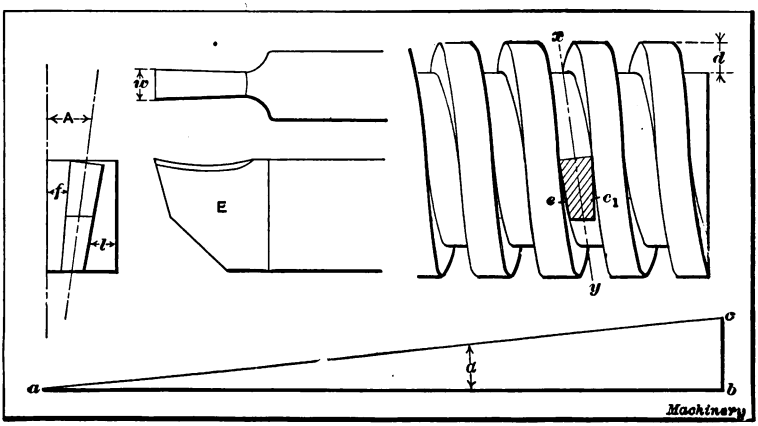


Fig. 27. End of Square Thread Tool, and Graphic Method of Determining Helix Angle of Thread

dotted lines. The second groove is then cut, producing a double thread as shown at *B*. In the case of a triple thread, the work would be indexed one-third of a revolution after turning the first groove, and then another third revolution to locate the tool for cutting the last groove. Similarly, for a quadruple thread, it would be turned one-quarter revolution after cutting each successive groove or thread.

There are different methods of indexing work when cutting multiple threads. Some machinists, when cutting a double thread, simply remove the work from the lathe and turn it one-half a revolution by placing the tail of the driving dog in the opposite slot of the faceplate. This is a very simple method, but if the slots are not directly opposite or 180 degrees apart, the last thread will not be central with the first. Another and better method is to disengage the idler gear from the gear on the stud, turn the spindle and work one-half, or one-third, of a revolution, as the case might be, and then connect the gears. For example, if the stud gear had 96 teeth, the tooth meshing with the idler gear would be marked with chalk, the gears disengaged, and the spindle turned until the chalked tooth had made the required part of a revolution, which could be determined by counting the teeth. When this method is used, the number of teeth in the stud gear must be evenly divisible by two if a double thread is being cut, or by three for a triple thread. If the stud is not geared to the spindle so that each makes the same number of revolutions, the ratio of the gearing must be considered.

Special faceplates are sometimes used for multiple thread cutting, that enable work to be easily and accurately indexed. One of these is illustrated in Fig. 29, and consists of two parts *A* and *B*, part *A* being free to rotate in relation to *B* when bolts *C* are loosened. The driving pin for the lathe dog is attached to plate *A*. When one

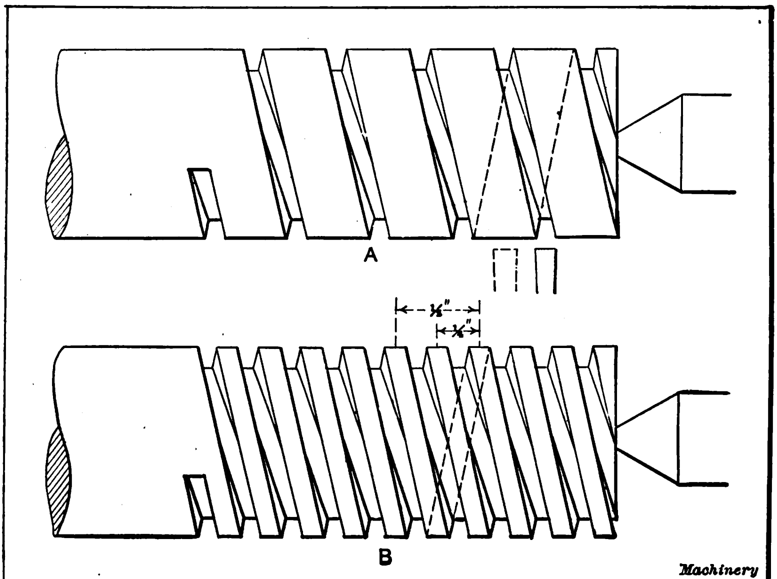


Fig. 28. Views illustrating how a Double Square Thread is Cut

groove of a multiple thread is finished, bolts *C* are loosened and plate *A* is turned around an amount corresponding to the type of thread being cut. The periphery of plate *A* is graduated in degrees, as shown, and for a double thread it will be turned one-half revolution or 180 degrees, for a triple thread 120 degrees, etc. This is a very good arrangement where multiple thread cutting is done frequently.

Taper Threading

When a taper thread is to be cut, the tool should be set square with axis *a—a* as at *A*, Fig. 30, and not by the tapering surface as at *B*. If there is a cylindrical part, the tool can be set as indicated by the dotted lines. All taper threads should be cut by the use of taper attachments. If the tailstock is set over to get the required taper, the curve of the thread will not be true. or in other words the thread will not advance at a uniform rate; this is referred to by machinists as a "drunken thread."

Internal Threading

Internal threading, or cutting threads in holes, is an operation performed on work held in the chuck or on a faceplate, as for boring. The tool used is similar to a boring tool except that the working end is shaped to conform to the thread to be cut. An internal threading

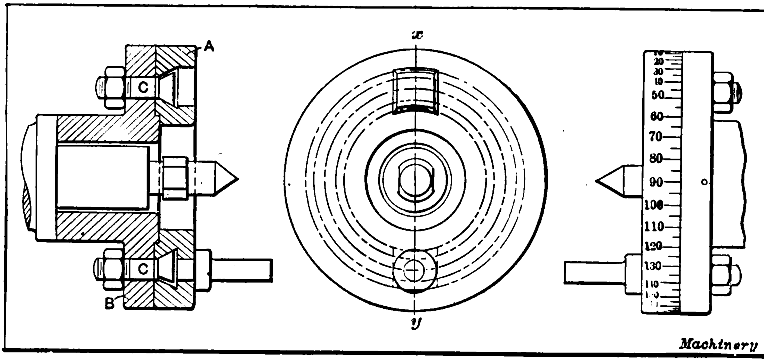


Fig. 29. Indexing Faceplate used for Multiple Thread Cutting

tool for cutting a V-thread is shown in Fig. 31. The method of procedure, when cutting an internal thread, is similar to that for outside work, as far as handling the lathe is concerned. The hole to be threaded is first bored to the root diameter of the thread that is to fit into it. The tool-point is then set square by holding a gage *G* against the true side of the work and adjusting the point to fit the notch in the gage as shown. Very often the size of a threaded hole

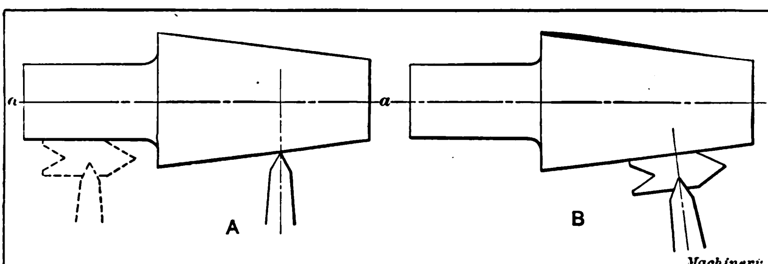


Fig. 30. Correct and Incorrect Positions of Tool for Taper Thread Cutting

can be tested by using as a gage the threaded part that is to fit into it. When making such a test, the tool is, of course, moved back out of the way. It is rather difficult to cut an accurate thread in a small hole, especially when quite deep, owing to the flexibility of the tool; for this reason threads are sometimes cut slightly under size with the tool, after which a tap with its shank end held straight by the tailstock center, is run through the hole. In such a case, the tap should be calipered and the thread made just small enough with the tool to give the tap a light cut. Small square-threaded holes are often finished in this way, and if a number of pieces are to be threaded, the use of a tap makes the holes uniform in size.

Stop for Thread Tools

When cutting a thread, it is rather difficult to feed in the tool just the right amount for each successive cut, because the tool is moved in before it feeds up to the work. A stop is sometimes used for threading which overcomes this difficulty. This stop consists of a screw which enters the tool slide and passes through a block clamped in front of the slide. The hole in the block through which the stop-screw passes is not threaded, but is large enough to permit the screw to move freely. When cutting a thread, the tool is set for the first cut and the

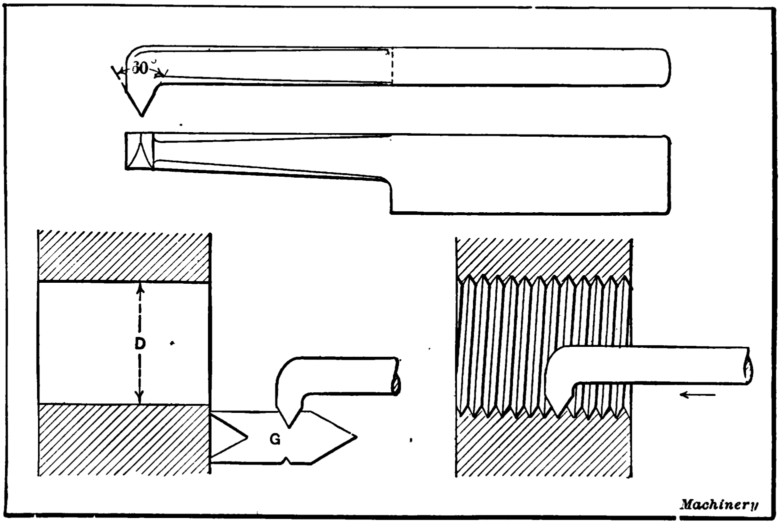


Fig. 31. Inside Thread Tool—Method of Setting and Using

screw is adjusted until the head is against the fixed block. After taking the first cut, the stop-screw is backed out, say one-half revolution, which allows the tool to be fed in far enough for a second cut. If this cut is about right for depth, the screw is again turned about one-half revolution and this is continued for each successive cut until the thread is finished. By using a stop of this kind, there is no danger of feeding the tool in too far as is often done when the tool is set by guess. If this form of stop is used for internal threading, the screw, instead of passing through the fixed block, is placed in the slide so that the end or head will come against the stop. This change is made because the tool is fed outward when cutting an internal thread.

Rivett-Dock Threading Tool

A special form of thread tool, which overcomes a number of disadvantages common to an ordinary single-point thread tool, is shown in Fig. 32. This tool has a circular-shaped cutter *C*, having ten teeth around its circumference, which, beginning with tooth No. 1, gradually increase in height, cutter No. 2 being higher than No. 1, etc. This cutter is mounted on a slide *S*, that is fitted to the frame *F*, and can be moved in or out by lever *L*. The hub of this lever has an eccentric stud which moves slide *S* and locks it when in the forward or cutting position. The action of the lever in moving the slide, engages the cutter with pawl *P*, thus rotating the cutter one tooth at a time and presenting a different tooth to the work for each movement of the lever. When the slide is moved forward, the heel or underside of the tooth which is in the working position, rests on a stop that takes the thrust of the cut. When the tool is in use, it is mounted on the tool-block of the lathe as shown in the illustration. The cutter is

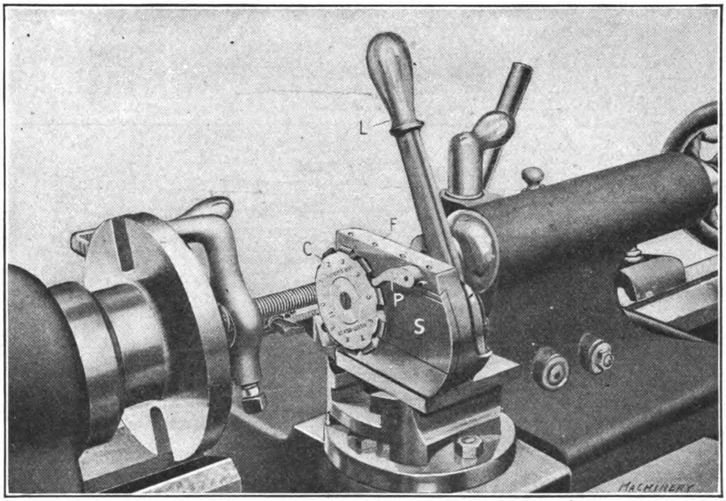


Fig. 32. Rivett-Dock Circular Threading Tool in Working Position

set for height by placing a tooth in the working position and setting the top level with the lathe center. The cutter is also set square with the work by using an ordinary square, and it is tilted slightly from the vertical to correspond with the angle of the thread to be cut, by adjusting frame *F*. At first a light cut is taken with lever *L* moved forward and tooth No. 1 on the stop. After this is completed, the lever is reversed which rotates the cutter one tooth, and the return movement places tooth No. 2 in the working position. This operation is repeated until the tenth tooth finishes the thread. It is often necessary, when using a single-point thread tool, to re-sharpen it before taking the finishing cut, but with a circular tool this is not necessary for by using the different teeth successively, the last tooth, which only takes finishing cuts, is kept in good condition. This tool has a micrometer adjustment which enables threads to be cut to the same size without the use of a gage.

CHAPTER V

TOOL GRINDING

In the grinding of lathe tools, there are three things of importance to be considered: First, the cutting edge of the tool (as viewed from the top) needs to be given a certain shape; second, there must be a sufficient amount of clearance; and third, tools, with certain exceptions, are ground with a backward slope or a side slope, or with a combination of these two slopes on that part against which the chip bears when the tool is in use.

Meaning of Terms Used in Tool Grinding

In Fig. 33 a few of the different types of tools which are used in connection with lathe work are shown. This illustration also

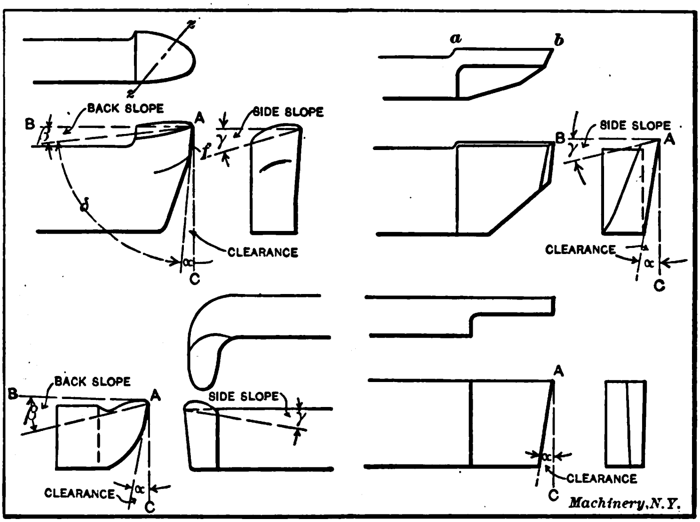


Fig. 33. Illustration showing the Meaning of Terms used in Tool Grinding as applied to Tools of Different Types

indicates the meaning of the various terms used in tool grinding. As shown, the clearance of the tool is represented by the angle α, the back slope is represented by the angle β, and the side slope by the angle γ. The angle δ for a tool without side slope, is known as the lip angle cr the angle of keenness. When, however, the tool has both back and side slopes, this lip angle would more properly be the angle between the flank f and the top of the tool, measured diagonally along a line z—z. It will be seen that the lines A—B and A—C from which the angles of clearance and back slope are measured, are parallel with the top and sides of the tool shank, respectively. For lathe tools, however, these lines are not necessarily located in this way when the tool is in use, as the height of the tool point with relation to the work center determines the position of these lines so that the *effective* angles of back slope, clearance and keenness are changed as the tool point is lowered or raised. The way the position of the tool effects these angles will be explained later.

While tools must, of necessity, be varied considerably in shape to adapt them to various purposes, there are certain underlying principles governing their shape which apply generally; so in what follows we shall not attempt to explain in detail just what the form of each tool used on the lathe should be, as it is more important to understand how the cutting action of the tool and its efficiency is affected when it is improperly ground. When the principle is understood, the grinding of tools of various types and shapes is comparatively easy.

Shape or Contour of Cutting Edge

In the first place we shall consider the shape or contour of the cutting edge of the tool as viewed from the top, and then take up the

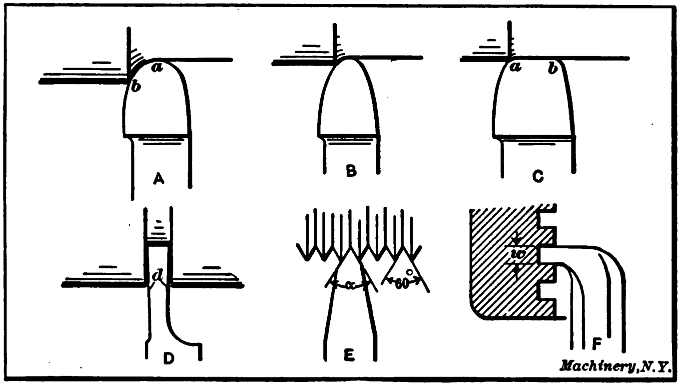


Fig. 34. Plan View of Lathe Turning and Threading Tools

question of clearance and slope, the different elements being considered separately to avoid confusion.

The contour of the cutting edge depends primarily upon the purpose for which the tool is intended. For example, the tool *A*, in Fig. 34, where a plan view of a number of different lathe tools is shown, has a very different shape from that of, say, tool *D*, as the first tool is used for rough turning, while tool *D* is intended for cutting grooves or severing a turned part. Similarly, tool *E* is V-shaped because it is used for cutting V-threads. Tools *A*, *B* and *C*, however, are regular turning tools, that is, they are all intended for turning plain cylindrical surfaces, but the contour of the cutting edges varies considerably, as shown. In this case it is the characteristics of the work and the cut that are the factors which determine the shape. To illustrate, tool *A* is of a shape suitable for rough turning large and rigid work, while tool *B* is adapted for smaller and more flexible parts. The first tool is well shaped for roughing because experiments have shown that a cutting edge of a large radius is capable of higher cutting speed than could be used with a tool like *B*, which has a smaller point. This increase in the cutting speed is due to the fact that the tool *A* removes a thinner chip for a given feed than tool *B*. Therefore, the speed may be increased without injuring the cutting edge to the same extent. If, however, tool *A* were to be used for turning a long and flexible part, chattering would result. Consequently, a tool *B* having a point with a smaller radius would be preferable, if not absolutely necessary. The character of the work also affects the shape of tools. The tool shown at *C* is used for taking light finishing cuts with a wide feed. Obviously, if the straight or flat part of the cutting edge is in line with the travel of the tool, the cut will be smooth and free from ridges, even though the feed is coarse, and by using a coarse feed the cut is taken in less time; but such a tool cannot be used on work that is not rigid, as chattering would result. Therefore, a smaller cutting point and a reduced feed would have to be employed. Tools with broad flat cutting edges and coarse feeds are often used for taking finishing cuts in cast iron, as this metal offers less resistance to cutting than steel, and is less conducive to chattering.

The shape of a tool (as viewed from the top) which is intended for a more specific purpose than regular turning, can be largely determined by simply considering the tool under working conditions. This point may be illustrated by the parting tool *D* which, as previously stated, is used for cutting grooves, squaring corners, etc. Evidently this tool should be widest at the cutting edge; that is, the sides *d* should have a slight amount of clearance so that they will not bind as the tool is fed into a groove. As the tool at *E* is for cutting a V-thread, the angle *a* between its cutting edges must equal the angle between the sides of a V-thread, or 60 degrees. The tool illustrated at *F* is for cutting inside square threads. In this case the width *w* should be made equal to one-half the pitch of the thread, and the sides should be given a slight amount of side clearance, the same as with the parting tool *D*. So we see that the outline of the tool, as viewed from the top, must conform to and be governed by its use.

Direction of Top Slope for Turning Tools

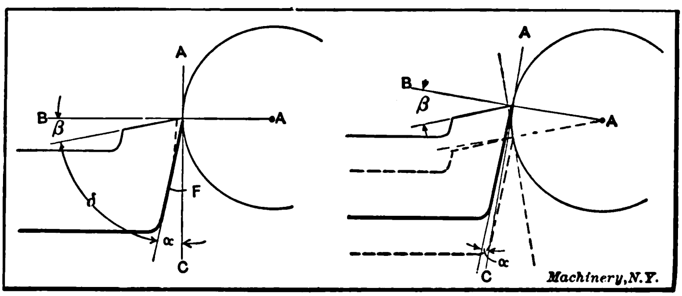
Aside from the question of the shape of the cutting edge as viewed from the top, there remains to be determined the amount of clearance that the tool shall have, and also the slope (and its direction) of the top of the tool. By the top is meant that surface against which the chip bears while it is being severed. It may be stated, in a general way, that the direction in which the top of the tool should slope should be away from what is to be the *working part* of the cutting edge. For example, the working edge of a roughing tool *A* (Fig. 34), which is used for heavy cuts, would be, practically speaking, between points *a* and *b*, or in other words, most of the work would be done by this part of the cutting edge; therefore the top should slope back from this part of the edge. Obviously, a tool ground in this way will have both a back and a side slope. When most of the work is done on the point or nose of the tool, as for example, with the lathe finishing tool *C* which takes light cuts, the slope should be back from the point or cutting edge *a–b*. As the side tool shown in

Fig. 33 does its cutting along the edge a–b, the top is given a slope back from this edge as shown in the end view. This point should be remembered, for when the top slopes in the right direction, less power is required for cutting. Tools for certain classes of work, such as thread tools, or those for turning brass or chilled iron, are ground flat on top, that is, without back or side slope.

Clearance for the Cutting Edge

Now, in order that the cutting edge may work without interference, it must have clearance; that is, the flank f (Fig. 33) must be ground to a certain angle α so that it will not rub against the work and make the cutting edge ineffective. This clearance should be just enough to permit the tool to cut freely. A clearance angle of eight or ten degrees is about right for lathe turning tools.

The back slope of a tool is measured from a line A–B which is parallel to the shank, and the clearance angle, from a line A–C at right angles to line A–B. These lines do not, however, always occupy this position with relation to the tool shank when the tool is in use.



Figs. 35 and 36. Illustrations showing how Effective Angles of Slope and Clearance change as Tool is raised or lowered

As shown in Fig. 35, the base line A–B for a turning tool in use, intersects with the point of the tool and center of the work, while the line A–C remains at right angles to the first. It will be seen then, that by raising the tool, as shown to the right (Fig. 36), the *effective* clearance angle α will be diminished, whereas lowering it, as shown by the dotted lines, will have the opposite effect.

A turning tool for brass or other soft metal, particularly where considerable hand manipulation is required, could advantageously have a clearance of twelve or fourteen degrees, as it would then be easier to feed the tool into the metal; but, generally speaking, the clearance for turning tools should be just enough to permit them to cut freely. Excessive clearance weakens the cutting edge and may cause it to crumble under the pressure of the cut.

Angle of Tool-point and Amount of Top Slope

The lip angle or the angle of keenness δ (Fig. 33) is another important consideration in connection with tool grinding, for it is upon this angle that the efficiency of the tool largely depends. By referring to the illustration it will be seen that this angle is governed by the clearance and the slope β, and as the clearance remains practically the same, it is the slope which is varied to meet different conditions. Now, the amount of slope a tool should have depends on the work for which it is intended. If, for example, a turning tool is to be used for roughing medium or soft steel, it should have a back slope of eight degrees and a side slope ranging from fourteen to twenty degrees, while a tool for cutting very hard steel should have a back slope of five degrees and a side slope of nine degrees. The reason for decreasing the slope and thus increasing the lip angle for harder metals is to give the necessary increased strength to the cutting edge to prevent it from crumbling under the pressure of the cut. The tool illustrated at *A*, Fig. 37, is much stronger than it would be if ground as shown at *B*, as the former is more blunt. If a tool ground as at *A*, however, were used for cutting very soft steel, there would be a greater chip pressure on the top and, consequently, a greater resistance to cutting, than if a keener tool had been employed; furthermore the cutting

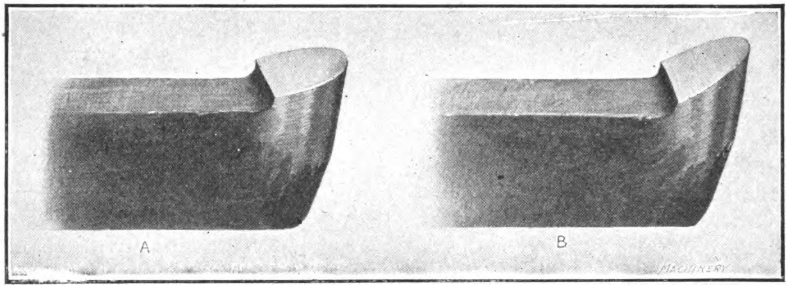


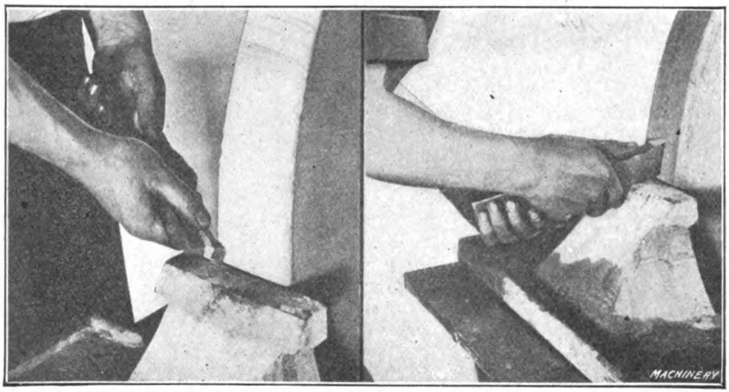
Fig. 37. (A) Blunt Tool for Turning Hard Steel. (B) Tool-point Ground to Give Keenness

speed would have to be lower, which is of even greater importance than the chip pressure; therefore, the lip angle, as a general rule, should be as small as possible without weakening the tool so that it cannot do the required work. In order to secure a strong and well-supported cutting edge, tools used for turning very hard metal, such as chilled rolls, etc., are ground with practically no slope and with very little clearance. Brass tools, while given considerable clearance, as previously stated, are ground flat on top or without slope; this is not done, however, to give strength to the cutting edge, but rather to prevent the tool from gouging into the work, which it is likely to do if the part being turned is at all flexible and the tool has top slope.

Experiments conducted by Mr. F. W. Taylor to determine the most efficient form for lathe roughing tools, the results of which have already been published in MACHINERY (January to August, 1907, engineering edition), showed that the nearer the lip angle approached sixty-one degrees, the higher the cutting speed. This, however, does not apply to tools for turning cast iron, as the latter will work more efficiently with a lip angle of about sixty-eight degrees. This is because the chip pressure, when turning cast iron, comes closer to the cutting edge which should, therefore, be more blunt to withstand the abrasive action and heat. Of course, the foregoing remarks concerning lip angles apply more particularly to tools used for roughing.

The way a turning tool is held while the top surface is being ground is shown in Fig. 38. By inclining the tool with the wheel face, it will be seen that both the back and side slopes may be ground at the same time. When grinding the flank of the tool it should be held on the tool-rest of the emery wheel or grindstone, as shown in Fig. 39. In order to form a curved cutting edge, the tool is turned about the face of the stone while it is being ground. This rotary movement can be effected by supporting the inner end of the tool with one hand while the shank is moved to and fro with the other.

Often a tool which has been ground properly in the first place, is greatly mis-shapen after it has been sharpened a few times. This is usually the result of attempts on the part of the workman to resharpen it hurriedly; for example, it is easier to secure a sharp edge on the turning tool shown in Fig. 35, by grinding the flank as indi-



Figs. 38 and 39. Grinding the Top and Flank of a Turning Tool

cated by the dotted line, than by grinding the entire flank. The clearance is, however, reduced and the lip angle changed.

There is great danger when grinding a tool of burning it or drawing the temper from the fine cutting edge, and, aside from the actual shape of the cutting end, this is the most important point in connection with tool grinding. If a tool is pressed hard against an emery or other abrasive wheel, even though the latter has a copious supply of water, the temper will sometimes be drawn.

When grinding a flat surface, to avoid burning, the tool should be frequently withdrawn from the stone so that the cooling water (a copious supply of which should be provided) can have access to the surface being ground. A moderate pressure should also be applied, as it is better to spend an extra minute or two in grinding, than to ruin the tool by burning it in an attempt to sharpen it quickly. Of course, what has been said about burning, applies more particularly to carbon steel, but even self-hardening steels are not improved by being overheated at the stone.

In some shops tools are ground to the theoretically correct shape in special machines instead of by hand. The sharpened tools are then kept in the tool-room and are given cut as they are needed.

CHAPTER VI

QUICK CHANGE-GEAR TYPE OF LATHE

A type of lathe that is much used at the present time is shown in Fig. 40. This is known as the quick change-gear type, because it has a system of gearing which makes it unnecessary to remove the change gears and replace them with different sizes for cutting threads of various pitches. Changes of feed are also obtained by the same mechanism, but the feeding movement is transmitted to the carriage by the rod R, whereas the screw S_1 is used for screw cutting. As previ-

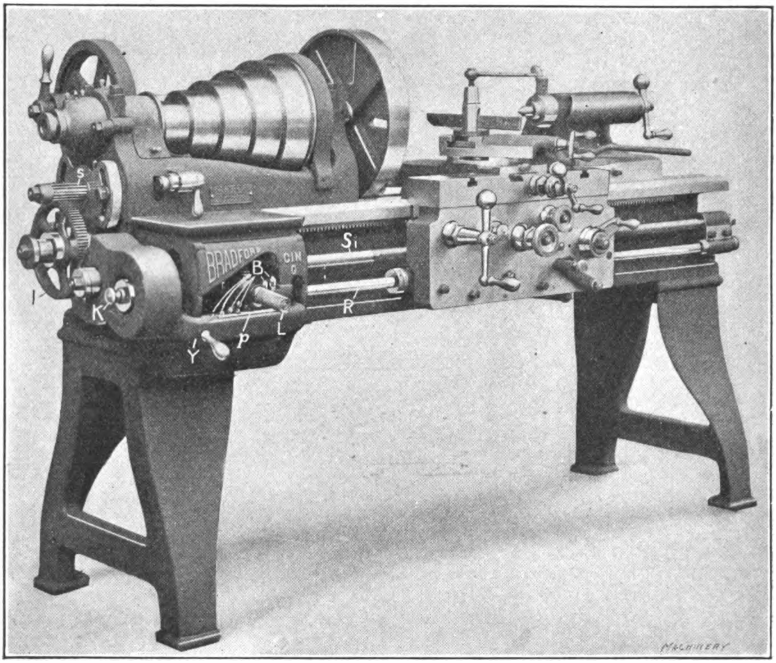


Fig. 40. Lathe Having Quick Change-gear Mechanism

ously explained the idea of using the screw exclusively for threading is to prevent it from being worn excessively, as it would be if continually used in place of rod R, for feeding the carriage when turning.

The general construction of this quick change gear mechanism and the way the changes are made for cutting threads of different pitch, will be explained in connection with Figs. 40, 41 and 42, which are marked with the same reference letters for corresponding parts. Referring to Fig. 40, the movement is transmitted from gear s on

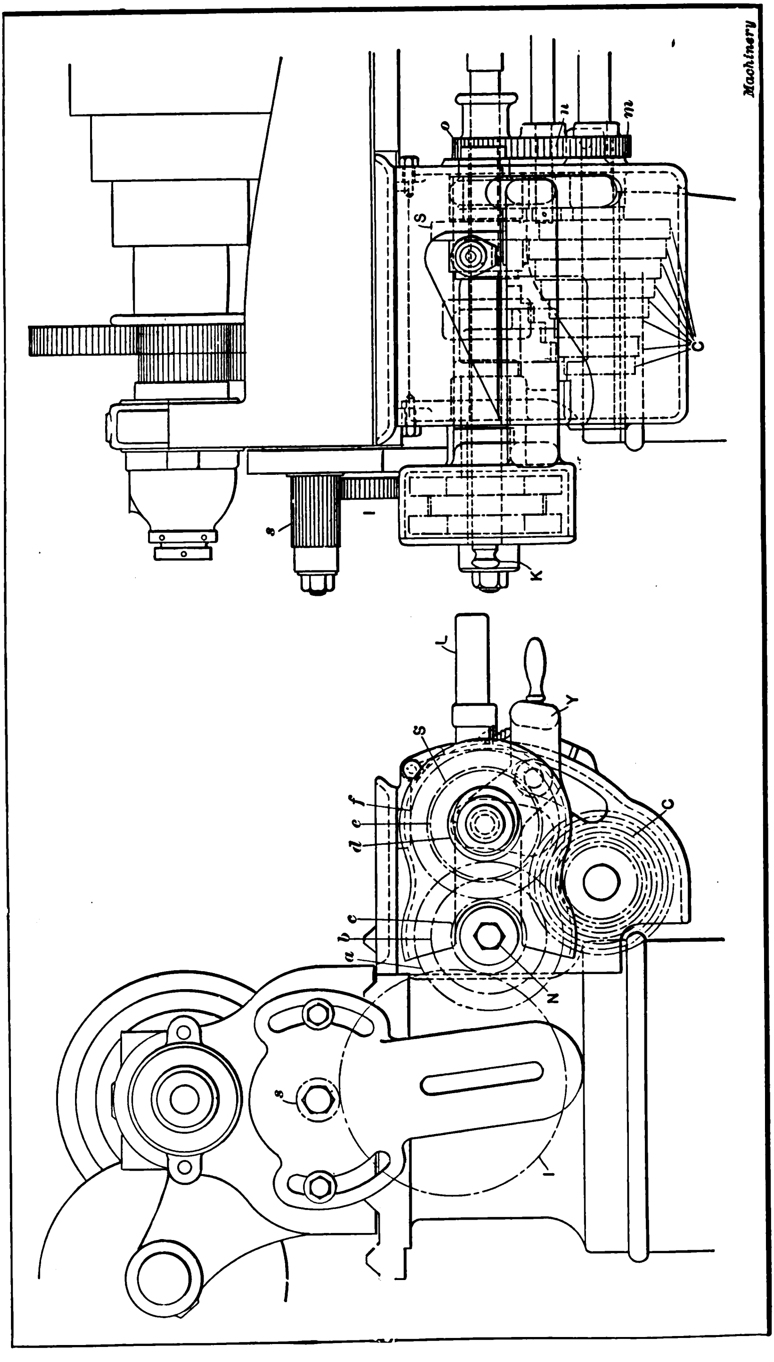


Fig. 41. End and Side Views of Quick Change-gear Mechanism

the spindle stud through idler gear *I*, which can be moved sidewise to mesh with either of the three gears *a*, *b* or *c*, Fig. 41. This cone of three gears engages gears *d*, *e* and *f*, any one of which can be locked with shaft *T* (Fig. 42) by changing the position of knob *K*. On shaft *T* there is a gear *S* which can be moved along the shaft by hand lever *L* and, owing to the spline or key *t*, both the sliding gear and shaft rotate together. Shaft *T*, carrying gears *d*, *e* and *f* and the sliding gear *S*, is mounted in a yoke *Y*, which can be turned about shaft *N*, thus making it possible to lower sliding gear *S* into mesh with any one of a cone of eight gears *C*, Fig. 41. The shaft on which the eight gears are mounted, has at the end a small gear *m* meshing with gear *n* on the feed-rod, and the latter, in turn, drives the lead-screw, unless gear *o* is shifted to the right out of engagement, which is its position

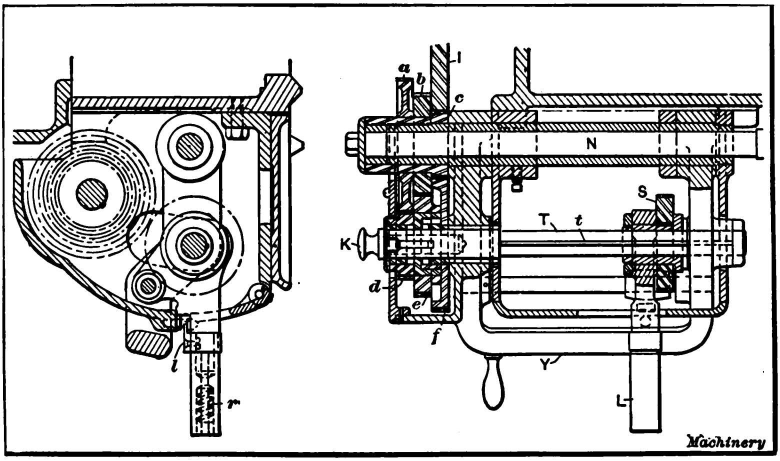
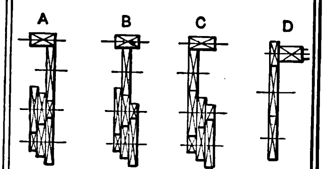


Fig. 42. Sectional Views of Quick Change-gear Mechanism

except when cutting threads. With this mechanism, eight changes for different threads or feeds are obtained by simply placing gear *S* into mesh with the various sized gears in cone *C*. As the speed of shaft *T* depends on which of the three gears *d*, *e* and *f* are locked to it, the eight changes are tripled by changing the position of knob *K*, making twenty-four. Now by shifting idler gear *I*, three speed changes may be obtained for gears *a*, *b* and *c*, which rotate together, so that the twenty-four changes are also tripled, giving a total of seventy-two variations without removing any gears, and if a different sized gear *s* were placed on the spindle stud, an entirely different range could be obtained, but such a change would rarely be necessary. As shown in Fig. 40, there are eight hardened steel buttons *B*, or one for each gear of the cone *C*, placed at different heights in the casing. When lever *L* is shifted sidewise to change the position of sliding gear *S*, it is lowered onto one of these buttons (which enters a pocket on the under side) and in this way gear *S* is brought into proper mesh with any gear of the cone *C*. To shift lever *L*, the handle is pulled outward against the tension of spring *r* (Fig. 42) which disengages latch *l* and enables the lever to be lifted clear of the button; yoke *Y* is then raised or lowered, as the case may be, and lever *L* with the sliding gear is shifted to the required position.

The position of lever *L* and knob *K* for cutting threads of different pitches, is shown by an index plate or table attached to the lathe and arranged as shown in Fig. 43. The upper section *a* of this table shows the different numbers of threads to the inch that can be obtained when idler gear *I* is in the position shown by the diagram *A*. Section *b* gives the changes when the idler gear is moved, as shown at *B*, and, similarly, section *c* gives the changes for position *C* of the idler. The horizontal row of figures from 1 to 8 below the word "stops", represents the eight positions for lever *L* which has a plate *p* (Fig. 40) just beneath it with corresponding numbers, and the column to the left shows whether knob *K* should be out, in a central position, or in. In order to find what the position of lever *L* and knob *K* should be for cutting any given number of threads to the inch, find what "stop" number is directly above the number of threads to be cut, which will indicate the location of lever *L*, and also what position should be occupied by knob *K*, as shown in the column to the left. For example, suppose the lathe is to be geared for cutting eight threads to the inch. By referring to section *a* we see that lever *L* should be in position 4 and knob *K* in the center, provided the idler gear *I* were in position *A*, as it would be ordinarily, because all standard numbers of threads per inch (U. S. standard) from ¼ inch up to and including 4 inches in diameter, can be cut with the idler gear in that position. As another illustration, suppose we want to cut twenty-eight threads per inch. This is listed in section *c*, which shows that lever *L* must be placed in position 3 with knob *K* pushed in and the idler gear shifted to the left as at *C*.

DIRECTIONS FOR COMPOUNDING



POSITION OF KNOB	STOPS								
	1	2	3	4	5	6	7	8	
	THREADS WITH COMP. A								
OUT	3	$3\frac{1}{4}$	$3\frac{1}{2}$	4	$4\frac{1}{2}$	5	$5\frac{1}{2}$	$5\frac{3}{4}$	*a*
CENT	6	$6\frac{1}{2}$	7	8	9	10	11	$11\frac{1}{2}$	
IN	12	13	14	16	18	20	22	23	
	THREADS WITH COMP. B								
OUT	$4\frac{1}{2}$	$4\frac{7}{8}$	$5\frac{1}{4}$	6	$6\frac{3}{4}$	$7\frac{1}{2}$	$8\frac{1}{4}$	$8\frac{5}{8}$	*b*
CENT	9	$9\frac{3}{4}$	$10\frac{1}{2}$	12	$13\frac{1}{2}$	15	$16\frac{1}{2}$	$17\frac{1}{4}$	
IN	18	$19\frac{1}{2}$	21	24	27	30	33	$34\frac{1}{2}$	
	THREADS WITH COMP. C								
OUT	6	$6\frac{1}{2}$	7	8	9	10	11	$11\frac{1}{2}$	*c*
CENT	12	13	14	16	18	20	22	23	
IN	24	26	28	32	36	40	44	46	

Fig. 43. Index Plate showing Positions of Control Levers for Cutting Threads of Different Pitch

The simplicity of this method as compared with the time-consuming operation of removing and changing gears, is apparent. The diagram *D* to the right, shows an arrangement of gearing for cutting nineteen threads per inch. A 20-tooth gear is placed on the spindle stud (in place of the regular one having 16 teeth) and one with 95 teeth on the end of the lead-screw, thus driving the latter direct as with ordinary change gears.

CHAPTER VII

MISCELLANEOUS POINTS ON LATHE WORK

The production of accurate lathe work depends partly on the condition of the lathe used and also on the care and judgment exercised by the man operating it. Even though a lathe is properly adjusted and in good condition otherwise, errors are often made which are due to other causes which should be carefully avoided.

If the turning tool is clamped so that the cutting end extends too far from the supporting block, the downward spring of the tool, owing to the thrust of the cut, sometimes results in spoiled work, especially when an attempt is made to turn close to the finished size by taking a heavy roughing cut. Suppose the end of a cylindrical part is first

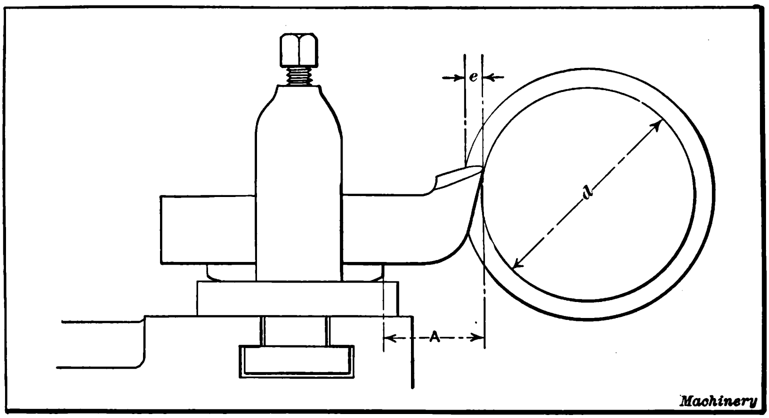


Fig. 44. To avoid springing, Overhang A of Tool should not be too great

reduced for a short distance by taking several trial cuts until the diameter *d*, Fig. 44, is slightly above the finished size and the power feed is then engaged. When the tool begins to take the full depth *e* of the cut, the point, which ordinarily would be set above the center, tends to spring downward into the work, and if there were considerable springing action, the part would probably be turned below the finished size, the increased reduction beginning at the point where the full cut started. This springing action, as far as the tool is concerned, can be practically eliminated by locating the tool so that the distance *A* between the tool-block and cutting end, or the "overhang," is as short as possible. Even though the tool has little overhang it may tilt downward because the tool-slide is loose on its ways, and for this reason the slide should have a snug adjustment that will permit an easy movement without unnecessary play.

When roughing cuts are to be taken, the tool should also be located so that any change in its position caused by the pressure of the cut, will not spoil the work. This point is illustrated at *A* in Fig. 45. Suppose the end of a rod has been reduced by taking a number of trial cuts, until it is 1/32 inch above the finished size. If the power feed is then engaged with the tool clamped in an oblique position, as shown, when the full cut is encountered at *c*, the tool, unless very tightly clamped, may be shifted backward by the lateral thrust of the cut, as indicated by the dotted lines. The point will then begin turning smaller than the finished size and the work will be spoiled. To prevent any change of position, it is good practice, especially when roughing, to clamp the tool square with the surface being turned, or in other words, at right angles to its direction of movement. Occasionally, however, there is a decided advantage in having the tool

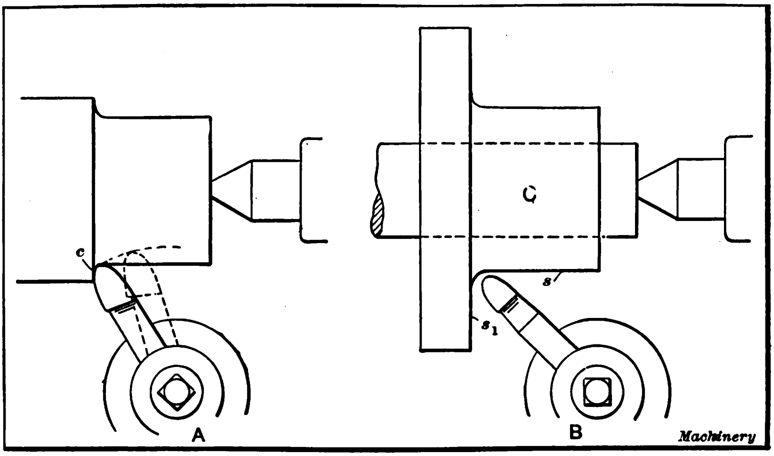


Fig. 45. (A) The Way in which Tool is sometimes displaced by Thrust of Cut, when set at an Angle. (B) Tool Set for Finishing Cylindrical and Radial Surfaces

set at an angle. For example, if it is held about as shown at *B*, when turning the flange casting *C*, the surfaces s and s_1 can be finished without changing the tool's position.

Work that is held in a chuck is sometimes sprung out of shape by the pressure of the chuck jaws so that when the part is bored or turned, the finished surfaces are untrue after the jaws are released and the work has resumed its normal shape. This applies more particularly to frail parts, such as rings, thin cylindrical parts, etc. Occasionally the distortion can be prevented by so locating the work with relation to the chuck jaws that the latter bear against a rigid part. When the work cannot be held tightly enough for the roughing cuts without springing it, the jaws should be released somewhat before taking the finishing cut, to permit the part to spring back to its natural shape.

Work that is turned between centers is sometimes driven by a dog which is so short for the faceplate that the bent driving end bears against the bottom *a* of the faceplate slot, as shown at *A*, Fig. 46. If the dog is nearly the right length, it may allow the headstock center to enter the center in the work part way, with the result that the turned surface is not true with the centers. When a driving dog of this type is used, care should be taken to see that it moves freely in the faceplate slot and does not bind against the bottom. By using a straight dog (*B*), which is driven by a pin *b* bolted to the faceplate, all danger from this source is eliminated. The straight dog, however, is used more particularly to do away with the leverage *l* of a bent dog, as this leverage tends to spring the part being turned. Straight dogs are also made with two driving ends which engage pins on opposite sides of the faceplate. This type is preferable because it applies the power required for turning, evenly to the work, which still further reduces the tendency to spring it out of shape. The principal objec-

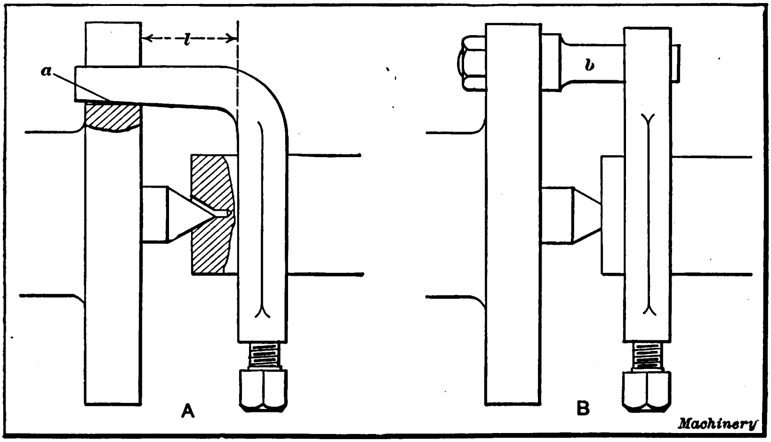


Fig. 46. (A) Dog that is too Short for Faceplate. (B) Straight Driving Dog

tion to the double-ended type lies in the difficulty of adjusting the driving pins so that each bears with equal pressure against the dog.

The lathe centers should receive careful attention especially when accurate work must be turned. If the headstock center does not run true as it revolves with the work, a round surface may be turned, but if the position of the driving dog with reference to the faceplate is changed, the turned surface will not run true because the turned surface is not true with the work centers. Furthermore, if it is necessary to reverse the work for finishing the dogged or driving end, the last part turned will be eccentric to the first. Therefore, the lathe centers should be kept true in order to produce turned surfaces that are true or concentric with the centered ends, as it is often necessary to change the part being turned "end for end" for finishing, and any eccentricity between the different surfaces would, in many cases, spoil the work.

Some lathes are equipped with hardened centers in both the head- and tail-stock and others have only one hardened center which is in the tailstock. The object in having a soft or unhardened headstock center is to permit its being trued by turning, but as a soft center is quite easily bruised and requires truing oftener than one that is hard, it is better to have both centers hardened. Special grinders are used for truing these hardened centers. One type that is very simple and easily applied to a lathe is shown in Fig. 47. This grinder is held in the lathe toolpost and is driven by a wheel *A* that is held in contact with the cone-pulley. The emery wheel *B* is moved to a position for grinding by adjusting the carriage and cross-slide, and it is traversed across the conical surface of the center by handle *C*. As the grinding proceeds, the wheel is fed inward slightly by manipulating the cross-slide. This grinder is set to the proper angle by placing the two centered ends *D* and D_1 between the lathe centers, which should be aligned as for straight turning. The grinding spindle will then be 30 degrees from the axis of the lathe spindle. The grinder should be carefully clamped in the toolpost so that it will remain

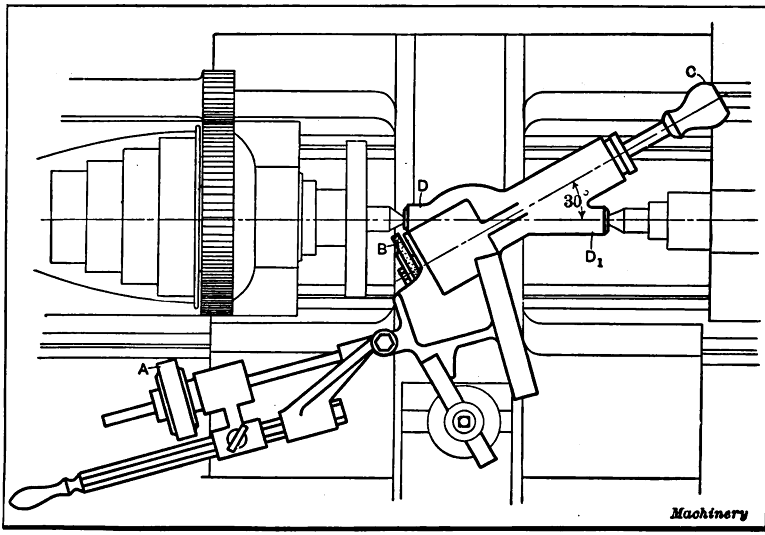


Fig. 47. Lathe Center Grinder

as located by the centered ends. The tailstock center is next withdrawn and the emery wheel is adjusted for grinding. As the wheel spindle is 30 degrees from the axis of the lathe spindle, the lathe center is not only ground true but to an angle of 60 degrees, which is the standard angle for lathe centers. There are many other styles of center grinders on the market, some of which are driven by a small belt from the cone-pulley and others by electric motors which are connected with ordinary lighting circuits. The tailstock center is ground by inserting it in the spindle in place of the headstock center. Before a center is replaced in its spindle, the hole should be perfectly clean as even a small particle of dirt may seriously affect the alignment.

When a rod or shaft must be turned cylindrical or to the same diameter throughout its entire length, it is good practice to test the alignment of the centers, before inserting the work. The position of the tailstock center for cylindrical turning may be indicated by the coincidence of graduation marks on the base, but if accuracy is necessary, the relative position of the two centers should be determined in a more positive way. A very simple and convenient method of testing the alignment is shown at *A* in Fig. 48. The work is first turned for a short distance, near the dogged end, as shown, and the tool is left as set for this cut; then the tailstock center is withdrawn and the work is moved sufficiently to permit running the tool back to the tailstock end without changing its original setting. A short cut is then taken at this end and the diameters d and d_1 are carefully compared. In case there is any variation, the tailstock center is

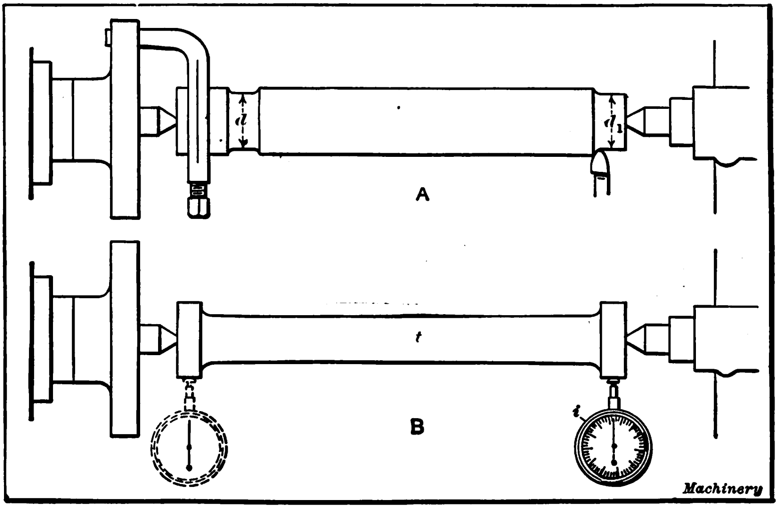


Fig. 48. Two Methods of Aligning Centers for Cylindrical Turning

adjusted laterally, other trial cuts are taken, and the test repeated.

Another method is illustrated at *B*, which requires the use of a test-bar *t*. This bar should have accurately made centers and the ends finished to exactly the same diameter. The lathe centers are aligned by placing the bar between them and then testing the position of the ends. This can be done by comparing each end with a tool held in the toolpost and moved from one to the other by shifting the carriage, but a better method is to clamp a test indicator *i* in the toolpost and bring it in contact with first one end of the bar and then the other. If the dial does not register the same at each end, it shows that the lathe centers are not in line.

Even when centers are correctly set, lathes that have been in use a long time do not always turn cylindrical or straight because if the ways that guide the carriage are worn unevenly, the tool as it moves along does not remain in the same plane and this causes a variation in the diameter of the part being turned.