Taper Turning with Taper Attachment

The taper attachment is used for turning and boring tapers in the lathe. It eliminates the necessity of setting over the tailstock, and if desired may be set permanently for a standard taper. The taper attachment does not interfere with using the lathe for straight turning.

The taper attachment is especially valuable for boring tapered holes. If the lathe is not equipped with a taper attachment, the compound rest top swivel may be set for the desired taper, but the length of the taper is limited to the comparatively short angular feed of the compound rest top when this method is used.

Graduations on one end of the taper attachment swivel bar indicate the total taper in inches per foot, and on the other end, the included angle of the taper is shown in degrees. See Fig. 182-A.

Plain Taper Attachment

The plain taper attachment (used on all 9" South Bend Lathes) shown at right consists of a bracket attached to the back of the lathe carriage, a compound slide with clamp for locking slide to lathe bed and a connecting bar to connect the slide block of the taper attachment to the compound rest base of the lathe.

When the plain taper attachment is to be used it is necessary to disconnect the cross feed screw by removing the bolt "A" which locks the cross feed nut to the compound rest base of the lathe. This leaves the compound rest base free to slide so that it may be controlled by the taper attachment. Binding screws "B" and "C" are tightened to engage the taper attachment.

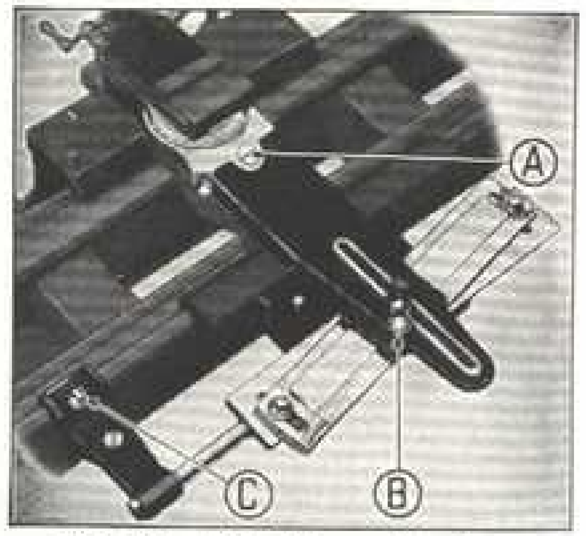


Fig. 181. Plain Taper Attachment

Telescopic Taper Attachment

The telescopic taper attachment (used on 10" to 16" swing South Bend Lathes) shown in the illustration below, Fig. 182, is similar to the plain taper attachment described above except that it is equipped with a telescopic cross feed screw. This feature eliminates the necessity of disconnecting the cross feed screw when the taper attachment is to be used.

The cross feed screw may be used to adjust the turning tool for the required diameter and the taper attachment may then be engaged by tightening binding screws "X" and "Y". To change back to straight turning, it is necessary to loosen binding screws "X" and "Y".

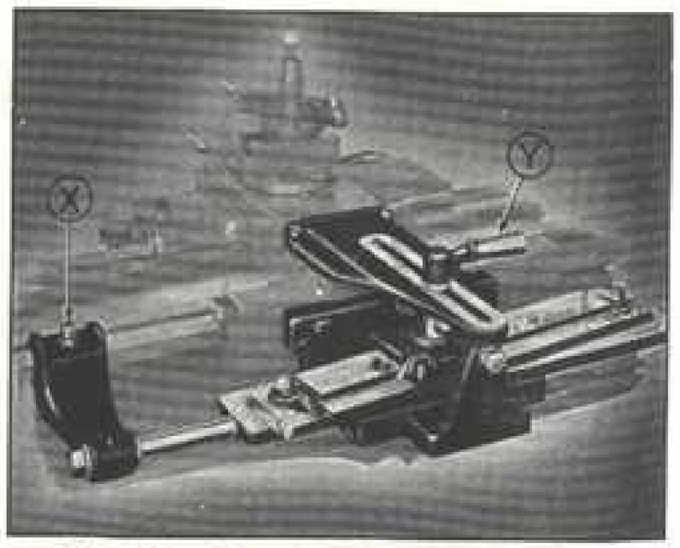


Fig. 182. Telescopic Taper Attachment

Setting the Taper Attachment Swivel Bar

Fig. 182A. Graduations in Inches per Foot on Swivel Bar

Tapers are usually specified in inches per foot or in degrees. When this information is not available the taper in inches per foot should be calculated before setting the taper attachment for taper turning.

To calculate the taper in inches per foot, subtract the diameter in inches at the small end of taper (B, Fig. 183) from the diameter in inches at the large end of the taper (A); divide the result by the length of tapered portion (C) in inches, and multiply by 12. The answer is the taper in inches per foot and indicates the graduation on the "taper per foot" end of the swivel bar which should be set in line with the witness mark in order to machine the desired taper. See Fig. 182A.

Each graduation on the "taper per foot" end of swivel bar represents a total taper of 1/16" per foot. If the taper per foot has been calculated or specified in decimal fractions instead of common fractions, refer to the decimal equivalent table on page 115 for the nearest fractional part of an inch.

When setting the swivel bar for taper turning, remember that the total taper is indicated by the graduations, either in inches per foot or in degrees. For example, if the swivel bar is set at 5° the taper machined will have a total included angle of 5°; that is 2½° (not 5°) each side of center line.

After setting the taper attachment swivel bar to the required angle, take a trial cut and test the taper with a taper gage or micrometers. Some readjustment of the swivel bar will probably be necessary, as it is difficult to align the graduations on swivel bar perfectly with the witness mark. See page 61 for information on fitting and testing tapers with taper gauges.

Standard tapered holes may be hand reamed after boring to standardize the taper and size the hole.

Turning an outside taper is shown in Fig. 183. Boring an inside taper with the aid of a center rest is shown in Fig. 184. See page 92.

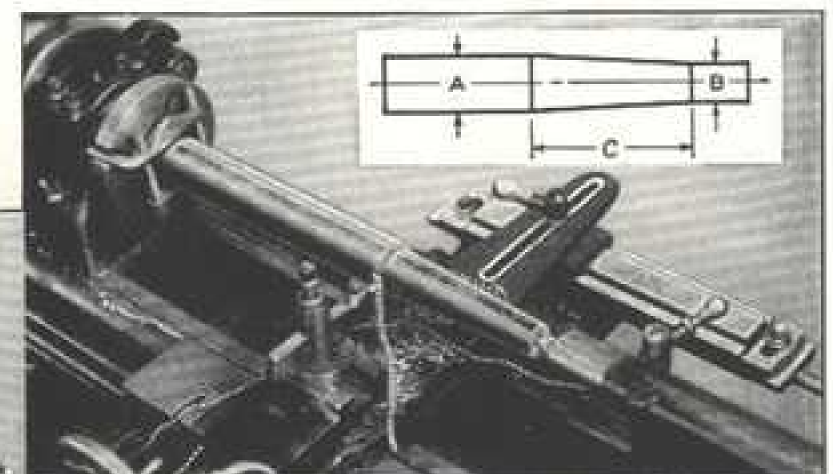


Fig. 183. Taper Turning with the Taper Attachment

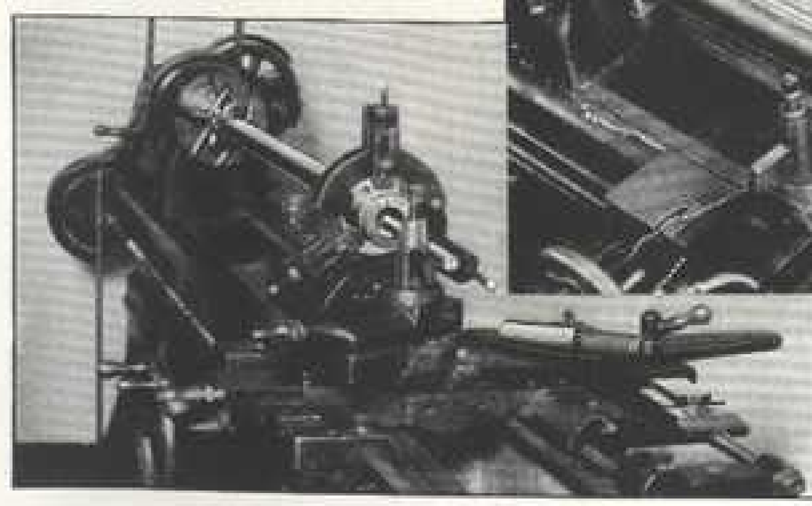


Fig. 184. Boring a Taper with the Taper Attachment

Morse Standard Tapers

Morse Standard Tapers are used for lathe and drill press spindles by most of the manufacturers of lathes and drill presses in the United States. South Bend Lathes have both head and tailstock spindles fitted for Morse Standard Tapers. The dimensions of various sizes of Morse Standard Tapers are listed in the tabulation below.

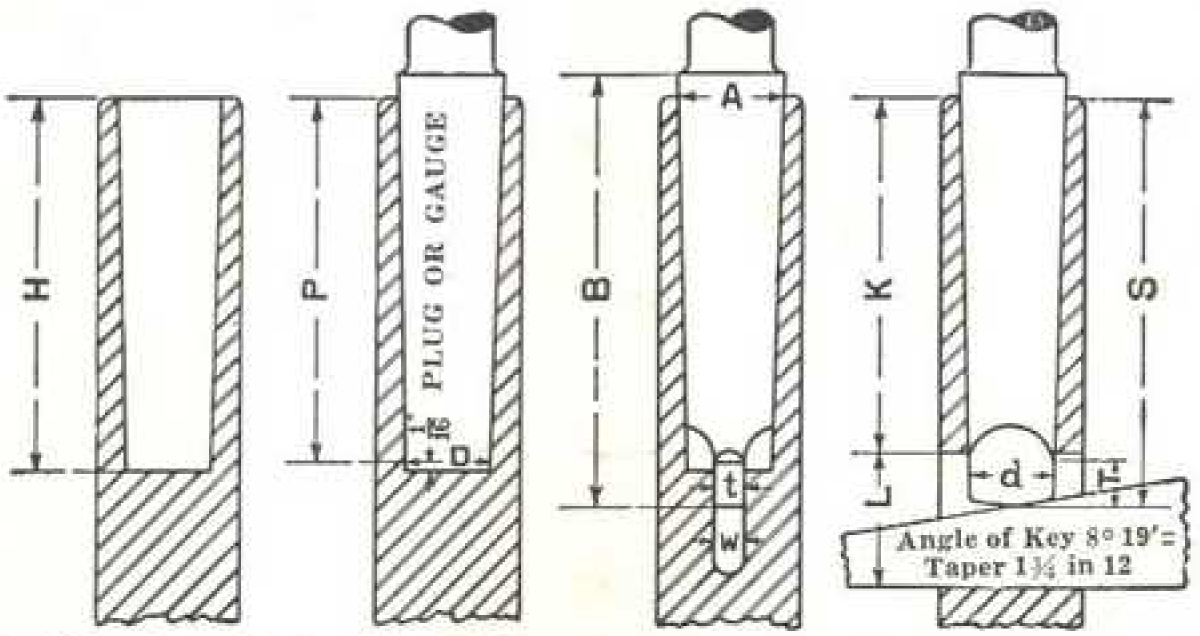


Fig. 185. Chart Showing Principal Dimensions of Morse Standard Tapers Which are Listed in Tabulation Below

DIMENSIONS OF MORSE STANDARD TAPERS

All Dimensions in Tabulations Below Are in Inches

Number of Taper	Diam. of Plug at Small End	Diam. at End of Socket	Shank		Depth of Hole	Standard Plug Depth	Tongue		Keyway		End of Socket to Keyway	Taper per Foot
			Whole Length	Depth			Thickness	Length	Width	Length		
	D	A	B	S	H	P	t	T	W	L	K	
0	0.252	0.3561	2 11/32	2 7/32	2 1/32	2	5/32	1/4	0.160	9/16	1 15/16	.6246
1	0.369	0.475	2 9/16	2 7/16	2 3/16	2 1/8	13/64	3/8	0.213	3/4	2 1/16	.5986
2	0.572	0.700	3 1/8	2 15/16	2 5/8	2 9/16	1/4	7/16	0.260	7/8	2 1/2	.5994
3	0.778	0.938	3 7/8	3 11/16	3 1/4	3 3/16	5/16	9/16	0.322	1 3/16	3 1/16	.6023
4	1.020	1.231	4 7/8	4 5/8	4 1/8	4 1/16	15/32	5/8	0.478	1 1/4	3 7/8	.6233
5	1.475	1.748	6 1/8	5 7/8	5 1/4	5 3/16	5/8	3/4	0.635	1 1/2	4 15/16	.6315
6	2.116	2.494	8 9/16	8 1/4	7 3/8	7 1/4	3/4	1 1/8	0.760	1 3/4	7	.6256
7	2.750	3.270	11 5/8	11 1/4	10 1/8	10	1 1/8	1 3/8	1.135	2 5/8	9 1/2	.6240

The figures in the "Taper per Foot" column have been revised to conform with the standard end diameters and lengths.

Brown & Sharpe and Jarno Tapers

Two other system of tapers are widely used. The Brown & Sharpe Tapers are used for milling machine spindles and the Jarno Tapers for some makes of lathe spindles. Specifications of these tapers can be obtained from technical hand books or from manufacturers using them.

Chapter IX

DRILLING, REAMING AND TAPPING

Many drilling, reaming and tapping jobs can be done more quickly and with greater accuracy in the lathe than by any other method.

Fig. 186 (at right) illustrates the use of the lathe as a drill press. A drill pad placed in the tailstock spindle of the lathe is used to support the work.

The tailstock hand wheel is turned as the hole is drilled through the work. The end of the work may rest on the lathe bed if desired.

Fig. 186. Using the Lathe as a Drill Press

The Location of the hole should be center punched to start the drill. The lathe should be operated at high speed when drilling small diameter holes.

Fig. 187. (Right) Crotch Center for Use in Tailstock of Lathe

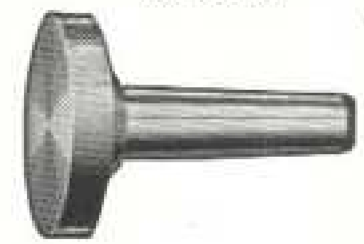


Fig. 187-A. (Left) Drill Pad for Use in Tailstock of Lathe

Drill Pad for Tailstock

A drill pad for the tailstock spindle of the lathe is shown in Fig. 187. The drill pad replaces the tailstock center and supports the work for drilling.

Crotch Center

The crotch center shown in Figs. 187 and 188 is similar to the drill pad except that it has a "V" so that round work may be accurately cross drilled. This is very convenient for drilling oil holes in bushings, drilling pin holes in shafts, etc.

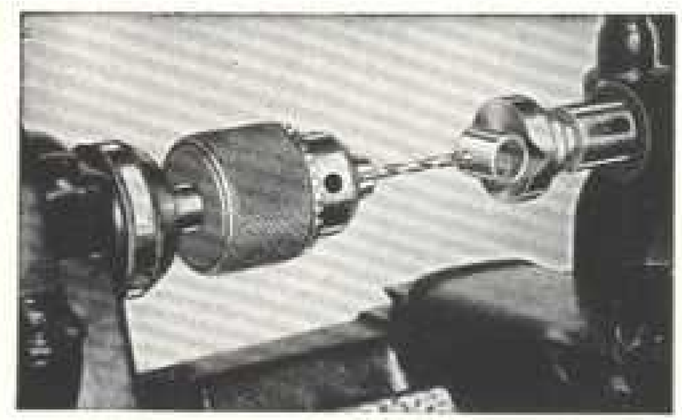


Fig. 188. Drilling an Oil Hole in a Bushing with Crotch Center in Tailstock

Drilling Work Held in the Chuck

Most of the drilling in the lathe is done with the work mounted in the lathe chuck (as shown in Fig. 189) or clamped to the face plate of the lathe. When this method is used it is important that the drill be started so that it will run true and the hole will be drilled concentric with the outside diameter of the work.

One method for starting the drill point true is illustrated in Fig. 190. The butt end of a lathe tool holder just touching the side of the drill will prevent the drill from bending and cause it to start approximately true in the center of the work.

Fig. 189. Drilling Work Held in Chuck

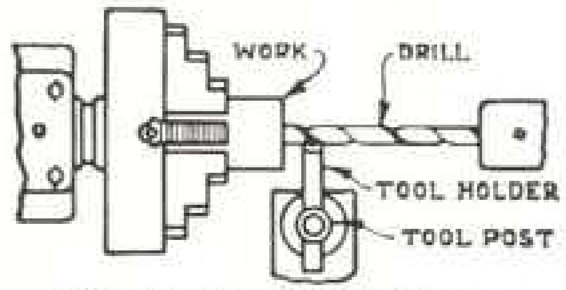


Fig. 190. Using Tool Holder to Steady Point of Drill

Center Drilling

When greater accuracy is required it is best to provide a true starting point for the drill. To do this the work should first be center drilled using a combination center drill and countersink, as shown in Fig. 191. The point of the center drill may be ground off as shown in Fig. 192 to prevent breaking.

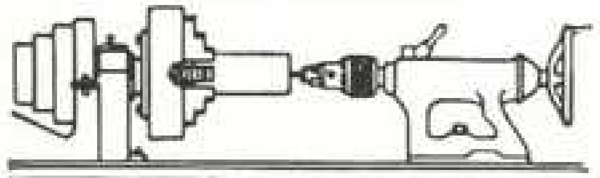


Fig. 191. Center Drilling

Grind off Tip of Drill, Shown by dotted line

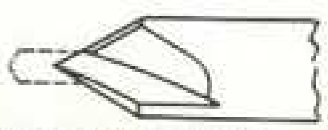


Fig. 192. Detail of Center Drill

Center Drill Holder

A special holder for mounting the combination center drill and countersink in the tailstock for drilling center holes is shown in Fig. 193. This provides an extremely rigid and accurate support for the drill, resulting in more accurately located center holes than can usually be obtained by mounting the center drill in a 3-jaw drill chuck. Large pieces that cannot be properly mounted in the chuck alone can be supported on the outer end by a steady rest, as shown in Fig. 194.

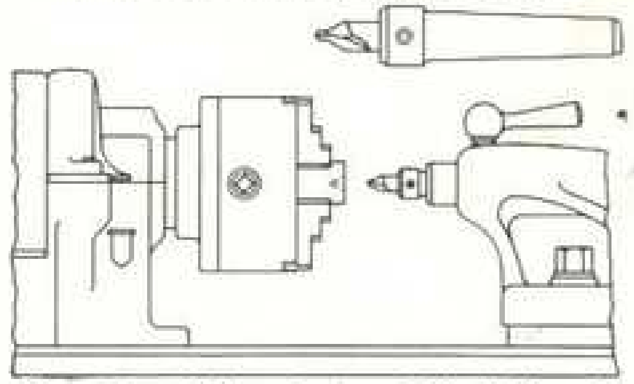


Fig. 193. Using Center Drill Holder in Tailstock to Drill Accurate Center Hole in Piece Held in Chuck

Drilling in Steel

When drilling in steel use plenty of lard oil on the point of the drill. If no lard oil is available, any good cutting oil or even machine oil may be used. However, lard oil is preferable, and for some deep hole drilling is the only satisfactory lubricant.

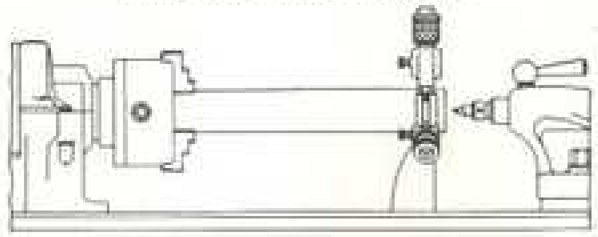


Fig. 194. Center Drilling Large Piece with the Aid of a Steady Rest

Drilling a Cored Hole

Castings having cored holes are usually drilled with a four lip drill. The hole in the casting should be beveled, as shown in Fig. 195, to start the drill true; otherwise, the drill will follow the cored hole and may be thrown off center. For accurate drilling it is advisable to counterbore the hole a short depth to give the drill point a perfectly concentric starting point.

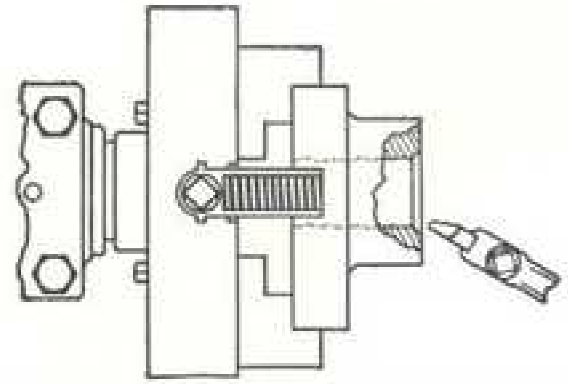


Fig. 195. Machining a Bevel in a Cored Hole to Start Drill True

How to Sharpen Drills

Correct grinding of the drill point is essential for accuracy and efficiency in all drilling operations. A medium grain grinding wheel that has been dressed true should be used for grinding drill points. The drill point should not be overheated by grinding or the temper may be drawn.

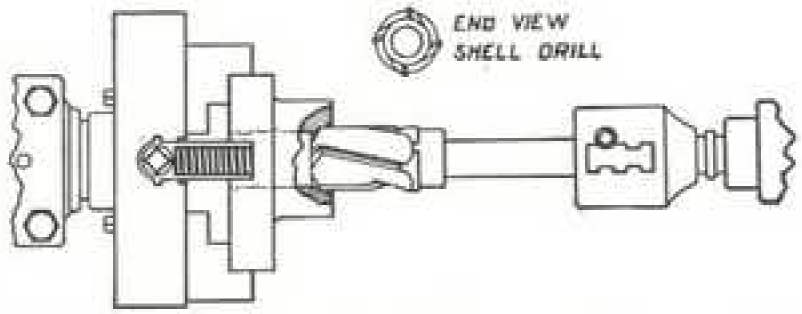


Fig. 196. Drilling a Cored Hole with a Four Lip Shell Drill

Before grinding a drill, study the point of a new drill as received from the manufacturer; then try to duplicate it. This can be accomplished by holding the drill at the correct angle with the grinding wheel and giving the drill point a wiping motion as it is ground, lowering the shank end of the drill and giving the drill a slight twist to the right simultaneously. It is very important that both lips of the drill be ground exactly the same.

Fig. 197. Correct Point Angle

Fig. 198. Correct Lip Angle

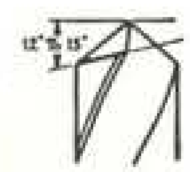


Fig. 199. Correct Clearance

The angle of the chisel point or dead center should be from 120° to 130°, as shown in Fig. 197. The cutting lips "L" Fig. 198, should be exactly the same length and angle; otherwise the drill will cut oversize. The best angle for general work is 59°, as indicated.

The clearance back of the cutting edge should be from 12° to 15°, as shown in Fig. 199. Less clearance may prevent the drill from cutting freely, and more clearance will cause the cutting edge to dull quickly.

A drill grinding gauge similar to the one shown in Fig. 200 will aid in grinding the correct angle and length of cutting lip on the drill point.

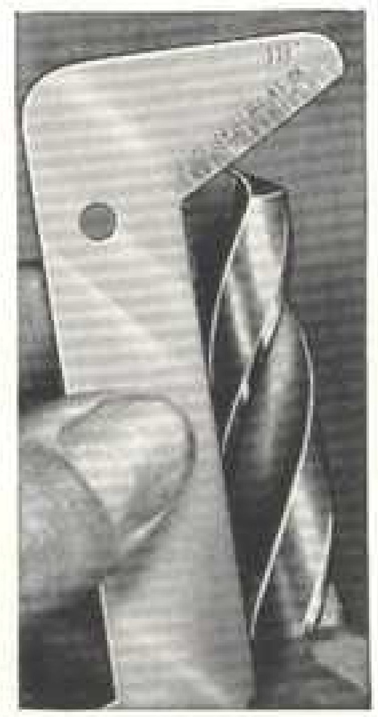


Fig. 200. Drill Grinding Gauge

Reaming in the Lathe

Reamers are used in the lathe to finish a number of holes quickly and accurately to the same diameter. Usually the hole is first drilled or bored roughly to size, allowing sufficient stock for reaming. Two types of reamers are used, the rose reamer and the fluted reamer.

Rose reamers are ground for cutting on the end only and are intended for rough reaming as they do not produce a good finish or an accurate diameter.

Fluted reamers are ground for cutting on both the ends and the sides of the blades and are usually used after the rose reamer to obtain an exact size and produce a good smooth finish. Fluted reamers should be used only for light cuts, removing not over .010 in. from the hole.

Reamer in Drill Chuck

Straight shank reamers are usually held in a drill chuck, as shown in Fig. 201. Taper shank reamers may be inserted direct in the tailstock spindle. The reamer is fed carefully through the hole by turning the tailstock handwheel. Always use a slow spindle speed and when reaming steel keep the reamer flooded with lard oil.

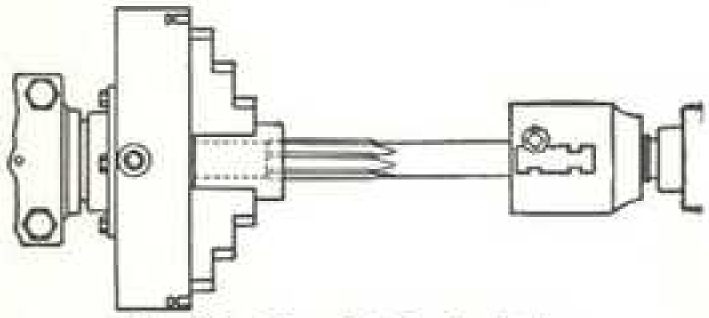


Fig. 201. Reaming in the Lathe

Floating Reamer Driver

For some reaming operations it is desirable for the reamer to follow a bored hole as accurately as possible, and for this type of work a floating reamer driver similar to the one shown in Fig. 202 is used.

Large reamers are sometimes supported on the tailstock center point. A lathe dog is attached to the reamer shank and a stick with one end resting against the lathe bed is placed between the reamer shank and the tail of the dog.

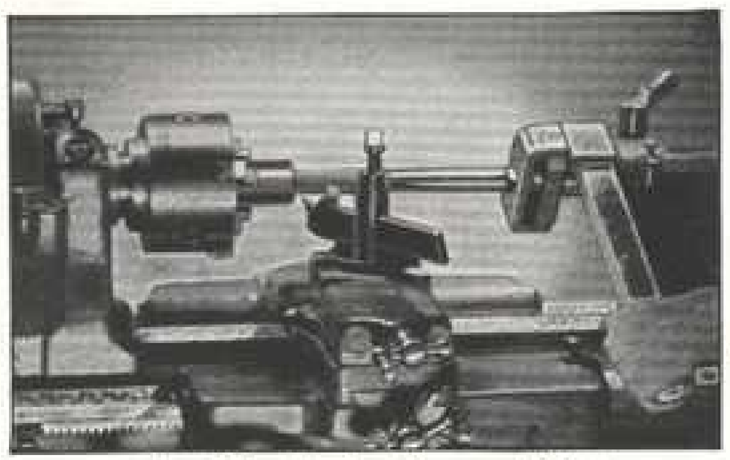


Fig. 202. Floating Reamer Driver

Tapping Threads

Threads may be tapped in the lathe, using a tap as shown in Fig. 203. The lathe spindle should be operated at slow speed and the tap fed to the work by turning the tailstock handwheel, or by sliding the entire tailstock on the lathe bed. Taps may also be held in a drill chuck.

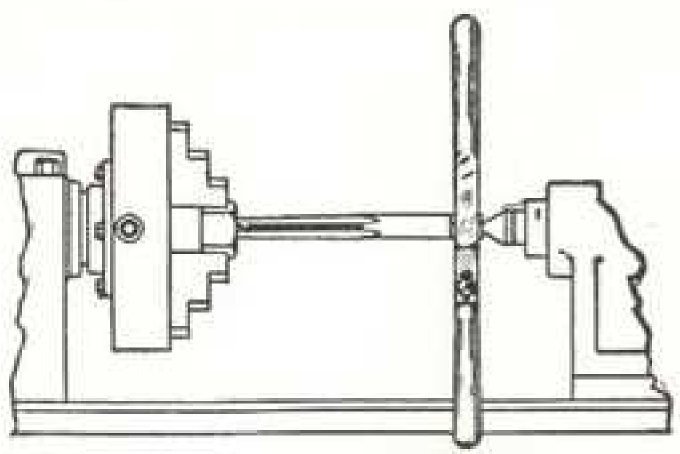


Fig. 203. Tapping in the Lathe

Fig. 204. Cutting Screw Threads in the Lathe

Chapter X

CUTTING SCREW THREADS

Cutting screw threads in the lathe is accomplished by connecting the headstock spindle of the lathe with the lead screw by a series of gears so that a positive carriage feed is obtained and the lead screw is driven at the required speed with relation to the headstock spindle.

The gearing between the headstock spindle and lead screw may be arranged so that any desired pitch of the thread may be cut. For example, if the lead screw has eight threads per inch and the gears are arranged so that the headstock spindle revolves four times while the lead screw revolves once, the thread cut will be four times as fine as the thread on the lead screw or 32 threads per inch.

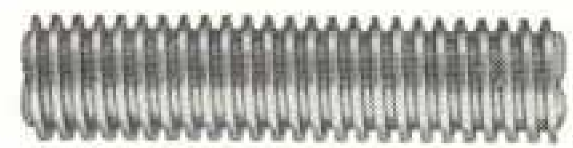


Fig. 205. Acme Screw Thread

The cutting tool is ground to the shape required for the form of the thread to be cut, that is, American National Form, "V," Acme, Square, Whitworth, International Metric, etc.

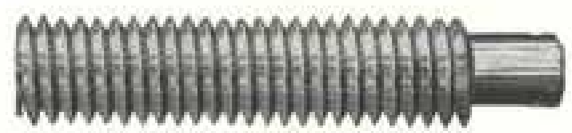
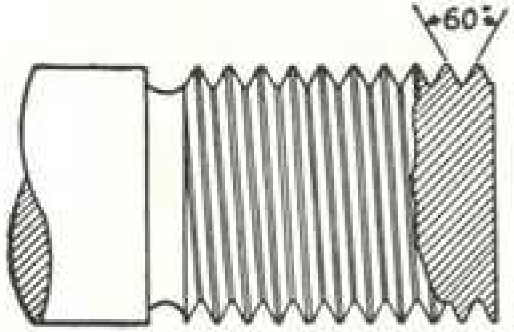


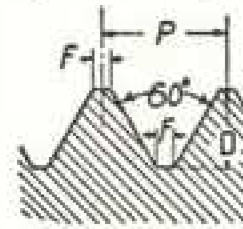
Fig. 206. National Coarse Thread

Either right hand or left hand threads may be cut by reversing the direction of rotation of the lead screw. This may be accomplished by shifting the reverse lever on the headstock.

Fig. 207. Double Square Thread



AMERICAN NATIONAL SCREW THREAD
(Formerly U.S. Standard Screw Thread)



FORMULA

$$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$$

$$D = \text{Depth} = P \times .64952$$

$$F = \text{Flat} = \frac{P}{8}$$

Fig. 208. American National Screw Thread Form

American National Screw Threads

The National Screw Thread Commission in 1918 was authorized by Congress to establish a standard system of screw threads for use in the United States. As a result this commission established the American National Screw Thread System which has been approved by the Secretary of War, Secretary of the Navy, and Congress, and is now generally used by all shops in the United States.

The form of the thread adopted is shown above and tables for both the Fine Thread Series and Coarse Thread Series are given on page 71. A report of the National Screw Thread Commission defines the follow ing terms:

Screw Thread. A ridge of uniform section in the form of a helix on the surface of a cylinder or cone.

External and Internal Threads. An external thread is a thread on the outside of a member. Example: A threaded plug. An internal thread is a thread on the inside of a member. Example: A threaded hole.

Major Diameter (formerly known as "outside diameter"). The largest diameter of the thread of the screw or nut. The term "major diameter" replaces the term "outside diameter" as applied to the thread of a screw and also the term "full diameter" as applied to the thread of a nut.

Minor Diameter (formerly known as "core diameter"). The smallest diameter of the thread of the screw or nut. The term "minor diameter" replaces the term "core diameter" as applied to the thread of a screw and also the term "inside diameter" as applied to the thread of a nut.

Pitch Diameter. On a straight screw thread, the diameter of an imaginary cylinder, the surface of which would pass through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder.

Pitch. The distance from a point on a screw thread to a corresponding point on the next thread measured parallel to the axis.

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

International Screw Thread Form

The need for an international screw thread form has been recognized for many years. Although several international missions have been made and hundreds of conferences held, little was accomplished until the summer of 1948 when United States and British representatives agreed on a 60° thread form with flat top and rounded bottom as shown in Fig. 208A. This thread form is a compromise between the 60° American National thread form with flat top and bottom shown in Fig. 208, and the British Whitworth 55° thread form with rounded top and bottom shown in Fig. 246. It is similar to the International Standard Metric thread form shown in Fig. 250.

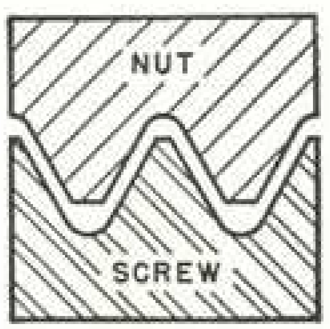


Fig. 208A. Diagram Showing International Screw Thread Form.

TABLES OF AMERICAN NATIONAL STANDARD SCREW THREAD PITCHES AND RECOMMENDED TAP DRILL SIZES

American National Coarse Standard Thread (N.C.)
Formerly U. S. Standard

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
1	64	.073	53	0.0595
2	56	.086	50	0.0700
3	48	.099	46	0.0810
4	40	.112	43	0.0890
5	40	.125	38	0.1015
6	32	.138	33	0.1130
8	32	.164	29	0.1360
10	24	.190	25	0.1495
12	24	.216	16	0.1770
1/4	20	.250	7	0.2010
5/16	18	.3125	F	0.2570
3/8	16	.375	5/16	0.3125
7/16	14	.4375	U	0.3680
1/2	13	.500	27/64	0.4219
9/16	12	.5625	31/64	0.4843
5/8	11	.625	17/32	0.5312
3/4	10	.750	21/32	0.6562
7/8	9	.875	49/64	0.7656
1	8	1.000	7/8	0.875
1 1/8	7	1.125	63/64	0.9843
1 1/4	7	1.250	1 7/64	1.1093

American National Fine Standard Thread (N.F.)
Formerly S. A. E. Thread

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
0	80	.060	3/64	0.0469
1	72	.073	53	0.0595
2	64	.086	49	0.0730
3	56	.099	44	0.0860
4	48	.112	42	0.0935
5	44	.125	37	0.1040
6	40	.138	32	0.1160
8	36	.164	29	0.1360
10	32	.190	21	0.1590
12	28	.216	14	0.1820
1/4	28	.250	7/32	0.2187
5/16	24	.3125	I	0.2720
3/8	24	.375	R	0.3390
7/16	20	.4375	25/64	0.3906
1/2	20	.500	29/64	0.4531
9/16	18	.5625	33/64	0.5156
5/8	18	.625	37/64	0.5781
3/4	16	.750	11/16	0.6875
7/8	14	.875	13/16	0.8125
1	14	1.000	15/16	0.9375
1 1/8	12	1.125	1 3/64	1.0468
1 1/4	12	1.250	1 11/64	1.1718

TABLES OF AMERICAN NATIONAL SPECIAL SCREW THREAD PITCHES (N.S.) AND RECOMMENDED TAP DRILL SIZES

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
1	56	.0730	54	0.0550
4	32	.1120	45	0.0820
4	36	.1120	44	0.0860
6	36	.1380	34	0.1110
8	40	.1640	28	0.1405
10	30	.1900	22	0.1570
12	32	.2160	13	0.1850
14	20	.2420	10	0.1935
14	24	.2420	7	0.2010
1/16	64	.0625	3/64	0.0469
3/32	48	.0938	49	0.0730
1/8	40	.1250	38	0.1015
5/32	32	.1563	1/8	0.1250
5/32	36	.1563	30	0.1285
3/16	24	.1875	26	0.1470
3/16	32	.1875	22	0.1570
7/32	24	.2188	16	0.1770
7/32	32	.2188	12	0 1890
1/4	24	.250	4	0.2090
	27		3	0.2130
	32		7/32	0.2187

Sizes	Threads Per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal Equivalent of Drill
5/16	20	.3125	17/64	0.2656
	27		J	0.2770
	32		9/32	0.2812
3/8	20	.375	21/64	0.3281
	27		R	0.3390
7/16	24	.4375	X	0.3970
	27		Y	0.4040
1/2	12	.500	27/64	0.4219
	24		29/64	0.4531
	27		15/32	0.4687
9/16	27	.5625	17/32	0.5312
5/8	12	.625	35/64	0.5469
	27		19/32	0.5937
11/16	11	.6875	19/32	0.5937
	16	.6875	5/8	0.6250
3/4	12	.750	43/64	0.6719
	27		23/32	0.7187
7/8	12	875	51/64	0.7969
	18		53/64	0.8281
	27		27/32	0.8437
1	12	1.000	59/64	0.9219
	27		31/32	0.9687

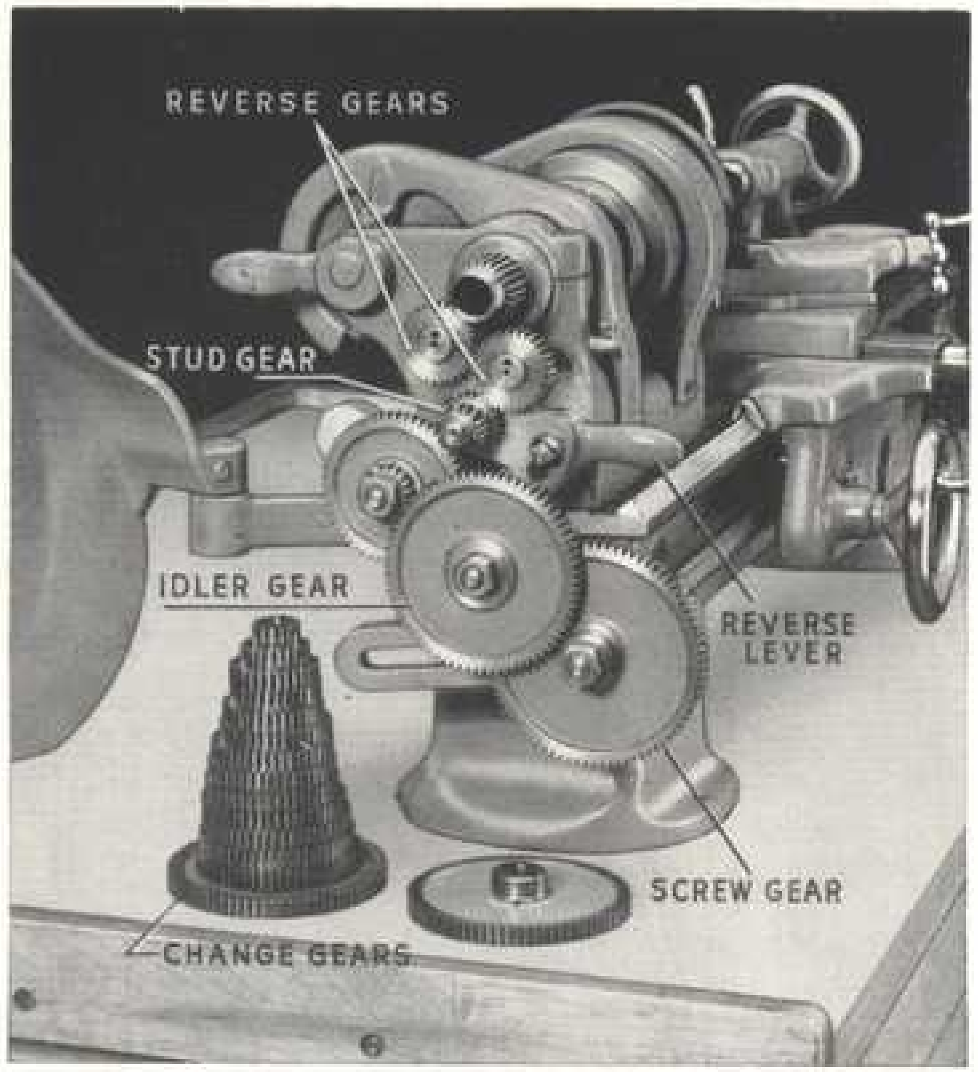


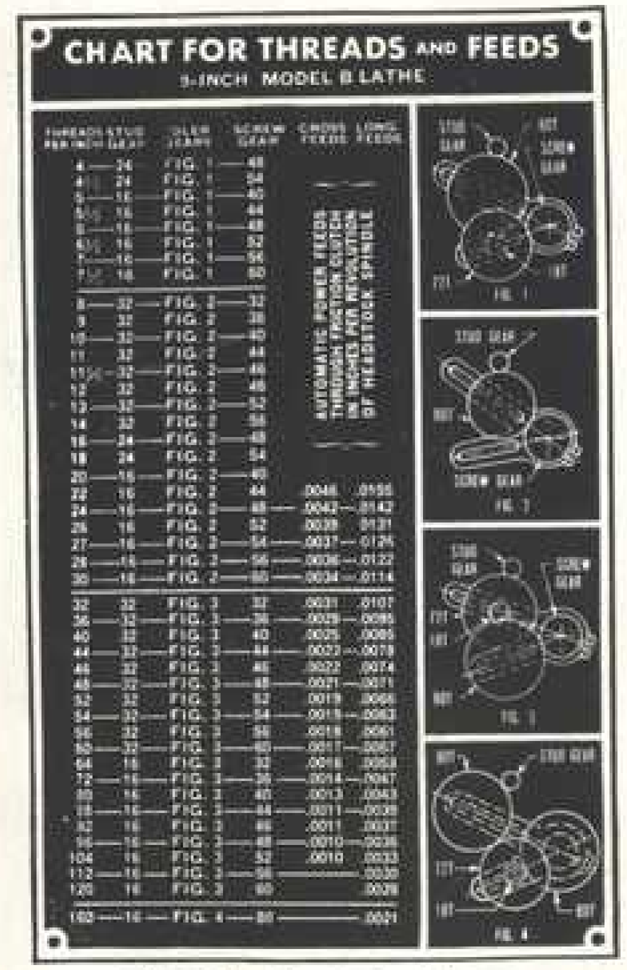
Fig. 209. Standard Change Gear Lathe Set Up for Cutting Screw Threads

Cutting Threads on Standard Change Gear Lathes

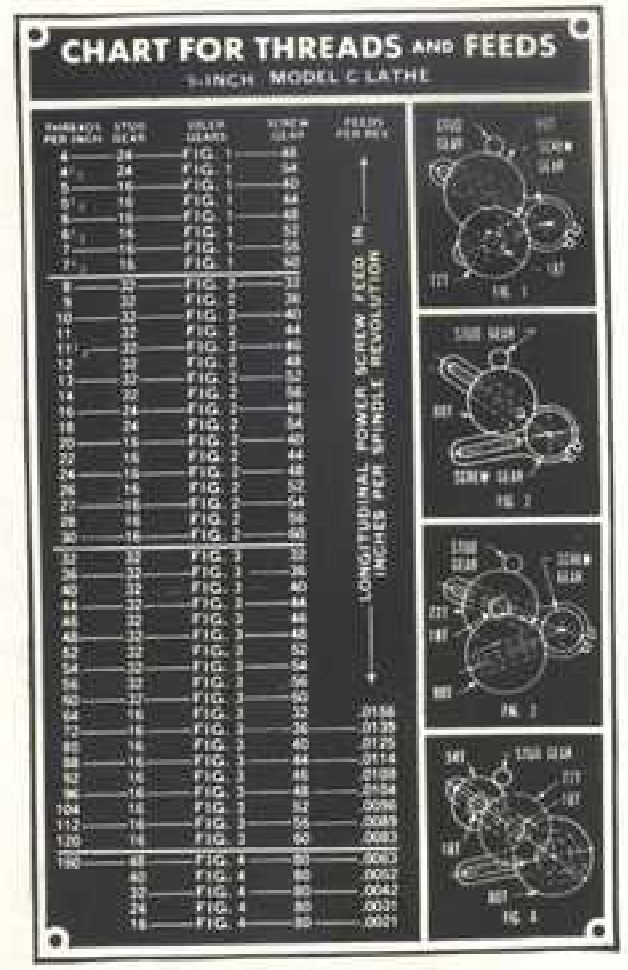
Screw threads are cut on Standard Change Gear Lathes by engaging the apron half nuts with the lead screw. The pitch of thread to be cut is determined by the number of teeth in the change gears used on the reverse stud and the lead screw, also the compound gears used.

To set up the lathe for cutting a screw thread, first determine the number of threads per inch to be cut. By referring to the change gear chart attached to the lathe (Fig. 210) the change gears required can be determined. The thread to be cut should be located in the first column under the heading "Threads Per Inch." In the second column under the heading "Stud Gear" is listed the number of teeth in the change gear which should be placed on the reverse stud of the lathe. (See Fig. 209.) In the third column under the heading "Idler Gear" is listed the figure number of the diagram on the index chart showing the arrangement of idler gear and compound gears. In the fourth column under the heading "Screw Gear" is listed the number of teeth in the gear to be placed on the lead screw "C".

After selecting the change gears necessary for cutting the desired thread, place them on the reverse stud and lead screw respectively and connect them with the idler gear and compound gears, as shown on the change gear chart.



9", Model B, Index Chart



9", Model C, Index Chart

Fig. 210. Change Gear Charts for Standard Change Gear Lathes

Position of Spacing Collar

The spacing collar on the lead screw must be placed outside the screw gear, as shown in Fig. 210A, when simple gearing (Fig. 2 on index charts) is used, and inside the screw gear, as shown in Fig. 210B, when compound gearing (Fig. 1 on index charts) is used. See page 114.

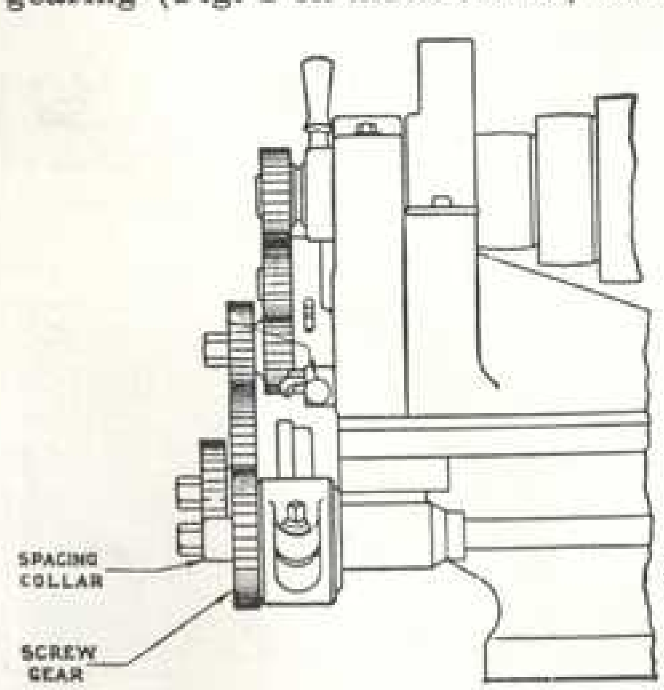


Fig. 210A. Position of Spacing Collar for Simple Gearing

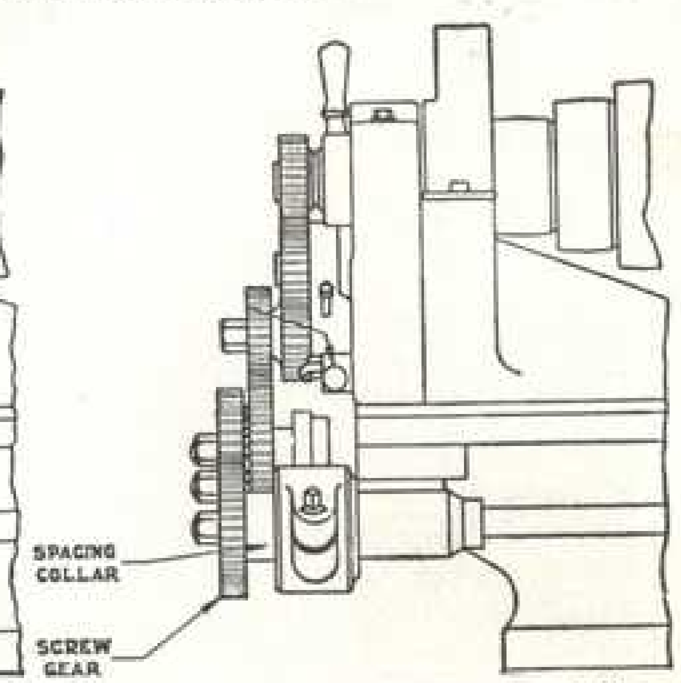


Fig. 210B. Position of Spacing Collar for Compound Gearing

Cutting Screw Threads on Quick Change Gear Lathe

The quick change gear lathe has a gear box, shown in Fig. 211, which eliminates the necessity of using loose change gears for cutting various pitches of screw threads and for obtaining various power cross-feeds and longitudinal feeds. Standard screw threads from 8 to 224* per inch are obtained by placing the two tumbler levers in the positions indicated by the direct reading index chart attached to the gear box. A close-up of this chart is shown in Fig. 211B.

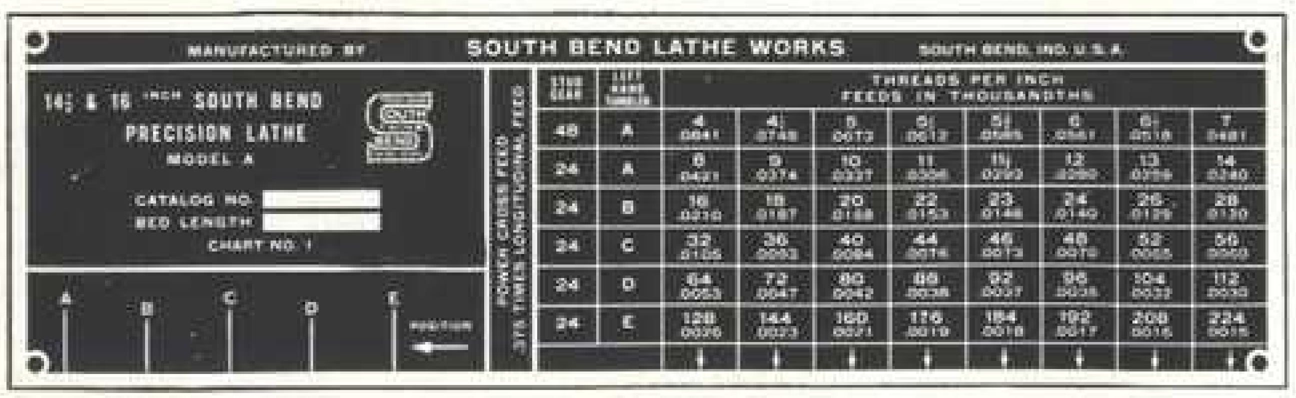
Fig. 211. Quick Change Gear Box

To cut coarse pitches of screw threads from 4 to 7 per inch, the small stud gear must be replaced by a large stud gear which, for safe keeping, is attached to the gear box drive shaft as shown in Fig. 211A. The number of teeth in the stud gear required is indicated on the index chart in the column of figures under the heading "Stud Gear."

Fig. 211A. Lathe set up for cutting threads 8 to 224 per inch

Almost any special thread (one not shown on the index chart) can be obtained by using a special stud gear in place of the standard gears. Special stud gears are usually made to order for the specific thread required.

*Gearbox on 10"-1" Collet Lathes cuts 70 threads, 4 to 480 per inch, without changing stud gear


MANUFACTURED BY SOUTH BEND LATHE WORKS SOUTH BEND, IND. U.S.A.

14½ & 16 INCH SOUTH BEND PRECISION LATHE MODEL A

CATALOG NO.
BED LENGTH
CHART NO. 1

POWER CROSS FEED .375 TIMES LONGITUDINAL FEED

STUD GEAR	LEFT HAND TUMBLER	THREADS PER INCH / FEEDS IN THOUSANDTHS							
48	A	4 .0841	4½ .0748	5 .0673	5½ .0612	5¾ .0585	6 .0561	6½ .0518	7 .0481
24	A	8 .0421	9 .0374	10 .0337	11 .0306	11½ .0293	12 .0280	13 .0259	14 .0240
24	B	16 .0210	18 .0187	20 .0168	22 .0153	23 .0146	24 .0140	26 .0129	28 .0120
24	C	32 .0105	36 .0093	40 .0084	44 .0076	46 .0073	48 .0070	52 .0065	56 .0060
24	D	64 .0053	72 .0047	80 .0042	88 .0038	92 .0037	96 .0035	104 .0032	112 .0030
24	E	128 .0026	144 .0023	160 .0021	176 .0019	184 .0018	192 .0017	208 .0016	224 .0015

A B C D E POSITION

Fig. 211B. Direct Reading Index Chart for Quick Change Gear Lathe

Tools for Cutting Screw Threads

The shape or form of a screw thread cut on the lathe is determined by the shape of the cutter bit, which must be carefully ground and set if an accurate thread form is to be obtained. The most common thread forms are shown on pages 70, 82, 83 and 84. A gauge should be used for grinding the lathe tool to the required shape for any form of screw thread.

Use of Center Gauge

The point of the cutter bit must be ground to an angle of 60° for cutting American National screw threads in the lathe, as shown in Fig. 213 at right. A center gauge having a 60° included angle is used for grinding the tool to the exact angle required. The top of the tool is usually ground flat, with no side rake or back rake. However, for cutting threads in steel, side rake is sometimes used.

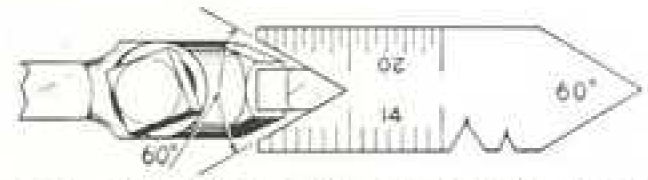


Fig. 213. Cutter Bit for Cutting Screw Threads is Ground to 60° Center Gauge

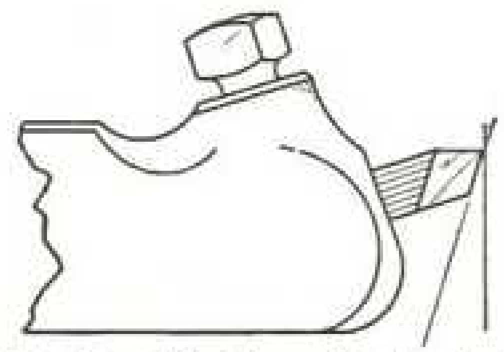


Fig. 214. Side View of Lathe Tool Cutter Bit Ground for Cutting Screw Threads

Front Clearance

There must be sufficient front clearance on the cutter bit to permit it to cut freely. Usually the front clearance is sufficient to prevent the tool from dragging in the helix angle of the thread so that except for very coarse pitches the helix angle may be ignored.

Formed Threading Tool

A formed threading tool is sometimes used if considerable threading is to be done. Fig. 215 illustrates a good type of formed threading tool. The formed threading tool requires grinding on top only to sharpen, and therefore always remains true to form and correct angle.

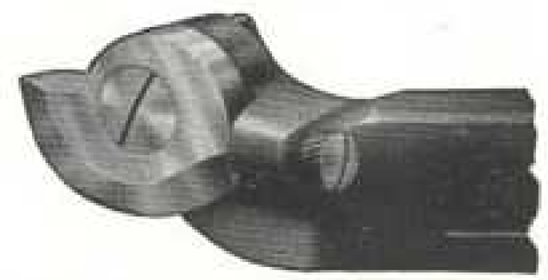


Fig. 215. Formed Thread Cutting Tool, Solid Type

Thread Tool Gauge

A gauge for grinding threading tools to the exact shape required for various pitches of American National screw threads is shown in Fig. 216.

For American National Screw Threads finer than 10 per inch, the point of the tool is usually left sharp or with a very small flat. However, for coarser pitches of threads and when maximum strength is desired, the flat on the point of the tool should be one-eighth of the pitch. (See Fig. 208, page 70.)

Fig. 216. Standard Screw Thread Tool Gauge for Grinding Thread Cutting Tools

Setting Cutter Bit

The top of the threading tool should be placed exactly on center, as shown in Fig. 217 at right, for cutting external screw threads. Note that the top of the tool is ground flat and is in exact alignment with the lathe center. This is necessary to obtain the correct angle of the thread.

The threading tool must be set square with the work, as shown in Fig. 218. The center gauge is used to adjust the point of the threading tool, and if the tool is carefully set a perfect thread will result. Of course, if the threading tool is not set perfectly square with the work, the angle of the thread will be incorrect.

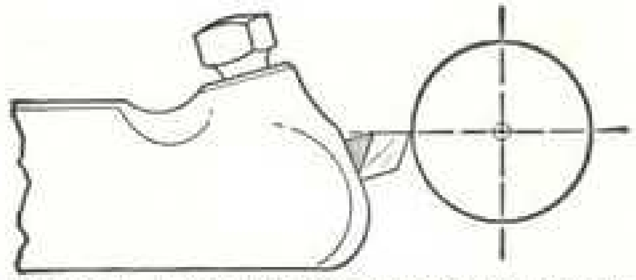


Fig. 217. Top of Cutter Bit Set On Center for Cutting Screw Threads

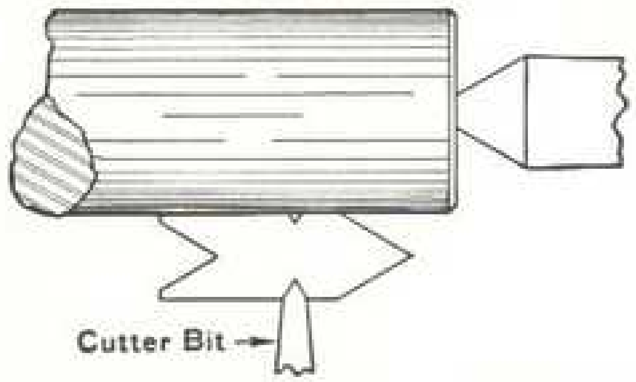


Fig. 218. Cutter Bit Set Square With Work, for Cutting External Screw Threads

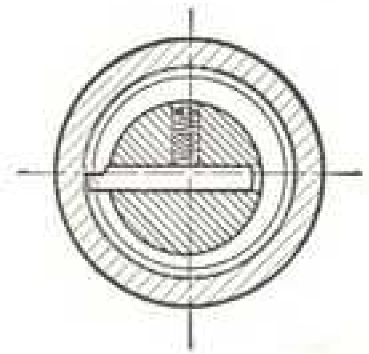
Internal Threads

The point of the threading tool is also placed exactly on center, as shown in Fig. 219 at right for cutting internal screw threads. The point of the tool must be set perfectly square with the work. This may be accomplished by fitting the point of the tool into the center gauge, as shown in Fig. 220 at right.

When adjusting the threading tool for cutting internal threads, allow sufficient clearance between the tool and the inside diameter of the hole to permit backing out the tool when the end of the cut has been reached. However, the boring bar should be as large in diameter and as short as possible to prevent springing.

When cutting internal screw threads more front clearance is required to prevent the heel of the tool from rubbing than when cutting external threads.

Fig. 219. For Cutting Internal Screw Threads, Top of Cutter Bit Should be Set Exactly on Center



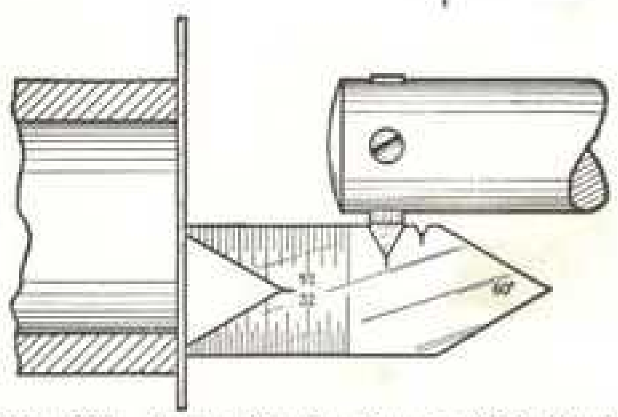


Fig. 220. Cutter Bit Set Square With Work for Cutting Internal Screw Threads

Engaging the Half-Nuts

After the work has been mounted in the lathe and the cutting tool properly adjusted, the half-nuts may be engaged with the lead screw for cutting the screw thread. Once the thread has been started, the half-nuts should not be disengaged from the lead screw, unless the thread dial indicator is used. See page 81.

Fig. 220A. Engaging the Half-Nuts

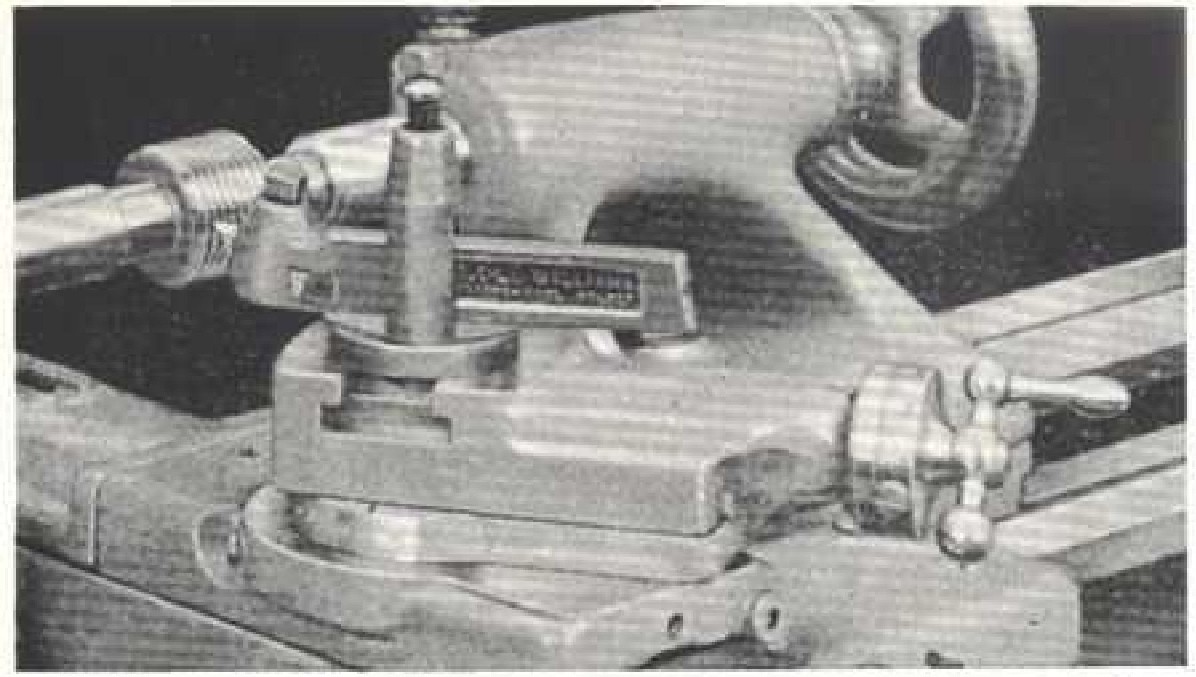


Fig. 221. Compound Rest Set at 29° Angle for Cutting 60° Screw Threads

Position of Compound Rest for Cutting 60° Screw Threads

In manufacturing plants where maximum production is desired, it is customary to place the compound rest of the lathe at an angle of 29° for cutting 60° screw threads.

The compound rest is swung around to the right, as shown in Figs. 221 and 224. The compound rest screw is used for adjusting the depth of cut and most of the metal is removed by the left side of the threading tool. (See Fig. 223.) This permits the chip to curl out of the way better than if tool is fed straight in.

The right side of the tool will shave the thread smooth and produce a good finish, although it does not remove enough metal to interfere with the main chip which is taken by the left side of the tool.

Fig. 222. Cutting a Screw Thread with Compound Rest Set at 29°

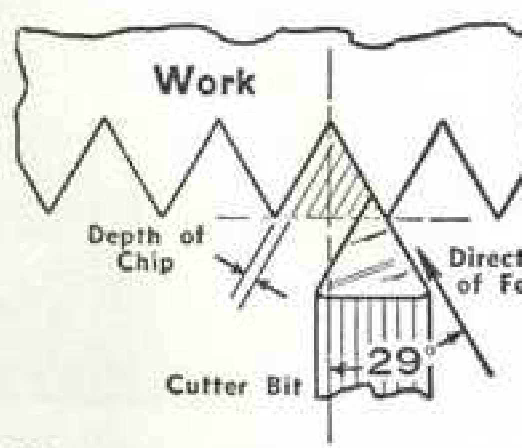


Fig. 223. Action of Thread Cutting Tool when Compound Rest is Set at 29° Angle

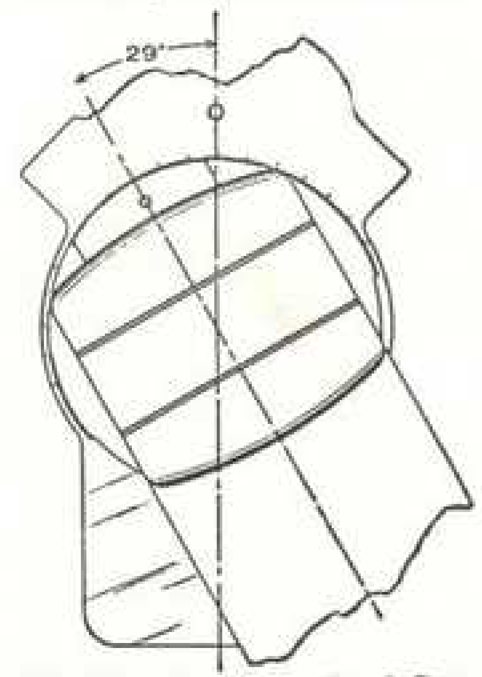


Fig. 224. Correct Angle of Compound Rest for Thread Cutting

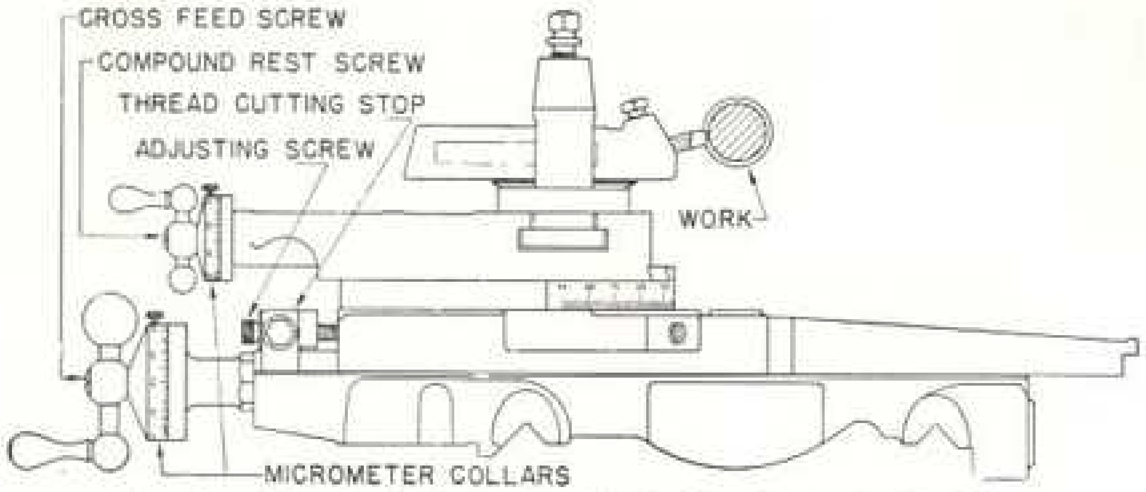


Fig. 225. Thread Cutting Stop Attached to Dovetail of Saddle

Use of Thread Cutting Stop

Because of the lost motion caused by the play necessary for smooth operation of the change gears, lead screw, half nuts, etc., the thread cutting tool must be withdrawn quickly at the end of each cut, before the lathe spindle is reversed to return the tool to the starting point. If this is not done, the point of the tool will dig into the thread and may be broken off. The thread cutting stop may be used for relocating the cutting tool for each successive chip.

The point of the tool should first be set so that it just touches the work; then lock the thread cutting stop to the saddle dovetail approximately ¼ of an inch from the compound rest base and turn the thread cutting stop screw until the shoulder is tight against the stop. When ready to take the first chip run the tool rest back by turning the cross feed screw to the left several turns and move the tool to the point where the thread is to start. Then turn the cross feed screw to the right until the thread cutting stop screw strikes the thread cutting stop. The tool rest is now in the original position, and by turning the compound rest feed screw in .002 in. or .003 in. the tool will be in a position to take the first cut.

Using Micrometer Collar

The micrometer collar on the cross feed screw of the lathe may be used in place of the thread cutting stop, if desired. To do this, first bring the point of threading tool up so that it just touches the work, then loosen small lock screw "A," adjust the micrometer collar on the cross feed screw to zero, and tighten lock screw.

All adjusting for obtaining the desired depth of cut should be done with the compound rest screw. Withdraw the tool at the end of each cut by turning the cross feed screw to the left one complete turn, return the tool to the starting point and turn the cross feed screw to the right one turn, stopping at zero. The compound rest feed screw may then be adjusted for any desired depth of chip.

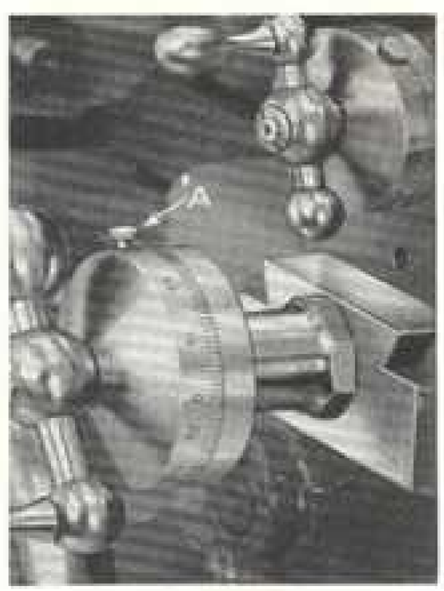


Fig. 226. Micrometer Collar on Cross Feed Screw of Lathe

Taking the First Cut

After setting up the lathe, as explained on the preceding pages, take a very light trial cut just deep enough to scribe a line on the surface of the work, as shown in Fig. 227. The purpose of this trial cut is to make sure the lathe is arranged for cutting the desired pitch of thread.

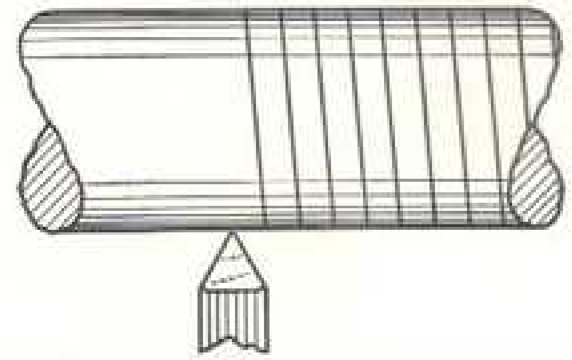


Fig. 227. Trial Cut to Check the Set-up for Thread Cutting

Measuring Screw Threads

To check the number of threads per inch, place a scale against the work, as shown in Fig. 228, so that the end of the scale rests on the point of the thread or on one of the scribed lines. Count the spaces between the end of the scale and the first inch mark, and this will give you the number of threads per inch. Fig. 228 shows eight threads per inch.

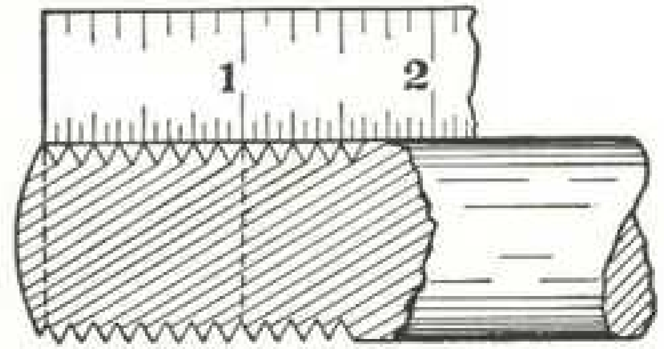


Fig. 228. Measuring Screw Threads

Screw Pitch Gauge

A screw thread gauge as illustrated in Fig. 229 is very convenient for checking the finer pitches of screw threads. This gauge consists of a number of sheet metal plates in which are cut the exact form of the various pitches of threads.

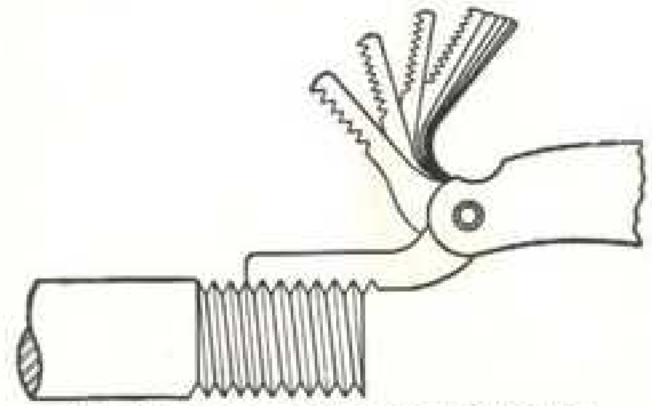


Fig. 229. Screw Thread Pitch Gauge

Fitting and Testing Threads

The final check for both the diameter and pitch of the thread may be made with the nut that is to be used or with a ring thread gauge, if one is available. Fig. 230 shows how the nut may be used for checking the thread. The nut should fit snugly without play or shake but should not bind on the thread at any point.

If the angle of the thread is correct and the thread is cut to the correct depth it will fit the nut perfectly. However, if the angle of the thread is incorrect, the thread may appear to fit the nut but will only be touching at a few points. For this reason, the thread should be checked by other methods in addition to the nut or ring gauge.

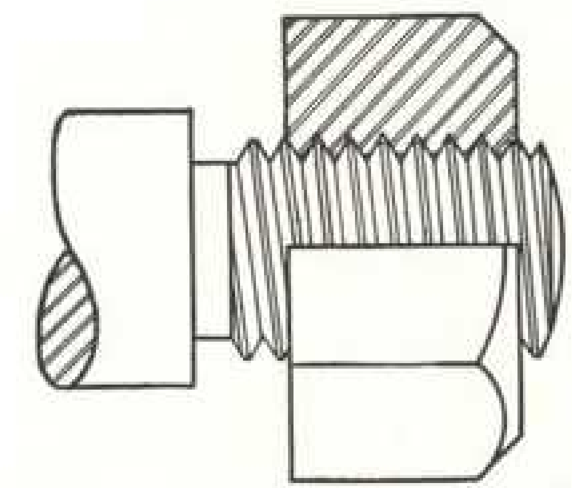


Fig. 230. Screw Thread Fitted to a Nut

Resetting Tool After Cut Has Been Started

If for any reason it is necessary to remove the thread cutting tool before the thread has been completed, the tool must be carefully readjusted so that it will follow the original groove when it is replaced in the lathe.

Before adjusting the tool, take up all the lost motion by pulling the belt forward by hand.

The compound rest top should be set at an angle, and by adjusting the cross feed screw and compound rest feed screw simultaneously the point of the tool can be made to enter exactly into the original groove.

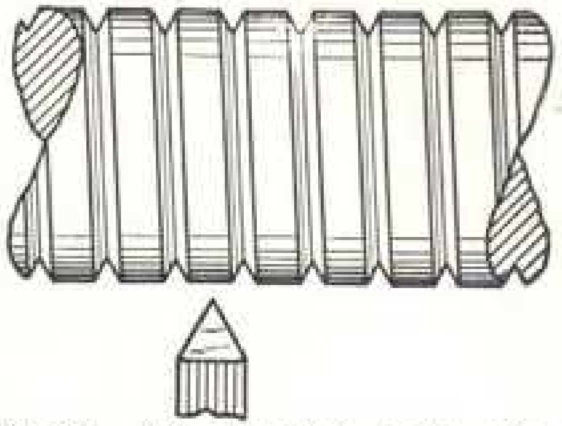


Fig. 231. Adjusting Point of Threading Tool to Conform with Thread

Finishing the End of a Thread

The end of the thread may be finished by any one of several methods. The 45° chamfer on the end of the thread, as shown in Fig. 232, is commonly used for bolts, cap screws, etc. For machine parts and special screws the end is often finished by rounding with a forming tool, as shown in Fig. 233.

It is difficult to stop the threading tool abruptly so some provision is usually made for clearance at the end of the cut. In Fig. 232 a hole has been drilled in the end of the shaft, and in Fig. 233 a neck or groove has been cut around the shaft. The groove is preferable as the lathe must be run very slowly in order to obtain satisfactory results with the drilled hole.

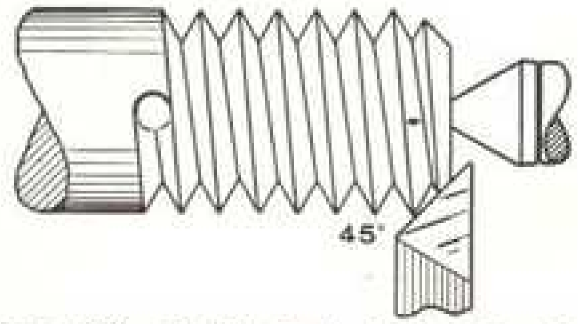


Fig. 232. Finish End of Thread with 45° Chamfer

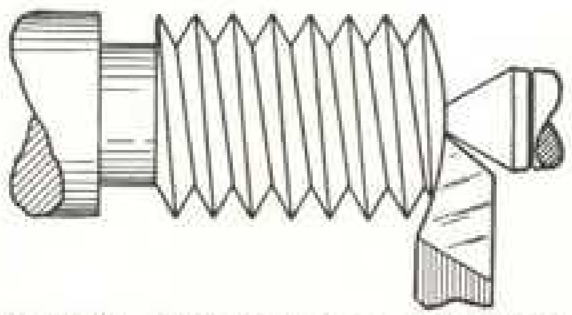


Fig. 233. Finishing End of Thread with Forming Tool

Cutting a Left Hand Screw Thread

A left hand screw is one that turns counter-clockwise when advancing (looking at head of screw) as shown in Fig. 235. This is just the opposite of a right hand screw. Left hand threads are used for the cross feed screws of lathes, the left hand end of axles for automobiles and wagons, one end of a turnbuckle, some pipe threads, etc.

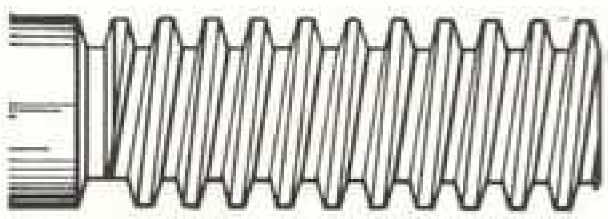


Fig. 234. A Left Hand Screw Thread

In cutting left hand screw threads the lathe is set up exactly the same as for cutting right hand screw threads, except that the lathe must be arranged to feed the tool from left to right, instead of from right to left, when the spindle is revolving forward.

Fig. 235. A Left Hand Screw Advances When Turned Counter-clockwise

Use of Thread Dial Indicator

The thread dial indicator, shown in Fig. 237, is used to save time, especially when cutting long screw threads. When the lathe is set up for cutting screw threads, the thread dial indicates the relative position of the lead screw, spindle, and carriage of the lathe. This permits dis-engaging the half-nuts from the lead screw at the end of a cut, returning the carriage quickly to the starting point by hand without reversing the lathe spindle, and re-engaging the half-nuts with the lead screw at a point which will assure the tool following exactly in the original cut.

The gear on the lower end of the thread dial shaft meshes with the lead screw and any movement of the carriage or lead screw is shown by a corresponding movement of the graduated dial at the top. Provided the thread is started when a numbered graduation on the dial coincides with the witness mark on the frame, the points at which the half-nuts may be engaged for successive cuts will be indicated as follows:

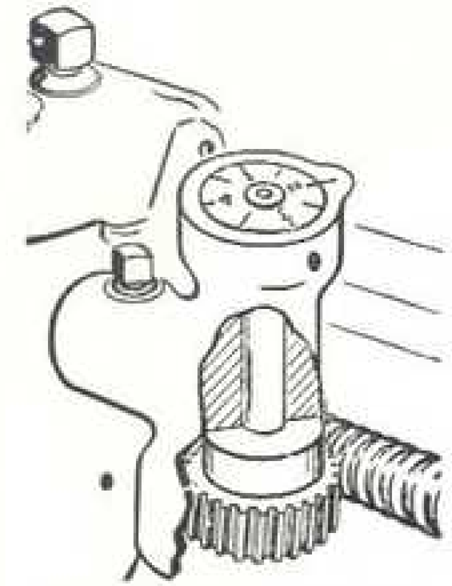


Fig. 237. Thread Dial Indicator Attached to Lathe Carriage

For all even numbered threads, close the half-nuts at any line on the dial, or each $\frac{1}{8}$ revolution.

For all odd numbered threads, close the half-nuts at any numbered line on the dial, or each $\frac{1}{4}$ revolution.

For all threads involving one-half of a thread in each inch, such as $11\frac{1}{2}$, close the half-nuts at any odd numbered line, or each $\frac{1}{2}$ revolution.

For all threads involving one-fourth of a thread in each inch, such as $4\frac{3}{4}$, return to the original starting point before closing the half-nuts.

Metric Thread Dial Indicator

Because metric threads are measured in mm pitch (the distance the thread advances each turn) instead of in threads per unit of measurement as is done in the English system, the thread dial indicator is a little more complicated. A special thread dial indicator is required for cutting metric screw threads. See page 85.

Use Oil When Cutting Threads in Steel

Lard oil (or machine oil) should be used when cutting screw threads in steel in order to produce a smooth thread. If oil is not used, a very rough finish will be caused by tearing of the steel by the cutting tool.

The oil should be applied generously preceding each cut. A small paint brush is ideal for applying the oil when cutting external screw threads, as illustrated in Fig. 238.

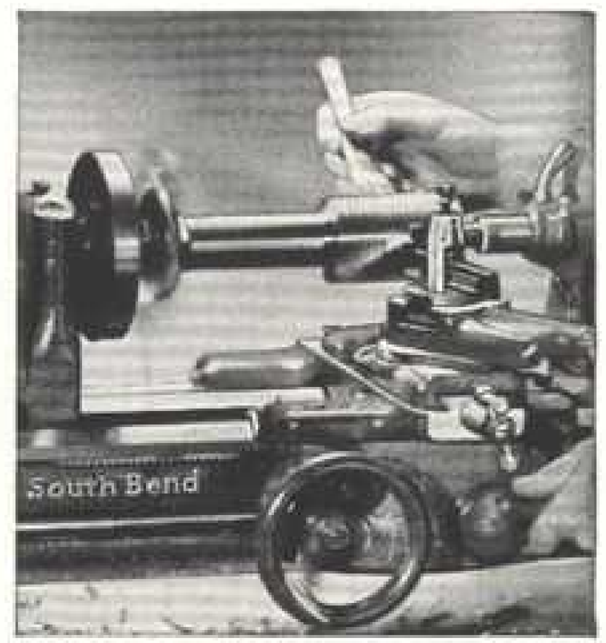


Fig. 238. Using a Small Brush to Apply Oil When Cutting Screw Threads

Tapered Screw Threads

Tapered screw threads such as pipe threads may be cut with the aid of a taper attachment, as shown in Fig. 239, or by setting over the tailstock off center, as shown in Fig. 240.

Regardless of which method is used, it is important that the thread tool is set square with the straight portion of the work, as shown in Figs. 239 and 240 and not to the tapered portion. The angle of the sides of the thread will be incorrect if the tool is not set as shown.

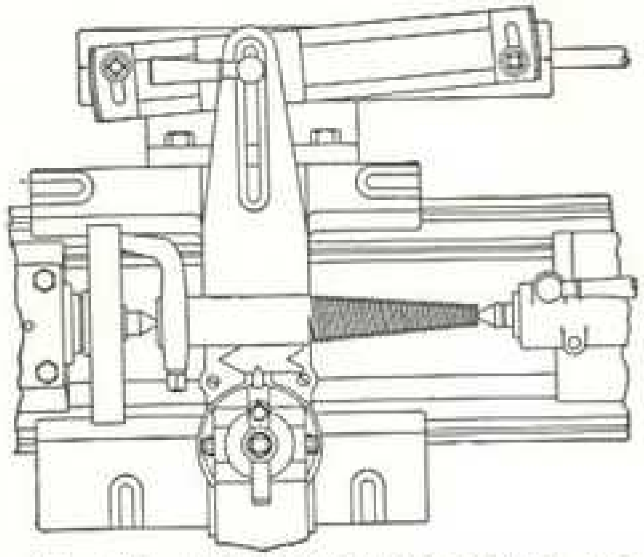


Fig. 239. Cutting Tapered Threads with Taper Attachment

Square Screw Threads

Square threads are used for vise screws, jack screws, etc. The sides of the tool for cutting square threads should be ground at an angle to conform with the helix angle of the thread, as shown in Fig. 241.

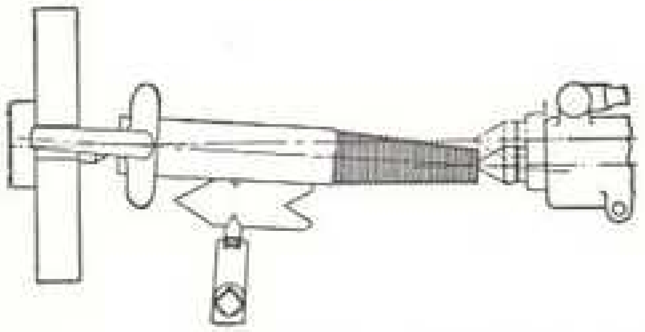


Fig. 240. Cutting Tapered Threads with Tailstock Center Set Over

To determine the helix angle of a screw thread, draw line A-C2 equal to the circumference of the thread to be cut. Draw line C2-C equal to the lead of the thread and at right angles to line A-C2. Complete the triangle by drawing line A-C. Angle B in the triangle is the helix angle of the thread. The sides of the tool E and F should be given a little clearance.

The width of the cutting edge of the tool for cutting square screw threads is exactly one-half the pitch, but the width of the tool for threading the nut should be from one thousandth to three thousandths of an inch larger, to permit a free fit on the screw.

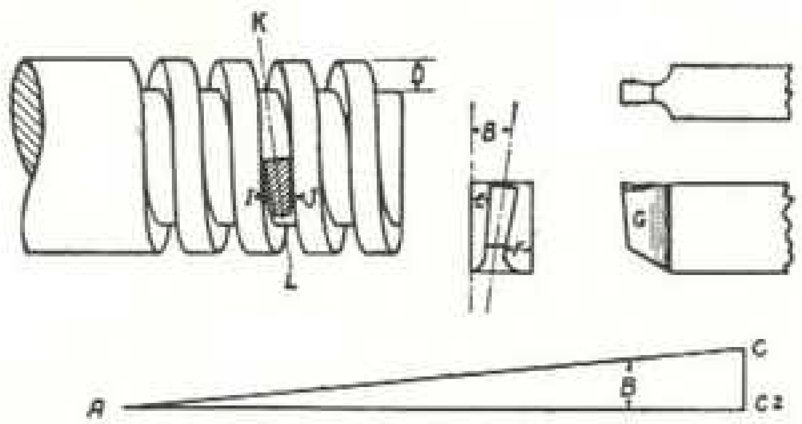
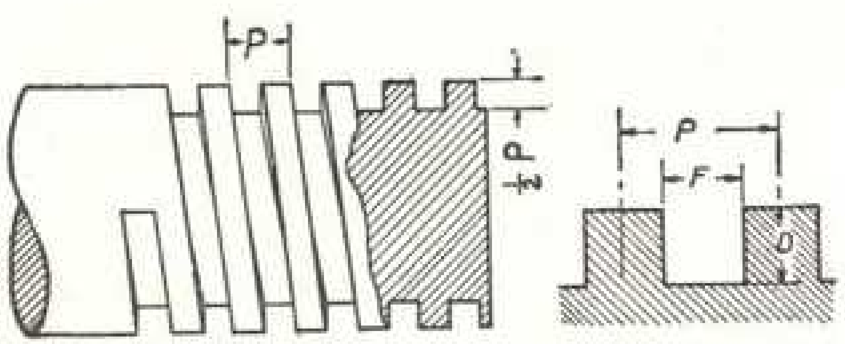


Fig. 241. Tool for Cutting Square Threads



FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

D = Depth = P x .500

F = Space = P x .500

Fig. 242. Design and Proportions of Square Screw Threads

Acme Screw Threads

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

D = Depth = ½ P + .010 In.

F = Flat = .3707 P

C = Flat = .3707 P — .0052 In.

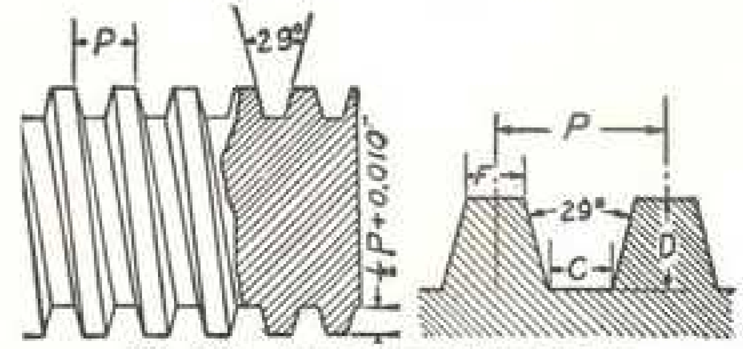


Fig. 243. Acme Screw Thread Form

Acme screw threads are used for the feed screws and adjusting screws of machine tools and machinery of all kinds. Acme threads are preferable to square threads because they are easier to cut.

While the top and the bottom of the threads are similar to a square thread in that they are flat, the sides of the thread have a 29° included angle, as shown in Fig. 243.

The method of setting a Threading Tool for cutting an Acme Thread is shown in Fig. 244.

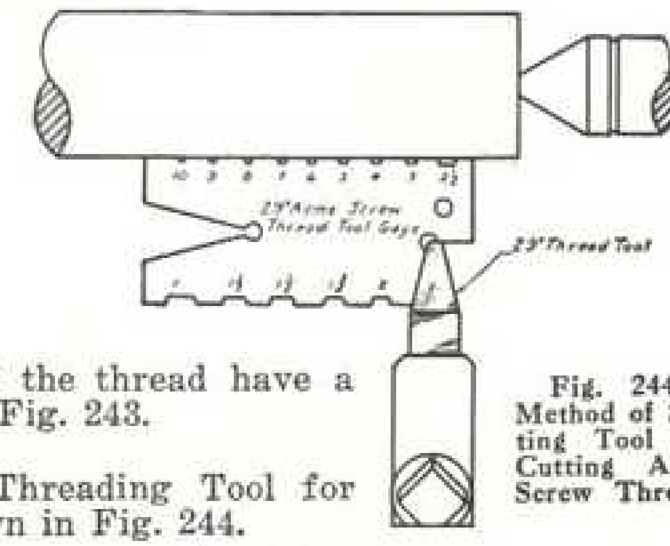


Fig. 244. Method of Setting Tool for Cutting Acme Screw Threads

29° Worm Thread
(Brown & Sharpe)

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

D = Depth = .6866 P

F = Flat = .31 P

C = Flat = .335 P

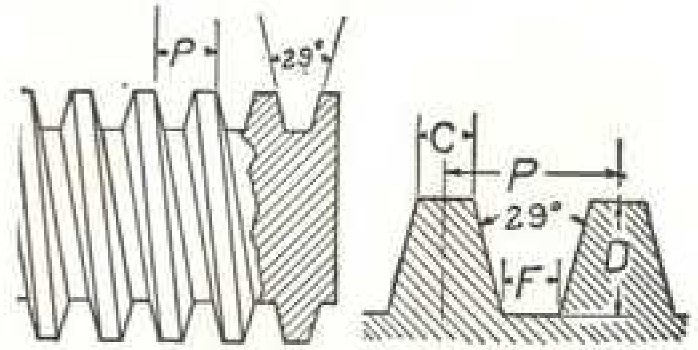


Fig. 245. 29° Worm Thread Form

A 29° Worm Thread should not be confused with the Acme Standard Thread because it differs in the depth of the thread, the width of the top of the tooth and the width of the bottom of the tooth, as shown above.

Whitworth Thread

FORMULA

$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$

D = Depth = P x .6403

R = Radius = .1373 x P

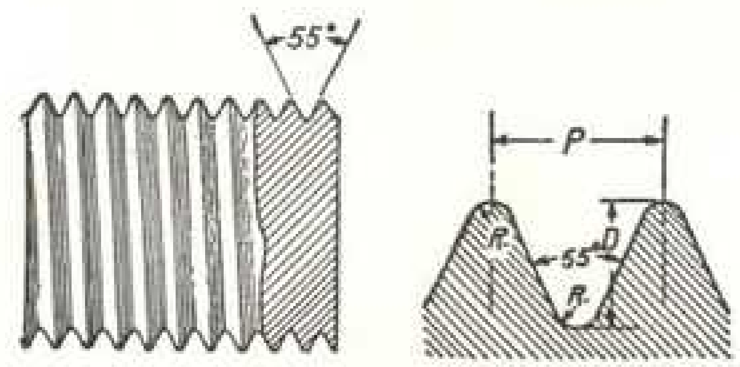


Fig. 246. Whitworth Screw Thread Form

The Whitworth thread form is used mostly in England. Great care is required in grinding the thread tool so that it will produce the radius at the top and bottom of the thread.

Metric and English Transposing Gears

When it is desired to cut both English and metric screw threads on the same lathe, transposing gears are required.

English transposing gears are used for cutting English screw threads on lathes having metric lead screws. Metric transposing gears are used for cutting metric screw threads on lathes having English lead screws.

The form of the metric thread is similar to the American National Screw Thread form, having a 60° included angle and a flat at the top of the thread, but a small radius at the root of the thread provides greater clearance. (See Fig. 250.)

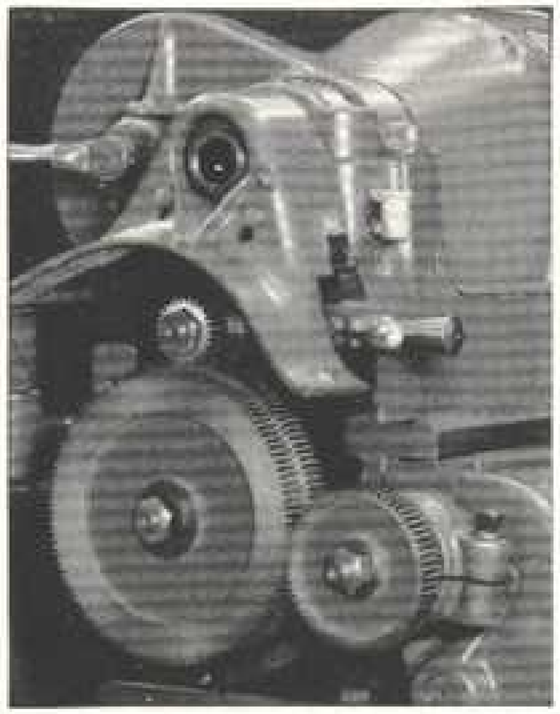
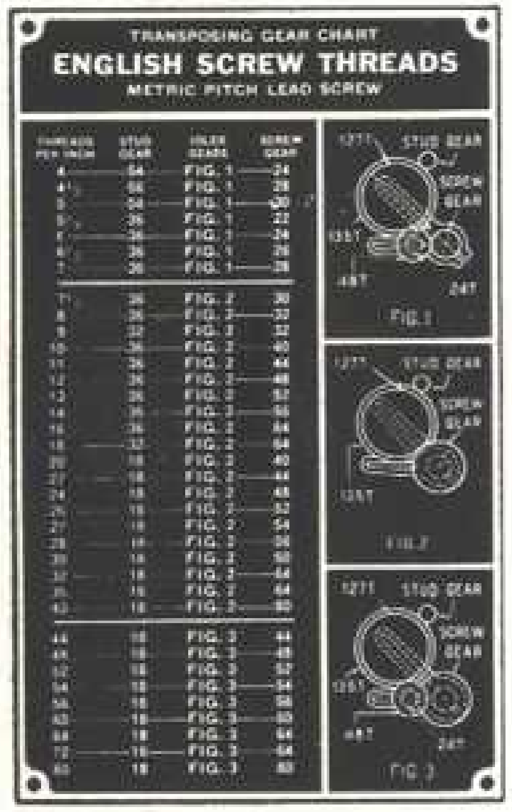
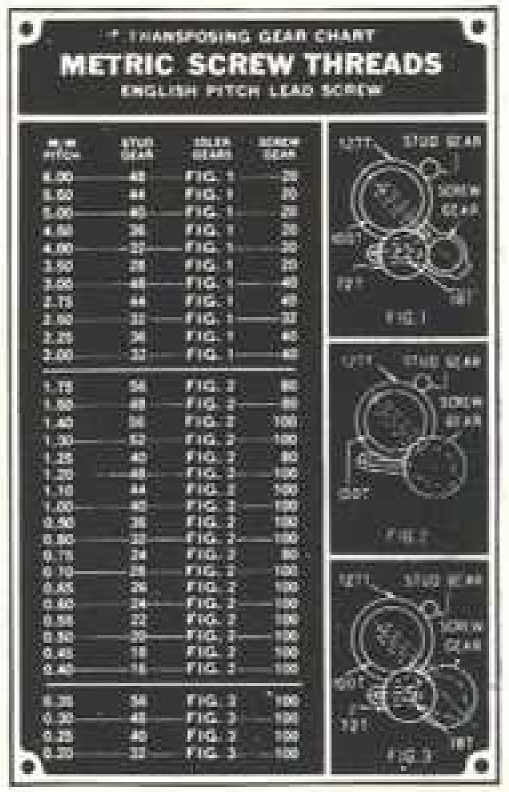


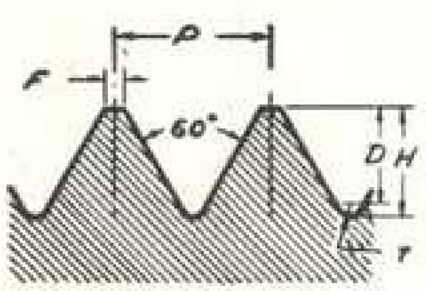
Fig. 247. Lathe Equipped with Transposing Gears



Left—Fig. 248 Index Chart Showing English Threads Cut on Metric Lathe with English Transposing Gears



Right—Fig. 249 Index Chart showing Metric Threads Cut on English Lathe with Metric Transposing Gears



FORMULA

P=Pitch in MM

D=Depth of Engagement=P x .6495

H=Depth of Thread =P x .6945

r=Maximum Radius =P x .0633

F=Flat=$\frac{P}{8}$

Fig. 250. International Standard Metric Screw Thread Form

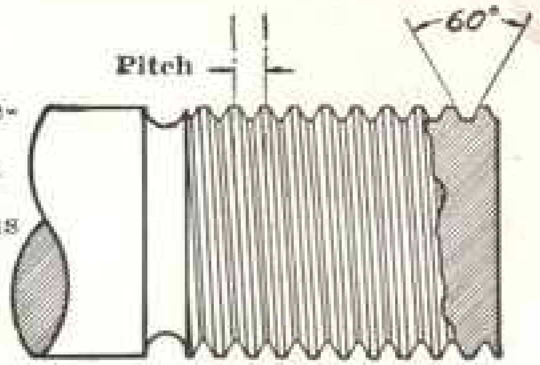
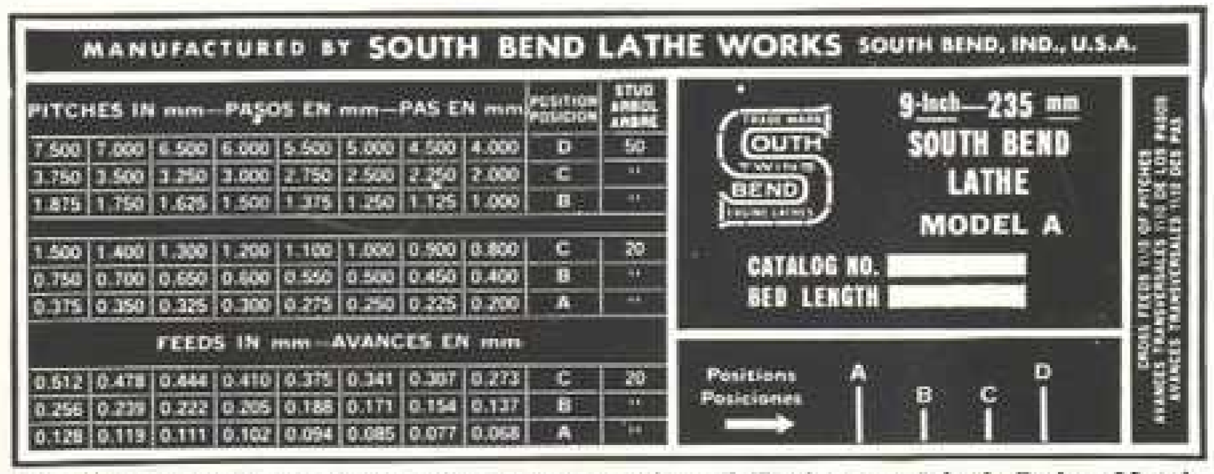


Fig. 251. 2.5 mm. Pitch Metric Screw Thread

Metric Lathe with Metric Lead Screw

Metric lathes equipped with metric lead screws are preferable in locations where metric screw threads are used exclusively. The metric lathe is identical with the English lathe, except that the lead screw, cross-feed screw and compound rest screw have metric threads, and all graduations are in the metric system.

Metric lathes are made in both the Standard Change Gear and Quick Change Gear types. Metric Quick Change Gear Lathes have a quick change gear box which permits cutting a wide range of metric screw threads and feeds, as listed on the index chart, which is illustrated below in Fig. 253. Metric Standard Change Gear Lathes have a similar range of metric screw threads and feeds.



PITCHES IN mm—PASOS EN mm—PAS EN mm								POSITION POSICION	STUD ARBOL ARBRE
7.500	7.000	6.500	6.000	5.500	5.000	4.500	4.000	D	50
3.750	3.500	3.250	3.000	2.750	2.500	2.250	2.000	C	"
1.875	1.750	1.625	1.500	1.375	1.250	1.125	1.000	B	"
1.500	1.400	1.300	1.200	1.100	1.000	0.900	0.800	C	20
0.750	0.700	0.650	0.600	0.550	0.500	0.450	0.400	B	"
0.375	0.350	0.325	0.300	0.275	0.250	0.225	0.200	A	"
FEEDS IN mm—AVANCES EN mm									
0.612	0.478	0.444	0.410	0.375	0.341	0.307	0.273	C	20
0.256	0.239	0.222	0.205	0.188	0.171	0.154	0.137	B	"
0.128	0.119	0.111	0.102	0.094	0.085	0.077	0.068	A	"

Fig. 253. Index Chart Showing Metric Threads and Feeds on a 9-inch Swing Metric Quick Change Gear Lathe

Metric Thread Dial Indicator

The thread dial used for cutting metric screw threads on lathes equipped with metric lead screws is shown in Fig. 254. To provide for the various pitches of metric threads, several gears having different numbers of teeth are mounted on the lower end of the shaft. The vertical position of the thread dial indicator is changed as required so that the correct gear for the pitch of the thread to be cut will mesh with the lead screw.

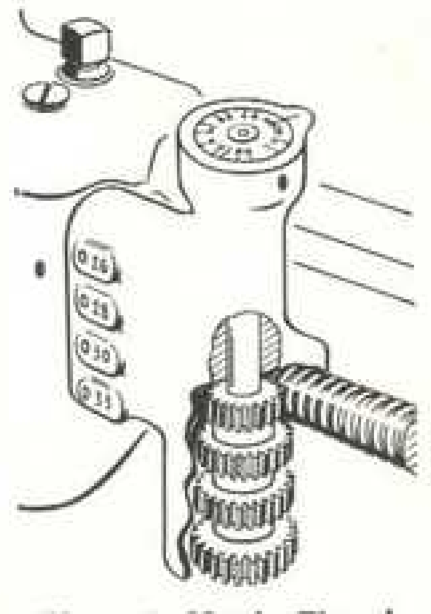


Fig. 254. Metric Thread Dial Indicator

Each graduation on the dial is marked with a letter which indicates the points at which the half-nuts may be engaged for certain threads. A chart (Fig. 254A) is supplied with the thread dial to show which gear and which graduations must be used for each pitch of metric screw thread.

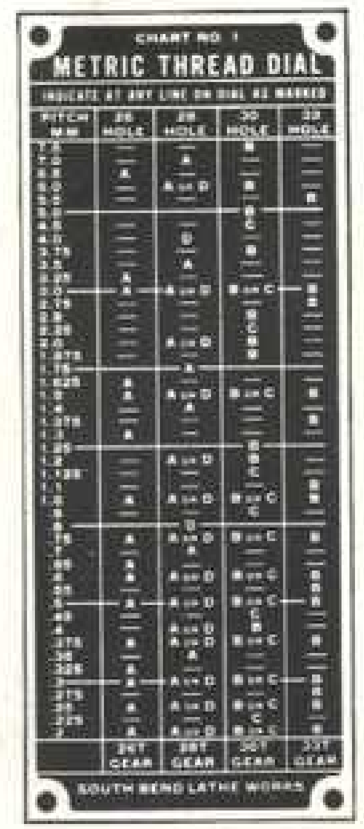


Fig. 254A. Index Chart for Metric Thread Dial

Cutting Multiple Screw Threads

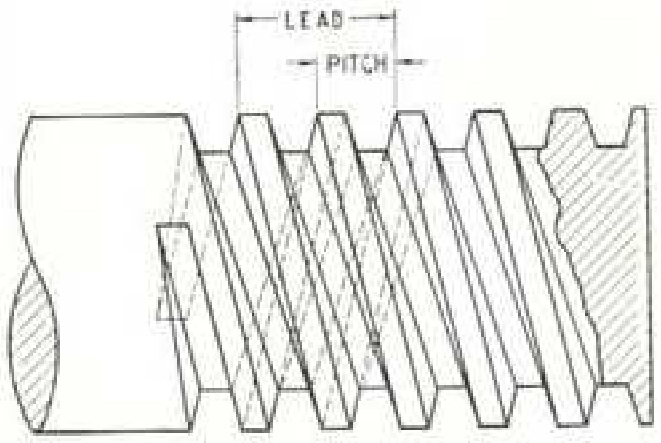


Fig. 255. A Multiple Screw Thread Having Two Grooves (Double Thread)

A multiple thread having two grooves is known as a double thread, three threads a triple thread, etc. (See Fig. 255.) The pitch and lead of a multiple thread should not be confused. The pitch is the distance from a point on one thread to the corresponding point on the next thread, while the lead is the distance a screw thread advances in one turn.

When cutting multiple threads in the lathe the first thread is cut to the desired depth. The work is then revolved part of a turn, and the second thread cut, etc. In order to obtain an exact spacing it is advisable to mill as many equally spaced slots in the face plate for the lathe dog as there are multiple threads to be cut. For a double thread, two slots; a triple thread, three slots, etc. If it is not convenient to cut slots in the face plate, equally spaced studs may be attached to the face plate and a straight tail lathe dog used.

Another method for indexing the work when cutting multiple threads is to disengage the change gears after the first thread has been completed and turn the spindle to the required position for starting the next cut.

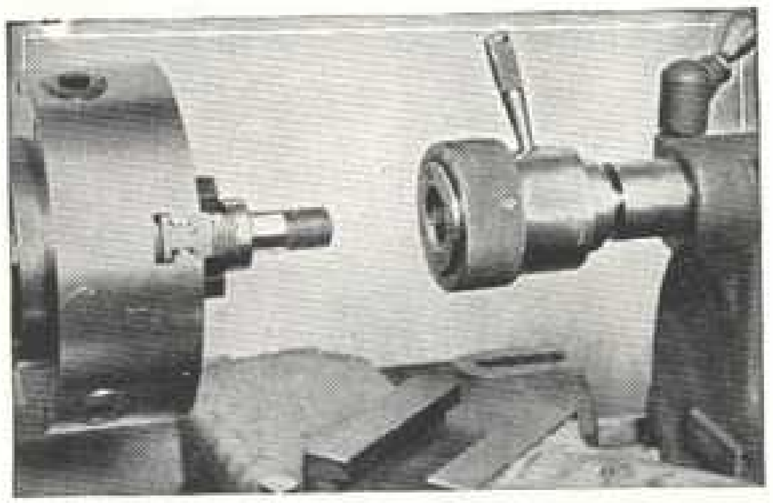


Fig. 256. Die Mounted in Tailstock of Lathe for Threading Studs

Cutting Threads with Die in Tailstock

A die may be mounted on tailstock of lathe for cutting screw threads, as shown in Fig. 256. A lathe tool may also be mounted in the tool post for turning operations or for cutting-off if desired. This method is often used for threading a large number of small pieces.

A die may be mounted on lathe carriage for cutting screw threads, as shown in Fig. 257. The lead screw and half nuts are used to feed the die so that threads with a perfect lead are obtained.

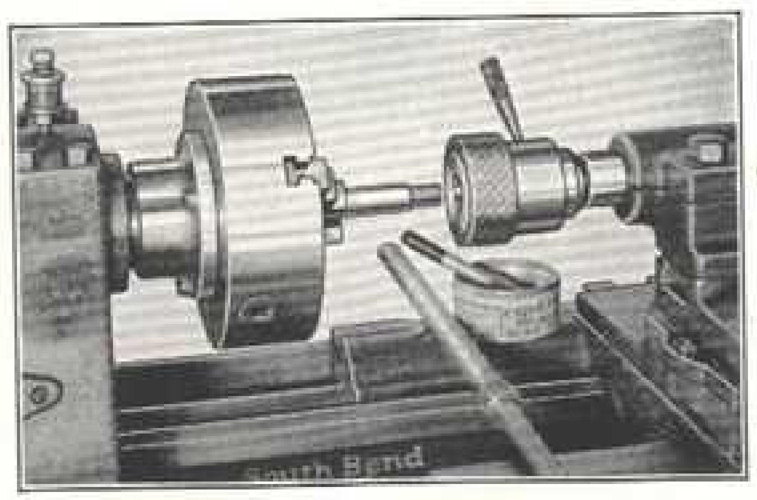


Fig. 257. Die Mounted on Lathe Carriage for Cutting Accurate Screw Threads

Chapter XI

SPECIAL CLASSES OF WORK

There are many special classes of lathe work such as knurling, filing, polishing, coil winding, etc. The most important are illustrated and described briefly on the following pages.

Knurling

Knurling is the process of embossing the surface of a piece of work in the lathe with a knurling tool (Fig. 258) in the tool post of the lathe.

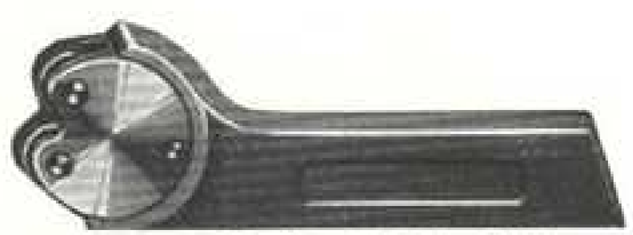


Fig. 258. Knurling Tool for Lathe

Three examples of knurling on a piece of steel are shown in Fig. 259. The pattern of the knurl is alike in all three cases but is of different grades, coarse, medium and fine.

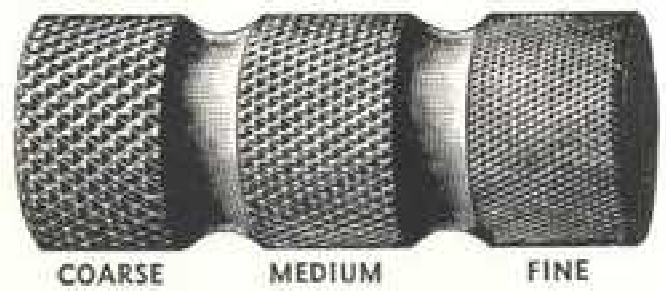


Fig. 259. Sample of Knurling

For all knurling operations the lathe should be arranged for the slowest back geared speed. After starting the lathe force the knurling tool slowly into the work at the right end until the knurl reaches a depth of about $\frac{1}{64}$ in. Then engage the longitudinal feed of the carriage and let the knurling tool feed across the face of the work. Plenty of oil should be used on the work during this operation.

When the left end of knurl roller has reached the end of work, reverse the lathe spindle and let the knurling tool feed back to the starting point. Do not remove the knurling tool from the impression but force it into the work another $\frac{1}{64}$ in., and let it feed back across the face of the work. Repeat this operation until the knurling is finished.

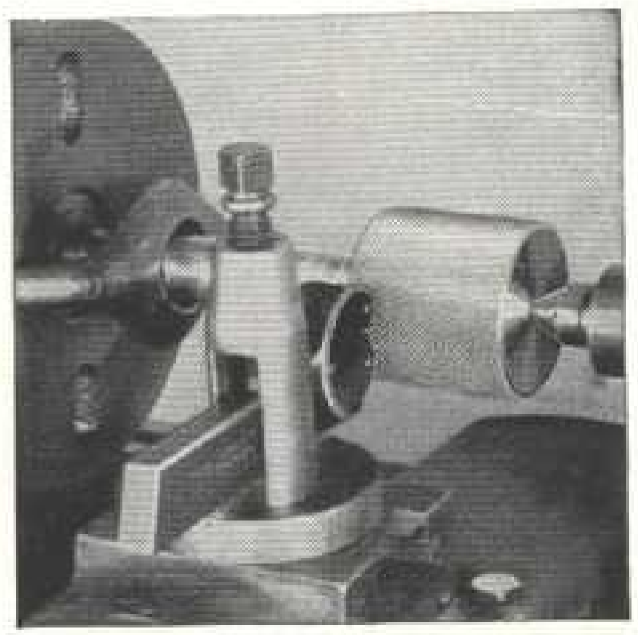


Fig. 260. Knurling a Steel Piece in the Lathe

Machining Work on the Face Plate

Before mounting a face plate on the spindle nose of the lathe, all dirt and chips should be removed from the threaded hole. Also clean the thread and the shoulder on the spindle nose because any dirt, chips or burrs will prevent the face plate from running true.

Oil the threads of the spindle so that the face plate will screw on easily and can be easily removed. If it seems difficult to screw on the face plate, unscrew the plate, remove the dirt, burrs, etc., and try again. The face plate hub should screw tight against the shoulder of the spindle but the face plate should not be spun up to the shoulder suddenly as this makes removal difficult.

The face plate is especially valuable in tool room work for machining holes in tools and jigs. In this class of work the holes must be accurately spaced, with an allowance usually not more than .001 of an inch.

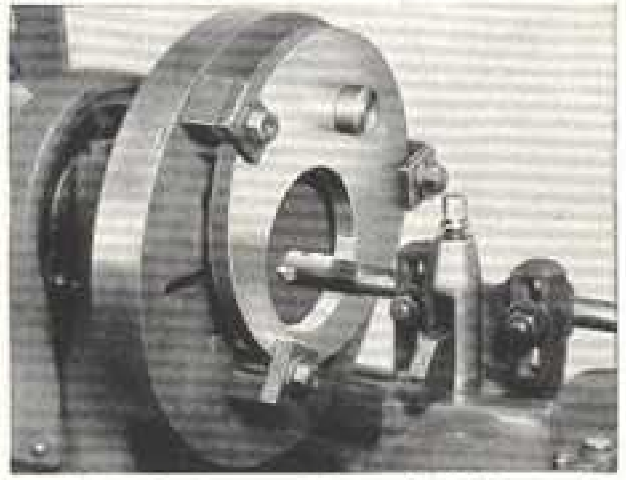


Fig. 261. Boring an Eccentric Hole on the Face Plate of the Lathe

Clamping Work on Face Plate

Some care should be exercised when clamping work on the face plate so that neither the work nor the face plate will be sprung. A piece of paper placed between the face plate and the work will reduce the danger of the work slipping. Balance weights should be used as shown in Fig. 264.

Centering the Work

A center indicator may be used, as shown in Fig. 263, for accurately locating work on the face plate for drilling and boring. A dial indicator may also be used, as shown in Fig. 264.

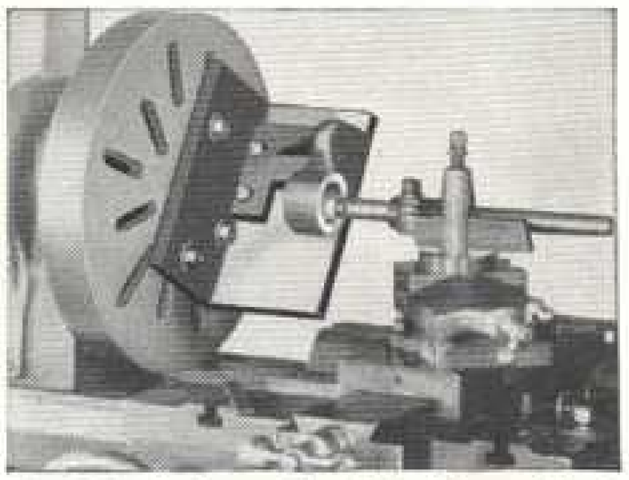


Fig. 262. Boring a Bracket with an Angle Plate Attached to the Face Plate

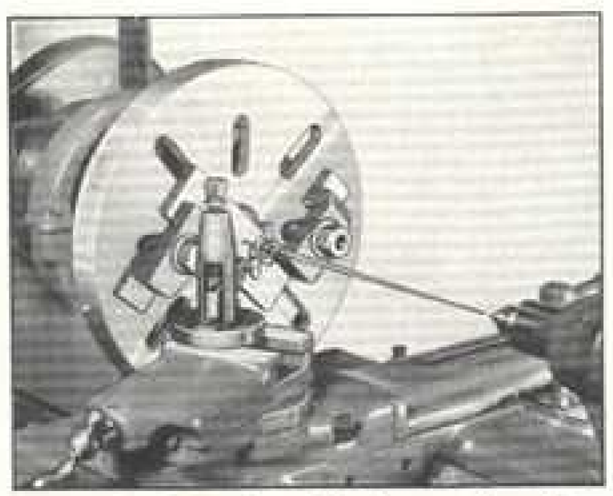


Fig. 263. Centering Work on the Face Plate with a Center Indicator

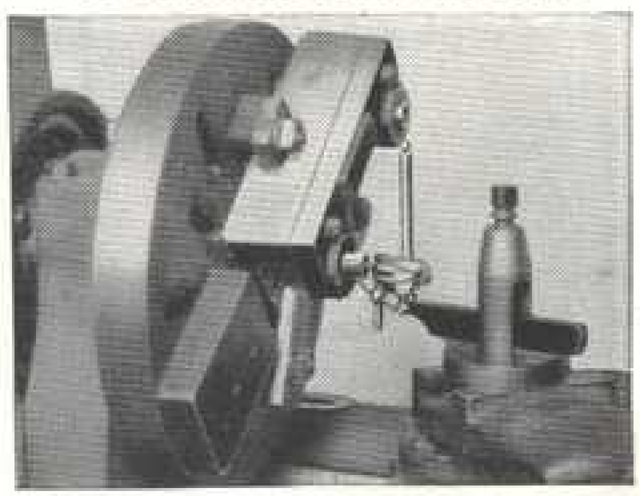


Fig. 264. Locating Work on the Face Plate with a Dial Indicator

Filing and Polishing

All tool marks can be removed and a smooth, bright finish obtained on the surface of a piece of work by filing and polishing, as shown in Figs. 265 and 266.

Use a fine mill file and file with the lathe running at a speed so that the work will make two or three revolutions for each stroke of the file. File just enough to obtain a smooth surface. If too much filing is done the work will be uneven and inaccurate. See page 119.

Keep the left elbow high and the sleeves rolled up so there will be no danger from the lathe dog.

Keep the file clean and free from chips, using a file card frequently.

A very smooth, bright finish may be obtained by polishing with several grades of emery cloth after filing. Use oil on the emery cloth and run the lathe at high speed. Be careful not to let the emery cloth wrap around the revolving work.

Fig. 265. Filing to Remove Tool Marks

Fig. 266. Polishing with Emery Cloth and Oil

Lapping

Hardened gauges, bushings and bearings are often finished in the lathe by lapping, as shown in Fig. 267. Emery cloth, emery dust and oil, diamond dust and other abrasives are used. Usually the lathe spindle is operated at high speed. See page 119.

The lap may be very simple, consisting of a strip of emery cloth attached to a shaft, or it may be elaborately constructed of lead, copper, cast iron, etc. Some very fine and precise work may be accomplished by careful lapping.

Fig. 267. Lapping the Inside of a Hardened Steel Bushing with Emery Dust and Oil

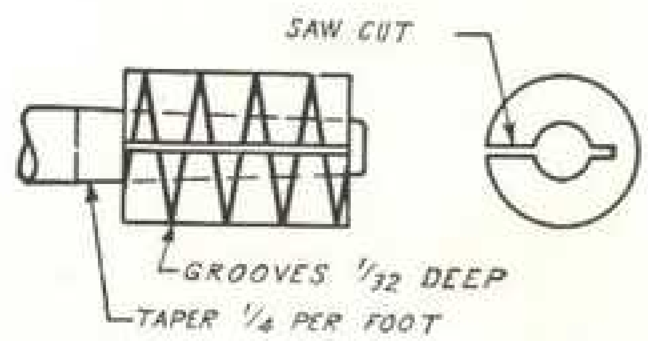


Fig. 268. A Cast Iron Lap for Emery Dust

Fig. 269. A Steel Lathe Mandrel

Machining Work on a Lathe Mandrel

Cylindrical work that has been bored and reamed in a chuck is usually further machined on a mandrel between the lathe centers, as shown in Figs. 270 and 271. The mandrel is slightly tapered and must be driven into the hole tight enough so that the work will not slip on the mandrel while it is being machined.

Large diameter work such as pulleys should be driven with a pin or driver attached to the lathe face plate if it can be arranged as this will eliminate possibility of the work slipping on the mandrel.

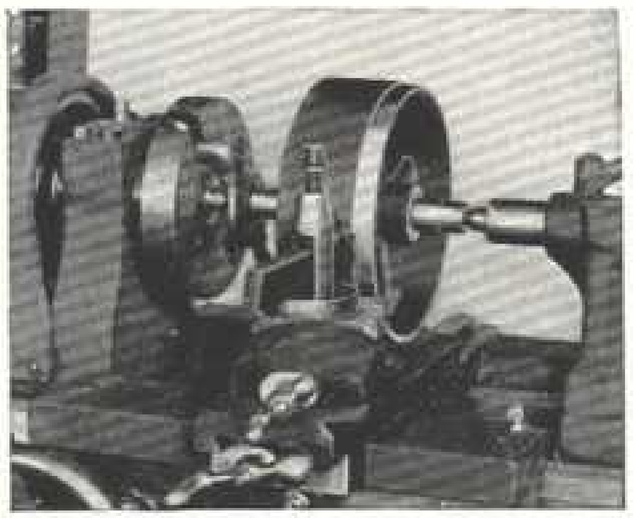


Fig. 270. Turning a Pulley on a Mandrel

Before driving the mandrel into the hole in the work, oil both the mandrel and the hole so that the work will be easy to remove from the mandrel. If there is no lubricant on the mandrel it may "freeze" in the work, in which case it cannot be driven out without ruining both the work and the mandrel.

In driving a mandrel out of a piece of work be sure that it is driven in the opposite direction from that which it entered the work.

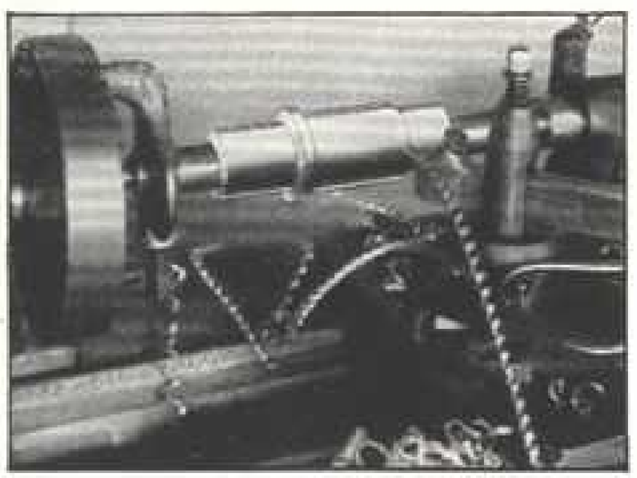


Fig. 271. Finishing a Bushing on a Mandrel

Standard lathe mandrels can be purchased in the various sizes. These mandrels are hardened and tempered and the surface that receives the work is ground usually to a taper of about .006 in. per foot.

In the case of special jobs having odd diameter holes, a soft mandrel may be used, turning and filing it to the proper diameter and tapered for a driving fit in the hole in the work. See page 102.

Special Mandrels

Special types of mandrels are often used for special classes of work. A nut mandrel for finishing the outside diameter of gear blanks is shown in Fig. 272. Expansion mandrels of various types are also available and are used where there is considerable variation in hole sizes.

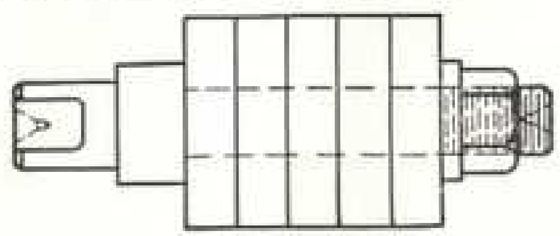


Fig. 272. Nut Mandrel for Finishing Gear Blanks

Winding Coils in the Lathe

The unusually wide range of positive power longitudinal feeds available on the lathe make it an ideal machine for winding electrical coils of all kinds. A revolution counter may be attached to register the number of turns, as shown in Fig. 273. Special gearing may be obtained for odd leads not in the usual thread cutting range of the lathe. Any type of coil form or wire guide required may be used.

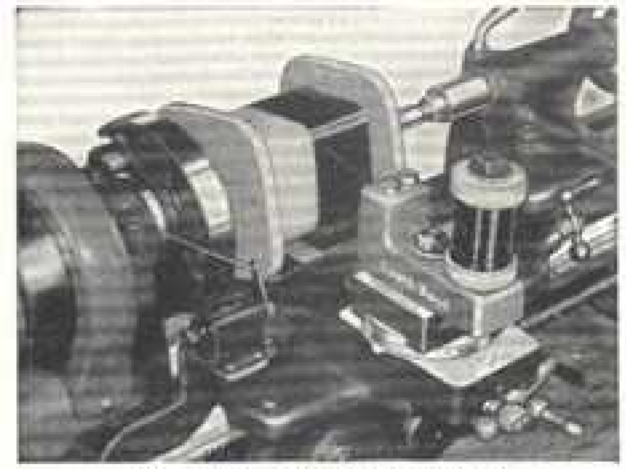


Fig. 273. Winding a Coil

Spring Winding

Coil springs of all kinds may be wound on the lathe, as shown in Fig. 274. Special mandrels are used for irregular shaped springs. The lead screw and half nuts of the lathe are usually used to obtain a uniform lead so that the coils are all equally spaced.

Fig. 274. Winding a Spring

Boring Work Mounted on Lathe Carriage

Large work may be mounted on the lathe carriage for boring, as shown in Fig. 277.

The boring bar is held between centers and driven by a lathe dog. The work is clamped to the top of the lathe saddle and is fed to the tool by the power longitudinal feed of the carriage.

Several good types of boring bars for this class of work are shown in Figs. 275, 276, and 278.

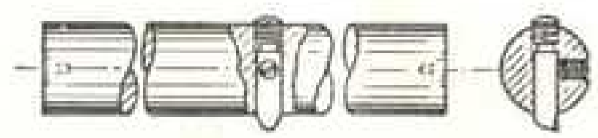


Fig. 275. Boring Bar with Fly Cutter

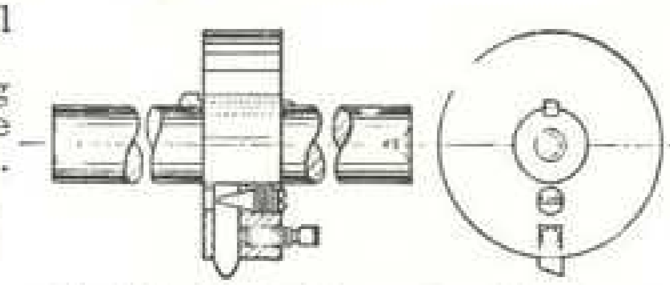


Fig. 276. Boring Bar with Boring Head

Fig. 277. Boring on the Lathe Carriage

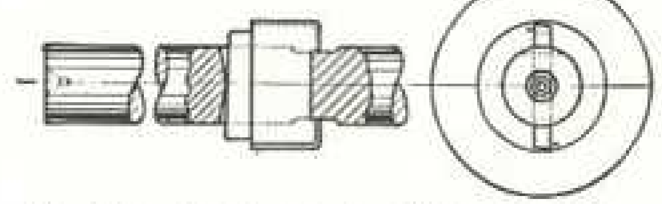


Fig. 278. Boring Bar for Sizing the Hole

Use of the Steady Rest (Center Rest)

The steady rest (or center rest) is used to support long slender shafts as shown in Fig. 280, or to support the outer end of shafts or spindles for drilling, facing, boring and similar operations as shown in Fig. 281. A steady rest with rectangular jaws is shown in Fig. 279. An improved steady rest with telescoping jaws is shown in Fig. 280.

To mount work in the steady rest, first clamp the open steady rest on the bed as shown in Fig. 279, then mount the work in the lathe. Close and lock the top portion of the steady rest. The jaws may then be adjusted to the work, which should be round and smooth at point of contact. Careful adjustment is necessary, as all three jaws should just touch the work. If one of the jaws is moved in too far, the work will be sprung and cannot be accurately machined. Make sure the work turns freely. Oil the jaws and work before starting the lathe.

For drilling, boring, and similar operations, one end of the work may be held in the chuck as shown in Figs. 281 and 282. However, for fine, accurate work most mechanics prefer to mount the left end of the work on the headstock center as shown in Fig. 283. To do this, the face plate is first unscrewed several turns. The work with lathe dog attached is then tied securely to the face plate with strong rawhide belt lacing, and the face plate is screwed onto the spindle. This tightens the lacing on the work and holds it firmly against the center point.

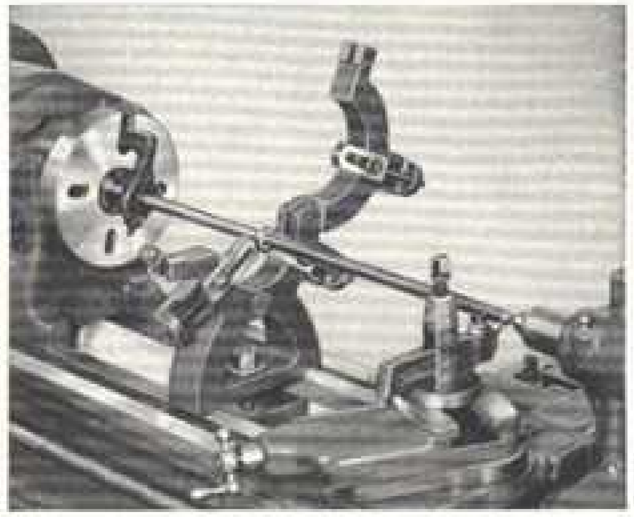


Fig. 279. Steady Rest Mounted on Lathe Bed with Top Open to Receive Work

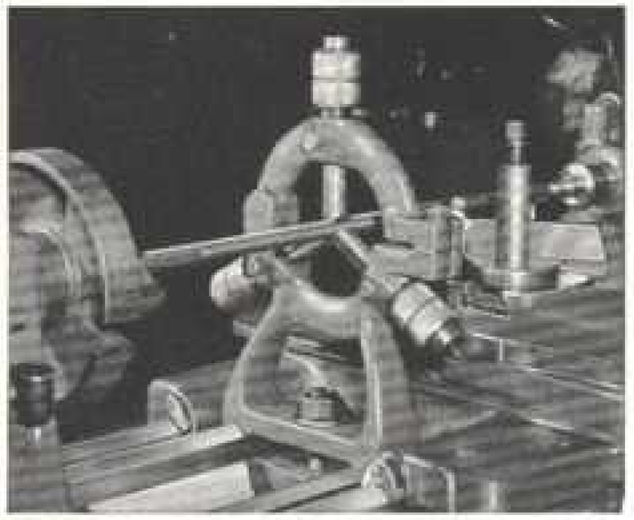


Fig. 280. Telescoping Jaw Steady Rest with Jaws Adjusted to Support Work

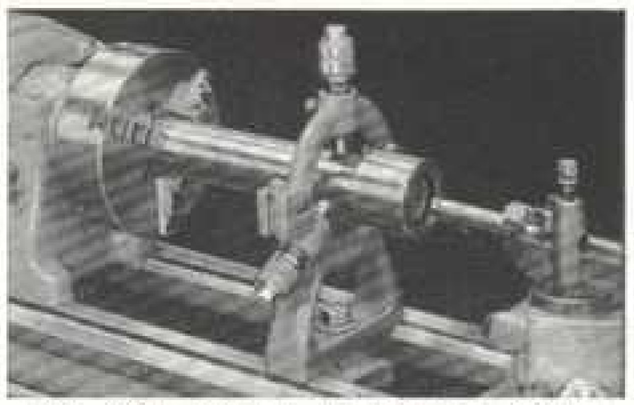


Fig. 281. Cutting an Internal Screw Thread with Work Supported in Steady Rest

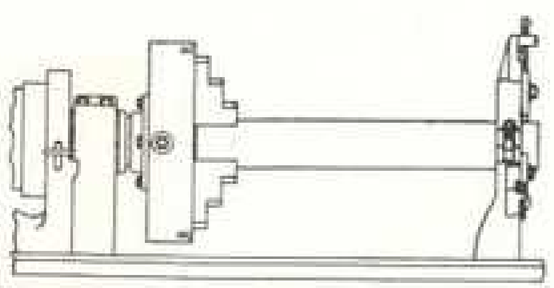


Fig. 282. Work Mounted in Chuck and Supported by Steady Rest

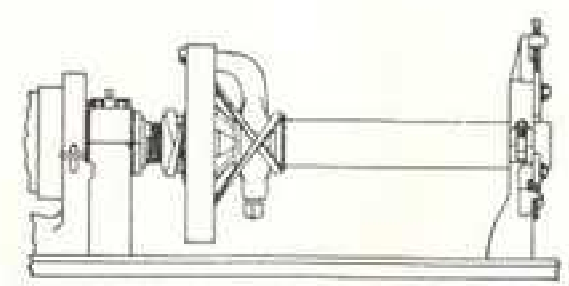


Fig. 283. Work Mounted on Center and Supported by Steady Rest

The Use of the Follower Rest

The follower rest is attached to the saddle of the lathe to support work of small diameter that is liable to spring away from the cutting tool. The adjustable jaws of the follower rest bear directly on the finished diameter of the work, as shown in Figs. 284 and 285. As the tool feeds along the work, the follower rest being attached to the saddle travels with the tool.

For the machining of small shafts in quantity, small rollers are sometimes substituted for the rigid adjustable jaws, and the device is then known as the roller bearing follower rest.

The application of both the steady rest and follower rest at the same time is shown in Fig. 286. The shaft to be machined, while very small in diameter is of considerable length, and in order to do an accurate job it is necessary to support the shaft with both the steady rest and follower rest.

Fig. 284. Threading a Long Slender Shaft with the Aid of a Follower Rest

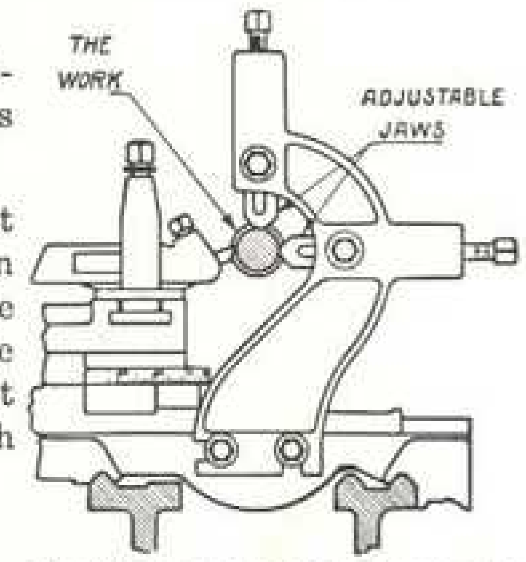


Fig. 285. Follower Rest Mounted on Lathe Saddle

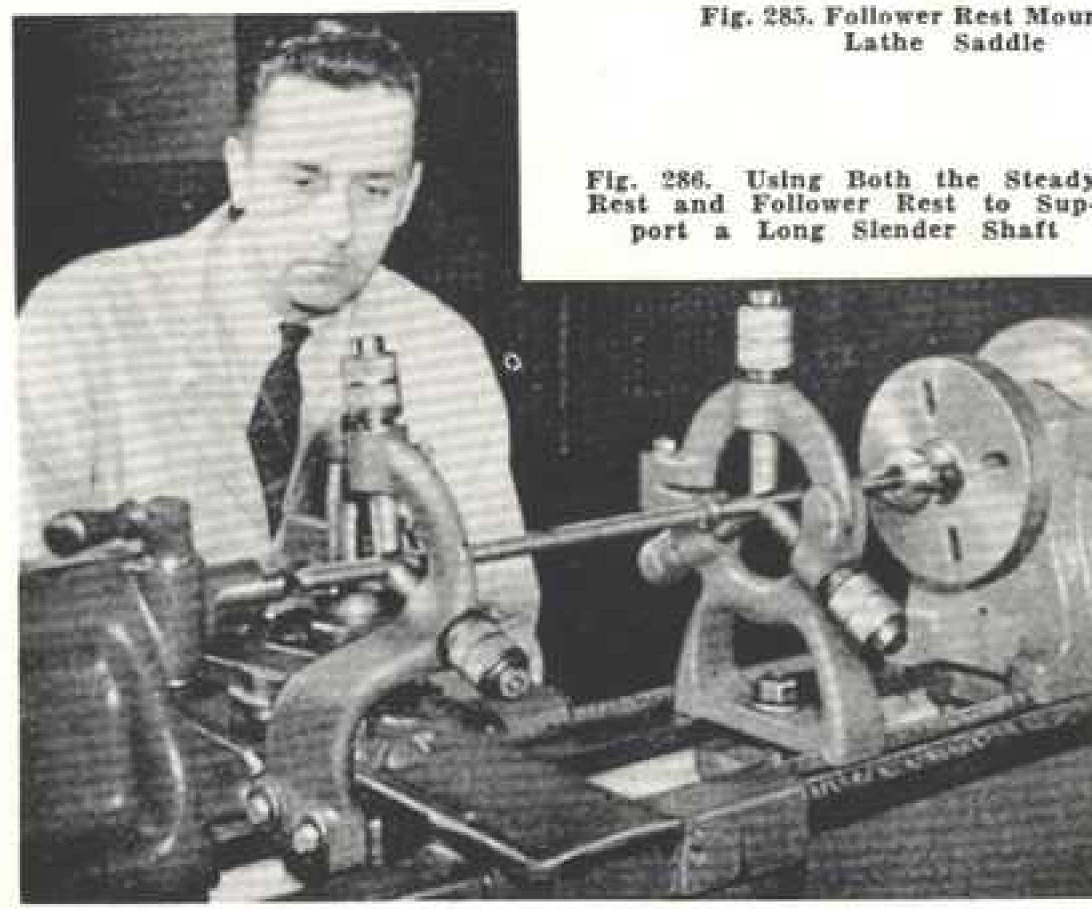


Fig. 286. Using Both the Steady Rest and Follower Rest to Support a Long Slender Shaft

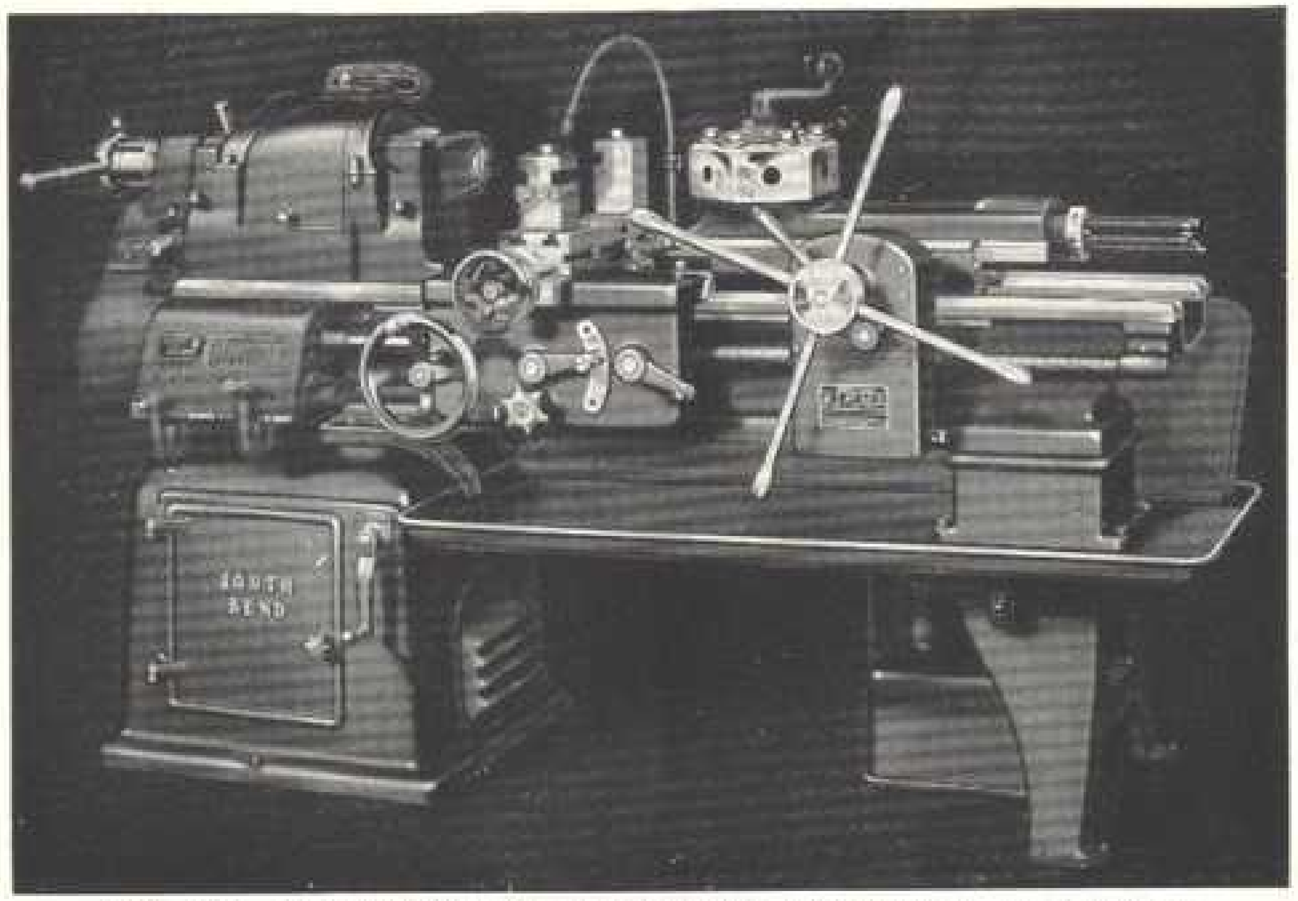


Fig. 287. No. 2-H South Bend Turret Lathe with Power Feed Bed Turret

Turret Lathes for Manufacturing

Turret Lathes are designed for the efficient production of duplicate parts. They are equipped with a power feed or hand lever feed turret having six faces, with automatic indexing and individual stop for each face. Cutting tools may be mounted in each of the six turret faces and indexed into position as required for performing various operations.

Turret lathes are usually equipped with either screw feed or hand lever operated double tool rest on the carriage cross slide. This permits using front and back tools for turning, facing, cutting off, and similar operations. A four-way turret tool post may be used on the cross slide.

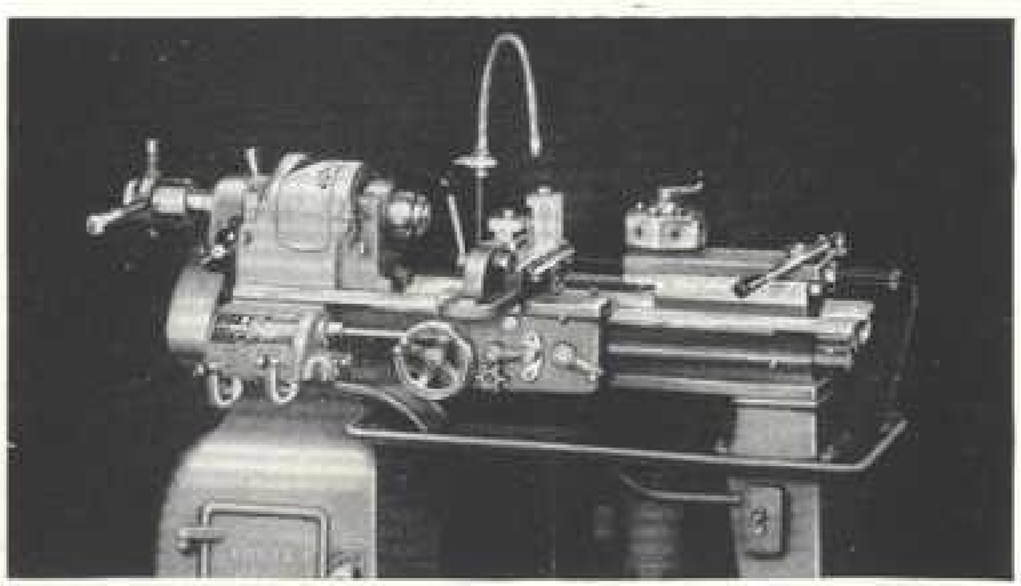


Fig. 288. 10-inch South Bend Turret Lathe with Hand-lever Operated Bed Turret

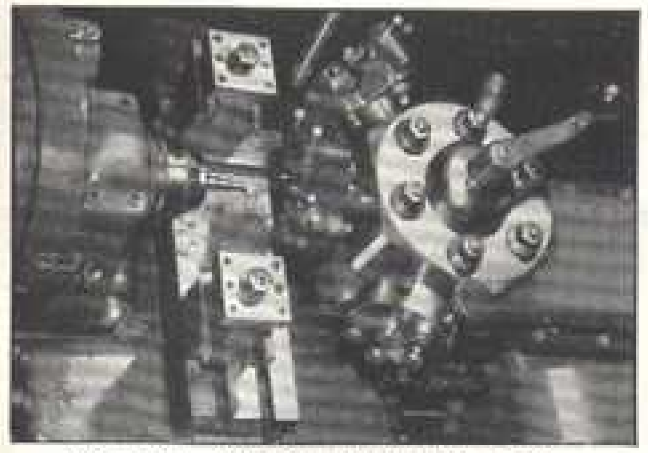


Fig. 289. Tooling on Turret Lathe

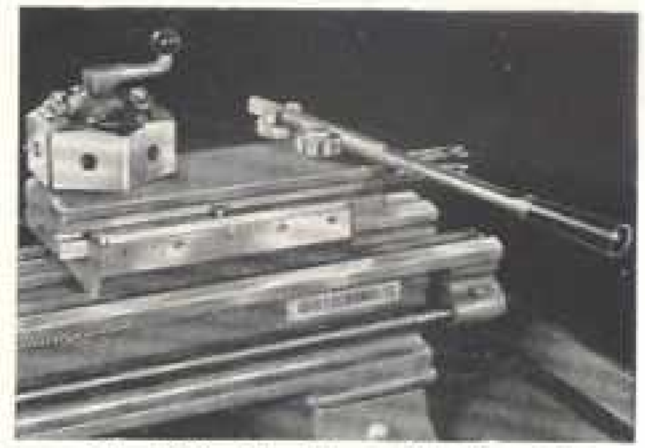


Fig. 290. Handlever Bed Turret

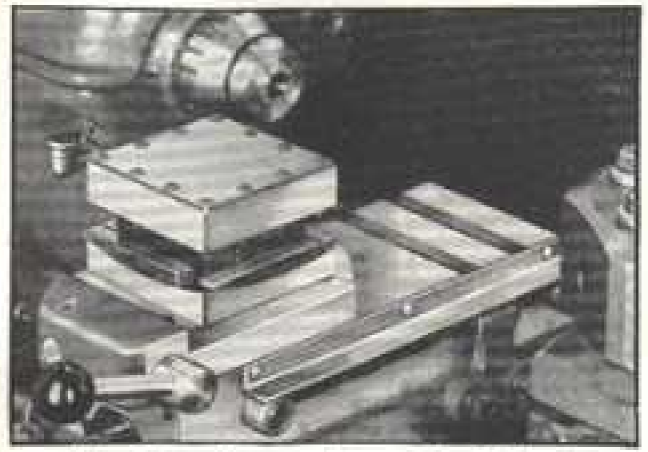


Fig. 291. Square Turret Tool Block

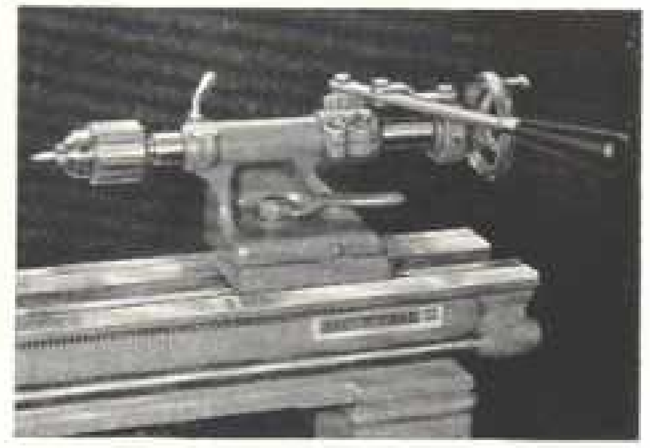


Fig. 292. Handlever Tailstock

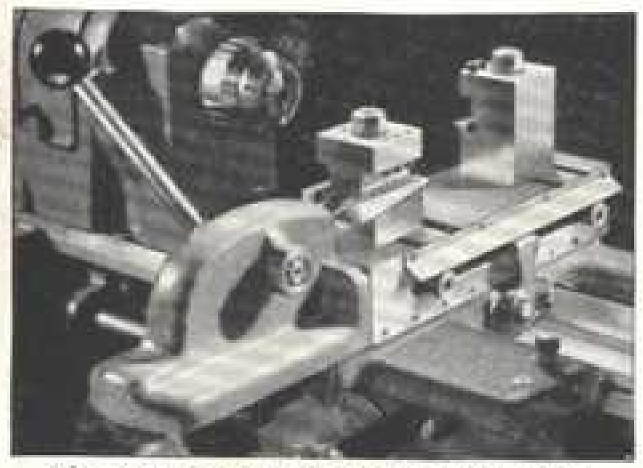


Fig. 293. Handlever Double Tool Rest

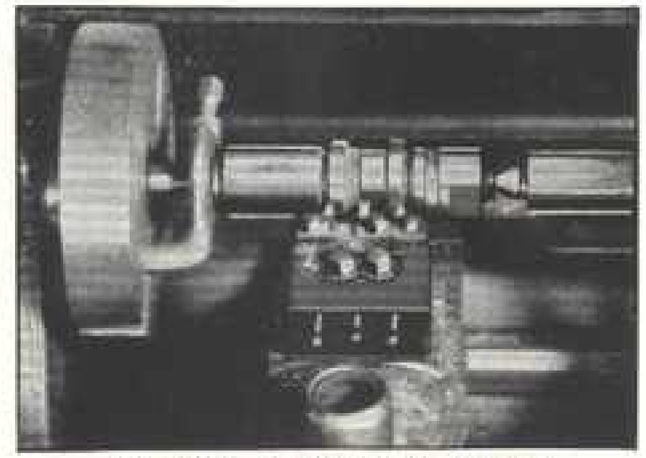


Fig. 294. Multiple Tool Block

Fig. 295. Two Tools Cutting Simultaneously on a Piece of Work

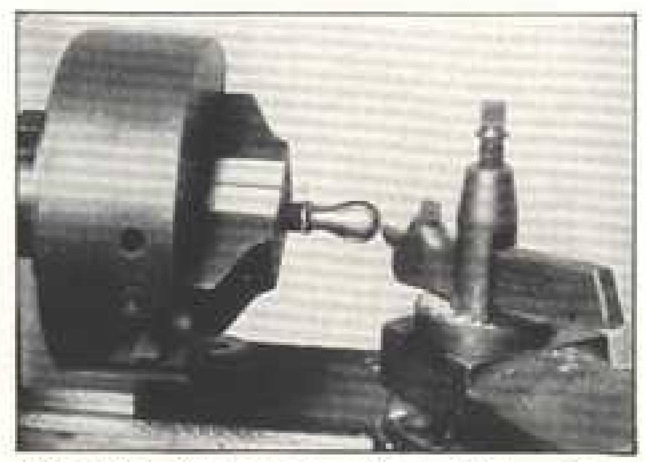


Fig. 296. An Irregular Shaped Piece Held in a Two-Jaw Chuck

Milling in the Lathe

The Milling and Keyway Cutting Attachment illustrated in Figs. 298 and 300 will take care of a great deal of milling in the small shop that does not have enough work to install an expensive milling machine.

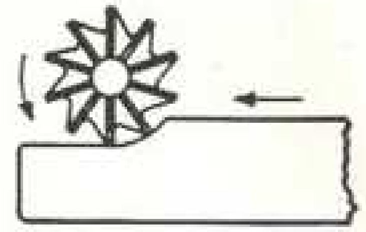


Fig. 297. Direction of Feed for Milling Operations

The cut is controlled by the hand wheel of the lathe carriage, the cross feed screw of the lathe and the vertical adjusting screw at the top of the milling attachment.

All milling cuts should be taken with the rotation of the cutter against the direction of the feed, shown in Fig. 297.


Fig. 298. Milling a Standard Keyway in a Shaft

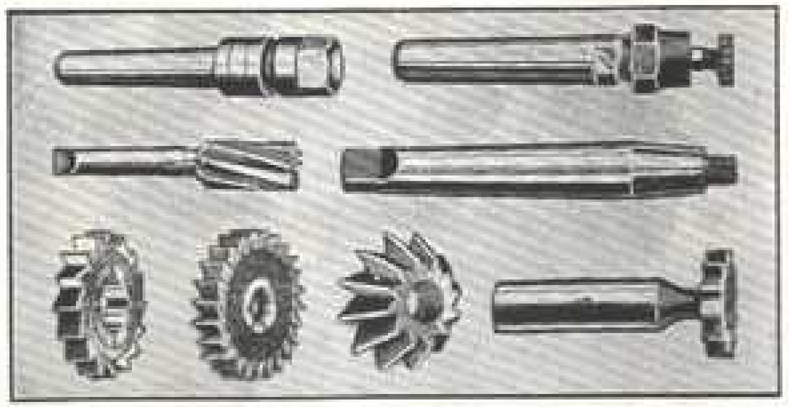


Fig. 299. An Assortment of Milling Cutters and Arbors

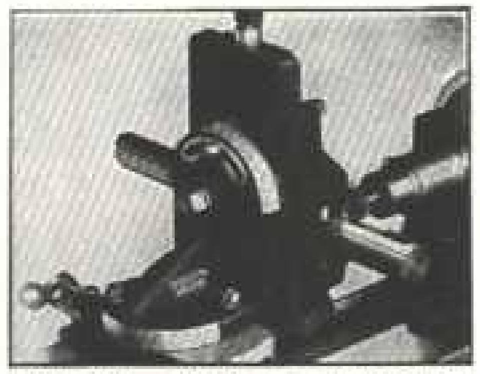


Fig. 300. Milling a Woodruff Keyway in a Shaft

Standard Keyways

The recognized standards for the depth and width of keyways in pulleys, gears, etc. are shown in Fig. 301 and the tabulation below. The same specifications are used for the depth and width of keyways in shafts.

The key should fit snugly in the keyway but must not be too tight.

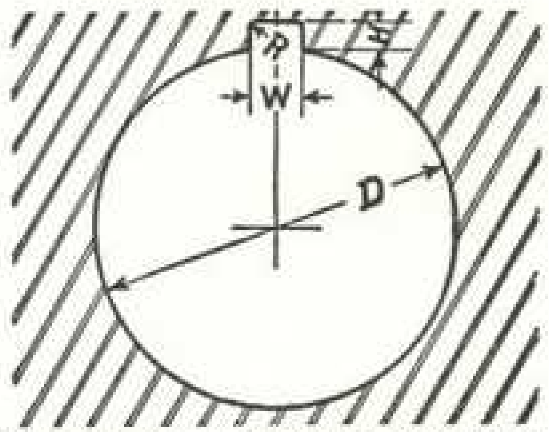


Fig. 301. Standard Keyway Dimensions

Specifications of American Standard Keyways

Diameter Hole D Inches	Width W Inches	Depth H Inches	Radius R Inches	Diameter Hole D Inches	Width W Inches	Depth H Inches	Radius R Inches
½	3/32	3/64	.020	2½	⅝	7/32	1/16
⅝ to ⅞	⅛	1/16	1/32	3	¾	¼	3/32
1	¼	3/32	3/64	3½	⅞	5/16	3/32
1¼	5/16	⅛	1/16	4	1	5/16	3/32
1½	⅜	5/32	1/16	4½	1⅛	7/16	⅛
1¾	7/16	3/16	1/16	5	1¼	½	⅛
2	½	3/16	1/16				

Cutting Gears on the Lathe

The gear cutting attachment for the lathe, shown in Fig. 302, will cut spur and bevel gears of all kinds. It will do graduating and milling, external key seating, cutting at angles, splining, slotting and all regular dividing head milling work.

This attachment is practical for cutting small gears and for milling small light work of various kinds on the screw cutting lathe.

The dividing head construction is based on the principle of interchangeable gears, the same as regularly used on gear cutting machines. The index plate shows the proper gears to use for divisions from 2 to 360.

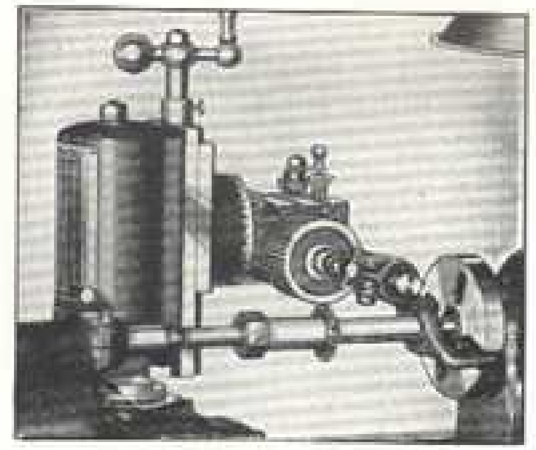


Fig. 302. Gear Cutting Attachment

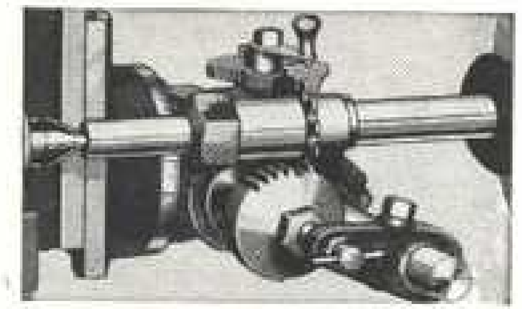


Fig. 303. Cutting a Gear on a Lathe

Turning Wood, Fiber, and Plastics

Turning wood in a metal working lathe is a very simple matter. Spur and cup centers are substituted for the 60° centers, a hand rest is attached and the lathe is ready for wood turning.

Special pulleys may be used on the motor and countershaft to provide a series of high spindle speeds for wood turning, in addition to the regular speeds for metal work.

Other materials may be machined as well. Alabaster, Catalin, Bakelite, fiber and other plastics, synthetic resins, etc., may be turned and polished with complete satisfaction.


Fig. 304. Hand Rest

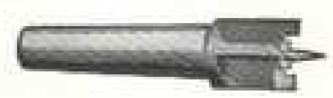
Fig. 305. Spur Center


Fig. 306. Cup Center



Fig. 307. Wood Turning in a Metal Working Lathe

Ten in One Tool Holder

The Ten in One Tool Holder replaces the conventional tool post and various tool holders ordinarily used for general lathe work. It provides rigid support for turning, boring, threading, and cutting-off tool bits. In addition, it is equipped with a self-aligning knurling head. Screw adjustments for tool height are easily made, and they stay put. No readjustment is required when replacing tools. The application of the Ten in One Tool Holder on various types of operations is shown in Fig. 308, and in the illustrations below.

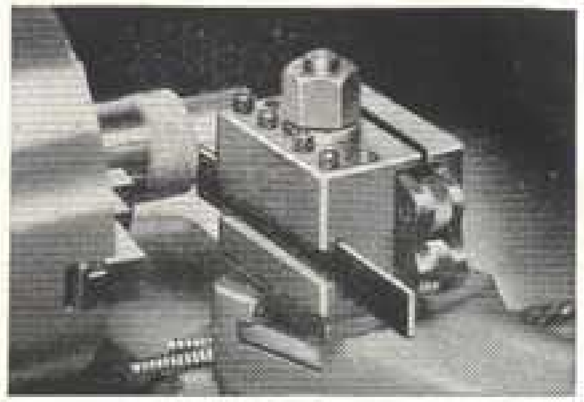
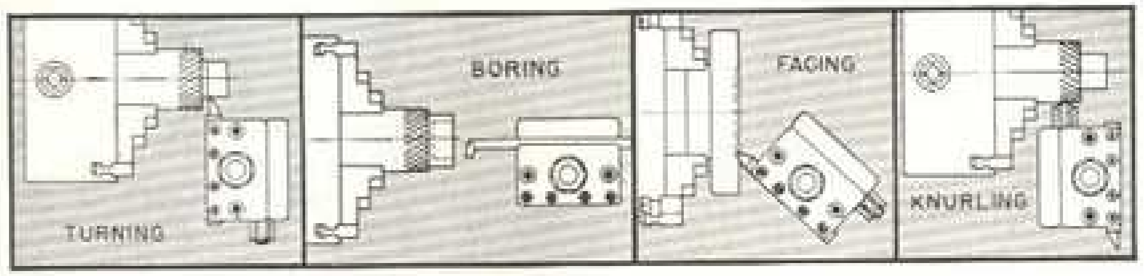


Fig. 308. Ten in One Tool Holder



Machinability of Various Grades of Steel

The machinability or free cutting quality of steel varies with the chemical analysis and the physical processes employed in its manufacture. The machinability ratings of popular grades of steel are listed in the table below. These ratings have been calculated to show the per cent of the relative cutting speeds based on a machinability rating of 100 for Bessemer Screw Steel No. B1112. The table shows the approximate cutting speeds in surface feet per minute for high speed steel cutting tools on average work. Higher or lower cutting speeds may be found more practical, depending on the cutting tool used, the rate of feed, the depth of cut, and the type of operation performed. See also pages 36 and 50.

Machinability Ratings and Cutting Speeds for Steel

AISI Number	Machinability Rating	Approximate Cutting Speed F.P.M.	AISI Number	Machinability Rating	Approximate Cutting Speed F.P.M.
B1111	94	155	3130*	72	120
B1112	100	165	3135*	70	115
B1113	136	225	3140*	66	110
C1117	91	150	4130*	72	120
C1118	91	150	4140*	66	110
C1119	100	165	4620	66	110
C1137	72	120	5120	76	125
C1141	70	115	5130	57	95
C1141*	81	135	5140*	70	115
C1144	76	125	6120	57	95
C1010†	...	120	8620	66	110
C1017	72	120	8630*	72	120
C1019	78	130	8640*	66	110
C1020	72	120	8650*	60	100
C1030	70	115	8720	66	110
C1035	70	115	8740*	66	110
C1040	64	105	E9310*	51	85
C1045	57	95	E9315*	49	80
2330*	70	115	E9317*	49	80
2340*	57	95	9763*	54	90

*Annealed †Light Feeds

Micrometer Carriage Stop

The micrometer carriage stop consists of a micrometer spindle mounted in a clamp which may be securely locked onto the front V-way of the lathe bed, as shown in Fig. 310. A lock screw is provided for locking the spindle at any point.

The micrometer carriage stop is used for facing shoulders to an exact length. It is convenient for many production operations and is usually included in the equipment of all toolroom lathes.

Fig. 310. Micrometer Carriage Stop

Four Position Carriage Stop

Much time can be saved in positioning the cutting tool for repetitive operations, by using the four position carriage stop shown in Fig. 311. It clamps onto the V-ways of the lathe bed, much the same as the micrometer stop. Each of the four adjustable stop screws may be set for a different tool position, and may be revolved into position to locate the carriage for each of four successive cuts. This attachment is especially desirable for spacing shoulders in shafts and similar operations.

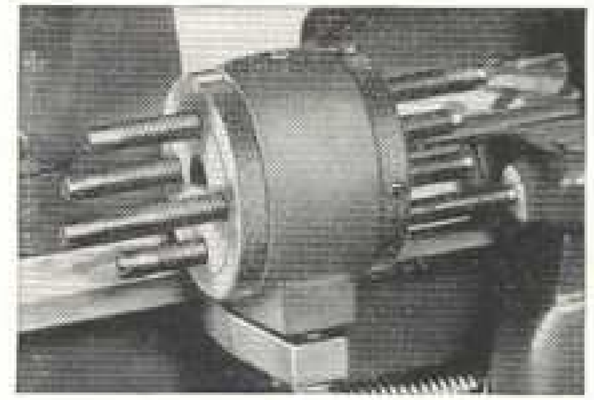


Fig. 311. Four Position Carriage Stop

Metric Graduated Collars

Metric cross-feed screw, metric compound rest screw and metric graduated collars are supplied on lathes that are to be used exclusively for working in the metric system.

The metric graduated collars read in tenths of a millimeter and are adjustable so that they may be set at zero whenever desired.

Fig. 312. Metric Graduated Collar

Metric Graduations on Taper Attachment

Taper attachments on lathes that are to be used for cutting tapers in the metric system are equipped with graduations reading in the metric system. Usually these graduations read in millimeters per centimeter and are in addition to the regular graduations.

Fig. 313. Taper Attachment with Metric Graduations

Metric Graduations on Tailstock Spindle

The tailstock spindle of the lathe may be graduated in centimeters, as shown in Fig. 314. The graduations are to aid in drilling accurately to the required depth.

Fig. 314. Metric Graduations on Tailstock Spindle

Grinding in the Lathe

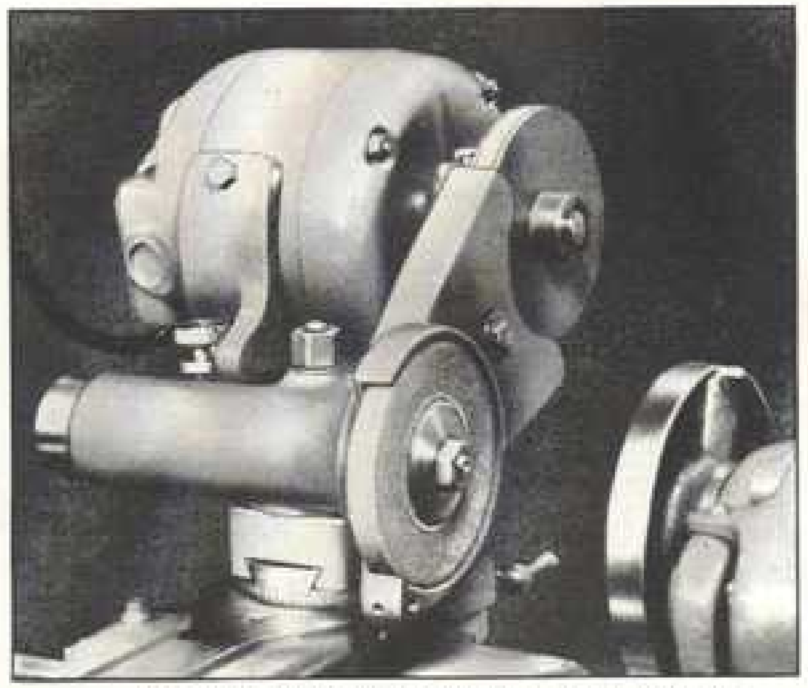


Fig. 315 External Grinding Attachment

When equipped with a good electric grinding attachment the lathe can be used for sharpening reamers and milling cutters, grinding hardened bushings and shafts and many other grinding operations.

The V-ways of the lathe bed should be covered with a heavy cloth or canvas to protect them from dust and grit from the grinding wheel, and the lathe spindle bearings should also be protected. A small pan of water or oil placed just below the grinding wheel will collect most of the grit.

A large, powerful grinder is most satisfactory for external grinding. The wheel should be at least four inches in diameter and the grinder should be mounted direct on the compound rest of the lathe, as shown in Fig. 315.

Internal Grinding

Fig. 315A. Internal Grinding Attachment

For grinding the inside of hardened drill jig bushings and other internal grinding, a high speed internal grinding attachment is used.

Fig. 315A shows an internal grinding attachment with compound V-belt drive which provides a spindle speed of 30,000 r.p.m.

GRINDING WHEELS FOR VARIOUS KINDS OF WORK

Tabulation shows grade of Norton Grinding Wheels.

Kind of Work	Rough Grind	Finish Grind
Cast Iron....................	37C36-KV Crystolon	37C60-JV Crystolon
Soft Steel....................	57A46-M8VBE	57A60-M8VBE
Hardened Steel..............	57A46-L8VBE	57A60-L8VBE
High Speed Steel............	57A46-K8VBE	57A60-K8VBE
Brass or Bronze..............	37C36-KV Crystolon	37C60-JV Crystolon
General Work................	57A46-M8VBE	57A46-M8VBE
Aluminum....................	A30-M3E Shellac	A36-M3E Shellac
Bakelite......................	37C36-KV Crystolon	37C46-KV Crystolon
Soft Rubber..................	37C20-K5B-2 Crystolon Resinoid	37C46-K5B-2 Crystolon Resinoid
Hard Rubber..................	37C30-K5B-2 Crystolon Resinoid	37C60-K5B-2 Crystolon Resinoid
Automobile Valves...........	57A60-M8VBE	57A80-L8VBE
Tungsten Carbide............	39C601-17V Crystolon	39C1002-H7V Crystolon

Diamond Dresser for Truing Grinding Wheel

Fig. 317. Truing a Grinding Wheel with a Diamond Dresser

The grinding wheel must be balanced and must be dressed with a diamond dresser if a smooth, accurate ground finish is to be obtained. The grinding wheel must be dressed frequently as it is used to keep it true and free from particles of metal which become embedded in the periphery of the wheel.

The diamond dresser consists of a small industrial diamond mounted in a steel shank, as shown in Fig. 318. The dresser must be rigidly supported in a fixture for truing the grinding wheel, as shown in Fig. 317.

Fig. 318. Diamond Dresser

The diamond point of the dresser should be placed on center, or slightly below center and the revolving grinding wheel passed back and forth across the diamond. Remove about .001 in. from the wheel at each cut and dress the wheel just enough to make it run true.

Grinding Hardened Steel Parts

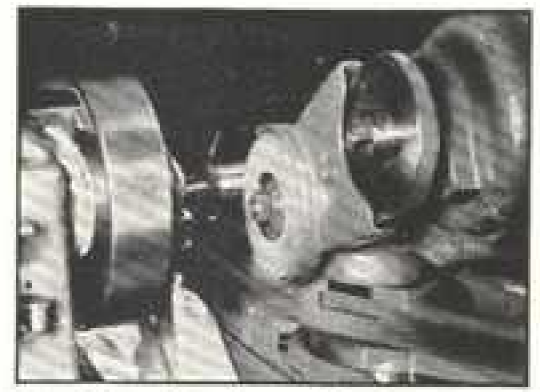


Fig. 319. Grinding a Hardened Steel Bushing

Hardened steel parts should be carefully ground in order to produce a smooth, accurate finish. The part should be machined to within a few thousandths of the finished size before it is hardened. After hardening, all scale should be removed before grinding. Remove only a few thousandths at each pass of the grinding wheel for if the part is ground too fast it may become overheated and warp, or the temper may be drawn.

Sharpening Reamers and Cutters

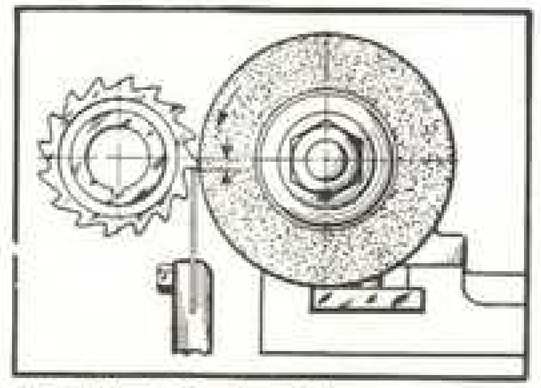


Fig. 320. Grinding Clearance on a Milling Cutter

Reamers and milling cutters may be sharpened by grinding in the lathe, as shown in Figs. 320, 321 and 322. Some reamers are first circular ground, then relieved by grinding with a tooth rest set slightly below center, as shown in Fig. 320, leaving a land .002 in. to .005 in. wide. Other reamers and most milling cutters are ground with about 2° relief.

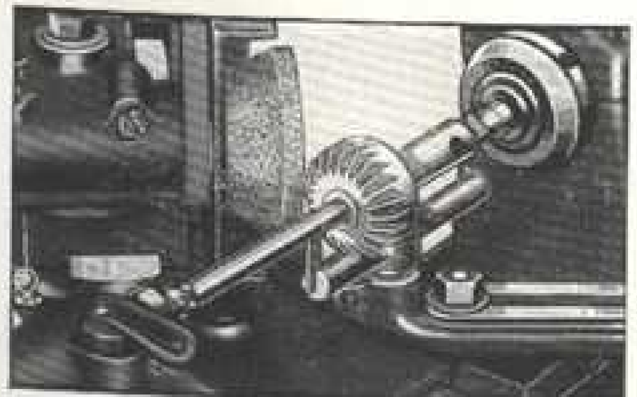


Fig. 321. Grinding an Angular Cutter in the Lathe

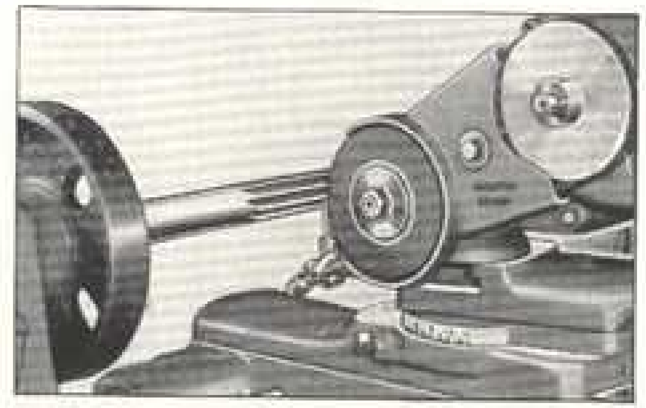


Fig. 322. Grinding a Straight Reamer in the Lathe

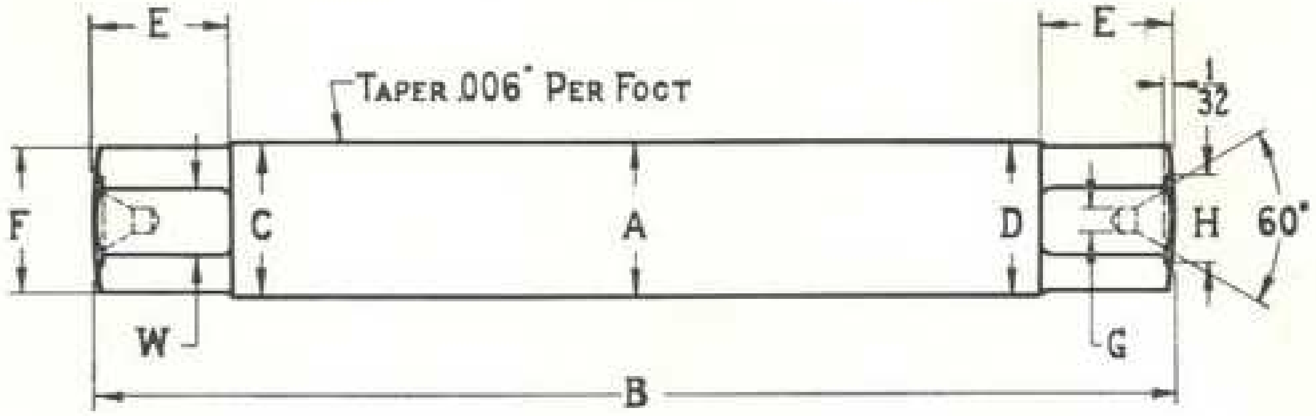


Fig. 323. Mandrel for machining work between centers in the lathe

How to Make Lathe Mandrels

Any good grade of machine steel can be used for making lathe mandrels. Old automobile axles are excellent. If only a few parts are to be made, the mandrel does not need to be hardened.

The tabulation below shows the dimensions recommended for standard lathe mandrels. A slight taper is required so that the mandrel can be pressed into the part tightly. The size of the mandrel is always stamped on the large end.

The center holes in the ends of the mandrel are very important. They should be large enough to provide a good bearing and they must be perfectly concentric with the outside diameter of the mandrel.

If a large quantity of parts are to be made, it is advisable to case-harden the center holes or harden the entire mandrel. The outside diameter of the mandrel must be finished after hardening; otherwise it will not run true because the steel will warp during the hardening process.

Before mounting a part on the mandrel, always oil both the inside of the part and the mandrel. See page 90.

All Dimensions Shown in Tabulation Below are in Inches

Nominal Diameter A	Total Length B	Small End C	Large End D	Undercut Length E	Undercut Diameter F	Center Drill G	Recess for Center H	Width of Flat W
3/16	3 3/4	.1870	.1884	7/16	11/64	3/64	1/8	5/64
1/4	3 3/4	.2495	.2509	7/16	15/64	3/64	1/8	7/64
5/16	4	.3120	.3136	7/16	9/32	3/64	3/16	1/8
3/8	4 1/4	.3745	.3760	1/2	11/32	1/16	1/4	5/32
7/16	4 1/2	.4370	.4387	9/16	13/32	1/16	9/32	1/4
1/2	5	.4995	.5014	5/8	15/32	3/32	5/16	1/4
9/16	5 1/4	.5620	.5640	5/8	17/32	3/32	5/16	1/4
5/8	5 1/2	.6245	.6265	3/4	19/32	3/32	3/8	1/4
11/16	5 3/4	.6870	.6891	3/4	5/8	3/32	3/8	5/16
3/4	6	.7495	.7517	13/16	11/16	1/8	3/8	5/16
13/16	6 1/4	.8120	.8142	7/8	3/4	1/8	3/8	7/16
7/8	6 1/2	.8740	.8764	7/8	13/16	1/8	3/8	7/16
15/16	6 3/4	.9370	.9394	15/16	7/8	1/8	3/8	1/2
1	7	.9995	1.0020	15/16	15/16	5/32	1/2	1/2
1 1/16	7 1/4	1.0615	1.0641	1	1	5/32	1/2	1/2
1 1/8	7 1/2	1.1240	1.1267	1	1 1/16	5/32	1/2	1/2
1 3/16	7 3/4	1.1865	1.1889	1	1 1/8	5/32	1/2	1/2
1 1/4	8	1.2490	1.2520	1	1 3/16	3/16	5/8	1/2
1 5/16	8 1/4	1.3115	1.3144	1 1/4	1 1/4	3/16	5/8	1/2
1 3/8	8 1/2	1.3740	1.3760	1 1/4	1 5/16	3/16	5/8	5/8
1 7/16	8 3/4	1.4365	1.4396	1 1/4	1 3/8	7/32	5/8	5/8
1 1/2	9	1.4990	1.5022	1 1/4	1 7/16	7/32	5/8	5/8

Press Fits and Running Fits

Standard tolerances for press fits and running fits, etc., are given in the tabulations below. The hole is usually made a standard size and the shaft made the required size for the desired type of fit. The figures shown in the tabulations below indicate the amount to increase or decrease the shaft diameter provided a standard hole size is maintained. The tolerance observed as standard for holes is usually +.000" −.001".

Since working conditions vary a great deal it may sometimes be advisable to increase or decrease the allowances shown in the tabulations. For example, the length of the bearing, the material used, and the speed should all be taken into consideration when calculating the tolerance for a running fit.

Standard Tolerances for Press Fits

Light Press Fit		Heavy Press Fit	
Dia. of Hole, in inches	Shaft Dia. More Than Hole, in inches	Dia. of Hole, in inches	Shaft Dia. More Than Hole, in inches
Up to 1/2	+.0004 to +.0006	Up to 1/2	+.0005 to +.001
1/2 to 1	+.0005 to +.0010	1/2 to 1	+.001 to +.003
1 to 2	+.00075 to +.0020	1 to 2	+.002 to +.004
2 to 3	+.0015 to +.0030	2 to 3	+.003 to +.006
3 to 4	+.0020 to +.0040	3 to 4	+.005 to +.008
4 to 5	+.0020 to +.0045	4 to 5	+.006 to +.010
5 to 6	+.0030 to +.0050	5 to 6	+.008 to +.012

Standard Tolerances for Running Fits

Speeds up to 1000 r.p.m.		Speeds over 1000 r.p.m.	
Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches	Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to 1/2	—.0005 to —.0010	Up to 1/2	—.0005 to —.0010
1/2 to 1	—.00075 to —.0015	1/2 to 1	—.0010 to —.0020
1 to 2	—.0015 to —.0025	1 to 2	—.0020 to —.0030
2 to 3	—.0020 to —.0025	2 to 3	—.0025 to —.0035
3 to 4	—.0025 to —.0030	3 to 4	—.0030 to —.0040
4 to 5	—.0030 to —.0035	4 to 5	—.0035 to —.0045
5 to 6	—.0035 to —.0040	5 to 6	—.0040 to —.0050

Standard Tolerances for Push Fits		Standard Tolerances for Sliding Fits	
Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches	Dia. of Hole, in inches	Shaft Dia. Less Than Hole, in inches
Up to 1/2	—.00025 to —.00075	Up to 1/2	—.0005 to —.001
1/2 to 1	—.0005 to —.0010	1/2 to 1	—.00075 to —.0015
1 to 2	—.0005 to —.0015	1 to 2	—.0015 to —.0025
2 to 3	—.0005 to —.0015	2 to 3	—.0020 to —.0030
3 to 4	—.00075 to —.0020	3 to 4	—.0025 to —.0030
4 to 5	—.00075 to —.0020	4 to 5	—.0025 to —.0035
5 to 6	—.00075 to —.0020	5 to 6	—.0025 to —.0040

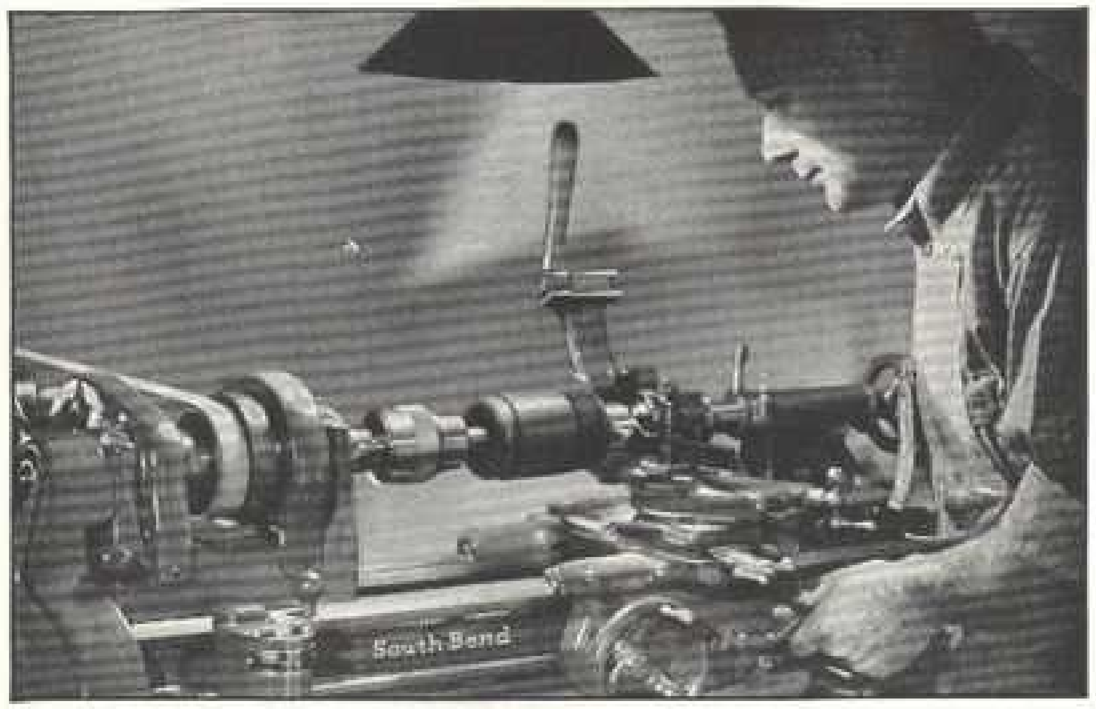


Fig. 327. Truing and Undercutting an Armature Commutator in the Lathe

The Lathe in the Auto Service Shop

The Back-Geared Screw Cutting Lathe is frequently called the "Universal Tool," and this applies in automotive service work as well as in general industry. Most of the mechanical parts of the automobile, bus, truck, tractor and airplane are originally made on lathes or in special machines which are adaptations of the lathe.

A lathe with 9-in. or 10-in. swing is very practical for handling such jobs as refacing valves; truing armature commutators and undercutting mica; finishing pistons; beveling piston skirts; reaming piston pin holes; making bushings, bearings and glands; cutting screw threads, testing and straightening bent shafts, and many other jobs. Special attachments used on the lathe greatly increase its versatility.

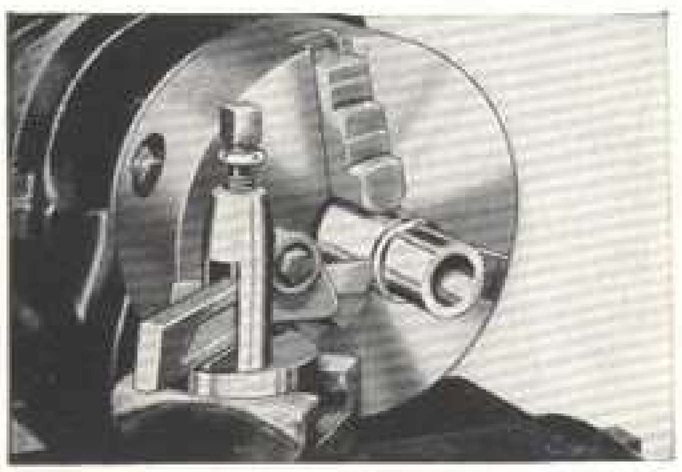


Fig. 328. Making a Replacement Bushing Complete in a 9-in. Lathe

Fig. 329. Turning a Semi-Machined Piston to Size in a 9-in. Lathe

Armature Truing

Machining the commutator of an armature true and under-cutting the mica are two of the most important jobs in auto electrical work, and these jobs are most easily handled in the lathe.

A small lathe equipped for these jobs is shown in Fig. 330. The undercutting attachment is mounted on the lathe in such a way that it is ready for instant use, yet it does not interfere with turning the commutator.

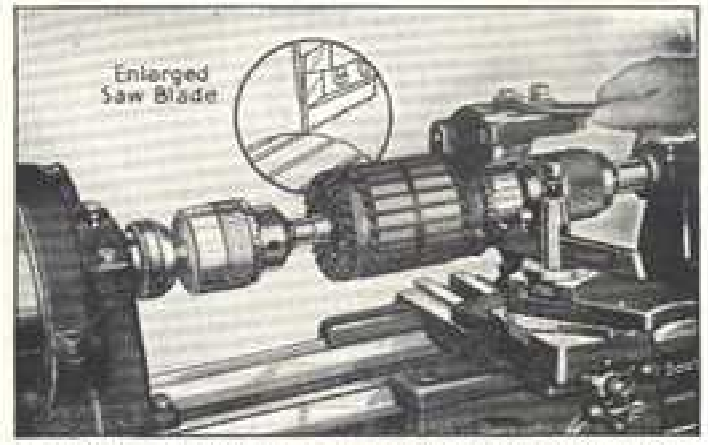


Fig. 330. Undercutting an Armature Commutator in the Lathe

Refacing Valves

A lathe equipped with a grinding attachment and a special hollow valve chuck for refacing valves is shown in Fig. 331.

Other valve jobs done in the lathe include: Truing the valve tappet face and rocker arm face; making valve guide bushings and valve seat replacement rings, etc.

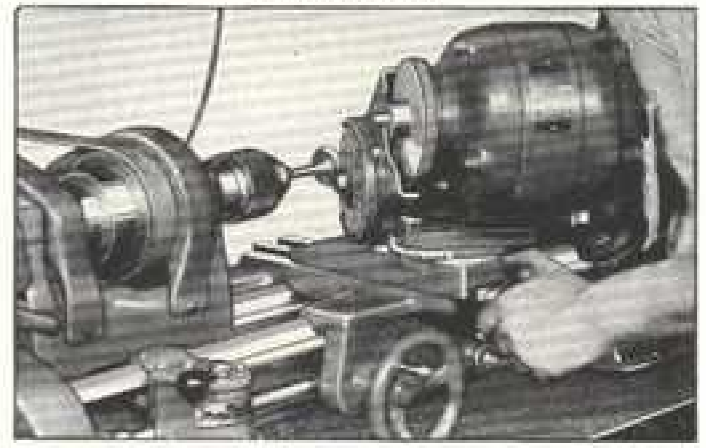


Fig. 331. Refacing a Valve by Grinding in the Lathe

Finishing Pistons

Pistons of all sizes and types can be rough and finish turned in the lathe, as shown in Fig. 332. The lathe can also be used for reaming and honing piston pin holes, cutting oil grooves in pistons, remachining piston ring grooves, beveling piston skirts, etc.

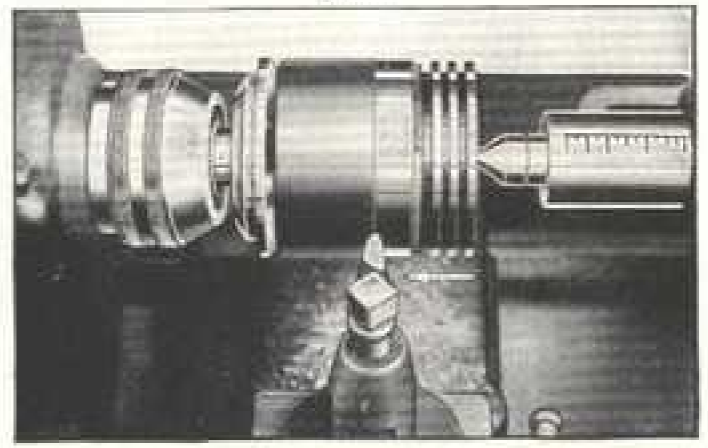


Fig. 332. Finishing a Piston in the Lathe

Straightening Bent Shafts

With the aid of a piece of chalk, the high spot on a bent shaft can easily be located and marked for straightening as it revolves between the lathe centers. The shaft should be removed from the lathe and placed in a press or on an anvil for straightening.

A dial test indicator mounted in the tool post of the lathe permits testing and straightening shafts to a high degree of accuracy. See Fig. 337.

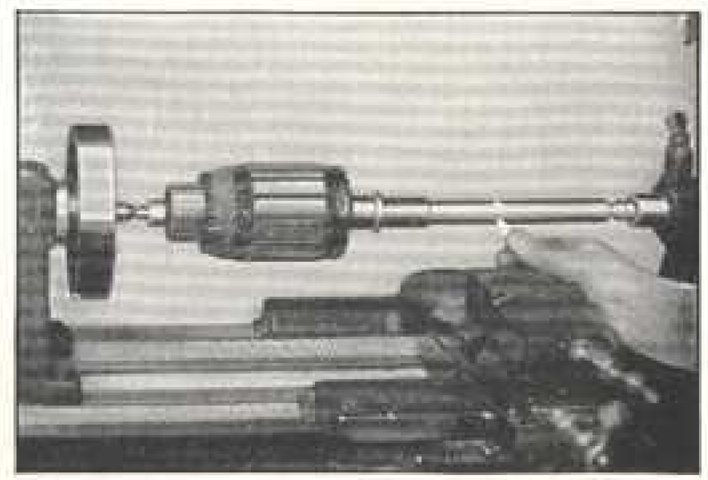


Fig. 333. Testing a Bent Armature Shaft in the Lathe

Machining Eccentrics

A simple eccentric can be machined on a straight mandrel having two sets of center holes as shown in Fig. 334. One set of centers is used for machining the concentric hub and the other set of centers is used for machining the eccentric part.

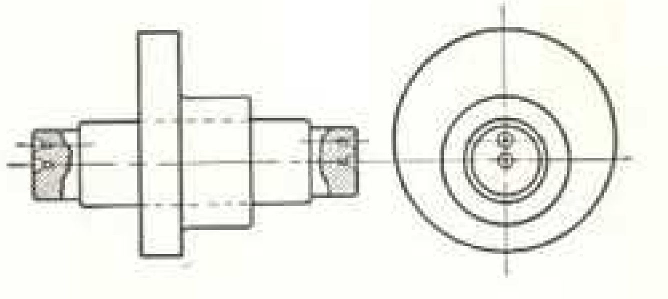


Fig. 334. A Mandrel with Two Sets of Center Holes for Machining an Eccentric

Crankshaft Turning

Crankshaft turning is an adaptation of eccentric machining. A single throw crankshaft mounted in the lathe for machining the throw bearing is shown in Fig. 335. The adapters attached to each end of the crankshaft have offset center holes corresponding to the throw of the crankshaft.

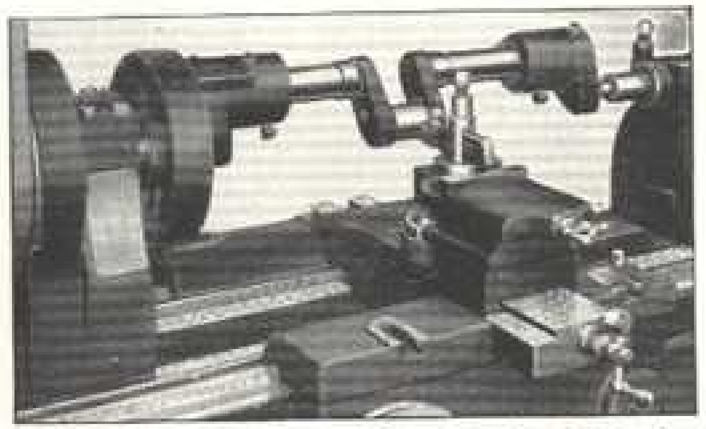


Fig. 335. A Crankshaft Mounted in the Lathe for Machining the Throw Bearing

Truing Crankshaft Bearings

Throw bearings of automobile crankshafts are often worn out of round or scored and must be reground. A special crank pin grinding attachment shown in Fig. 336, permits truing the throw bearings without the use of offset centers. The grinding wheel travels around with the throw bearing in such a way that it will grind the bearing round and straight. The lathe spindle must revolve very slowly (about 10 r.p.m.) while this tool is being used.

Fig. 336. Truing the Throw Bearings of a Crankshaft by Regrinding in the Lathe

Testing Crankshafts

Crankshafts may be tested between the lathe centers, as shown in Fig. 337. The dial indicator mounted in the tool post of the lathe reads in thousandths of an inch and will show exactly how much the crankshaft is sprung and will also disclose any out-of-round condition of the bearing. Straightening a crankshaft is a delicate job and should be attempted only by an experienced mechanic.

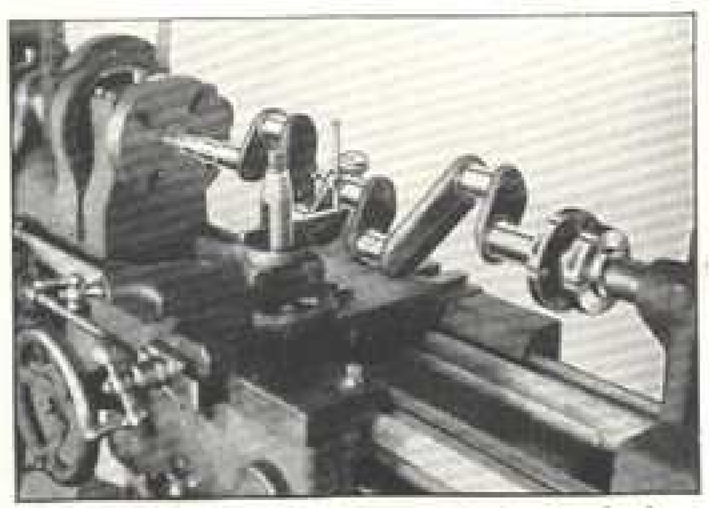


Fig. 337. Testing a Crankshaft in the Lathe

Fig. 338. A Portable Machine Shop Built in a Large Truck

Portable Machine Shop

The portable machine shop shown in Fig. 338 is rapidly gaining in popularity. This type of shop is especially valuable for service in oil fields, construction camps, air ports, army posts, etc., also for the maintenance of road building equipment and for repairing construction machinery and equipment on large engineering projects. The advantage in taking the shop to the job is obvious when the delay and difficulty involved in transporting heavy, awkward parts to and from the shop are taken into consideration.

The equipment of the portable machine shop may be quite complete, consisting of a 16 in. by 8 ft. lathe, a 20-in. drill press, a forge, anvil, grinder, welding outfit, etc., as shown above, or it may be limited to a small lathe and a good assortment of small tools. The equipment will vary with the purpose of the shop and the amount to be invested.

The lathe is the most important of all of the tools in the portable machine shop because it can be used for so many classes of work. When equipped with the necessary attachments, the lathe may be used as a milling machine, gear cutting machine, grinding machine, drill press, etc.

Regardless of the size of the lathe and other equipment, it is important that the truck or trailer in which the shop is installed be of substantial construction with a good solid floor. Provision should be made for blocking and leveling the floor while the machinery is in use. All parts must be securely fastened in place so there will be no danger of damage while the shop is being transported from one location to another.

Power for operating the lathe and other machinery is usually obtained through a generator installed in such a way that it can be operated by the truck engine. This same generator also supplies current for electric lights to illuminate the shop and also large flood lights which permit working near the unit after night.

Cutting Speeds for Turning—Drilling—Tapping With High Speed Steel Cutting Tools

Material	Turning Speeds		Drilling Speeds		Tapping Speeds	
	Ft. per Minute	Lubricant	Ft. per Minute	Lubricant	Ft. per Minute	Lubricant
Aluminum	300-400	Comp. or Kerosene	200-330	Comp. or Kerosene	90-110	Kerosene & Lard Oil
Brass, leaded	300-700	Dry or Comp.	200-500	Comp.	150-250	Comp. or Lt. Base Oil
Brass, red and yellow	150-300	Comp.	75-250	Comp.	60-150	Comp. or Lt. Base Oil
Bronze, leaded	300-700	Comp.	200-500	Comp.	150-250	Comp. or Lt. Base Oil
Bronze, phosphor	75-150	Comp.	50-125	Comp.	30- 60	Comp. or Lt. Base Oil
Cast Iron	50-110	Dry	100-165	Dry	70- 90	Dry or Comp.
Cast Steel	45- 90	Comp.	35- 45	Comp.	20- 35	Sul. Base Oil
Copper, leaded	300-700	Comp.	200-500	Comp.	150-250	Lt. Base Oil
Copper, electro.	75-150	Comp.	50-125	Comp.	30- 60	Lt. Base Oil
Chrome Steel	65-115	Comp.	50- 65	Comp.	20- 35	Sul. Base Oil
Die Castings	225-350	Compound	200-330	Compound	60- 80	Kerosene & Lard Oil
Duralumin	275-400	Compound	250-375	Compound	90-110	Comp. or Ker. and Lard Oil
Fiber	200-300	Dry	175-275	Dry	80-100	Dry
Machine Steel	115-225	Compound	80-120	Compound	40- 70	Comp., Sul. Base Oil or Kero. & Para
Malleable Iron	80-130	Dry or Comp.	80-100	Dry or Comp.	35- 70	Comp. or Sul. Base Oil
Mang. Bronze	150-300	Comp.	75-250	Comp.	60-150	Lt. Base Oil
Mang. Steel	20- 40	Comp.	15- 25	Comp.	10- 20	Comp. or Sul. Base Oil or Ker. & Para
Moly. Steel	100-120	Comp.	50- 65	Comp.	20- 35	Sul. Base Oil
Monel Metal	100-125	Comp. or Sul. Base	40- 55	Sul. Base	20- 30	Sul. Base or Kero. and Lard Oil
Nickel Silver 18%	75-150	Comp.	50-125	Comp.	30- 60	Sul. Base or Kero. and Lard Oil
Nickel Silver, leaded	150-300	Comp.	75-250	Comp.	60-150	Sul. Base or Kero and Lard Oil
Nickel Steel	85-110	Comp. or Sul. Base	40- 65	Sul. Base Oil	25- 40	Sul. Base Oil
Plastics, hot-set molded	200-600	Dry	75-300	Dry	40- 54	Dry or Water
Rubber, Hard	200-300	Dry	175-275	Dry	80-100	Dry
Stainless Steel	100-150	Sul. Base	30- 45	Sul. Base	15- 30	Sul. Base
Tool Steel	70-130	Comp.	50- 65	Comp.	25- 40	Sul. Base or Kero. and Lard Oil
Tungsten Steel	70-130	Comp.	50- 65	Comp.	20- 35	Sul. Base
Vanadium Steel	85-120	Comp.	45- 65	Sul. Base	25- 40	Sul. Base

The above speeds have been collected from several sources and are suggested as practical for average work. Special conditions may necessitate the use of higher or lower speeds for maximum efficiency.

Use of Coolant

Coolants are used extensively in the machining of steel parts in order to permit higher cutting speeds, produce a better finish, and increase tool life. The principal function of a coolant is, as its name implies, to cool the work and the cutting tool. The coolant also facilitates production by lubricating the cutting tool, flushing away chips, and preventing rust.

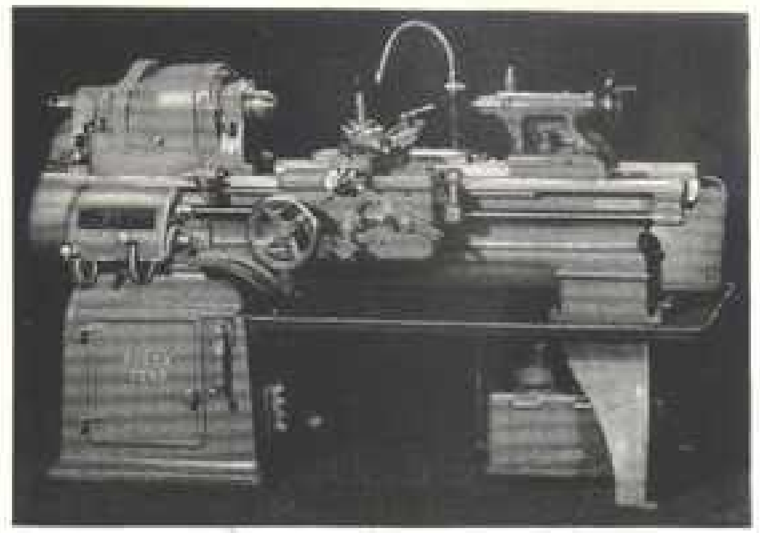


Fig. 341. Lathe Equipped with Coolant Pan, Pump, Reservoir, and Piping

Coolant Equipment

Very simple equipment can sometimes be used effectively in applying a coolant. For example, a small paint brush may be used as shown in Fig. 238, page 81. An ordinary oil can is also convenient for applying a limited amount of coolant. However, on continuous high speed production the lathe should be equipped with an oil pan, coolant pump, and reservoir as shown in Fig. 341.

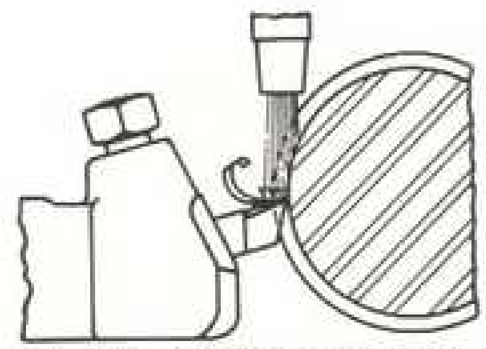


Fig. 342. Coolant Should Make Contact with Work Where Cutting Action Takes Place

Application of Coolant

For effective cooling, it is important that the coolant be properly applied to the work. A large stream at slow velocity is preferable to a small, high velocity stream. The coolant should make contact with the work at the exact spot where the cutting action takes place, not above or to one side of the cutting tool. See Fig. 342.

Types of Coolants

Coolants considered most effective for various classes of work are listed in the table on page 108. Each coolant has some qualities which make it especially desirable for certain applications. The characteristics of the most popular coolants are as follows:

Lard Oil—One of the oldest and best coolants, and most expensive. Especially good for cutting screw threads, drilling deep holes, and reaming. Provides excellent lubrication, increases tool life, and produces a smooth finish on the work. Prevents rust.

Mineral Lard Oil Mixtures—Various mixtures of lard oil and petroleum base mineral oils are used in lieu of lard oil because they are more fluid, less expensive and are almost as effective.

Mineral Oils—Petroleum base oils compounded with chemicals to improve their lubricating and anti-welding qualities. Less expensive than lard oil and mineral oil mixtures.

Soluble Oils—Mineral oils which have been treated so that they may be mixed with water to form an emulsion and provide an excellent low cost coolant. Although they carry away heat better than lard oil or mineral oil, their lubricating qualities are comparatively poor. Their use is usually limited to rough turning operations. Even though they are mixed with water, they leave a protective film on metal which resists rusting.

Soda Water Mixtures—Cheapest of all coolants, soda water mixtures are very effective for cooling but have practically no lubricating qualities and will cause steel or iron to rust. A popular mixture consists of 1 pound of sal-soda (carbonate of soda), 1 quart of lard oil, 1 quart of soft soap, and 10 gallons of water, boiled together for ½ hour.

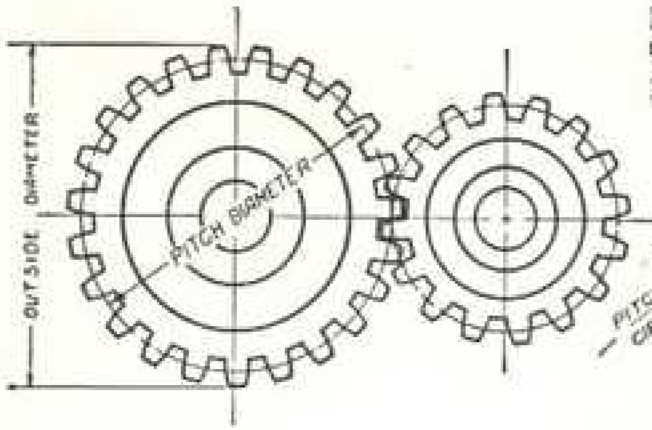
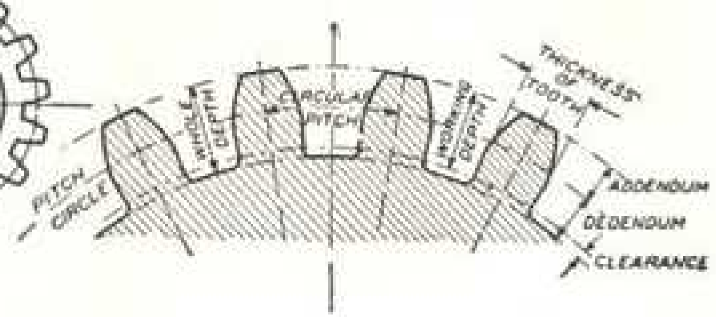


Fig. 343. At Left. Illustration Showing Outside Diameter and Pitch Diameter of Gear Teeth

Fig. 344 Below. Names of Gear Tooth Parts



Information on Gears

The rules and formulas listed below may be used for calculating the dimensions of involute spur gears.

Diametral Pitch—Number of teeth divided by pitch diameter, or 3.1416 divided by circular pitch.

Example: If a gear has 40 teeth and the pitch diameter is 4 in., the diametral pitch is 40 divided by 4, and the diametral pitch is 10, or in other words there are 10 teeth to each inch of the pitch diameter, and the gear is 10 diametral pitch.

Circular Pitch—Distance from center to center of two adjacent teeth along pitch circle, or 3.1416 divided by diametral pitch.

Pitch Diameter—Number of teeth divided by diametral pitch.

Example: If the number of teeth is 40 and the diametral pitch is 4, divide 40 by 4, and the quotient, 10, is the pitch diameter.

Outside Diameter—Number of teeth plus two divided by diametral pitch.

Example: If the number of teeth is 40 and the diametral pitch is 4, add 2 to the 40, making 42, and divide by 4; the quotient, 10½, is the outside diameter of gear or blank.

Addendum—1 divided by diametral pitch.

Whole Depth of Tooth—2.157 divided by diametral pitch.

Thickness of Tooth—1.5708 divided by diametral pitch.

Number of Teeth—Pitch diameter multiplied by diametral pitch, or multiply outside diameter by diametral pitch and subtract 2.

Example: If the diameter of the pitch circle is 10 in. and the diametral pitch is 4, multiply 10 by 4 and the product, 40, will be the number of teeth in the gear.

Example: If the outside diameter is 10½ and the diametral pitch is 4, multiply 10½ by 4 and the product, 42, less 2, or 40, is the number of teeth.

Center Distance—Total number of teeth in both gears divided by two times the diametral pitch.

Example: If the two gears have 50 and 30 teeth respectively, and are 5 pitch, add 50 and 30, making 80, divide by 2, and then divide the quotient, 40, by the diametral pitch, 5, and the result, 8 in., is the center distance.

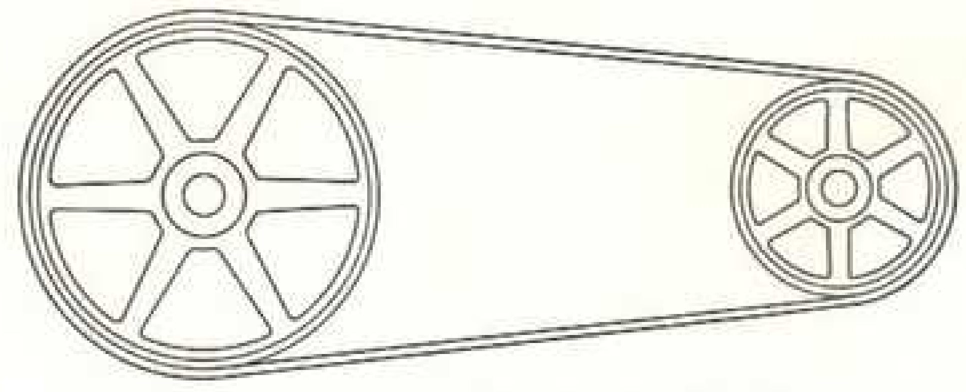


Fig. 345. A Pair of Pulleys for Flat Belt Drive

Calculating the Speed and Size of Pulleys

Diameter of Driving Pulley—Multiply the diameter of the driven pulley by its number of revolutions, and divide by the number of revolutions of the driver.

Diameter of Driven Pulley—Multiply the diameter of the driving pulley by its number of revolutions, and divide the product by the number of revolutions of the driven pulley.

Speed of Driven Pulley—Multiply the diameter of the driving pulley by its number of revolutions, and divide by the diameter of the driven pulley.

Speed of Driving Pulley—Multiply the diameter of the driven pulley by its number of revolutions, and divide by the diameter of the driving pulley.

The driving pulley is called the driver and the driven pulley is the driven or follower.

R.P.M. indicates the number of revolutions per minute.

Example: Problem 1.

Given: Speed of the driving pulley 260 R.P.M. Speed of the driven pulley 390 R.P.M. Diameter of the driven pulley 8 in.

To find the diameter of the driving pulley.

$$390 \times 8 = 3120$$
$$3120 \div 260 = 12$$

The diameter of the driving pulley is 12 in.

Width of Pulleys—Pulleys for flat belts should be about 10% wider than the width of the belt used.

Types of Pulleys—Two types of pulleys are used for flat belts, the crowned face pulley and the flat face pulley. Crowned pulleys should always be used if possible, as it is the crown that keeps the belt on the pulley. Flat face pulleys should be used only when it is necessary to shift the belt from one position on the pulley to another, as in a drum pulley or a wide faced pulley on a machine used to match a tight and loose pulley on a countershaft.

Fitting a Chuck Plate to a Chuck

Before a chuck can be used on a lathe it must be fitted with a chuck plate that has been threaded to fit the spindle nose of the lathe. Semi-machined chuck plates that have been accurately threaded to fit the lathe spindle can be obtained from the lathe manufacturer.

Mounting Chuck Plate on Spindle

Before screwing the chuck plate on to the spindle nose of the lathe, clean the threads of the chuck plate and the spindle nose thoroughly. Make sure that there are no chips, burrs or small particles of dirt lodged in the screw threads, or on face of hub, and also make sure that the shoulder on the headstock spindle is perfectly clean and free from chips or burrs.

Oil the threads of the headstock spindle and the chuck plate and screw the chuck plate onto the spindle nose. Do not jam the threads tight or it may be difficult to remove the chuck plate after it has been finished.

Fig. 346. Semi-Machined Chuck Plate, Threaded to Fit the Spindle Nose of the Lathe

Finishing the Flange

First, machine the face of the chuck plate, taking one roughing cut about $\frac{1}{32}$ in. deep, and then one or two finishing cuts, removing not over .001 in. in the last cut.

Measure the diameter of the recess in the back of the chuck carefully with inside calipers, and set outside calipers to correspond with the inside calipers. Machine the diameter of the chuck plate flange very carefully. Take very light finishing cuts and try the chuck on the chuck plate frequently, as the chuck plate must fit snugly into the recess in the back of the chuck.

Fig. 347. Rear View of the Lathe Chuck

After the chuck plate has been finished to fit the recess in the back of the chuck, remove it from the lathe spindle and chalk the face of the flange thoroughly. Place the flange in the recess in the back of the chuck and tap lightly on the chuck plate so that the edge of the bolt holes in the chuck will mark the location of bolt holes.

Drill the holes $\frac{1}{16}$ in. larger in diameter than the bolts used to secure the chuck plate onto the chuck. It is very important that the bolt holes be large enough to eliminate all possibility of the bolts binding.

Fig. 348. Chuck with Chuck Plate Attached

Hardening and Tempering Lathe Tools

After a forged lathe tool has been used for some time, it should be re-forged, hardened and tempered. If carefully done, this will make the tool as good as new. Before attempting to harden and temper the tool, make sure of the kind of steel from which it is made.

Fig. 349. Hardening a Lathe Tool

To Distinguish Carbon Tool Steel

To distinguish carbon tool steel from high speed steel, touch against emery wheel. Carbon steel gives off a shower of bright yellow sparks; high speed steel gives off a few dark red sparks.

To Harden Carbon Tool Steel

To harden a forged lathe tool made of carbon tool steel, heat the end of the tool slowly to a bright cherry red for a distance of at least an inch back of the cutting edge; then immerse the point about 1½ in. deep in cold water, but do not cool the shank. When the point is cool remove from water, polish the cutting edge with emery cloth and wipe with oily rag.

The tool is now hardened, and as the heat in the shank passes into the point it will discolor the polished surface, indicating the amount the temper is drawn. When a light straw color appears, cool the entire tool quickly in water and it will have the correct strength and toughness for metal turning.

The method outlined above may be followed for hardening and tempering any tool made of carbon tool steel. For wood cutting tools, taps and dies, draw to a dark straw color. For hatchets, screw drivers, cold chisels, etc., draw to a brown yellow; for springs dark purple.

Case Hardening

To case harden a piece of machinery steel, heat the steel to a cherry red; then remove from fire and apply cyanide of potassium to the surface you wish to case harden. The cyanide will dissolve slowly and be absorbed by the steel. After the surface has received a thorough coat of cyanide return the steel to the fire and heat slowly for about one minute so that the cyanide will be thoroughly absorbed by the steel. Remove from the fire and quench in cold water.

How to Anneal Tool Steel

Carbon tool steel may be annealed by heating slowly and evenly to a cherry red and then placing in a box of lime or ashes to cool slowly. The steel should be completely covered and when it is cooled to room temperature will be ready for machining.

How to Anneal Brass

Brass that has been hardened through cold working may be annealed by heating to a dull red when held in dark shadow and plunging into cold water. Care must be taken not to overheat the brass.

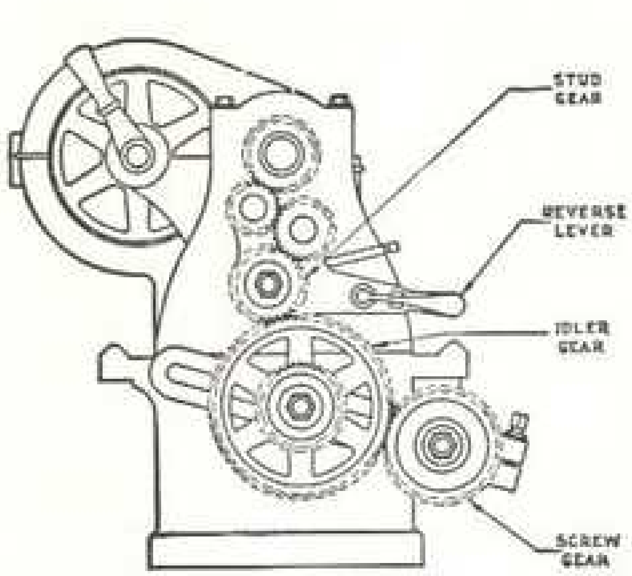


Fig. 350. Simple gearing

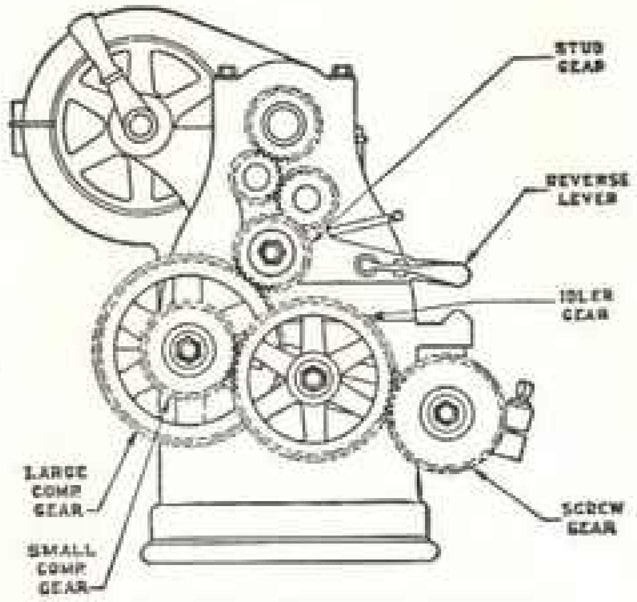


Fig. 351. Compound gearing

How to Calculate Change Gears for Thread Cutting

If it is necessary to cut a special thread that does not appear on the index chart of a lathe or if no index chart is available, the gears required can easily be calculated. All South Bend Lathes are even geared; that is, the stud gear revolves the same number of revolutions as the headstock spindle, and when gears of the same size are used on both the lead screw and stud, the lead screw and spindle revolve the same number of revolutions, so it is not necessary to consider the gearing between the headstock spindle and the stud gear when calculating change gears.

If simple gearing is to be used, as shown in Fig. 350, the ratio of the number of teeth in the change gears used will be the same as the ratio between the thread to be cut and the thread on the lead screw. For example, if 10 threads per inch are to be cut on a lathe having a lead screw with 6 threads per inch, the ratio of the change gears would be 6 to 10. These numbers may be multiplied by any common multiplier to obtain the number of teeth in the change gears that should be used.

Rule—To calculate change gears, multiply the number of threads per inch to be cut and the number of threads per inch in the lead screw by the same number.

Example: Problem—To cut 10 threads per inch on lathe having lead screw with 6 threads per inch.

Solution— 6 x 8 = 48 — No. of teeth in gear on stud.
10 x 8 = 80 — No. of teeth in gear on lead screw.

If these gears are not to be found in the change gear set, any other number may be used as a common multiplier, such as 3, 5, 7, etc.

When compound gearing, as shown in Fig. 351, is used, the ratio of the compound idler gears must also be taken into consideration, but otherwise the calculations are the same as for simple gearing. Usually, the compound idler gear ratio is 2 to 1, so that the threads cut are just twice the number per inch as when simple gearing is used.

DECIMAL EQUIVALENTS OF FRACTIONAL PARTS OF AN INCH

1/64	.015625	11/32	.34375	11/16	.6875
1/32	.03125	23/64	.359375	45/64	.703125
3/64	.046875	3/8	.375	23/32	.71875
1/16	.0625	25/64	.390625	47/64	.734375
5/64	.078125	13/32	.40625	3/4	.75
3/32	.09375	27/64	.421875	49/64	.765625
7/64	.109375	7/16	.4375	25/32	.78125
1/8	.125	29/64	.453125	51/64	.796875
9/64	.140625	15/32	.46875	13/16	.8125
5/32	.15625	31/64	.484375	53/64	.828125
11/64	.171875	1/2	.5	27/32	.84375
3/16	.1875	33/64	.515625	55/64	.859375
13/64	.203125	17/32	.53125	7/8	.875
7/32	.21875	35/64	.546875	57/64	.890625
15/64	.234375	9/16	.5625	29/32	.90625
1/4	.25	37/64	.578125	59/64	.921875
17/64	.265625	19/32	.59375	15/16	.9375
9/32	.28125	39/64	.609375	61/64	.953125
19/64	.296875	5/8	.625	31/32	.96875
5/16	.3125	41/64	.640625	63/64	.984375
21/64	.328125	21/32	.65625	1	1.
		43/64	.671875		

METRIC AND ENGLISH LINEAR MEASURE

The measuring rules shown below are graduated, in the Metric system and in the English system. This shows at a glance the comparison of the fractions of the Metric and English units, the meter and the inch. It will be noticed that one inch is equal to 25.4001 + mm.

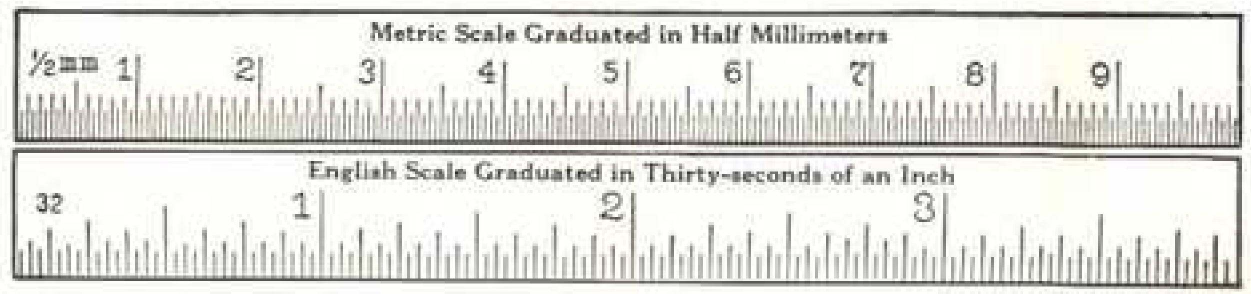


Fig. 352. Comparison of English and Metric Scales, Actual Size

TABLE OF METRIC LINEAR MEASURE

10 Millimeters	= 1 Centimeter	1 Centimeter	= .3937 inch
10 Centimeters	= 1 Decimeter	1 Decimeter	= 3.937 inches
10 Decimeters	= 1 Meter	1 Meter	= 39.37 inches

Equivalents of Millimeters in Decimals of Inches

1/10 mm = .00394 in.	8 mm = .31496 in.	18 mm = .70866 in.
1/5 mm = .00787 in.	9 mm = .35433 in.	19 mm = .74803 in.
1/2 mm = .01969 in.	10 mm = .39370 in.	20 mm = .78740 in.
1 mm = .03937 in.	11 mm = .43307 in.	21 mm = .82677 in.
2 mm = .07874 in.	12 mm = .47244 in.	22 mm = .86614 in.
3 mm = .11811 in.	13 mm = .51181 in.	23 mm = .90551 in.
4 mm = .15748 in.	14 mm = .55118 in.	24 mm = .94488 in.
5 mm = .19685 in.	15 mm = .59055 in.	25 mm = .98425 in.
6 mm = .23622 in.	16 mm = .62992 in.	26 mm = 1.02362 in.
7 mm = .27559 in.	17 mm = .66929 in.	27 mm = 1.06299 in.

SHOP KINKS

From "American Machinist"

A good lubricant for turning, boring and milling aluminum is made of equal parts of lard oil and kerosene. Kerosene alone is also good. and cheap.

When drilling or turning hard steel (not hardened steel) in the lathe, run slowly and lubricate the tool with turpentine, or turpentine and spirits of camphor.

Red lead and graphite are good lubricants for the tail center. On heavy work, make the countersinks for the center as large as possible without making the job unsightly.

Probably the handiest lathe chuck for the jobbing shop is the four-jawed independent chuck having stepped, reversible jaws. These will hold almost any shaped piece, and hold it firmly.

Before complaining that the lathe is not in line, be sure that the bed is carefully leveled up. Don't twist the lathe bed out of shape by pulling it down on to an uneven floor with lagscrews, and then expect it to turn straight.

Squared ends and a uniform depth of lathe center are desirable for accurate and economical work. Don't make the mistake of thinking that any boy can do the centering well enough without proper instruction and supervision.

A little square of mica with a tin rim makes a good chip guard when turning brass. A simple wire spring clip can be fastened to the tin rim so that the guard can be readily fastened to or detached from the tool or tool post.

If a $\frac{1}{32}$-in. slot is made on the top of the tail center running from the point back to a little beyond the large part of the conical end, the center can be oiled without slacking it back. This is better than grinding half the conical end away.

The indicator, of the dial variety, is an extremely handy tool around the engine lathe, particularly if accurate work is to be done. Don't get the idea that this is an unnecessary frill, but get accustomed to using it if you wish to become an accurate workman.

Universal lathe chucks are very convenient to have in the shop but can rarely be depended on to run true if accuracy is desired. They may be plenty good enough for a large proportion of the work to be done, but accurate work demands separate adjustment of the jaws.

The lathe centers should be well cared for, the point ground in place, if possible, and always put into place in the same position. This may sound finicky but it is the only way to produce accurate work. Don't make the mistake of using a lathe center either for a hammer or a center punch.

DON'TS FOR MACHINISTS

From "Machinery"

Don't run a lathe with the belt too loose.

Don't run the point of your lathe tool into the mandrel.

Don't rap the chips out of your file on the lathe shears.

Don't set a lathe tool below the center for external work.

Don't start up a lathe without seeing that the tail stock spindle is locked.

Don't put an arbor or shaft on lathe centers without lubricant on them.

Don't leave too much stock on a piece of work to take off with the finishing cut.

Don't try a steel gauge or an expensive caliper on a shaft while it is running.

Don't put a mandrel into a newly bored hole without a lubricant of some kind on it.

Don't put a piece of work on centers unless you know that the internal centers are clean.

Don't try to straighten a shaft on lathe centers, and expect that the centers will run true afterwards.

Don't put a piece of work on lathe centers unless you know that all your centers are at the same angles.

Don't take a lathe center out of its socket without having a witness mark on it, and put it back again according to the mark.

Don't start polishing a shaft on lathe centers without having it loose enough to allow for the expansion by heat from the polishing process.

Don't run your lathe tool into the faceplate.

Don't try to knurl a piece of work without oiling it.

Don't run a lathe an instant after the center begins to squeal.

Don't forget to oil your machine every morning; it works better.

Don't forget that a fairly good center-punch may be made from a piece of round file.

Don't forget that a surface polished with oil will keep clean much longer than one polished dry.

Don't start to turn up a job on lathe centers unless you know that the centers are both in line with the ways.

Don't cross your belt laces on the side next the pulley, for that makes them cut themselves in two.

Don't try to cut threads on steel or wrought iron dry; use lard oil or a cutting compound.

Don't run a chuck or faceplate up to the shoulder suddenly; it strains the spindle and threads and makes removal difficult.

Don't screw a tool post screw any tighter than is absolutely necessary; many mechanics have a false idea as to how tight a lathe tool should be to do its work.

To drive the center out of head spindle use a rod and drive through the hole in spindle.

When putting a lathe chuck on the head spindle, always remove the center.

When the center is removed from the head spindle of the lathe, always put a piece of rag in spindle hole to prevent any dirt from collecting.

Fig. 353. A Trade School Shop.

INDUSTRIAL APPRENTICE TRAINING

In the United States

The industrial plants of the United States are very much interested in the Vocational and Trade Schools. These schools are doing remarkable work for the individual boys of their own communities and also for the industries of the entire United States as a whole.

The metal-working industry is aware that well equipped vocational schools can teach young men the fundamentals of the machinist's trade much better than they can be taught to apprentices in factories. Under the guidance of a capable and thoroughly trained instructor, the boys not only receive practical instruction in the operation of various machine tools, but are also taught the necessary shop mathematics, mechanical drawing, business English, economics, etc.

Industry looks to vocational and trades training schools to supply young men with sufficient vocational training so that they can be further trained in the factory for positions of expert workmen, specialty mechanics, foremen, superintendents, salesmen and advertising men—not for the ordinary jobs of operating production machines.

A number of large industrial plants in the United States have established training courses in their own plants to supplement and continue the fundamental training now being given in the Vocational and Trade Schools.

Allowance for Finish

Fig. 356. Taking a Finish Turning Cut Following a Rough Turning Operation

The amount of stock which should be left when rough turning or boring a surface that is to be finish turned, reamed, ground, or otherwise finished, varies with the size of the work and the process used. Generally speaking, a minimum amount of stock should be left for the finishing cut. However, it is important that sufficient stock be left to make sure that the work will "clean up." Parts that are to be heat-treated must have ample finish to allow for warping during the heat-treating process. Allowance must also be made for warping which may result from the removal of an outer surface in which there are likely to be strains, such as castings or steel bars which have not been normalized.

Finish Turning

It has been found that an allowance of 1/64" to 1/32" on the diameter is usually sufficient for finish turning work of normal length up to 2" in diameter. For work over 2" in diameter, the allowance is usually between 1/32" and 1/16".

File Finish

When work is to be finished by filing in the lathe, the stock left for filing should be as small as possible. This not only saves time, but produces more accurate results. Very little stock need be left if the cutting tool is properly ground and set so that it will produce a smooth finish and will not scratch or tear the work. Usually about .001" to .0015" on the diameter will be ample, but if only .0005" is necessary, so much the better. See page 89.

Polishing

The amount of stock removed by polishing with emery cloth is so small that usually no allowance is made. Only a few tenths of a thousandth are required for a polishing operation, less than can be measured with ordinary micrometers. See page 89.

Grinding

The customary allowance for grinding is .010" to .012" on the diameter. Less may be required on small parts. However, if parts are to be heat-treated between the turning and grinding operations, an allowance of .015" to .020" on the diameter may be necessary to allow for warping.

Reaming

Holes that are to be finished by reaming must be drilled or bored undersize. The allowance for machine reaming in steel is usually .005" to .010", for cast iron .010" to .015". For hand reaming in steel or cast iron .0005" to .0015" is sufficient.

Lapping and Honing

Very little stock is removed by the lapping or honing processes. Usually less than .0005" on the diameter is allowed. See page 89.

MOTORS FOR OPERATING LATHES

Many different types of motors are available for operating individual motor driven lathes. In some cases any one of several different motors may be used, but usually there is one particular type of motor that will give more satisfactory service than any other. A brief study of the various types will aid in selecting the most desirable motor.

Reversing Motor Required—Several important lathe operations necessitate reversing the rotation of the lathe spindle. For this reason it is important that the motor be of the reversible type. The motor should conform exactly with the electric current that is to be used. Dual rated motors may be operated on two different voltages.

When more than one type of current is available, the motor should be selected for use with the most desirable type of current. Two or three-phase alternating current is generally considered more desirable for power service than single phase current or direct current.

The Size of Motor specified by the lathe manufacturer should always be used unless the lathe is to be operated at spindle speeds higher than standard. More power is required for high speed operation, necessitating a larger motor. The lathe manufacturer should always be consulted before equipping a lathe for high speed operation. The speed of the motor should be, as nearly as possible, the speed specified.

Polyphase Motors for 3-phase or 2-phase alternating current are instantly reversible, have a high starting torque and constant speed. These motors have no brushes or commutators to cause radio interference. They are very satisfactory for lathe operation.

Capacitor Motors for single phase alternating current are made in instant reversing, and start-stop reversing types. The instant reversing type is preferable but the start-stop reversing type is less expensive and may be used satisfactorily on certain classes of lathe work. This type of motor has a fairly high starting torque and constant speed. It has a starting switch but has no commutator to cause radio interference.

Repulsion Induction Motors for single phase alternating current have a high starting torque and fairly constant speed. They are made to order in the instant reversing type for lathe operation. Having a commutator and brushes, this motor may cause radio interference.

Split Phase Motors for single phase alternating current do not have sufficient starting torque for satisfactory use with a lathe.

Shunt Wound Motors for direct current are instantly reversible, have a good starting torque and a fairly constant speed. They are entirely satisfactory for lathe operation, although the commutator and brushes may cause radio interference.

Compound Wound Motors for direct current are instantly reversible. They have a high starting torque and a fairly constant speed. The compound wound motors are satisfactory for lathe operation but may cause radio interference due to the commutator and brush construction.

Universal Motors are designed for operating on either alternating current or direct current. This type of motor is seldom used for operating a metal working lathe.

Decimal Equivalents of Number Drills

Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent	Gauge Size	Decimal Equivalent
1	.2280	14	.1820	26	.1470	39	.0995
2	.2210	15	.1800	27	.1440	40	.0980
3	.2130	16	.1770	28	.1405	41	.0960
4	.2090	17	.1730	29	.1360	42	.0935
5	.2055	18	.1695	30	.1285	43	.0890
6	.2040	19	.1660	31	.1200	44	.0860
7	.2010	20	.1610	32	.1160	45	.0820
8	.1990	21	.1590	33	.1130	46	.0810
9	.1960	22	.1570	34	.1110	47	.0785
10	.1935	23	.1540	35	.1100	48	.0760
11	.1910	24	.1520	36	.1065	49	.0730
12	.1890	25	.1495	37	.1040	50	.0700
13	.1850			38	.1015		

Decimal Equivalents of Letter Drills

Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent	Letter Size	Decimal Equivalent
A	.234	H	.266	N	.302	U	.368
B	.238	I	.272	O	.316	V	.377
C	.242	J	.277	P	.323	W	.386
D	.246	K	.281	Q	.332	X	.397
E	.250	L	.290	R	.339	Y	.404
F	.257	M	.295	S	.348	Z	.413
G	.261			T	.358		

Use of Gauge Blocks

Precision gauge blocks are used for making extremely accurate measurements, usually for toolroom or inspection operations. Each hardened steel block has two opposite parallel faces ground and lapped to a fine finish, with the distance between them equal to the size marked on the block within a few millionths of an inch. A set of blocks includes an assortment of sizes which permits assembling various combinations to obtain any dimension within the capacity of the set, advancing in increments of one ten-thousandth of an inch. Assemblies up to 6″ in length are said to be accurate within one hundred-thousandth part of an inch at 66° F.

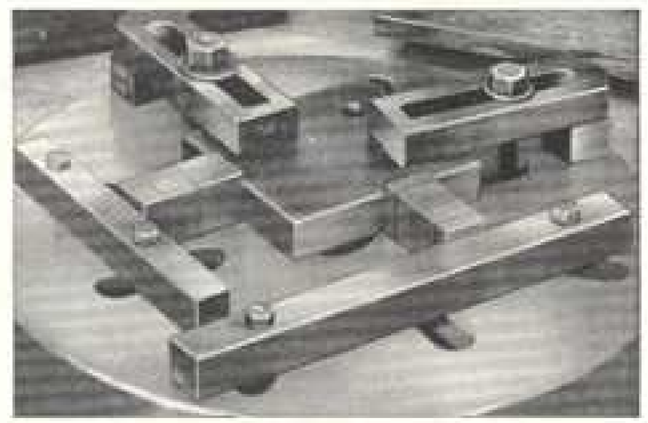


Fig. 359. Using Gauge Blocks to Position Work on Face Plate of Lathe

Gauge blocks may be used to locate jig plates and other work on the face plate of the lathe for boring accurately spaced holes to known vertical and horizontal dimensions. Accurate parallels must first be attached to the face plate in a position which will locate the hole farthest from the working edges of the work piece in line with the center of the lathe spindle. After the first hole is bored, the location of each subsequent hole is obtained by inserting gauge blocks of the required size between the work and the parallels.

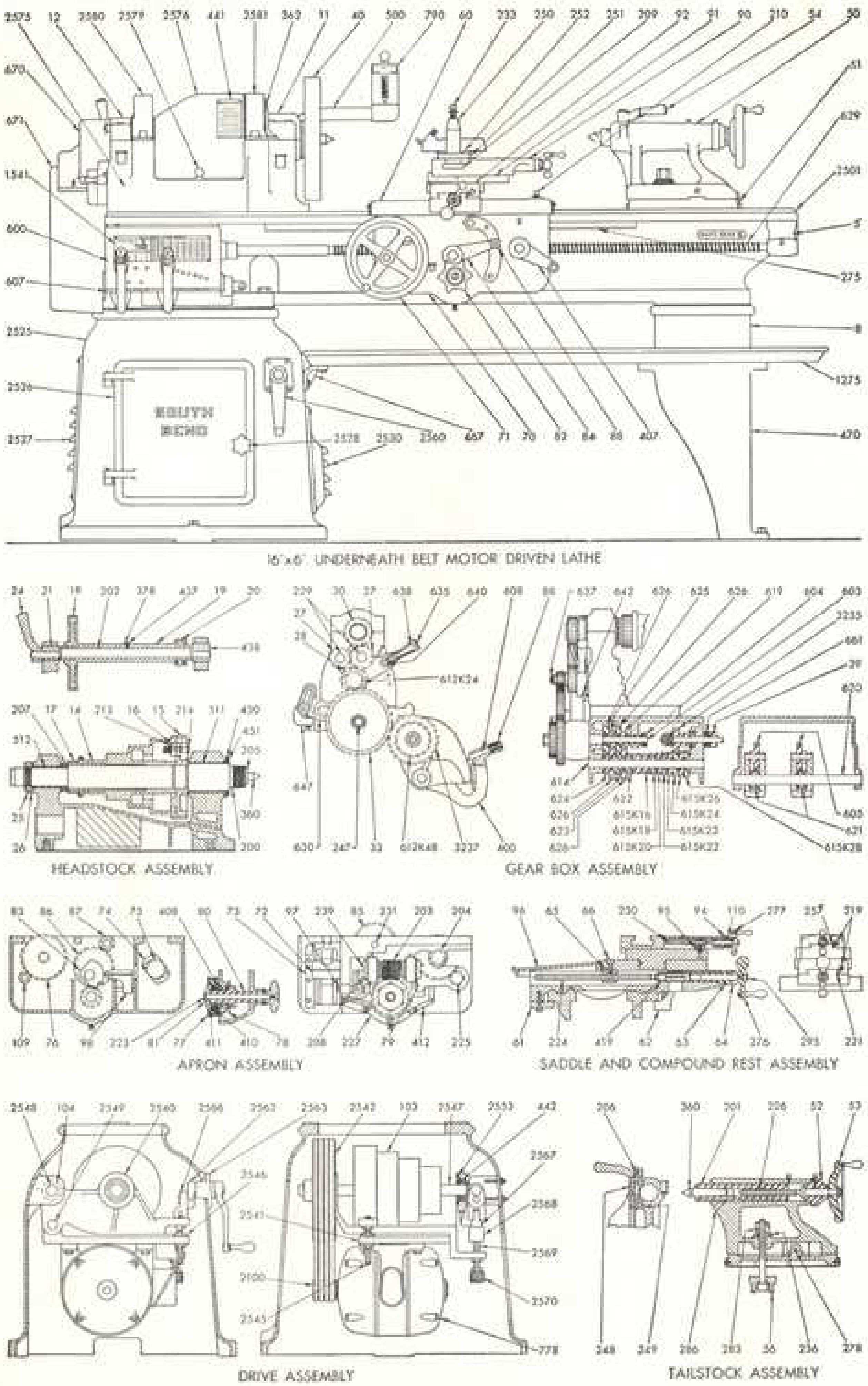


Fig. 362. Principal Parts of a South Bend Underneath Motor Drive Lathe

BASE NUMBERS AND NAMES OF LATHE PARTS

The base numbers of the parts of an Underneath Motor Drive Lathe are shown in Fig. 362. The names of the various parts are listed opposite the corresponding numbers, on this and the following page.

The base numbers identify each part in a general way, but do not designate the size or model of the lathe. A prefix or suffix is used with the base number to establish the exact part required to fit a particular size and model of lathe. However, if the serial number of the lathe is known, it is sometimes possible to identify a lathe part by the base number only.

Part No.	Name of Part	Part No.	Name of Part
5	Rear Lead Screw Bracket	80	Clutch Sleeve
8	Tailstock Leg (Bed to Chip Pan)	81	Apron Clutch Plate
11	Headstock Cap, Large	82	Apron Clutch Knob
12	Headstock Cap, Small	83	Apron Feed Change Gear Shaft
14	Spindle Cone Pulley	84	Apron Feed Change Lever
15	Bull Gear	85	Cross-Feed Compound Gear
16	Bull Gear Lock	86	Apron Feed Change Idler Gear
17	Cone Pulley Pinion	87	Cross-Feed Compound Pinion
18	Quill Gear	88	Knob
19	Quill Sleeve	90	Compound Rest Top
20	Quill Sleeve Pinion	91	Compound Rest Swivel
21	Eccentric Shaft Bushing	92	Compound Rest Base
24	Back-Gear Lever	94	Compound Rest Bushing
25	Spindle Take-up Nut	95	Compound Rest Nut
26	Spindle Take-up Nut Washer	96	Compound Rest Chip Guard
27	Twin Gears	97	Apron Feed Interlock Plunger
28	Reverse Gear	98	Apron Feed Interlock Arm
30	Spindle Gear	103	Countershaft Cone Pulley
33	Idler Gear	104	Cone Cradle Shaft Collar
39	Lead Screw Thrust Collar	110	Compound Rest Graduated Collar
40	Large Face Plate	200	Headstock Spindle
50	Tailstock Top	201	Tailstock Spindle
51	Tailstock Base	202	Eccentric Shaft
52	Tailstock Nut	203	Apron Feed Worm
53	Tailstock Handwheel	204	Rack Pinion
54	Tailstock Binding Lever	205	Headstock Spindle Sleeve
56	Tailstock Clamp	206	Tailstock Binding Lever Stud
60	Saddle	207	Spindle Thrust Bearing
61	Saddle Gib	208	Apron Worm Collars
62	Saddle Lock	209	Tool Post Block
63	Cross-Feed Bushing	210	Carriage Lock Screw
64	Cross-Feed Graduated Collar	213	Bull Gear Lock Gib
65	Cross-Feed Nut	214	Bull Gear Lock Plunger
66	Cross-Feed Nut Screw	219	Dovetail Gib Adjusting Screws
70	Apron	221	Compound Rest Base Gib
71	Apron Handwheel	223	Clutch Screw
72	Half-Nuts	224	Cross-Feed Screw
73	Half-Nut Gib	225	Apron Handwheel Pinion
74	Half-Nut Cam	226	Tailstock Screw
75	Half-Nut Friction Washer	227	Oil Reservoir Plate
76	Rack Pinion Gear	229	Twin Gear Studs
77	Worm Gear	230	Compound Rest Screw
78	Oil Distributing Washer	231	Cross-Feed Compound Gear Shaft
79	Worm Bushing	233	Tool Post Screw
		236	Tailstock Clamp Bolt Washer

Part No.	Name of Part
239	Apron Worm Key
247	Idler Gear Bolt
248	Tailstock Binding Lever Upper Plug
249	Tailstock Binding Lever Lower Plug
250	Tool Post
251	Tool Post Ring
252	Tool Post Wedge
257	Compound Rest Top Gib
275	Rack
276	Cross-Feed Ball Crank
277	Compound Rest Ball Crank
278	Tailstock Set-Over Screws
283	Tailstock Clamping Nut
286	Tailstock Spindle Key
295	Cross-Feed Ball Crank Retaining Nut
360	Center
362	Bull Gear Lock Guard
378	Quill Sleeve Oil Hole Washer
400	Tumbler
407	Half-Nut Lever
408	Worm Gear Lock Ring
409	Handwheel Pinion Thrust Spring
410	Inside Clutch Plates
411	Outside Clutch Plates
412	Oil Reservoir Gasket
419	Cross-Feed Screw Chip Collar
437	Quill Sleeve Oil Hole Plug
438	Eccentric Shaft Bushing, Large
441	Lubrication Plate
442	Countershaft Cone Cradle Bearings
450	Spindle Protecting Ring Gasket
451	Spindle Protecting Ring
467	Chip Pan Brackets
470	Tailstock Leg (Chip Pan to Floor)
500	Switch Arm
511	Headstock Spindle Bearing, Large
512	Headstock Spindle Bearing, Small
600	Gear Box
603	Main Drive Shaft
604	Main Drive Collar
605	Tumbler Idler Gear
607	Tumbler Guide Bar
608	Tumbler Plunger
612K24	Stud Gear
612K48	Stud Gear
614	Cone Gear Shaft
615K16	Gear Box Cone Gear—16 Teeth
615K18	Gear Box Cone Gear—18 Teeth
615K20	Gear Box Cone Gear—20 Teeth
615K22	Gear Box Cone Gear—22 Teeth
615K23	Gear Box Cone Gear—23 Teeth
615K24	Gear Box Cone Gear—24 Teeth
615K26	Gear Box Cone Gear—26 Teeth
615K28	Gear Box Cone Gear—28 Teeth
619	Small Compound Gear

Part No.	Name of Part
620	Tumbler Pinion Shaft
621	Main Drive Gear
622	Small Compound Gear
623	Small Compound Gear
624	Cone Shaft Gear
625	Small Compound Gear
626	Large Compound Gear
629	Lead Screw
630	Primary Gear Bracket
635	Reverse Bracket
637	Reverse Shaft
638	Reverse Latch Lever
640	Reverse Plunger
642	Reverse Lock
647	Primary Gear Bracket Support
661	Thrust Ball Bearing
670	Upper Gear Guard
671	Lower Gear Guard
778	Motor
790	Drum Switch
1275	Chip Pan
1541	Gear Box Index Plate
2100	Motor Pulley
2501	Lathe Bed
2525	Cabinet Leg
2526	Cabinet Leg Door
2527	Left Ventilator Cover
2528	Leg Door Knob
2530	Right Ventilator Cover
2540	Countershaft Cone Cradle
2541	Motor Mounting Plate
2542	Countershaft V-Belt Pulley
2545	Motor Belt Adjusting Stud
2546	Motor Belt Adjusting Stud Washers
2547	Countershaft Cone Pulley Shaft
2548	Cone Cradle Hinge Shaft
2549	Motor Plate Hinge Shaft
2553	Countershaft Bearing Retainer Washers
2560	Belt Tension Release Crank
2562	Belt Tension Eccentric
2563	Belt Tension Eccentric Bushing
2566	Belt Tension Eccentric Clevis
2567	Belt Tension Eccentric Ball Socket
2568	Belt Tension Eccentric Ball Socket Sleeve
2569	Countershaft Cone Cradle Adjusting Screw
2570	Countershaft Cone Cradle Adjusting Screw Nut
2575	Headstock
2576	Headstock Cone Pulley Cover
2579	Cone Pulley Cover Knob
2580	Quill Gear Guard
2581	Bull Gear Guard
3235	Lead Screw Gear
3237	Screw Gear

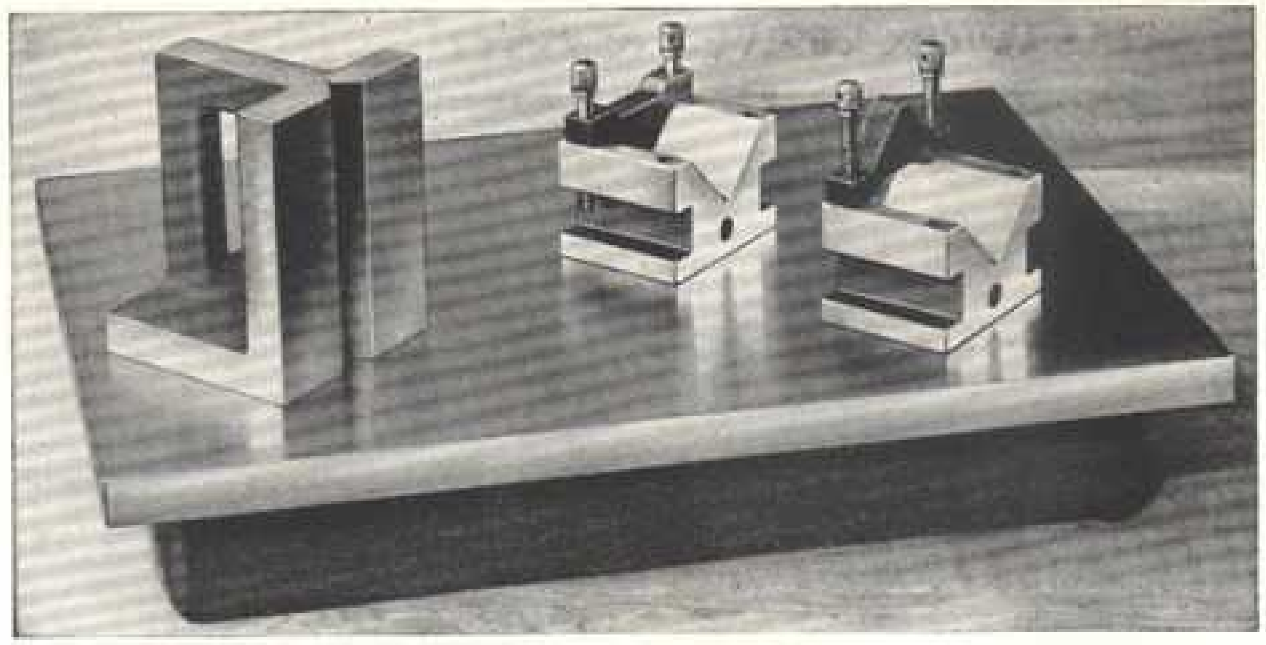


Fig. 363. Surface Plate with V-Blocks and Angle Plate for Laying Out Work

The Surface Plate

The surface plate, shown in the illustration above, is a flat cast iron plate used in the building of fine machinery to test plane surfaces. The surface plate is coated with a very thin film of pigment such as prussian blue or red lead and the surface to be tested is rubbed over it. The high spots on the surface tested will be marked with the pigment, and can be filed or scraped off. The surface is then re-tested and re-scraped until satisfactory.

Fig. 364. Toolmaker Locating Holes in Jig Plate with Aid of Surface Plate, Surface Gauge, and Gauge Blocks

Surface plates are also used by toolmakers for locating and checking holes in jig plates, laying out work, testing, inspecting, and similar operations. With the aid of a surface gauge and a set of gauge blocks very accurate work can be done on the surface plate.

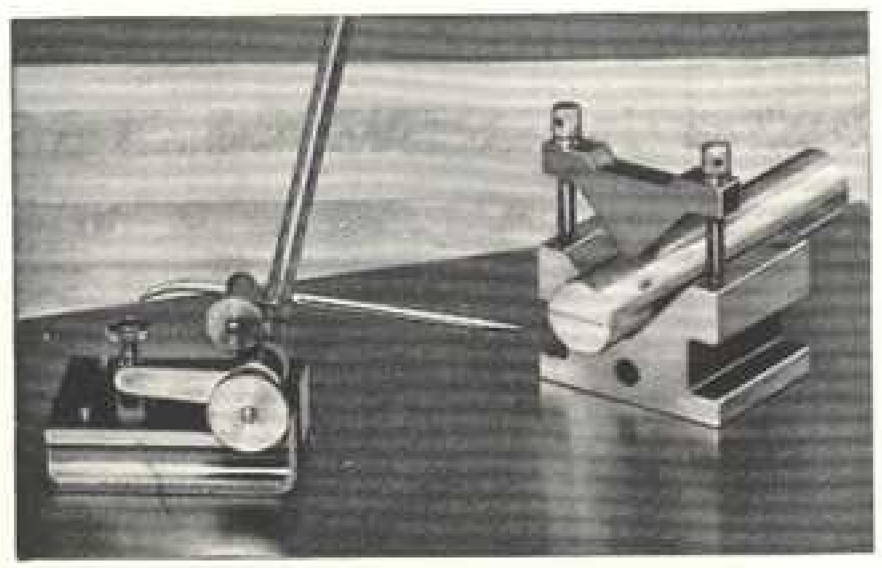
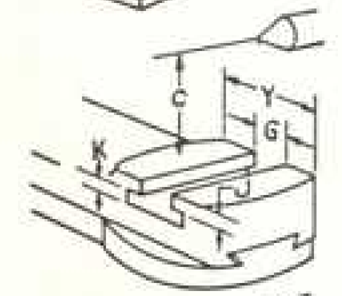
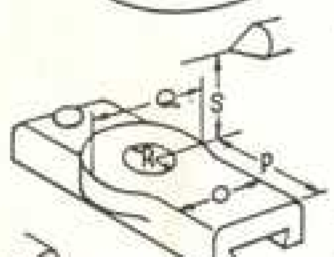
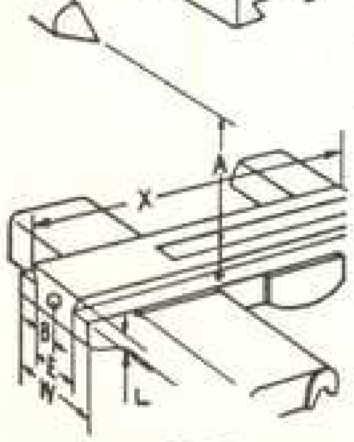
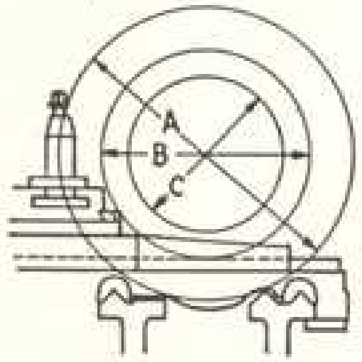


Fig. 365. Laying Out Work with Surface Plate, V-Block, and Surface Gauge









DIMENSIONS FOR TOOLING SOUTH BEND LATHES

All dimensions are in inches

SPINDLE NOSE THREAD

Size Lathe	9" and 10K	10" & 13" 1" Collet	10"-1 1/16" Collet	14½"-1" Collet	16" & 16-24"
E	1½—8	2¼—8	1⅞—8	2⅜—6	2⅜—6
F	11/16	15/16	15/16	1⅜	1⅜
G	1.509	2.259	1.884	2.384	2.384
H	⅞	1 3/16	1 3/16	1¾	1¾

SPINDLE TAPER, SPINDLE HOLE, SPINDLE LENGTH

Size Lathe	9"	10K	10"-1" Collet	10"-1 1/16" Collet	13"-1" Collet	14½"-1" Collet	16" & 16-24"
B	.938	.938	1.629	1.231	1.629	1.629	1.629
D	.602	.602	.602	.623	.602	.602	.602
K	¾	27/32	1⅜	1	1⅜	1⅜	1⅜
N	2	2	2	2	3	3	3
J	12¼	12¼	13 23/32	13⅝	21⅛	24 23/64	24 11/16

Note: Tailstock spindle taper is same as N above.

TOOL POST OPENING

Size Lathe	9" and 10K	10"	13"	14½"	16" & 16-24"
D	19/32	19/32	1 3/64	1 7/32	1 7/32
M	13/32	15/32	19/32	21/32	23/32
N	⅞	1	1¼	1¾	1¾

COMPOUND REST TOP

Size Lathe	9" and 10K	10"	13"	14½"	16" & 16-24"
C	1 1/32	1 1/32	1⅝	1 17/32	2 1/16
G	⅞	1	1¼	1 7/16	1⅝
J	9/32	21/64	7/16	7/16	7/16
K	9/32	9/32	13/32	17/32	17/32
Y	2½	2⅞	3⅝	4	4½
T	1 3/32	1 13/32	1 49/64	2 1/64	2 17/64

COMPOUND REST BASE

Size Lathe	9" and 10K	10"	13"	14½"	16" & 16-24"
O	2⅝	2⅞	4	4⅛	4⅝
P	3 5/16	3 5/16	4½	5⅛	5¾
Q	3½	3⅝	5¼	5⅝	6
R	1⅜	1⅝	2¼	2½	2¾
S	2 9/16	2 51/64	3¾	4 5/16	4 5/16

60° SADDLE DOVETAIL

Size Lathe	9" and 10K	10"	13"	14½"	16"	16-24"
A	3 19/32	3 61/64	5 7/32	5¾	6 5/16	10 13/16
B	.7655	.8328	1.1753	1.2690	1.4253	1.4253
E	1.5253	1.6032	2.1631	2.4756	2.7256	2.7256
L	23/64	7/16	9/16	9/16	9/16	9/16
W	3¼	3⅝	4 5/16	4⅞	5⅜	5⅜
X	10 1/16	11⅞	16 1/32	18 5/16	19⅞	10⅞

SWING OVER BED, SADDLE, COMPOUND REST

Size Lathe	9"	10K	10"	13"	14½"	16"	16-24"
A	9¼	10	10⅛	13⅛	14⅝	16¼	25⅛
B	...	...	6¾	8¾	10¼	11⅛	18¾
C	5½	6¼	5⅞	7¾	8¾	9⅝	19¼

INDEX

INDEX

D5824-ROXM-10-58

How to Become a Machinist

1. Keep your cutting tools sharp.
2. Look at your drawing carefully before starting your job.
3. Be sure your machine is set up right before starting the work.
4. Take your measurements accurately.
5. Keep your machine well oiled, clean and neat. Personal neatness will give you personality.
6. Take an interest in your job; don't feel that you are forced to work.
7. Learn the fundamentals of mechanical drawing.
8. Keep your belts tight and free from oil.
9. Take as heavy a cut as the machine and cutting tool will stand until you are near the finished size; then finish carefully and accurately.
10. Try to understand the mechanism of the machine you are operating.
11. Hold yourself responsible for the job you are working on.
12. Keep your eyes on the man ahead of you; you may be called on to take his place some day.
13. Have a place for everything, and keep everything in its place.
14. Read one or two of the technical magazines relating to your line of work.
15. If a boy learns a trade properly he becomes a first-class mechanic, but if he has ability he need not stop at that.

    Henry Ford, George Westinghouse and others got their start because they were mechanics.
16. If you have spoiled a job, admit your carelessness to your foreman, and don't offer any excuses.
17. Before starting to work on a lathe, roll up your sleeves and remove your necktie—safety pays.