MANUAL OF
MACHINE SHOP PRACTICE

BY

OTIS BENEDICT, JR.

Assistant Professor, Department of Shop Practice, Pratt Institute, School of Science and Technology, Brooklyn, N.Y.

McGRAW-HILL BOOK COMPANY, INC.

NEW YORK AND LONDON

1944

PRINTED IN THE UNITED STATES OF AMERICA

VII

PREFACE

This text is the outgrowth of notes prepared by the author, and the publication of this material in book form is the result of many requests that it be put into a more convenient and permanent form.

This is not another book on machine-tool operation, as there are many good works available on that subject, but rather a presentation of material that is helpful to the student pursuing a course in engineering shop practice. The very nature and diversity of the material are such that there must be some lack of continuity. A group of 10 experiments is provided and serves as a valuable method of instruction when the required equipment is available.

The author is sincerely grateful to the following publishers and manufacturers for their permission to use material from their publications: The Industrial Press, McGraw-Hill Publishing Company, Inc., Brown & Sharpe Manufacturing Company, South Bend Lathe Works, and The Taft-Peirce Manufacturing Company.

He also wishes to express his appreciation to his students for their criticism and suggestions.

OTIS BENEDICT, JR.

BROOKLYN, N.Y.,
August, 1944.

CONTENTS

EXPERIMENTS

APPENDIX

MANUAL OF MACHINE SHOP PRACTICE

CHAPTER I

CHISELS AND FILES

Machinist's chisels are generally made of $\frac{3}{4}$-in. octagonal tool steel and are about 8 in. long when new, although for delicate work they are sometimes made of $\frac{1}{2}$-in. steel.

The Flat Chisel.—The flat chisel (Fig. 1) is tapered and flattened about one-third its length to the cutting edge, which is about $\frac{3}{32}$ in. thick on the $\frac{3}{4}$-in. steel, and proportionally thinner on smaller steel. The flat chisel should be forged about $\frac{1}{16}$ in. wider at the cutting end and may be wider when used on soft metals. It will pay to grind the chisel to a sharper angle for the softer materials, a 30-deg. included angle being about right for babbitt metal, lead, and copper. For chipping brass and reasonably soft cast iron 45 deg. will answer, while for average steel, 60 deg. would be about right.

The experienced workman will not require a gauge to test the chisel angle, but common angles may be conveniently measured (by those who prefer) by using a gauge. Thus the chisel for steel may be tested by a center gauge.

The flat chisel should be so ground that the center line *CL* in Fig. 1 will bisect the angle of the cutting edge, or angle *A*. Looking at the other view, the end of the chisel may be square with the center line or slightly rounded as at *R*.

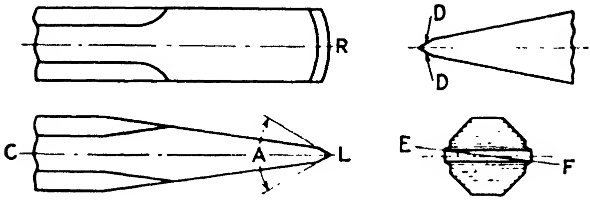


FIG. 1.

A fault that the beginner is very likely to fall into is that of grinding the extreme end at an angle with the flat sides, as at *EF* in Fig. 1; the line formed at *EF* should, of course, be parallel with the sides and the two facets at *D* should be quite flat.

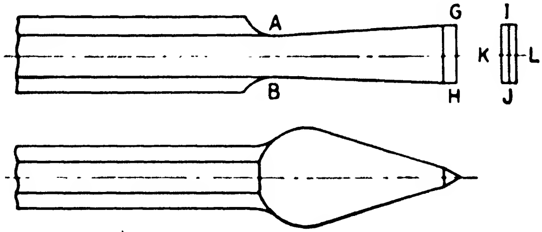


FIG. 2.

The Cape Chisel.—All that has been said respecting the flat chisel applies equally well to the cape chisel (Fig. 2), except that the sides at right angles to the cutting edge are narrower than the shank (octagonal part), and that the sides at right angles to these are spread wider where they join the shank. The cape chisel will cut up some ugly capers if not properly forged and ground. If not made narrower at *AB* than at *GH* (Fig. 2), it will, when the corners wear dull or tapering at *GH*, wedge and possibly break open a frail piece in which it is being used to cut a slot. Referring to the end view, if the sides *IJ* are not ground approximately at right angles with *KL*, the chisel will twist and hang and cannot be accurately guided in a slot or a keyway.

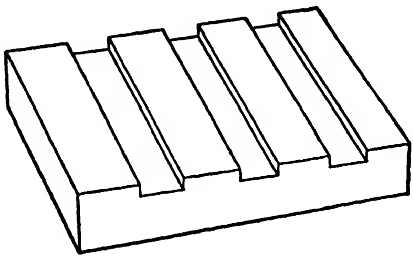


Fig. 3.

The Uses of Flat and Cape Chisels.—The flat chisel is used principally on flat surfaces, but it is also used for general chipping. If we have a light cut to take from a metal surface, we use the flat chisel. If we are to cut $\frac{1}{8}$ in. deep or deeper, it is best to precede the flat chisel by grooves cut with the cape chisel.

The distance between these grooves should be less than the width of the flat chisel, thus leaving on narrow strips to be chipped by the latter. This method is used in chipping broad surfaces, but it is unnecessary when the area is quite small. The surface shown in Fig. 3 represents this preparatory grooving with the cape chisel; the intervening strips are to be cut away with the flat chisel.

There are other uses of the cape chisel than that just described. It is used for cutting key-ways in shafts, pulleys, gears, etc., and also for cutting slots.

Smooth Chipping.—For chipping, the chisel should not be held near the cutting end, but near the head. To do smooth work it is necessary to maintain a constant inclination of the chisel to the surface being chipped. The proper inclination is easily determined during the first few blows of the hammer. If the angle is too great, the chisel will cut too deep; if too small, the chisel soon ceases to cut. The smaller the angle between the center line of the chisel and the surface of the work within proper limitations, the more effective the hammer blows; consequently a greater amount of work is accomplished.

The chisel should be kept well up against the shoulder formed by the cut. When the chisel approaches the edge of a surface, it should be reversed, or a cut taken at right angles to the preceding cut. Otherwise the edge of the metal, especially if it is cast iron, is likely to be broken.

Precautions in Grinding the Chisel.—No matter how carefully the chisel may be tempered, it may be softened in a few minutes by overheating the edge at the grinding wheel. To avoid this, a constant flow of water must be directed to the point of the chisel when grinding it. Grinding the temper from a chisel is a common fault with beginners, and sometimes they botch their work by attempting to use chisels in this condition.

Files.—A file (Fig. 4) may be defined as a bar of tool steel pointed at one end to receive a handle and having cutting edges or teeth extending from near the handle to the opposite end.

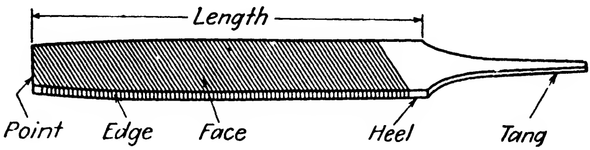


FIG. 4.

Classification of Files.—There is a bewildering array of names, shapes, and peculiarities connected with the subject of files. We shall consider only a few of them:

1. Single-cut.
    a. Rough.
    b. Coarse.
    c. Bastard.
    d. Second-cut.
    e. Smooth.
2. Double-cut.
    a. Coarse.
    b. Bastard.
    c. Second-cut.
    d. Smooth.
    e. Dead-smooth.
3. Rasps.
    a. Coarse.
    b. Bastard.
    c. Second-cut.
    d. Smooth.

Single-cut files have one series of teeth; double-cut files have a second series cut diagonally across the first series. Files are cut with a kind of chisel-edged tool. Rasps are cut with a pointed punch. The teeth of rasps are therefore disconnected cutting points, rather than continuous cutting edges.

Pitch of Teeth.—The terms rough, coarse, bastard, second-cut, etc., refer to the pitch of the teeth, or degree of fineness, the coarsest being about 20 and the finest about 120 to the inch. It is important to observe, however, that these terms do not, independently of the length of the file, definitely express the number of teeth per

4 in. 16 in.

FIG. 5.—Hand bastard file. (*Courtesy of Nicholson File Company, New York.*)

inch for a given file. They rather indicate the range of pitches for a given nominal cut. To express the pitch definitely, the length of the file must be coupled with the name of the cut. Thus when we speak of the bastard file, we refer to a subclass of files, the pitches of which vary within certain limits as the length of the file varies. But when we speak of a 12-in. bastard we mean a file having a definite number of teeth per lineal inch. Figure 5 shows the difference in pitch due to the difference in length between the longest and shortest files of the same cut. To avoid confusion, which has often arisen, one should note that the terms double-cut and second-cut are not synonymous. One should also remember that, as distinguished from double-cut, which refers to a file having two courses of chisel cuts crossing each other, single-cut means one course of teeth.

Convexity of Files.—Most files are made with the faces slightly curved lengthwise or "bellied." There are good reasons for this. If when filing a broad surface all the teeth were in contact, it would require too much pressure downward to make the file bite as well as forward to make it cut. This would mean practically double work and also make it more difficult to control the file. If the face of the file were straight to produce a flat surface, every part of the stroke would have to be perfectly straight. This is impossible.

Note.—An old file does not readily take hold of brass and cast iron, and it is proper to start a new file on these and other cast metals, except cast surfaces. After the extreme points of the teeth are dulled somewhat, the file may be used on fibrous materials such as steel and wrought iron.

Oil may be removed from a file by filling the teeth with chalk and then brushing the chalk out with a file brush.

In filing work having two plane surfaces at right angles it is sometimes necessary to file one surface without cutting the other. For this purpose, we use a file having one blank side or edge. Such files are called "safe-edge" files, and they may be purchased from dealers.

The files most used in the machine shop are the hand file, pillar file, mill file, equaling file, and the round file.

CHAPTER II

CUTTING SPEEDS

Cutting tools are essentially wedges that are used to remove excess material. This excess material must be removed as rapidly and as economically as possible, and at the same time a surface must be produced, the grade of finish depending on the place where the part or article is to be used.

In order to fulfill the above conditions, certain factors must be taken into consideration. Some of these factors are

1. Kind of cutting material.
2. Kind of material to be machined.
3. Size and shape of material to be machined.
4. Shapes of cutting tools, cutting angles, clearance angles, angles of side rake or back rake.
5. Rate of removal of material.
    a. Cutting speed.
    b. Feed.
    c. Depth of cut.

In general, the following rules apply:

For heavy roughing cuts, use slow speeds and heavy feeds.

For light finishing cuts, use fast speeds and light feeds.

The harder the material to be machined, the larger the cutting angle necessary to support the cutting edge. An increase of the cutting angle usually results in smaller clearance and rake angles. As the material decreases in hardness, the clearance and rake angles are increased, giving a smaller cutting angle to the tool and hence a much keener cutting tool.

No hard and fast rules can be laid down for speeds and feeds, but those cutting speeds given in Table I, page 201, are representative of good practice.

The cutting speed is usually considered as a constant, therefore the following general statement holds true.

If the diameter of the work is decreased, the revolutions per minute (r.p.m.) will be increased.

If the diameter of the work is increased, the r.p.m. will be decreased.

The cutting action is much the same, whether it takes place in the lathe when turning or boring, in the drill press when drilling, or in the milling machine when milling.

In the removal of the surplus material by the cutting tool, heat is generated. Some of the heat passes into the metal, and some of it is absorbed by the cutting tool.

If the rate of removal is such that the heat generated causes the cutting tool to lose its hardness, it ceases to cut and may be made useless as a cutting tool thereafter. Many cutting tools are ruined by being operated at too high a speed.

When metal is cut in a lathe arranged to vary the speed 1 or 2 r.p.m. at a time, the speed can be increased until the heat generated in cutting will cause the cutting tool to lose its hardness, and cutting will cease.

If the diameter of the work and its speed are recorded when the cutting tool loses its hardness, the cutting speed at which the tool failed can be calculated.

In practice, a cutting speed would be used somewhat lower than that at which the tool failed.

The cutting speed of a tool is the distance in feet that the tool point cuts in 1 min. If the point of a lathe tool (Fig. 6) cuts 60 ft., measured around the circumference of the work turned in

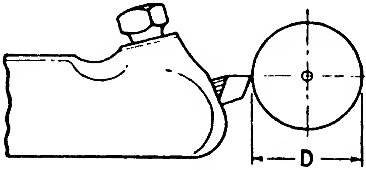


FIG. 6.

the lathe, in 1 min., the cutting speed is said to be 60 ft. per min. (see Table I).

On the planer and shaper, the cutting speed is the length of cut that would be taken in 1 min. Thus, if 15 sec. (or ¼ min.) is required to take a cut 10 ft. long, in 1 min. the cut would be 4 × 10, or 40 ft. The length of the cut is 40 ft. in 1 min., and the cutting speed is 40 ft. per min.

When drilling a hole in the drill press, the cutting speed is the number of feet that the outer corners of the cutting edges travel in 1 min.

Lathe, Boring Mill, and Drill Press.—The problems in cutting speeds in the lathe or boring mill are divided into two groups.

1. *To Find the Cutting Speed.*—The diameter of the work and the number of revolutions that

the work makes in the lathe or boring mill are known. What is the cutting speed?

A brass rod 2 in. in diameter is to be turned in the lathe. By counting the number of revolutions of the spindle by means of a speed indicator (Fig. 7) it is found that the work revolves 200 r.p.m. To find the cutting speed, the circumfer-

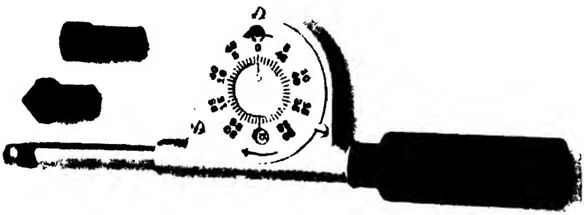


FIG. 7.—(*From Brown & Sharpe Manufacturing Company.*)

ence is first computed and changed to feet. The circumference in inches is 2 × 3.1416 = 6.2832, and 6.2832 ÷ 12 = 0.5236, the circumference in feet, or the distance passed over by the tool point for each revolution. During 200 revolutions, the distance passed over is 200 × 0.5236 = 104.72 ft., which is the cutting speed in feet per minute.

The formula for this calculation is written

$$\text{Cutting speed in feet per minute} = \frac{\text{diameter of work in inches} \times 3.1416}{12} \times \text{r.p.m.} \qquad (1)$$

If S = cutting speed in feet per minute, D = diameter of work in inches, and N = r.p.m., this equation can be written

$$S = \frac{D \times 3.1416}{12} \times N \qquad (2)$$

If in this formula D = diameter of work or diameter of bored or drilled hole in inches, the equation can be used to find the cutting speeds for drills and boring tools.

(If the cut taken on a piece being turned is deep in proportion to the diameter of the work, it is preferable in calculations for the cutting speed and r.p.m. to consider the mean diameter of the cut instead of the outside diameter of the work and use the value for the mean diameter in the rules and equations given. When the outside diameter and the depth of the cut are known, the mean diameter equals the outside diameter minus the depth of the cut.)

2. *To Find the Revolutions per Minute of the Work.*—The diameter of the work turned in the lathe (Fig. 6) or in the boring mill and the required cutting speed are known. How many r.p.m. should the work make?

The diameter of the work is given as 3 in., and a cutting speed of 40 ft. per min. is required. Find the speed (r.p.m.) of the work. When the diameter of the work is known, its circumference equals the diameter times 3.1416. Therefore the circumference of the work is

$$3 \times 3.1416 = 9.4248 \text{ in}$$

(9.43 in. is near enough for calculations of this kind). For each revolution of the work, the length of its circumference passes the tool point once; thus for each revolution a length of 9.43 in. passes the tool. As the cutting speed is always expressed in feet, the length 9.43 in. should also be expressed in feet. This is done by dividing by 12; thus, $9.43 \div 12 = 0.786$ ft., the circumference of the work. Now the question is, how many revolutions, each equal to 0.786 ft., does it require to get a cutting speed of 40 ft.? The answer is obtained by dividing 40 by 0.786. The result of this division is 50.9, and 50.9 is the required number of r.p.m. to give a cutting speed of 40 ft. per min. In practice, 51 revolutions would be used.

The formula for this calculation is written

$$\text{R.p.m.} = \frac{\text{cutting speed in feet per minute}}{(\text{diameter of work in inches} \times 3.1416) \div 12} \quad (3)$$

Using the same letters to denote the quantities in this formula as in Eq. (2), the formula can be written

$$N = \frac{S}{(D \times 3.1416) \div 12} = \frac{S \times 12}{D \times 3.1416} \quad (4)$$

If instead of turning work 3 in. in diameter, it is required to bore a hole 3 in. in diameter

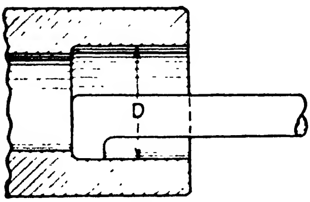


Fig. 8.

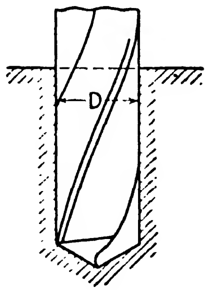


Fig. 9.

(Fig. 8) and the same cutting speed of 40 ft. per min. is required, the calculation for the r.p.m. is carried out in the same manner as above. Equations (3) and (4) are used, except that in the formulas we write "diameter of the bored hole in inches" instead of "diameter of the work in inches."

For work done in the drill press, Eqs. (3) and (4) can also be used by substituting "diameter of hole to be drilled in inches" for "diameter of work in inches" (Fig. 9).

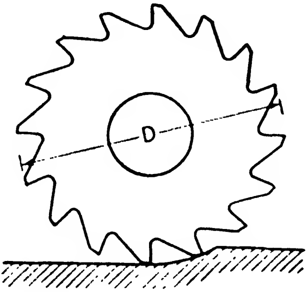


FIG. 10.

Thus, if D = diameter of the work to be turned or diameter of the hole to be bored or drilled in inches, then Eqs. (3) and (4) apply to turned, bored, or drilled work.

Milling-machine Cutters.—In the milling machine, the milling cutter (Fig. 10) is usually mounted on an arbor and rotates. The cutting speeds of milling cutters can be calculated when the diameter of the cutter and the r.p.m. are known. It is necessary to find the cutting speed in feet per minute for a cutter 4 in. in diameter that makes 50 r.p.m.

To find the cutting speed in feet per minute, first find the circumference of the cutter; thus, $4 \times 3.1416 = 12.5664$, or 12.6 in. Change this to feet; thus, $12.6 \div 12 = 1.05$ ft. Since the cutter makes 50 r.p.m., the cutting speed is

50 times the circumference, or $50 \times 1.05 = 52.5$ ft. per min.

If, in Eq. (2), D = diameter of cutter, this formula can be used to find the cutting speed of milling cutters.

If the cutting speed of a cutter is given and its diameter known, the number of revolutions (r.p.m.) at which it should be run can be found by Eq. (4). In this case, D is the diameter of the milling cutter in inches.

The Planer.—In the planer the platen is given a reciprocating motion. The speed at which the platen returns when the cutting stroke is completed is usually two or more times the cutting speed. When the return speed is twice as fast as the cutting speed, the ratio of the return speed to the cutting speed is said to be 2 to 1. When the return speed is three times as fast as the cutting speed, the ratio between the speeds is 3 to 1. These ratios are usually designated 2, 3, etc. If the return speed is 60 ft. and the cutting speed 30 ft. per min., the ratio is 2, while, if the return speed is 90 ft. per min., the ratio is 3. When the length of the cutting stroke, the number of strokes per minute, and the ratio of the speeds are known, the cutting speed and the return speed can be calculated. The length of the stroke can be measured and the number of strokes per minute can be counted. For long strokes, the time required for the cutting stroke and the return stroke can be determined with a stop clock. The ratio of the return speed to the cutting speed is determined by the design of the planer. Thus the cutting speed and the return speed of a planer can be readily determined.

The number of strokes is six, the length of the cutting stroke is 4 ft., and the ratio of the return speed to the cutting speed is 3. The time required for the platen to make one complete stroke (cutting stroke plus return stroke) is $\frac{1}{6}$ min. In $\frac{1}{6}$ min., the platen travels 4 ft. in cutting and 4 ft. in returning. Its rate of travel in cutting is not 4 ft. in $\frac{1}{6}$ min. but greater than 4 ft. in $\frac{1}{6}$ min. If no return stroke were to take place, the platen would travel 4 ft. plus $\frac{1}{3} \times 4$ ft., or a total of $4 + 1\frac{1}{3} = 5\frac{1}{3}$ ft. (The return stroke takes place in one-third the time of the cutting stroke.) Thus, the platen travels at the rate of $5\frac{1}{3}$ ft. in $\frac{1}{6}$ min., and the true cutting speed is $5\frac{1}{3}$ ft. per stroke. The number of strokes per minute of the platen times the true cutting speed per stroke equals the cutting speed of the planer in feet per minute; thus, $6 \times 5\frac{1}{3} = 32$ ft. per min. cutting speed.

The formula for the cutting speed for the planer can be written

$$\text{Cutting speed in feet per minute} = \text{number of strokes per minute} \times \left(\text{length of stroke in feet} + \frac{\text{length of stroke}}{\text{ratio of return speed to cutting speed}}\right) \quad (5)$$

If S = cutting speed in feet per minute, N = number of strokes per minute, L = length of stroke in feet, and P = ratio of the return speed to the cutting speed, this formula can be written

$$S = N \times \left(L + \frac{L}{P}\right) \quad (6)$$

The return speed in feet per minute can be found by multiplying the cutting speed by the ratio of the return speed to the cutting speed. The cutting speed from Eq. (6) is 32 ft. per min. and the ratio is 3. Thus, $3 \times 32 = 96$ ft. per min. return speed.

If R = return speed in feet per minute, the formula for return speed can be written

$$R = S \times P \qquad (7)$$

In Eqs. (5), (6), and (7), the time lost in reversing is not considered.

The number of strokes per minute can be found when the cutting speed, return speed, and length of stroke are known.

Given a cutting speed of 32 ft. per min., the length of stroke 4 ft., and the ratio of the return speed to the cutting speed as 3, find the number of strokes per minute.

This problem can be solved in two ways:

1. The ratio of the return speed to the cutting speed is 3; thus, the return speed is three times the cutting speed, or 96 ft. per min. As the length of the cutting stroke is 4 ft. and the cutting speed is 32 ft. per min., the time necessary to complete one forward stroke is equal to $4 \div 32 = \frac{1}{8}$ min.

In a like manner, the time for one return stroke is $4 \div 96 = \frac{1}{24}$ min.

The time for one complete stroke, therefore, is $\frac{1}{8} + \frac{1}{24} = \frac{4}{24}$, or $\frac{1}{6}$ min. The number of strokes per minute is obtained by finding how many times $\frac{1}{6}$ is contained in 1 min., or by dividing 1 by $\frac{1}{6}$; thus, $1 \div \frac{1}{6} = 6$, the number of strokes per minute.

In the above calculation, the time lost at the moment of reversal is not considered, and the formula can be written as follows:

$$N = \frac{1}{\frac{L}{S} + \frac{L}{R}} \qquad (8)$$

where N = number of strokes per minute.
L = length of stroke, ft.
S = cutting speed, ft. per min.
R = return speed, ft. per min.
P = R/S, ratio of return speed to cutting speed.

2. When the formula for cutting speed

$$S = N \times \left(L + \frac{L}{P}\right)$$

is solved for N, we get

$$N = \frac{S}{L + \frac{L}{P}}$$

Substituting the proper values from above in this formula, we have

$$N = \frac{32}{4 + \frac{4}{3}} = \frac{32}{5\frac{1}{3}} = 6 \text{ strokes per minute.}$$

The Crank Shaper.—The cutting tool in the crank shaper is given a reciprocating motion by means of a crank arrangement built into the shaper.

In Fig. 11, the path of the crankpin is represented by the circle. The arc C represents the drive for the forward, or cutting, stroke of the cutting tool, and arc R represents the return stroke. To find the cutting speed in feet per minute, when the length of stroke in inches and the number of strokes per minute are known, proceed as follows.

The length of the stroke is 18 in. and the number of strokes per minute is 30. When the cutting tool travels a distance of 18 in., the crankpin will have traveled the arc C, or three-fifths of a revolution. In one-fifth of a revolution of the crankpin, the cutting tool will travel 6 in., and in five-fifths (or one revolution) the tool will travel 5 × 6 in., or 30 in.

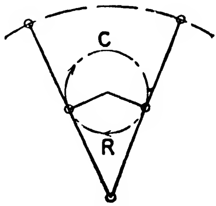


FIG. 11.

From Fig. 11, we know that the tool travels 18 in. for one revolution of the crankpin. But the tool would travel 30 in. if it moved forward only, when the crankpin makes one revolution. This same result can be obtained by multiplying the length of the stroke by the reciprocal of the part of a revolution of the crankpin required to produce the forward movement of the tool; thus, $\frac{5}{3} \times 18 = 30$ in. Changing this to feet, $30 \div 12 = \frac{5}{2}$, or $2\frac{1}{2}$ ft. Since the cutting tool makes 30 strokes per minute, the cutting speed is thirty times the number of feet per stroke, or $30 \times 2\frac{1}{2} = 75$ ft.

This calculation is expressed by the formula

$$\text{Cutting speed in feet per minute} = \frac{\text{length of stroke in inches}}{12} \times \text{number of strokes per minute} \times \frac{5}{3} \quad (9)$$

If S = cutting speed in feet per minute, L = length of stroke in inches, and N = number of strokes per minute, the formula can be written

$$S = \frac{L}{12} \times N \times \frac{5}{3} \qquad (10)$$

In Fig. 11, the arc R represents two-fifths of a revolution of the crankpin. Thus, the ratio of the return speed to the cutting speed is

$$\frac{\frac{3}{5}}{\frac{2}{5}} = \frac{3}{5} \times \frac{5}{2} = \frac{3}{2} = \frac{1\frac{1}{2}}{1}, \text{ or } 1\tfrac{1}{2} \text{ to } 1$$

Thus, if the ratio P equals $1\frac{1}{2}$, the cutting speed can be found by using Eq. (6).

$$\begin{aligned} S &= N \times \left(L + \frac{L}{P}\right) \\ &= 30 \times \left(1\tfrac{1}{2} + \frac{1\frac{1}{2}}{1\frac{1}{2}}\right) \\ &= 30 \times (1\tfrac{1}{2} + 1) \\ &= 30 \times 2\tfrac{1}{2} \\ &= 75 \end{aligned}$$

The number of strokes per minute can be found when the length of the stroke and the cutting speed in feet per minute are known. The length of the stroke is 18 in., and the cutting speed in feet per minute is 75. Divide the length of the stroke in inches by 12 to change to feet; thus, $18 \div 12 = 1\frac{1}{2}$ ft. The length of the stroke in feet times the fraction $\frac{5}{3}$ will give the feet cut in one stroke; thus, $1\frac{1}{2} \times \frac{5}{3} = 2\frac{1}{2}$ ft. The number of strokes per minute is found by dividing the cutting speed in feet per minute by the feet cut in one stroke; thus, $75 \div 2\frac{1}{2} = 30$ strokes

This calculation can be expressed by the formula

$$\text{No. of strokes per min.} = \frac{\text{cutting speed, ft. per min.}}{\left(\frac{\text{length of stroke, in.}}{12} \times \frac{5}{3}\right)} \qquad (11)$$

Using the same letters to denote the quantities in this formula as in Eq. (10), we may write

$$N = \frac{S}{\frac{L}{12} \times \frac{5}{3}} \qquad (12)$$

Problems

1. Find the cutting speed for turning a brass rod $1\frac{1}{4}$ in. in diameter running at 252 r.p.m.

2. A piece of tool steel $1\frac{1}{2}$ in. in diameter is turned in a lathe. Find the r.p.m. to give a cutting speed of 40 ft. per min.

3. Find the r.p.m. to turn a 14-in.-diameter cast-iron pulley in a lathe when a high-speed steel tool is used.

4. A hole 4 in. in diameter is to be bored in the hub of a gear mounted in a chuck on a lathe. Find the r.p.m. if the cutting speed is 60 ft. per min.

5. A cast-iron flywheel for a diesel engine is to be machined in a vertical boring mill. Find the r.p.m. for turning the outside diameter when a stellite-tipped tool is used.

6. A brass rod $1\frac{3}{4}$ in. in diameter is turned in the lathe at 152 r.p.m. Find its cutting speed.

7. A $1\frac{1}{4}$-in.-diameter drill makes 110 r.p.m. What is its cutting speed?

8. A $1\frac{3}{4}$-in.-diameter drill operates at a cutting speed of 70 ft. per min. Find its r.p.m.

9. A $\frac{3}{4}$-in.-diameter drill is used to drill a hole in a gray-iron casting. Find its r.p.m. if the cutting speed is 35 ft. per min.

10. A 1-in. diameter drill is operated at a speed of 115 r.p.m. What is its cutting speed?

11. A No. 40 high-speed steel drill has a diameter of 0.098 in. Find the r.p.m. of the drill for drilling aluminum.

12. A milling cutter 8 in. in diameter operates at a speed of 30 r.p.m. What is its cutting speed?

13. A $2\frac{1}{2}$-in.-diameter milling cutter is used to mill a gray-iron casting at a cutting speed of 35 ft. per min. At what r.p.m. should the cutter rotate?

14. The aluminum crankcase of an airplane engine is milled with a 10-in.-diameter cemented-carbide-tipped face milling cutter. At what r.p.m. should the cutter operate?

15. A $2\frac{1}{2}$-in-diameter 14-diametral-pitch high-speed steel gear cutter is used in cutting a 132-tooth cast-iron gear. Find the r.p.m. of the cutter.

16. Find the diameter of a milling cutter operating at 25 r.p.m. and a cutting speed of 35 ft. per min.

17. Find the r.p.m. of a milling cutter 4 in. in diameter if the cutting speed is 80 ft. per min.

18. A $4\frac{1}{2}$-in.-diameter milling cutter is used to machine a soft gray-iron casting. Select the proper cutting speed, and find the r.p.m. of the cutter.

19. The length of the stroke is 68 in., the number of strokes per minute is $3\frac{1}{4}$, and the ratio of the return speed to the cutting speed is 2.4. Find the cutting speed of the planer.

20. Find the return speed for the planer in Prob. 19.

21. Find the number of strokes for planing when the cutting speed is 60 ft per min., the stroke is 10 ft., and the ratio is 2.

22. In planing a casting the cutting speed is 40 ft. per min., the return speed is 80 ft. per min., and the length of the cutting stroke is 15 in. Find the number of strokes per minute of the platen.

23. Find the cutting speed of a planer that makes six strokes per minute when the length of the stroke is 3 ft. and the ratio of the return speed to the cutting speed is 3.

24. The return speed of a planer is 54 ft. per min. Find the number of strokes per minute if the ratio is 2 and the length of the stroke is 3 ft.

25. What is the cutting speed of a shaper making 50 strokes per minute when the length of the stroke is 6 in.

26. Find the number of strokes per minute for shaping when the length of the cutting stroke is 14 in. and the cutting speed is 80 ft. per min.

27. Find the cutting speed for shaping when the length of the cutting stroke is 12 in. and the number of strokes per minute is 60.

28. Find the return speed for the shaper in Prob. 27.

29. The stroke of a shaper is 6 in., and the number of strokes is 80 per minute. Find the cutting speed if the ratio of the cutting stroke to the return stroke is 2 to 3.

CHAPTER III

TIME FOR MACHINING

The time required to perform various machining operations is important. It is often necessary to know the time to machine a casting or forging without actually performing the machining operation. Given the necessary data, results that compare favorably with actual machining time can be computed.

Feed of Cutting Tools.—The feed of a lathe tool is the amount of side movement of the tool for each revolution of the work. If the feed is $\frac{1}{64}$ in., it means that for each revolution of the work the lathe carriage and tool move $\frac{1}{64}$ in. along the lathe bed, cutting a chip $\frac{1}{64}$ in. wide.

The feed of a drill in the drill press is its downward movement per revolution.

The feed of a milling cutter is the movement of the milling-machine table for each revolution of the cutter.

Time Required for Turning or Boring Work in the Lathe.—To find the time required to turn or bore a piece of work in the lathe when the feed, the cutting speed, and the diameter of the work are known, first find the r.p.m. of the work, using Eq. (4).

A steel bar $1\frac{3}{4}$ in. in diameter is to be turned. The length of the turned part is 10 in. The cutting speed is 35 ft. per min., and the feed of the cutting tool is 0.015 in. per revolution. How long will it take to make one cut over the surface of the work?

$$N = \frac{S \times 12}{D \times 3.1416} = \frac{35 \times 12}{1\frac{3}{4} \times 3.1416} = 76.5$$

As the tool feeds forward 0.015 in. for each revolution of the work, it is fed forward 76.5 × 0.015, or 1.15 in. in 1 min. The time required to traverse the whole length of the work, 10 in., is obtained by finding how many times 1.15 is contained in 10, or by dividing 10 by 1.15. The result of this division is 8.69 min. It would take 8.7 min. to traverse the work once with the feed and speed given.

Expressed in a formula, the calculation takes the form

Time to take one cut over the work

$$= \frac{\text{length of cut in inches}}{\text{feed in inches per revolution} \times \text{r.p.m.}} \qquad (13)$$

If T = time in minutes to take one cut over the work, L = length of cut in inches, N = r.p.m. of the work, and F = feed in inches per revolution, then the formula can be written

$$T = \frac{L}{F \times N} \qquad (14)$$

Using the formula for this calculation

$$T = \frac{10}{0.015 \times 76.5} = \frac{10}{1.15} = 8.69 \text{ min.}$$

Time Required for Drilling.—The time required to drill a hole through a piece of metal or to drill a blind hole of a given depth can be calculated when the diameter of the drill, the feed per revolution, and the cutting speed are known.

A 1-in. diameter high-speed steel drill is to operate at a cutting speed of 60 ft. per min. and a feed of 0.010 in. per revolution. Find the time required to drill a hole 3 in. deep. Figure 12 shows that the total distance the drill travels

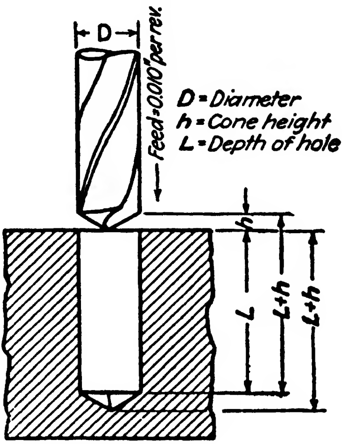


FIG. 12.

downward is equal to the depth of the hole L plus the cone height h of the drill, or $L + h$. The cone height can be calculated, but it is more convenient to refer to a table. From Table III (page 203), the cone height for a 1-in. drill is 0.301 in.; thus, the distance the drill travels downward is 3 in. + 0.301 in., or 3.301 in. For calculations of this kind, 3.3 in. is accurate enough.

The r.p.m. can be found from Eq. (4).

$$N = \frac{S \times 12}{D \times 3.1416}$$

$$N = \frac{60 \times 12}{1 \times 3.1416} = 229.18$$

The distance the drill feeds downward in 1 min. is equal to the feed in inches per revolution times the r.p.m. the drill makes; thus,

$$0.010 \times 229 = 2.29 \text{ in.}$$

As the feed of the drill is 2.29 in. in 1 min., the time required for drilling a hole 3.3 in. deep is found by dividing 3.3 by 2.29; thus,

$$3.3 \div 2.29 = 1.44 \text{ min.}$$

The time for drilling the hole is 1.5 min., nearly.

If T = time required for drilling in minutes, L = depth of drilled hole in inches, N = r.p.m. of the drill, F = feed in inches per revolution, and h = cone height for drill, the formula for this calculation can be written

$$T = \frac{L + h}{F \times N} \qquad (15)$$

Using this formula,

$$T = \frac{3 + 0.301}{0.010 \times 229} = 1.44 \text{ min.}$$

In the Eq. (15), $F \times N = F_M$, the feed in inches per minute. The formula can then be written

$$T = \frac{L + h}{F_M} \qquad (16)$$

Time Required for Milling.—The time required to slab mill a surface with a plain milling cutter can be found if the diameter of the cutter, feed per revolution, and the cutting speed are known.

It is necessary to machine a surface 6 in. long with a milling cutter $2\frac{1}{2}$ in. in diameter. The depth of cut is $\frac{1}{16}$ in., the cutting speed is 60 ft. per min., and the feed is 0.046 in. per revolution.

In Fig. 13*a*, it is seen that the total distance the table (also the work) travels to take one cut over the work is equal to its length *L* plus the

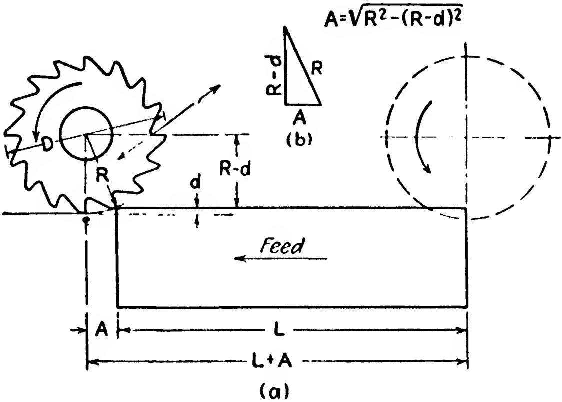


Fig. 13.

distance *A*, the approach of the cutter to the work. *L* can be measured, but *A* must be calculated. In Fig. 13*b*, the side *A* of the triangle is equal to the approach of the cutter to the work.

In the triangle (Fig. 13*b*),

$$R^2 = A^2 + (R - d)^2$$

Then $$A^2 = R^2 - (R - d)^2$$

and $$A = \sqrt{R^2 - (R - d)^2}$$

where R = radius of the cutter = $1\frac{1}{4}$ in.
d = depth of cut = $\frac{1}{16}$ in.
A = approach of the cutter to work, in.

$$A = \sqrt{(1\tfrac{1}{4})^2 - (1\tfrac{1}{4} - \tfrac{1}{16})^2}$$
$$= \sqrt{(1\tfrac{1}{4})^2 - (1\tfrac{3}{16})^2}$$
$$= \sqrt{1.56 - 1.41}$$
$$= \sqrt{0.15}$$
$$= 0.390 \text{ in.}$$

The total travel of the table is $L + A$; thus, 6 in. + 0.390 in. = 6.390 in.

The r.p.m. of the cutter can be found from Eq. (4).

$$N = \frac{S \times 12}{D \times 3.1416} = \frac{60 \times 12}{2.5 \times 3.1416} = \frac{720}{7.85}$$
$$= 91.7 \text{ or } 92$$

The distance the table travels in 1 min. is equal to the feed in inches per revolution times the number of revolutions the cutter makes in 1 min.; thus, 0.046 × 92 = 4.232 in. As the table travels 4.232 in. in 1 min., the time required to travel 6.390 is found by dividing 6.390 by 4.232; thus, 6.390 ÷ 4.232 = 1.5 min. The time for milling is 1½ min.

If T = time for the cutter to traverse the work in minutes, L = length of the cut in inches, A = approach of cutter to work, N = r.p.m. of the cutter, and F = feed per revolution in inches, the formula can be written

$$T = \frac{L + A}{F \times N} \qquad (17)$$

Using this formula,

$$T = \frac{6 + 0.390}{0.046 \times 92} = \frac{6.390}{4.232} = 1.5 \text{ min.}$$

In the Eq. (17), $F \times N = F_M$, the feed in inches per minute. The formula can then be written

$$T = \frac{L + A}{F_M} \qquad (18)$$

When using an end mill (Fig. 14) or a face milling cutter, it is usually necessary to allow the work to pass the cutter to get the desired result. Thus, the total distance the work must travel is equal to the length of the work L plus the diameter

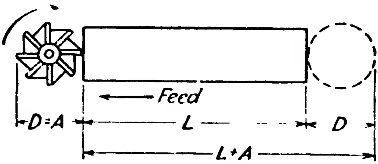


FIG. 14.

ter D of the cutter. The formula for the time for the cutter to traverse the work in minutes is

$$T = \frac{L + A}{F \times N} = \frac{L + D}{F \times N}$$

or

$$T = \frac{L + D}{F_u} \qquad (19)$$

The Time Required for Planing and Shaping. The feed of a planer tool is its sidewise movement for each cutting stroke of the platen. If the tool-carrying head moves along the crossrail $\frac{1}{16}$ in. for each cutting stroke, we say the feed is $\frac{1}{16}$ in. For each cutting stroke, there is necessarily a return stroke. In the following, when the expression number of strokes is used, it means the number of cutting strokes per minute. The time required to plane a piece of work is readily determined if the width of the work, the number of strokes of the platen per minute, and the feed per stroke are known.

A planer makes six strokes per minute. The feed per stroke is 0.060 in., and the width of the work is 12 in. Find the time required for planing.

As the feed per stroke is 0.060 in. and the platen makes six strokes per minute, the feed per minute is 0.060 × 6, or 0.360 in. The number of minutes required for the tool to traverse the work is found by dividing 12 by 0.360; thus, 12 ÷ 0.360 = 33.3 min. The time required for planing is 34 min., nearly.

If T = time required for planing in minutes, W = width of the work in inches, F = feed per stroke in inches, and N = number of strokes per minute, then the formula can be written

$$T = \frac{W}{F \times N} \tag{20}$$

Using Eq. (20),

$$T = \frac{12}{0.060 \times 6} = \frac{12}{0.360} = 33.3 \text{ min.}$$

In the shaper, the work-carrying table is given a sidewise movement for each cutting stroke of the tool. If the table moves along the crossrail 0.010 in. for each cutting stroke of the tool, we say the feed is 0.010 in. In the shaper as in the planer, there is a return stroke for each cutting stroke, and the expression number of strokes means the number of cutting strokes. Thus, if T = time required for shaping in minutes, W = width of the work in inches, F = the feed per stroke in inches, and N = the number of strokes per minute, the formula for the time required for shaping can be written

$$T = \frac{W}{F \times N} \tag{20}$$

Thus, the same formula can be used to find the time for planing and shaping.

Problems

1. A bar of tool steel 2 in. in diameter is to be turned for a length of 8 in. How long will it take for one cut when the cutting speed is 35 ft. per min. and a feed of $\frac{1}{32}$ in. is used?

2. A gray-iron casting 12 in. in diameter is to be turned for a length of 5 in. Find the time to take one cut when the feed is $\frac{1}{16}$ in. and the cutting speed is 60 ft. per min.

3. A $1\frac{1}{2}$-in.-diameter drill is used to drill a hole $2\frac{3}{4}$ in. deep. If the feed is 0.012 in. per revolution and the cutting speed is 40 ft. per min., find the time required to drill the hole.

4. It is required to machine a surface $10\frac{1}{2}$ in. long with a milling cutter $2\frac{1}{2}$ in. in diameter. Find the time it will take if the cutting speed is 40 ft. per min., the depth of cut $\frac{3}{16}$ in., and the feed 0.032 in.

5. Find the time for planing a surface $4\frac{1}{2}$ in. wide when the number of strokes per minute is six and the feed is $\frac{1}{16}$ in. per stroke.

6. Find the time for shaping the surface of a casting 2 in. wide when the number of strokes per minute is 60 and the feed is 0.010 in. per stroke.

7. A $\frac{3}{4}$-in.-diameter high-speed drill is used to drill a hole in a soft gray-iron casting 6 in. deep. How long will it take to drill the hole if the feed is 0.010 in. per revolution? (Hint: Select cutting speed from Table I, page 201.)

8. A surface 2 in. long is machined with a milling cutter 3 in. in diameter. How long will it take if the cutting speed is 35 ft. per min., the depth of the cut is $\frac{1}{16}$ in., and the feed is 0.046 in. per revolution?

9. Find the time for planing a casting 18 in. wide when the length of stroke is 3 ft., the feed is $\frac{5}{32}$ in. per stroke, the cutting speed is 65 ft. per min., and the return-speed ratio is 2 to 1.

10. A drill $1\frac{1}{4}$ in. in diameter makes 110 r.p.m. and has a feed of 0.012 in. per revolution. How long will it take to drill 16 holes in a cylinder head $1\frac{1}{2}$ in. thick if 1 min. is allowed for locating each hole?

11. The surface of a gray-iron casting measuring $2\frac{1}{2}$ by 8 in. is to be finished in the milling machine with a $2\frac{1}{2}$-in.-diameter high-speed steel plain milling cutter. The feed is approximately 5 in. per min. Find the time to machine 50 castings if the depth of the cut is $\frac{1}{16}$ in.

12. How long will it take to mill the above lot of castings if a 3-in.-diameter high-speed steel shell end milling cutter were used, other conditions remaining unchanged?

13. A high-speed steel tool bit is used to turn the face of a 14-in.-diameter gray-iron pulley. The face of the pulley is 5 in. wide. Find the time required to machine 100 pulleys if a total of 2 min. is allowed for loading and unloading each pulley and a feed of $\frac{1}{32}$ in. is used.

14. Four $1\frac{1}{4}$-in.-diameter holes are required to be drilled in a cast-steel frame 8 in. thick. Select the drill, feed, and cutting speed, and find the time required to drill the four holes.

15. In milling the top and bottom of a six-cylinder engine block, a 10-in. diameter stellite-tipped face milling cutter is used having 30 teeth. If the allowable feed per tooth is 0.015 in. per revolution of the cutter, how long will it take to mill a cylinder block 30 in. long? (The top and bottom are milled simultaneously.)

16. Find the cutting speed, the return speed, and the time to take one cut on a gray-iron bedplate casting 20 in. wide, when the ratio of the return speed to the cutting speed is 2.4 to 1, the feed 0.060 in. per stroke, the number of strokes per minute $3\frac{1}{4}$, and the length of stroke 68 in.

CHAPTER IV

SCREW THREADS

Thread Nomenclature.—A thread is formed by cutting a helical groove around the body of the screw. Threads are used as fasteners and for communicating motion. The threads are formed by various methods depending on the use to which they are put. Thus, threads on fasteners

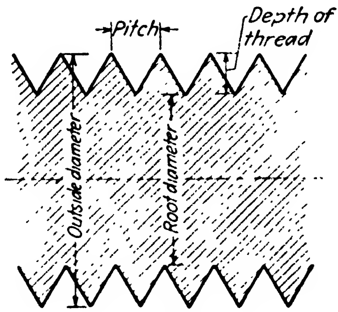


Fig. 15.

are produced by dies and taps, while threads for parts communicating motion are made in the lathe.

The terms "pitch" and "lead" of screw threads are often used interchangeably and with much confusion. The pitch of a thread is the distance from the top of one thread to the top of the next thread, as shown in Fig. 15. In the American National form of thread (Fig. 18), the pitch is defined as the distance from a point on one thread to a corresponding point on the next thread. Whether the screw has a single or a multiple thread, the pitch is always the distance from the top of one thread to the top of the next as stated above.

The lead of a screw thread is the distance the screw will move forward in one complete turn, or it is the distance the nut will advance on the screw for one full revolution of the nut. In a single-threaded screw, the pitch and the lead are equal, because the nut would move forward the distance from one thread to the next, if turned around once. In a double-threaded screw, however, the nut will move forward two threads, or twice the pitch, so that in a double-threaded screw the lead equals twice the pitch. In a triple-threaded screw, the lead is equal to three times the pitch.

In Fig. 16 is shown the lead and pitch for three screws with American Acme Standard threads. The first is a single-threaded screw, the second is double-threaded, and the third is triple-threaded. In a single-threaded screw, the lead is the distance to the next thread from the first one considered. In a double-threaded screw, there are two threads running side by side around the screw, so that the lead is the distance to the second thread from the one first considered. In a triple-threaded screw, the lead is the distance to the third thread; in a quadruple-threaded screw, it is the distance to the fourth thread. The term pitch is often improperly used to denote the number of threads per inch. We hear of screws having six-pitch thread or eight-pitch thread, when six threads per inch and eight

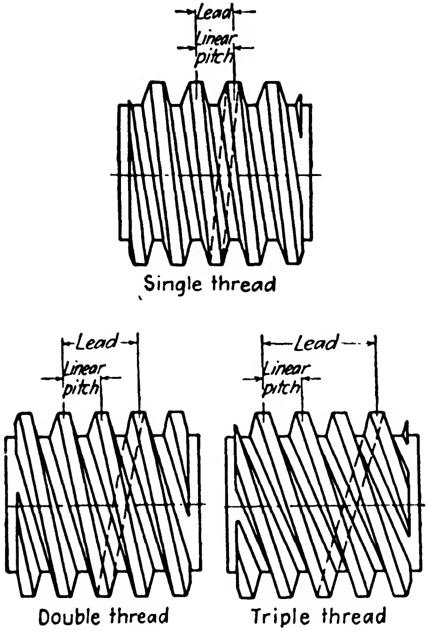

Lead
Linear pitch
Single thread
Lead
Linear pitch
Double thread
Lead
Linear pitch
Triple thread

FIG. 16.

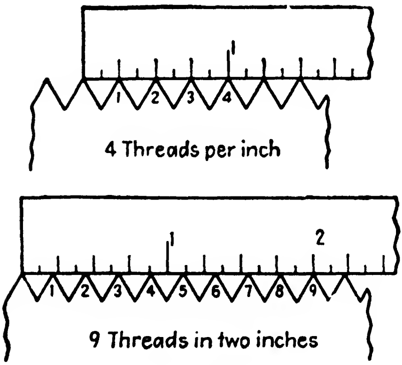

1
1 2 3 4
4 Threads per inch
1
2
1 2 3 4 5 6 7 8 9
9 Threads in two inches

FIG. 17.

threads per inch are meant. The number of threads per inch is the number of threads counted in 1 in. of length. When a steel scale is placed against the screw as shown in Fig. 17, we count four threads per inch. The thread directly under the end of the scale is not counted. If there is not a whole number of threads in 1 in., count the number of threads in two or more inches, until the top of a thread comes opposite an inch mark. The number of threads counted divided by the number of inches will give the number of threads per inch. In Fig. 17, we count nine threads in 2 in.; thus, $9 \div 2 = 4\frac{1}{2}$ threads per inch.

The pitch of a screw equals 1 divided by the number of threads per inch.

Expressed in a formula

$$\text{Pitch} = \frac{1}{\text{number of threads per inch}} \qquad (21)$$

The number of threads per inch equals 1 divided by the pitch of the screw.

Expressed in a formula

$$\text{Number of threads per inch} = \frac{1}{\text{pitch}} \qquad (22)$$

Then, if p = pitch and n = number of threads per inch, the formulas can be written

$$p = \frac{1}{n} \qquad (23)$$

and

$$n = \frac{1}{p} \qquad (24)$$

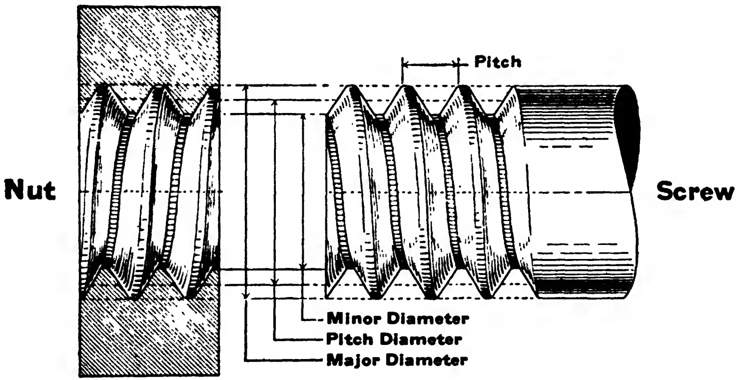
Thus, if the number of threads per inch is 10, the pitch equals $\frac{1}{10}$. If the pitch equals 0.050

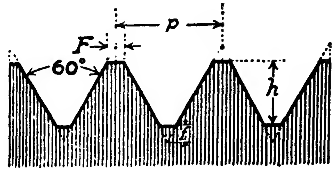
in., the number of threads per inch equals $1 \div 0.050 = 20$.

Much confusion is caused by the indefinite designation of multiple-threaded screws. "Two threads per inch double" is one way to express that a double-threaded screw is required. This means that two separate and distinct threads are cut on the screw. The threads are side by side, and the number of threads per inch, counting the threads with a scale as in Fig. 17, is four. The pitch of the screw is $\frac{1}{4}$ in. The lead for a double-threaded screw is twice the pitch; thus, $2 \times \frac{1}{4} = \frac{1}{2}$ in.

To cut this screw, the lathe would be geared to cut two threads per inch, but the thread will be cut only to the depth required for four threads per inch. "Four threads per inch triple" means that there are 4 times 3, or 12, threads per inch when counted by the scale. The pitch of the screw is $\frac{1}{12}$ in., but as the screw is triple-threaded the lead of the thread is three times the pitch; thus, $3 \times \frac{1}{12} = \frac{1}{4}$ in. The best way to express that a multiple-threaded screw is to be cut is to state the pitch and lead of the screw, that is, $\frac{1}{12}$-in. pitch, $\frac{1}{4}$-in. lead, triple-threaded.

American National Standard Screw Thread.—The American National form of thread, formerly known as the "United States Standard," is one of the most widely used thread forms in the world. There are two standard series in commercial use: the National coarse (NC) and the National fine (NF). The depth of the National form thread equals 0.6495 times pitch. The width of the flat of the thread at the top and bottom equals $\frac{1}{8}$ times pitch. The minor diameter





Formula

p = pitch $= \dfrac{1}{\text{No. of threads per inch.}}$

h = depth = $p \times .649519$

F and f = flat = $p \times \frac{1}{8}$

FIG. 18.

(root diameter) is found by subtracting twice the depth of the thread from the major diameter (outside diameter) of the screw.

Tap Drills.—The tap drills used for drilling holes previous to tapping are usually somewhat larger in diameter than the minor diameter (root diameter) of the thread.

Table IV (page 204) gives the depths of threads for the National coarse form threads. If twice the depth (figure in table opposite number of threads per inch) is subtracted from the major diameter (outside diameter), the minor diameter is obtained.

To find the minor diameter for a $\frac{7}{8}$–9 NC.

In the table, under National coarse-thread series, opposite $\frac{7}{8}$ in. diameter and nine threads per inch, the depth h is 0.07217 in. Twice this depth is $2 \times 0.07217 = 0.14434$ in. The minor diameter equals the major diameter minus 0.14434; thus, $0.875 - 0.14434 = 0.73066$ in. From a table of decimal equivalents, the nearest commercial size drill to the minor diameter is 0.734375 in., or $\frac{47}{64}$ in. The tap-drill diameter for a $\frac{7}{8}$–9 NC tap to produce a thread of nearly full depth is $\frac{47}{64}$ in.

Fewer tapping troubles are encountered and lower power consumption is required if the tap-drill diameter is chosen to produce 75 per cent of the thread depth. The thread percentage times twice the depth equals

$$0.75 \times 2 \times 0.07217 = 0.1083 \text{ in.}$$

The tap-drill diameter for 75 per cent of the thread depth equals $0.875 - 0.1083 = 0.7667$ in.

The nearest commercial size drill to this is 0.765625, or $\frac{49}{64}$ in.

To calculate the size of hole prior to tapping, use the following formula:

$$\text{Tap drill size} = \text{major diameter} - \text{thread percentage} \times 2h \quad (25)$$

Using Eq. (25), we get

$$\begin{aligned} \text{Tap drill size} &= 0.875 - (0.75 \times 2 \times 0.07217) \\ &= 0.875 - (0.75 \times 0.1443) \\ &= 0.875 - 0.1083 \\ &= 0.7667 \text{ in.} \end{aligned}$$

The nearest commercial size drill is $\frac{49}{64}$ in.

Cutting Threads in the Lathe.—Cutting a thread in an engine lathe is accomplished by feeding the lathe carriage a definite distance along the bed for each revolution of the work while the tool is cutting. If the work revolves eight times while the carriage feeds 1 in. along the bed of the lathe, eight threads per inch will be cut on the work.

The number of times the spindle revolves while the carriage feeds 1 in. along the lathe bed depends upon the ratio of the gears on the stud shaft and the lead screw. Since the gears can be changed at will by the operator, they are called "change" gears. By employing different ratios of change gears, therefore, different numbers of threads per inch can be cut.

The change gears may be arranged to form either a simple gear train or a compound gear train and are usually designated as simple gearing or compound gearing. This is shown in Figs. 19

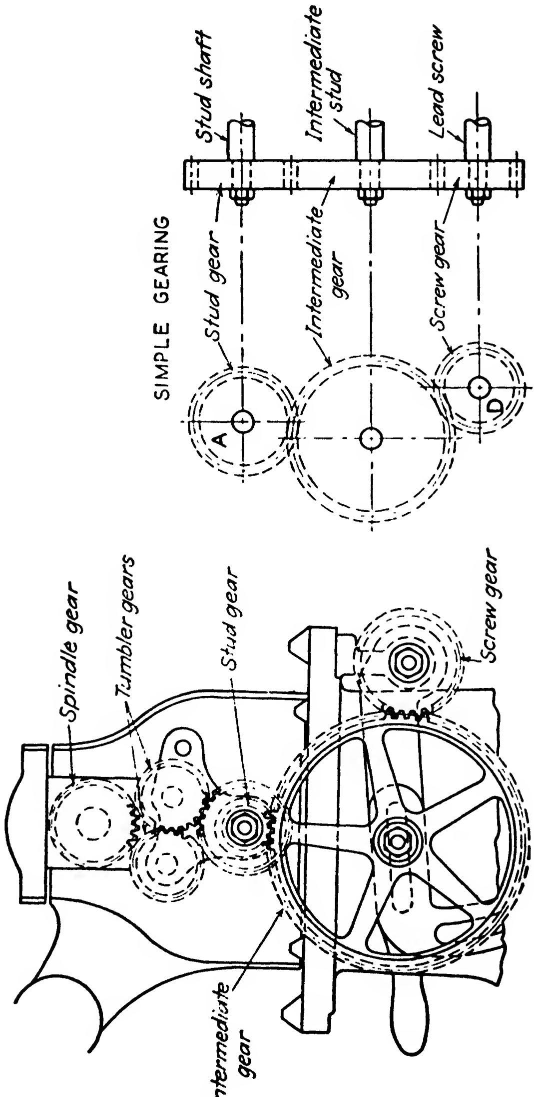

Stud shaft
Intermediate stud
Lead screw
SIMPLE GEARING
Stud gear
Intermediate gear
Screw gear
A
D
Spindle gear
Tumbler gears
Stud gear
Screw gear
Intermediate gear

Fig. 19.

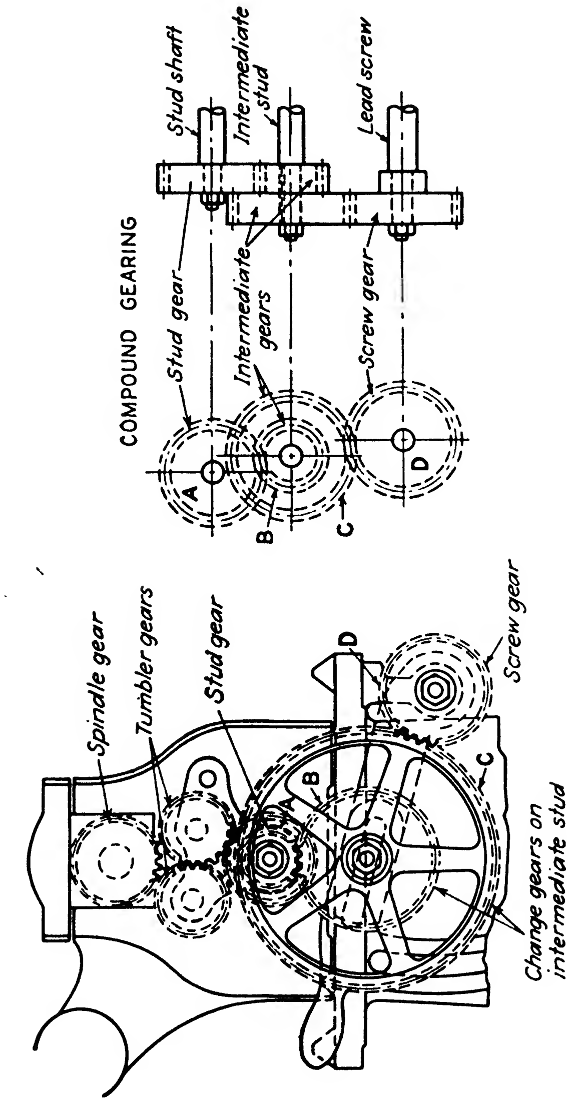
and 20. With simple gearing it is always necessary to use an intermediate gear between the gear on the stud shaft and the gear on the lead screw. The intermediate gear has no influence on the ratio of the gearing and can have any number of teeth.

The Lathe Constant.—To make change-gear calculations for the engine lathe, it is necessary to find the lathe constant.

To find the lathe constant of a lathe, place gears with the same number of teeth on the stud shaft and on the lead screw. Then cut a thread on a piece of work in the lathe. The number of threads per inch cut on the work is called the "lathe constant" of the lathe.

For a given lathe, place gears with 44 teeth on the stud shaft and on the lead screw and any convenient gear on the intermediate stud. Then cut a thread on a piece of work between centers. If the number of threads per inch when counted is found to be nine, the lathe constant of the lathe is said to be 9. The lathe constant for a given lathe is always the same, and once found it should be recorded.

Thread Cutting with Simple Gearing.—When the lathe constant has been found, the number of teeth in the change gears for cutting any number of threads within the capacity of the lathe can be determined. The rate of travel of the lathe carriage compared with the revolution of the spindle is a fraction. The numerator of the fraction is the same as the lathe constant, and its denominator is the same as the number of threads per inch to be cut. Multiply the numerator and denominator of this fraction by the same



COMPOUND GEARING
Stud shaft
Intermediate stud
Lead screw
Stud gear
Intermediate gears
Screw gear
A
B
C
D
Spindle gear
Tumbler gears
Stud gear
Screw gear
Change gears on intermediate stud

Fig. 20.

number (any number) to get a new fraction. In this new fraction, the numerator gives the number of teeth in the gear on the stud shaft and the denominator the number of teeth in the gear on the lead screw. This can be written as a formula as follows:

$$\frac{\text{Lathe constant}}{\text{Threads per inch to be cut}} = \frac{\text{teeth in gear on stud shaft}}{\text{teeth in gear on lead screw}} \qquad (26)$$

The gears supplied with the lathe are varied in size by adding the same number of teeth to the number of teeth in the gear next below in size. The number of teeth added is known as the "gear progression." In most lathes, the gear progression is 4; in a few lathes 5 and 7 are used. When the gear progression is 4, the smallest gear usually has 24 teeth and proceeds to 28, 32, 36, and so on, up to 100 teeth.

Nine threads per inch are to be cut in a lathe that has a lathe constant of 8. The gear progression is 4, and the gears available are 24, 28, . . . , 100.

$$\frac{\text{Lathe constant}}{\text{Threads per inch to be cut}} = \frac{8}{9} = \frac{8 \times 4}{9 \times 4}$$

$$= \frac{32}{36} \frac{\text{stud gear}}{\text{lead-screw gear}}$$

By multiplying both numerator and denominator, we obtain two gears that are available, with 32 and 36 teeth, respectively. Place the 32-tooth gear on the stud shaft and the 36-tooth gear on the lead screw.

If both numerator and denominator were multiplied by 5, we would have

$$\frac{8}{9} = \frac{8 \times 5}{9 \times 5} = \frac{40}{45}$$

These gears are not available in the change gears supplied with this lathe.

It is necessary to cut $11\frac{1}{2}$ threads per inch in a lathe that has a lathe constant of 8. The gear progression is 4, and the gears available are 24, 28, . . . , 100.

$$\frac{8}{11.5} = \frac{8 \times 4}{11.5 \times 4} = \frac{32}{46}$$

These gears are not available.

$$\frac{8}{11.5} = \frac{8 \times 8}{11.5 \times 8} = \frac{64}{92}$$

These gears are available.

$$\frac{8}{11.5} = \frac{8 \times 6}{11.5 \times 6} = \frac{48}{69}$$

These gears are not available on the above lathe, but as $11\frac{1}{2}$ threads per inch is widely used, many lathes are supplied with a 69-tooth gear for this purpose.

Thread Cutting with Compound Gearing.—It is not always possible to cut all numbers of threads per inch by simple gearing, because the change gears of a lathe are limited in number, and compound gearing must be used.

The method for finding the change gears in compound gearing is the same as for simple gearing, except that both the numerator and denominator of the fraction are divided into two factors, one factor in the numerator and one in the denominator making a pair.

Thirty-six threads per inch are to be cut in a lathe with the same lathe constant and gear progression as stated above. Our fraction is $\frac{8}{36}$.

Dividing the numerator and denominator of the fraction into two factors and multiplying the numerator and denominator of each pair by the same number, as shown below, we get

$$\frac{8}{36} = \frac{2 \times 4}{4 \times 9} = \frac{(2 \times 14)(4 \times 8)}{(4 \times 14)(9 \times 8)} = \frac{28 \times 32}{56 \times 72}$$

The four numbers in the last fraction give the numbers of teeth in the change gears to be used. These gears are available and are placed as follows: the gears in the numerator, with 28 and 32 teeth, are the driving gears, and those in the denominator, with 56 and 72 teeth, are driven gears. In Fig. 20, the driving gears are gear *A* on the stud shaft, and gear *C*, the second gear on the intermediate stud shaft, which meshes with the screw gear. Driven gears are gear *B* on the intermediate stud shaft, which meshes with the stud gear, and the screw gear *D*, which meshes with the second gear *C* on the intermediate stud shaft.

The formula for the calculation above can be written as follows:

$$\frac{\text{Lathe constant}}{\text{Threads per inch to be cut}} = \frac{\text{product of teeth in driving gears}}{\text{product of teeth in driven gears}} \quad (27)$$

Find the change gears to cut 40 threads per inch compound gearing in the lathe above.

$$\frac{8}{40} = \frac{2 \times 4}{4 \times 10} = \frac{(2 \times 14)(4 \times 8)}{(4 \times 14)(10 \times 8)} = \frac{28 \times 32}{56 \times 80}$$

The gears with 28 and 32 teeth are the driving gears, and the 56- and 80-tooth gears are the driven gears.

In Fig. 21, the gears with 28 and 56 teeth are mounted on the intermediate stud and revolve together, that is, when one makes a complete

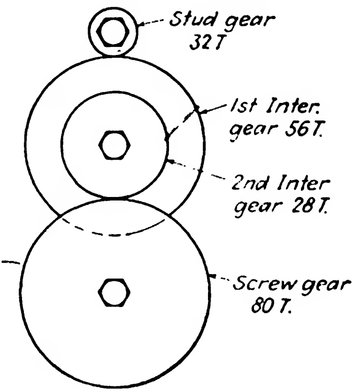


Fig. 21.

turn, the other also makes a complete turn. These two gears, one having twice as many teeth as the other, make what is called the "compound gear." The effect of the compound gear is to change the speed ratio 2 to 1. With the compound gear in the gear train, the lead screw will turn only one-half as fast as it did before and the number of threads cut per inch will be twice as great as when the same gears are used on the stud shaft and lead screw in simple gearing. Thus, to cut 40 threads per inch compound gearing, divide the number of threads per inch to be cut by 2 and use the result in the formula for simple gearing.

$$40 \div 2 = 20$$

$$\frac{8}{20} = \frac{8 \times 4}{20 \times 4} = \frac{32}{80}\,\frac{\text{stud gear}}{\text{screw gear}}.$$

The compound gear can be made up of any two gears, the large gear having twice as many teeth as the small gear; thus the compound gear could be 24 and 48, 28 and 56, 32 and 64, etc.

Fractional Threads.—Occasionally the lead of a thread is given as a fraction of an inch instead of stating the number of threads per inch. It is required to cut a thread having a $\frac{5}{16}$-in. lead. Find the number of threads per inch.

$$\text{Number of threads per inch } \frac{1}{\frac{5}{16}} = 1 \div \tfrac{5}{16}$$

$$= 1 \times \tfrac{16}{5} = \tfrac{16}{5} = 3\tfrac{1}{5}$$

Find the change gears to cut $3\frac{1}{5}$ threads per inch in a lathe with a screw constant of 8. The gear progression is 4, and the gears available are 24, 28, . . . , 100.

Using the formula for compound gearing

$$\frac{8}{3\frac{1}{5}} = \frac{2 \times 4}{1 \times 3\frac{1}{5}} = \frac{(2 \times 24)(4 \times 20)}{(1 \times 24)(3\frac{1}{5} \times 20)} = \frac{48 \times 80}{24 \times 64}$$

Metric Screw Threads.—The metric system is in use in practically all countries except the United States, Great Britain, and the British colonies.

One inch in the English system is equal to 25.4 mm. in the metric system.

Screws made in accordance with the metric system usually have the lead of the thread given in millimeters. A screw thread is said to have a 2-mm. lead, 4-mm. lead, 5.5-mm. lead, etc.

It frequently happens that screws and taps with metric threads are required. Metric threads can be cut on a lathe having an English lead screw, provided that change gears with the necessary number of teeth are used.

Cutting Metric Threads, Simple Gearing.—A screw with a 2-mm. lead is to be cut in a lathe with an English lead screw. The screw constant is 8, the gear progression is 4, and the gears available are 24, 28, . . . , 100. Find the change gears to cut this thread.

Since there are 25.4 mm. in 1 in., the number of threads per inch is found by dividing 25.4 by 2. It is not necessary to carry out the division; simply write it in the form of a fraction 25.4/2. This fraction now gives us the number of threads per inch to be cut. The method for finding the change gears is the same as for cutting threads with English pitches.

$$\frac{8}{\frac{25.4}{2}} = \text{fraction}$$

Performing the indicated division, we get

$$8 \div \frac{25.4}{2} = 8 \times \frac{2}{25.4} = \frac{8 \times 2}{25.4}$$

ratio of change gears

To find the change gears, multiply the numerator and denominator by 5 so as to get whole numbers

$$\frac{8 \times 2 \times 5}{25.4 \times 5} = \frac{16 \times 5}{127} = \frac{80}{127} \quad \frac{\text{stud gear}}{\text{screw gear}}$$

Thus, we must have one gear with 127 teeth to cut a metric thread in a lathe with an English lead screw.

The formula for the calculations above can be written

$$\frac{\text{Lathe constant} \times \text{lead of thread to be cut in millimeters} \times 5}{127} = \frac{\text{teeth in gear on stud shaft}}{\text{teeth in gear on lead screw}} \quad (28)$$

Applying Eq. (28) to cut a thread with a 3-mm. lead on a lathe with a constant of 6, we have

$$\frac{6 \times 3 \times 5}{127} = \frac{90}{127} \quad \frac{\text{stud gear}}{\text{screw gear}}$$

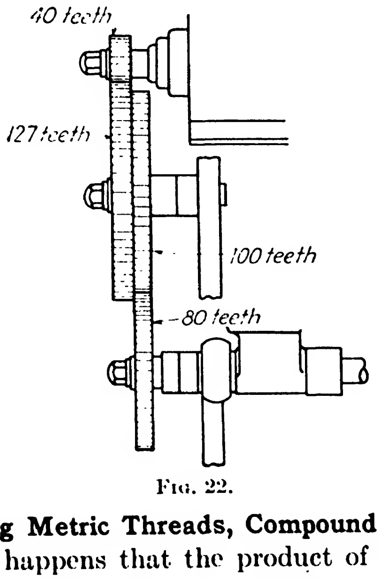


FIG. 22.

Cutting Metric Threads, Compound Gearing. It often happens that the product of the lathe constant times the lead in millimeters times 5 is not a whole number or that the stud gear is larger than any supplied with the lathe, in which case it would be necessary to use compound gearing.

It is required to cut a screw of 3-mm. lead on a lathe with a constant of 8. Thus

$$\frac{8 \times 3 \times 5}{127} = \frac{120}{127} = \frac{60 \times 2}{127 \times 1} =$$

$$\frac{(60 \times 1)(2 \times 48)}{(127 \times 1)(1 \times 48)} = \frac{60 \times 96}{127 \times 48} \quad \frac{\text{driving gears}}{\text{driven gears}}$$

Figure 22 shows a lathe with an English lead screw and gears arranged to cut a screw of 1.25-mm. lead. The lathe constant is 8.

To Cut American National Screw Threads with a Metric Lead Screw.—If it is required to cut a screw with a given number of threads per inch in a lathe with a metric lead screw, it is first necessary to find the metric lathe constant of the lathe. Place gears (simple gearing) with the same number of teeth on the stud shaft and on the lead screw and a gear with any number of teeth on the intermediate stud between them to complete the train. Then cut a thread on a piece in the lathe. The lead of the thread cut, in millimeters, is the metric lathe constant of the lathe. Now the method for finding the change gears, when a screw with a given number of threads per inch is to be cut with a metric lead screw, is just the reverse of the method used for cutting a metric thread with an English lead screw.

Thus, to find the change gears for cutting the American National Standard screw threads with a metric lead screw, place 127 in the numerator and the metric lathe constant times threads per inch to be cut times 5 in the denominator of the fraction; 127 is the number of teeth in the stud gear, and the product of the numbers in the denominator gives the number of teeth in the lead-screw gear.

The formula for finding the change gears can be expressed as follows:

$$\frac{127}{\text{Threads per inch to be cut} \times \text{metric screw constant} \times 5} = \frac{\text{teeth in gear on stud shaft}}{\text{teeth in gear on lead screw}} \quad (29)$$

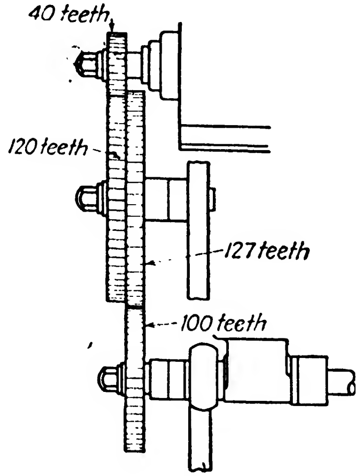


FIG. 23.

To cut five threads per inch in a lathe having a metric lathe constant of 4 mm., the gears are found directly by using Eq. (29).

$$\frac{127}{5 \times 4 \times 5} = \frac{127}{100} \quad \frac{\text{stud gear}}{\text{screw gear}}$$

It is often necessary to use compound gearing in order to obtain gears that are found in the set of change gears supplied with the lathe.

Find the change gears to cut nine threads per inch in a lathe with a metric lathe constant of 4 mm.

$$\frac{127}{9 \times 4 \times 5} = \frac{127}{180} = \frac{127 \times 1}{90 \times 2} =$$

$$\frac{(127 \times 1)(1 \times 40)}{(90 \times 1)(2 \times 40)} = \frac{127 \times 40}{90 \times 80} \quad \frac{\text{driving gears}}{\text{driven gears}}$$

In Fig. 23 is shown the arrangement of the gears (compound gearing) to cut a screw having 12 threads per inch in a lathe with a metric lead screw, the metric lathe constant being 5 mm.

$$\frac{127}{12 \times 5 \times 5} = \frac{127}{300} = \frac{127 \times 1}{100 \times 3}$$

$$= \frac{(127 \times 1)(1 \times 40)}{(100 \times 1)(3 \times 40)} = \frac{127 \times 40}{100 \times 120}$$

Problems

1. Find the tap drill size for a 1–8 NC tapped hole to produce a thread 65 per cent of full depth.

2. Find the tap drill size for a $\frac{3}{4}$-16 NF tapped hole to produce a thread 75 per cent of full depth.

3. In the automotive and aircraft industries it is recommended that parts with threads be made to the National fine thread specifications. Why?

4. The pitch of a double thread is $\frac{1}{8}$ in. What is the lead of the thread?

5. In a certain make of aircraft engine, one end of the crankshaft is $2\frac{1}{8}$ in. in diameter and has 12 threads per inch cut on it. Find the pitch of the thread and the minor diameter.

6. In one type of aircraft engine the starter is secured to the crankcase by six bolts $\frac{3}{8}$ in. in diameter with 24 threads per inch. What is the minor diameter of the thread? What size drill should be used to drill the crankcase for tapping?

7. The cross-feed on a lathe has a single-threaded screw of $\frac{1}{4}$-in. pitch. If the dial on it has 25 divisions, how far will the cross slide move when the dial is moved one division?

8. What is meant by the lathe constant of an engine lathe? By means of a sketch, show the arrangement of the change gears for determining the lathe constant.

9. A lathe has a constant of 6. If the gear progression is 4 and the gears available are 24, 28, . . . , 100, find the change gears for cutting 12 threads per inch.

10. On valves and certain other pipe fittings it is required to cut $11\frac{1}{2}$ threads per inch. If the threads are to be cut on the lathe in Prob. 9, what change gears are required?

11. Find the change gears for cutting 20 threads per inch when the lathe constant is 8. The gear progression and gears available are the same as in Prob. 9.

12. An engine lathe has a lathe constant of 6. Find the change gears for cutting 16 threads per inch. The gear progression is 4, and the change gears available are 24, 28, . . . , 100.

13. Find the change gears for cutting a $\frac{1}{20}$-in. pitch thread on a lathe with a lathe constant of 6 and a gear progression of 4. The change gears available are the same as in Prob. 12.

14. It is required to cut 30 threads per inch on an engine lathe with a constant of 8 and a gear progression of 4. The change gears available are 24, 28, . . . , 80. Find the change gears for compound gearing.

15. Find the change gears to cut 40 threads per inch on the lathe in Prob. 14. Use compound gearing.

CHAPTER V

TAPERS

TAPER CALCULATIONS

Taper.—Taper is the difference in the diameters of a conical piece of work. The taper is usually expressed in terms of taper per inch or taper per foot.

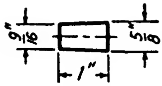


FIG. 24.

Taper per inch is the difference between the two diameters in a tapering piece 1 in. long.

In Fig. 24, the diameter at the small end is $\frac{9}{16}$ in., the diameter at the large end is $\frac{5}{8}$ in., and

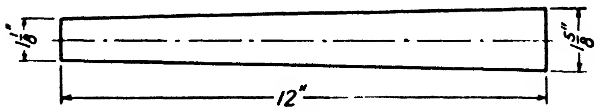


FIG. 25.

the length of the piece is 1 in. The taper, therefore, is $\frac{1}{16}$ in. per in.

Taper per foot is the difference between the two diameters in a tapering piece 1 ft. long.

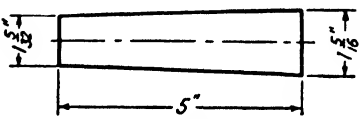


FIG. 26.

In Fig. 25, the diameter at the small end is $1\frac{1}{8}$ in., the diameter at the large end is $1\frac{5}{8}$ in., and the length of the piece is 1 ft. The taper is $\frac{1}{2}$ in. in 1 ft. of length, or $\frac{1}{2}$ in. per ft.

In Fig. 26, the diameter at the small end is $1\frac{5}{32}$ in., the diameter at the large end is $1\frac{5}{16}$ in., and the length is 5 in. The difference in the diameters is $1\frac{5}{16} - 1\frac{5}{32}$, or $\frac{5}{32}$ in., and the taper is $\frac{5}{32}$ in. in 5 in.

If the taper in a given length is known, the taper per inch can be found. Thus if the taper is $\frac{5}{32}$ in. and the length is 5 in., the taper per inch

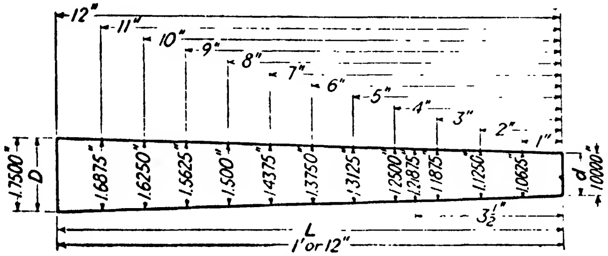


Fig. 27.

equals the taper in 5 in. divided by 5, or in this case $\frac{5}{32} \div 5 = \frac{1}{32}$, which is the taper per inch.

The taper per foot is found by multiplying the taper per inch by 12. The taper per foot for the above is $12 \times \frac{1}{32} = \frac{3}{8}$ in.

The length of the taper is always measured along the center line (axis) of the work or parallel to it, and never along the tapered surface.

Figure 27 shows very clearly what has been stated above. The length of the tapered piece is 1 ft. and the diameters at the small and large ends 1.000 in. and 1.750 in., respectively. The difference in the diameters is 0.750 in. in 1 ft. of length, and the taper per foot is said to be 0.750 in. per ft. The taper per inch is equal to the taper per foot divided by 12; thus, 0.750 ÷ 12 = 0.0625 in. The taper per inch is said to be 0.0625 in. per in.

Figure 27 shows that at a distance 1 in. from the small end the diameter is increased by an amount equal to the taper per inch, or

$$1.000 + 0.0625 = 1.0625 \text{ in.}$$

At 2 in. from the small end, the diameter will be increased by 2 × 0.0625 = 0.125 in. At 2 in. from the small end, the diameter will be

$$1.000 + 0.125 = 1.125 \text{ in.}$$

Thus, it is seen that the diameter increases 0.0625 in. for each inch of length, and in 1 ft., or 12 in., the diameter increases 12 × 0.0625 in., or 0.750 in. Therefore, the amount of taper for any given length is equal to the taper per inch times the length of the taper in inches. From Fig. 27, it is seen that the taper in 3½ in. is 3.5 × 0.0625, or 0.21875 in., and the diameter at 3½ in. from the small end is 1.000 + 0.21875 = 1.21875 in.

Formulas for Taper Calculations.—When all the dimensions are given in inches, the formulas for taper per inch and taper per foot can be written

$$\text{Taper per inch} = \frac{\text{large diameter} - \text{small diameter}}{\text{length of work}} \qquad (30)$$

$$\text{Taper per foot} = \frac{\text{large diameter} - \text{small diameter}}{\text{length of work}} \times 12 \qquad (31)$$

If T_{ft} = taper per foot, T_{in} = taper per inch, L = length of work in inches, D = diameter at large end in inches, and d = diameter at small end in inches, then Eqs. (30) and (31) can be written

$$T_{in} = \frac{D - d}{L} \tag{32}$$

$$T_{ft} = \frac{D - d}{L} \times 12 \tag{33}$$

To Find the Diameter at the Small End of the Taper.—When the length of the taper, the diameter at the large end, the taper per foot are known, and the diameter at the small end is required, the following formula can be used:

$$\text{Diameter small end} = \text{diameter large end} - \left(\frac{\text{taper per foot}}{12} \times \text{length}\right) \tag{34}$$

or

$$d = D - \left(\frac{T_{ft}}{12} \times L\right) \tag{35}$$

To Find the Diameter at the Large End of the Taper.—When the length of the taper, the diameter at the small end, the taper per foot are known, and the diameter at the large end is required, the following formula can be used:

$$\text{Diameter large end} = \text{diameter small end} + \left(\frac{\text{taper per foot}}{12} \times \text{length}\right) \tag{36}$$

or

$$D = d + \left(\frac{T_{ft}}{12} \times L\right) \tag{37}$$

To Find the Distance between Two Diameters on a Tapered Piece.—If the diameters at the small and the large ends of the work are known and the taper per foot is given, the length of the piece can be found from Eq. (38)

$$\text{Length} = \frac{\text{diameter large end} - \text{diameter small end}}{\text{taper per foot} \div 12} \qquad (38)$$

or

$$L = \frac{D - d}{T_{ft} \div 12} \qquad (39)$$

A tapering piece of work measures $1\frac{1}{8}$ in. in diameter at the small end and $1\frac{3}{8}$ in. in diameter at the large end. The taper per foot is 0.75 in. Find the length of the taper.

Substituting the given values in the formula for the length between two diameters,

$$L = \frac{1\frac{3}{8} - 1\frac{1}{8}}{0.75 \div 12} = \frac{0.250}{0.0625} = 4 \text{ in.}$$

In the denominator of this formula, taper per foot divided 12 equals taper per inch. This formula may also be written

$$\text{Length} = \frac{\text{diameter large end} - \text{diameter small end}}{\text{taper per inch}} \qquad (40)$$

or simply,

$$L = \frac{D - d}{T_{in}} \qquad (41)$$

TAPER TURNING IN THE LATHE

Setting the Taper Attachment.—The taper attachment (Fig. 28) is a lathe accessory developed especially for turning tapers. The taper attachment permits a wide range of tapers to

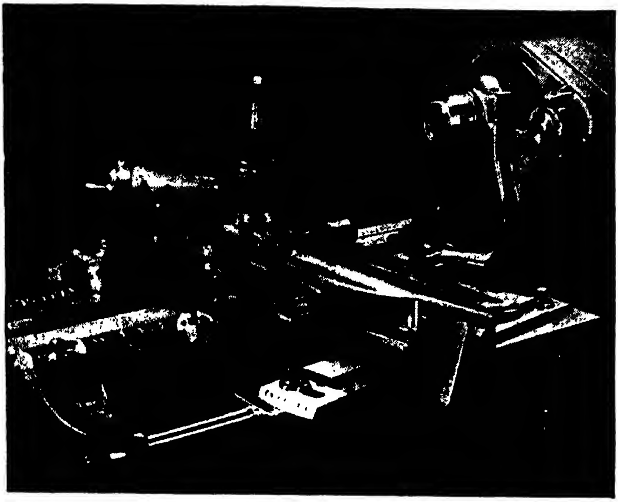


FIG. 28.—(*Courtesy of South Bend Lathe Works, South Bend, Ind.*)

FIG. 29.—(*Courtesy of South Bend Lathe Works, South Bend, Ind.*)

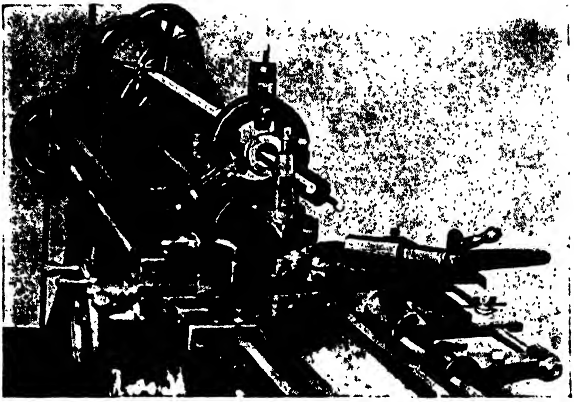


FIG. 30.—(*Courtesy of South Bend Lathe Works, South Bend, Ind.*)

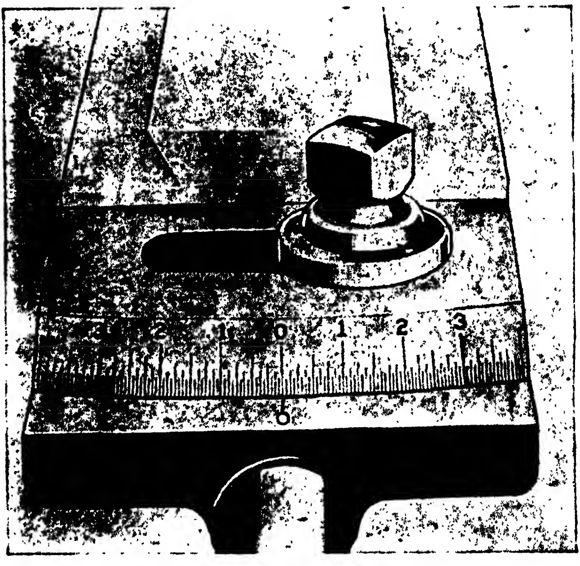


FIG. 31.—(*Courtesy of South Bend Lathe Works, South Bend, Ind.*)

be cut, and its use has largely displaced the method of offsetting the tailstock (setover method) for turning tapers. The lathe centers are always in alignment, and the taper attachment, once set, reproduces the same taper no matter what the length of the work. It can be used to produce either outside or inside tapers, these operations being referred to as taper turning and taper boring (see Figs. 29 and 30). The swivel bar, which controls the taper, is graduated at one end in inches per foot of taper (taper per foot) (see Fig. 31). On the other end the included angle of the taper is shown in degrees.

To set the taper attachment, it is necessary to know the taper per foot so that the swivel bar can be set.

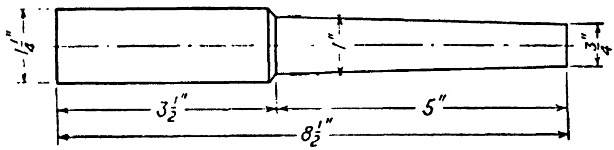


FIG. 32.

To produce the taper in Fig. 32 by means of the taper attachment, first find the taper per foot by using Eq. (33).

$$T_{ft} = \frac{D - d}{L} \times 12 \tag{33}$$

$$= \frac{1 - \frac{3}{4}}{5} \times 12$$

$$= \frac{0.250}{5} \times 12$$

$$= 0.050 \times 12$$

$$= 0.600$$

A table of decimal equivalents shows that 0.600 in. equals $\frac{38}{64}$ in., almost.

To set the taper attachment to cut the above taper, adjust the swivel slide until the zero mark on the stationary base coincides with $\frac{38}{64}$-in. mark on the swivel slide. A trial cut is taken, and if necessary a slight adjustment of the swivel slide is made to produce a taper of the desired accuracy.

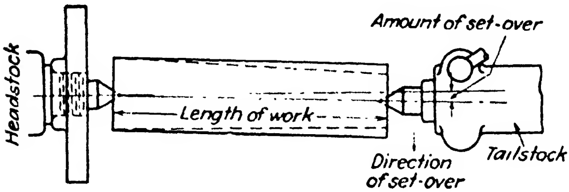


FIG. 33.

Figure 33 shows the tailstock of the lathe set over for turning a taper. The live center and the tail center of a lathe are in perfect alignment when the cutting tool, mounted in the tool post of the lathe carriage, travels in a direction parallel to a line connecting the points of the two centers. If a piece of work is then placed between centers and revolved and a cut taken over it, a cylindrical piece is generated. The cylinder generated will have the same diameter throughout its whole length, or we say it is turned "straight."

If the tailstock center (dead center) is moved out of alignment with the headstock center (live center) any amount A, as shown in Fig. 34, then the center of the work at the dead-center end will be nearer to the line of travel BC of the tool than the center of the work at the live center end and the diameter of the piece, when turned, will be smaller at the dead center than at the live center. Setting over the tailstock is a common method for turning tapered work, especially when the lathe is not equipped with a taper attachment.

The amount of taper depends on the length of the work between centers and the setover of

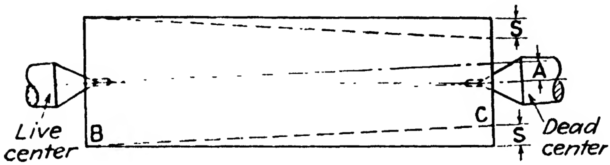


FIG. 34.

the dead center in each case. When the dead center is set over the amount *A* as in Fig. 34, the radius at the small end will be smaller than the radius at the large end by the amount *S*. The distance *S* is equal to *A*, the amount the dead center has been set over, and the taper of the work in the length between centers is two times the amount the dead center is set over, or the setover is one-half of the taper in the length of the work between centers. The length of the taper is always measured along the center line of the work or parallel to it, and never along the tapered surface.

To Find the Setover.—When the work is tapered throughout its length, the setover can be found if the length and the taper foot are known.

The taper in Fig. 35 is $\frac{3}{4}$ in. per foot, the length is 9 in., and the setover is required.

First find how much the work tapers in 9 in. This is found by dividing $\frac{3}{4}$ in. by 12 and multiplying the result by 9.

$$\left(\frac{3}{4} \div 12\right) \times 9 = \frac{9}{16}$$

The taper of 9 in. is $\frac{9}{16}$ in., and the dead center is set over one-half of this amount. The setover is $\frac{9}{32}$ in.

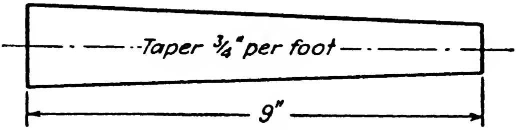


Fig. 35.

Thus, when the taper per foot and the length of the work are known, the formula for finding the setover is

$$\text{Setover} = \frac{1}{2}\left(\frac{\text{taper per foot}}{12} \times \text{length of work}\right) \qquad (42)$$

If S = amount of setover, T_{ft} = taper per foot, and L = length of the work, the formula can be written

$$S = \frac{1}{2}\left(\frac{T_{ft}}{12} \times L\right) \qquad (43)$$

It is not practical to determine the setover of the dead center so accurately that the taper can be turned to the exact dimensions without taking a trial cut. This is because the lathe centers enter in the work to support it. In the calculation, no allowance has been made for this, the assumption being that the distance between the points of the centers and the length of the work are the same. The calculation for the setover gives a close approximation; after a trial cut, the necessary adjustments of the dead center can be made to produce the correct taper.

When the work tapers throughout its length and the diameters at both the large and the small ends are known, the setover can be determined without knowing the taper per foot. All that it is necessary to know is the taper in the length between centers in the lathe.

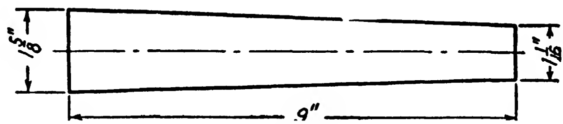


FIG. 36.

In Fig. 36, the diameters at the large and small ends are $1\frac{5}{8}$ and $1\frac{1}{16}$ in. respectively, and the length is 9 in.

The taper is the difference between the diameter at the large end and the small end. Thus, the taper in the entire length (9 in.) is $1\frac{5}{8} - 1\frac{1}{16}$, or $\frac{9}{16}$ in. The setover is one-half the taper. The setover is $\frac{9}{32}$ in., and the formula for this calculation can be written

Setover

$$= \tfrac{1}{2}(\text{large diameter} - \text{small diameter}) \qquad (44)$$

or

$$S = \tfrac{1}{2}(D - d) \qquad (45)$$

If part of the work is turned straight and part of it turned tapered, the taper in the whole length of the work must be found and then the dead center set over one-half of this amount.

In Fig. 37, the diameter at the small end of the taper is $1\frac{1}{16}$ in. The tapered part is 5 in. long, and the diameter at the large end of the taper is $1\frac{3}{8}$ in. The remainder of the length is turned straight for 4 in. to a diameter of $1\frac{5}{8}$ in., the total length of the work being 9 in.

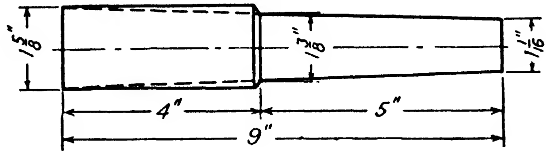


FIG. 37.

First determine what the taper would be in 9 in. when cut to the same taper as is required in the length of 5 in. The setover is one-half of this amount. The taper in 5 in. is $1\frac{3}{8} - 1\frac{1}{16} = \frac{5}{16}$ in. The taper per inch equals $\frac{5}{16} \div 5$, or $\frac{1}{16}$ in. The taper in 9 in. equals $9 \times \frac{1}{16}$, or $\frac{9}{16}$ in. The setover is one-half of this amount; thus, $\frac{9}{16} \div 2 = \frac{9}{32}$ in. If the taper in Fig. 37 is continued beyond the 5-in. length, as shown by the broken line, Fig. 37 would become the same as Fig. 36. The formulas that follow can be used when part of the work is straight and part tapered:

Setover

$$= \frac{1}{2} \times (\text{taper per inch} \times \text{total length}) \quad (46)$$

or

$$S = \frac{1}{2}(T_{in} \times L) \quad (47)$$

Setover

$$= \frac{1}{2}\left(\frac{\text{taper per foot}}{12} \times \text{total length}\right) \quad (48)$$

or

$$S = \frac{1}{2}\left(\frac{T_{ft}}{12} \times L\right) \quad (49)$$

In the setover method it is important to remember that the whole length of the work must be used in calculating the taper. The setover is one-half of this amount.

Problems

1. A tapered piece of work measures $\frac{7}{8}$ in. in diameter at the small end and $1\frac{1}{8}$ in. at the large end. The taper is 0.600 in. per ft. Find the length of the taper.

2. A tapered piece is 9 in. long. The diameter at the small end is $1\frac{1}{2}$ in., and the rate of taper is $\frac{3}{4}$ in. per ft. Find the diameter at the large end.

3. A tapered key is $5\frac{7}{8}$ in. long. The thickness at the small end is $\frac{3}{8}$ in. and at the large end 1 in. What is the taper per foot? Taper per inch?

4. The taper on a No. 4 Morse taper reamer is 0.623 in. per ft. What is the length of the tapered part, if the diameter at the small end is 1.020 in. and the diameter at the large end is 1.293 in.?

5. A tapered ring gauge for an aircraft-engine part is $1\frac{3}{8}$ in. long. The small diameter is 0.760 in., and the large diameter is 1.250 in. What is the rate of taper per foot of the gauge?

6. A piston rod is $38\frac{1}{2}$ in. long and tapered at one end for a length of 6 in. If the diameter at the large end of the taper is $3\frac{3}{4}$ in. and the diameter at the small end is $3\frac{1}{2}$ in., find the setting for the taper attachment.

7. A piece of work is $8\frac{1}{2}$ in. long and tapered at one end for a length of 5 in. If the diameter at the small end of the taper is $1\frac{3}{4}$ in. and the diameter at the large end is $2\frac{1}{4}$ in., find the setting for the taper attachment.

8. Find the setover for turning the taper in Prob. 7.

9. A piece of work 24 in. long is tapered at one end. The diameters at the large and the small ends of the taper are 2 and $1\frac{1}{2}$ in., respectively. If the taper per foot is $1\frac{1}{2}$ in., what setover is required?

10. Find the setover for turning the taper in Prob. 2.

11. A piece of work $9\frac{1}{2}$ in. long is tapered for 6 in. from one end. Find the setover for the tailstock if the diameters at the large and the small ends of the taper are $1\frac{5}{8}$ and $1\frac{1}{4}$ in., respectively.

CHAPTER VI

THE SPIRAL HEAD

The spiral head is an attachment for the milling machine, and its use enables a large variety of operations to be performed that would other-

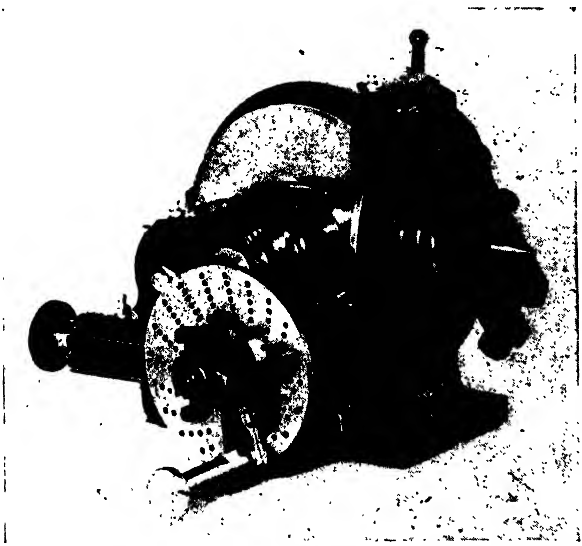


FIG. 38.—(*From Brown & Sharpe Manufacturing Company, "Practical Treatise on Milling.*")

wise be impossible. The spiral head (**Fig. 38**) is used principally for indexing and spiral cutting.

Indexing.—When cutting the teeth in a gear, each tooth must be properly spaced in relation to the other teeth in the gear. This process of spacing the teeth in a gear or of dividing the periphery of a piece of work into a number of parts is called "indexing." The spiral head is sometimes called the "index head" or "dividing head." An elevation and a section of the spiral head is shown in Fig. 39. A worm gear (worm wheel) *A* is mounted on the index head spindle *B*. The worm *C* which is an integral part of shaft *D* meshes with the worm gear *A*. Secured to the opposite end of the shaft *D* is the crank *E* in the outer end of which is fitted a pin *F*. The end of the pin has a cylindrical projection that fits into the holes of the index plate *G*. When the crank *E* is moved, the worm *C* is rotated and imparts motion to the worm wheel *A*, and the work held in the spiral head is rotated. By moving the crank with its index pin a certain number of holes in one of the index circles, a certain angular movement is imparted to the spindle and the work. The calculation for the indexing movement consists of finding how much the crank *E* is required to be turned to produce the required movement for indexing the work.

Indexing Calculations.—Most spiral heads are constructed with a single-threaded worm engaging a worm gear having 40 teeth. When the index crank *E* is turned one full revolution, the worm *C* is also revolved one full turn, and this moves the worm gear one tooth, or $\frac{1}{40}$ of its circumference. Therefore, in order to turn the worm gear and the spindle on which it is mounted one whole revolution, it is necessary to turn the index crank 40 revolutions; thus, the ratio

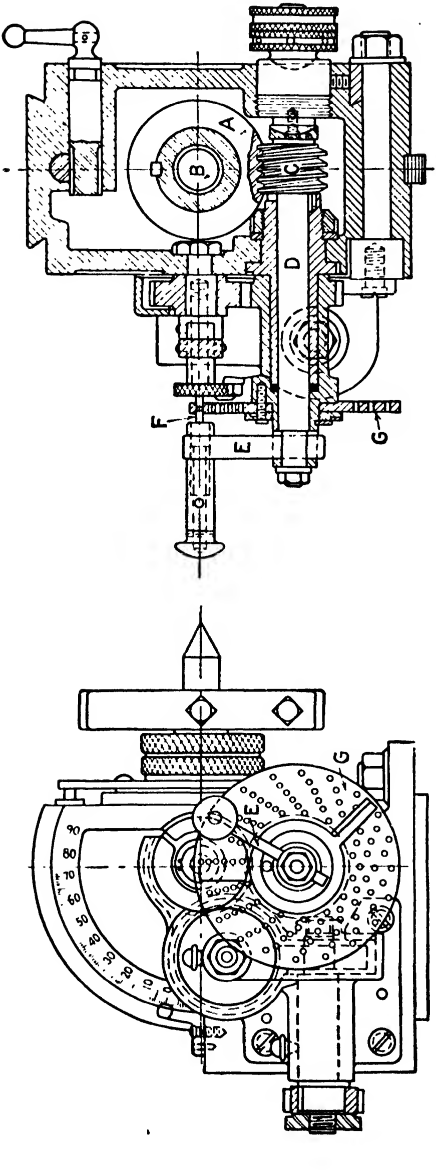


FIG. 39.

between the index crank and the spindle is 40 to 1. If it is required to revolve the spindle (work) one-fourth revolution, the index crank is turned $\frac{1}{4} \times 40$, or 10 revolutions. If it is required to revolve the spindle $\frac{1}{2}$ revolution, the index crank is turned $\frac{1}{2} \times 40$, or 20 revolutions.

The circular piece of work shown in Fig. 40 has eight equally spaced cuts about its periphery. To revolve the spindle (work) once requires 40 revolutions of the index crank. To revolve the spindle $\frac{1}{8}$ revolution would require as many turns of the index crank as $40 \div 8$, or 5. After the first cut is made, the index crank is turned 5 revolutions, thus causing the spindle (work) to be revolved $\frac{1}{8}$ revolution. The second cut is then made and the process repeated until all the cuts have been made. In the same manner, a piece of work with 10 cuts would require $40 \div 10$, or 4, turns of the index crank to revolve the work $\frac{1}{10}$ revolution; if 60 cuts were required, the movement of the index crank would be $\frac{40}{60}$, or $\frac{2}{3}$ turn, to revolve the work $\frac{1}{60}$ revolution.

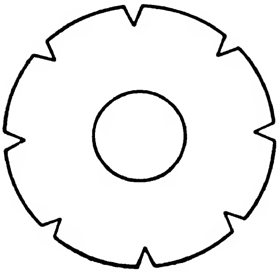


FIG. 40.

Thus, it is seen that 40 is the spiral-head constant, and 40 divided by the number of divisions

Constant		Divisions Required		Turns of Index Crank for Each Division
40	÷	8	=	5
40	÷	10	=	4
40	÷	40	=	1
40	÷	60	=	$\frac{2}{3}$
40	÷	80	=	$\frac{1}{2}$

required equals the revolutions of the index crank to move the spindle (work) the required amount.

The Index Plate and Sector.—In indexing operations, it is often required to make only a part of a turn of the index crank to get the required movement of the spindle.

It is necessary to cut a gear with 45 teeth. Find the index circle and the setting of the sector.

$$40 \div 45 = \tfrac{40}{45}$$

The division indicated by the fraction $\frac{40}{45}$ means that $\frac{40}{45}$ of a whole turn of the index crank is required to revolve the gear $\frac{1}{45}$ revolution. This fractional turn of the index crank is accomplished by making use of the index plates supplied with the spiral head.

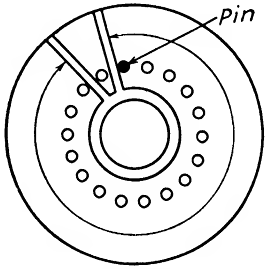


FIG. 41.

Most spiral heads are supplied with three index plates, each having six index circles. The plates and circles regularly supplied with the Brown & Sharpe Manufacturing Company's spiral head are given below.

Plate	Circles
1	15, 16, 17, 18, 19, 20
2	21, 23, 27, 29, 31, 33
3	37, 39, 41, 43, 47, 49

From the circles available, 45 is missing. However it is possible to change the fraction $\frac{40}{45}$ to a new fraction whose denominator is the same as one of the index circles available. Thus

$$\frac{40 \div 5}{45 \div 5} = \frac{8}{9} \quad \text{and} \quad \frac{8 \times 2}{9 \times 2} = \frac{16}{18}$$

Referring to the table of index circles above, circle 18 is available. Plate 1 is mounted on the index head and the index crank moved 16 holes in the 18-hole circle. Figure 41 shows the arms of the sector set for 16 holes in the 18-hole circle. In setting the sector, never count the hole in which the pin of the index crank is placed. For 16 holes, there should actually be 17 holes between the arms of the sector, but one is filled by the pin.

Constant		Required divisions		Crank turns	Whole turns	Number of holes	Circle
40	÷	45	=	$\frac{40}{45}$	0	16	18
40	÷	6	=	$6\frac{4}{6}$	6	12	18
40	÷	65	=	$\frac{40}{65}$	0	24	39
40	÷	5	=	8	8	0	Any
40	÷	9	=	$4\frac{4}{9}$	4	12	27
40	÷	85	=	$\frac{8}{17}$	0	8	17
40	÷	8	=	5	5	0	Any

Indexing Calculations for Angles.—When indexing for an angle, the work is not usually required to make a complete revolution, as is done when a definite number of divisions are required. Instead, it is required to find the movement of the index crank necessary to produce the required movement of the work in degrees before another cut is taken. Indexing for angles is required whenever the angle given is not a simple fraction of the whole circle (as, for example, 36 deg., which is $\frac{1}{10}$ revolution; or 45 deg., which is $\frac{1}{8}$ revolution; or 72 deg., which is $\frac{1}{5}$ revolution). The number of turns of the index crank in these cases is determined as explained previously, but if the angle required is 35½ deg. (Fig. 42), the calculations for the indexing movement are carried out as follows:

There are 360 deg. in one circle. When the spindle (work) makes one revolution, the spindle

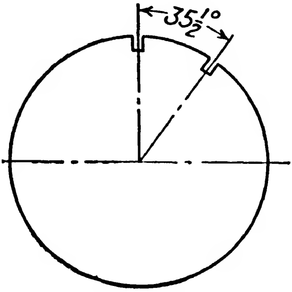


FIG. 42.

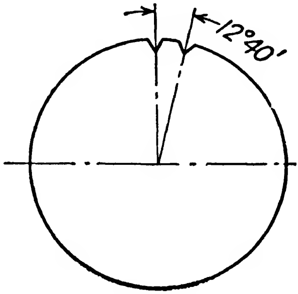


FIG. 43.

is revolved through 360 deg.; thus, if it requires 40 turns of the crank to produce one revolution of the spindle (360 deg.), one turn of the index crank must move the spindle through an angle equal to 360 deg. divided by 40, or 9 deg. Of the index circles available, 18 and 27 are divisible by 9; thus, $18 \div 9 = 2$ and $27 \div 9 = 3$ give the number of holes to move the index crank in the respective circles to move the spindle 1 deg.

If the index-crank movement is two holes in the 18-hole circle to index the spindle 1 deg., then one hole in the 18-hole circle indexes the spindle ½ deg. Likewise three holes in the 27-hole circle indexes the spindle 1 deg.; one hole, ⅓ deg.; and two holes, ⅔ deg.; thus, if the angle contains a half degree, the plate with the 18-hole circle is used. To index 35½ deg., proceed as follows:

$$
\begin{array}{rl}
35\tfrac{1}{2} \text{ deg.} &= \text{required angle} \\
-27 \text{ deg.} &= 3 \text{ full turns of crank} \\
\hline
8\tfrac{1}{2} \text{ deg.} &= \text{fractional turn of crank}
\end{array}
$$

$$8\tfrac{1}{2} \text{ deg.} = 8 \text{ deg.} + \tfrac{1}{2} \text{ deg.} = 16 \text{ holes} + 1 \text{ hole}$$

$$35\tfrac{1}{2} \text{ deg.} = 3 \text{ turns} + 17 \text{ holes in the 18-hole circle}$$

The angle required in Fig. 43 is 12°40′. As 40 minutes is $\frac{2}{3}$ deg., the plate with the 27-hole circle is used.

$$
\begin{array}{rl}
12\tfrac{2}{3} \text{ deg.} &= \text{required angle} \\
-9 \text{ deg.} &= 1 \text{ full turn of crank} \\
\hline
3\tfrac{2}{3} \text{ deg.} &= \text{fractional turn of crank}
\end{array}
$$

$$3\tfrac{2}{3} \text{ deg.} = 3 \text{ deg.} + \tfrac{2}{3} \text{ deg.} = 9 \text{ holes} + 2 \text{ holes}$$

$$12\tfrac{2}{3} \text{ deg.} = 1 \text{ turn} + 11 \text{ holes in the 27-hole circle}$$

This method, which makes use of the 18- and 27-hole circles, gives the exact angle only for whole degrees and $\frac{1}{3}$, $\frac{1}{2}$, and $\frac{2}{3}$ deg. (20, 30, and 40 minutes).

The indexing movement for all other angles containing minutes cannot be found to give the exact angle. But results that give the angle approximately are sometimes of value and can be readily determined as follows.

It is necessary to index for an angle of 55 minutes.

One whole turn of the index crank equals 9 deg., or 540 minutes ($9 \times 60 = 540$). The index movement for 55 minutes is $540 \div 55 = 10$, nearly, and a movement of the index crank of one hole in a 10-hole circle gives the angle approximately. In this case there is no index circle with 10 holes, but there is an index circle with 20 holes available. This circle is used, and the index crank is moved two holes in that circle instead of one.

The indexing movement for 3°25′ is required. First change the angle to minutes.

$$\begin{array}{rl} 60 & = \text{minutes in 1 deg.} \\ \times 3 & = \text{degrees given} \\ \hline 180 & = \text{minutes in 3 deg.} \\ +25 & = \text{minutes given} \\ \hline 205 & = \text{total minutes in } 3°25' \end{array}$$

Dividing 540 by 205 gives the quotient 2.634. Now multiply this quotient by a number (any number) to obtain a product that equals the number of holes in any one of the index circles available.

$$\begin{array}{rl} 2.634 & \\ \times 8 & \\ \hline 21.072 & \text{or a 21-hole circle} \\ 2.634 & \\ \times 11 & \\ \hline 28.974 & \text{or a 29-hole circle} \\ 2.634 & \\ \times 14 & \\ \hline 36.876 & \text{or a 37-hole circle} \end{array}$$

In the above multiplications the product is the circle to use and the multiplier is the number of holes to move in this circle.

To find the circle that gives the approximate angle nearest to the required angle, it is simply necessary to test the values as follows:

$$\tfrac{8}{21} \times 540 = 205.7 \text{ minutes}$$
$$\tfrac{11}{29} \times 540 = 204.8 \text{ minutes}$$
$$\tfrac{14}{37} \times 540 = 204.6 \text{ minutes}$$

An indexing movement of 11 holes in the 29-hole circle gives an angle of 204.8 minutes, 0.2-minute error.

Compound Indexing.—It is often required to obtain divisions beyond the scope of simple indexing, and to do this a method known as compound indexing is used.

In this method, the crankpin is moved a definite number of holes in the regular way, the stop pin holding the index plate is then disengaged, and the index plate with crank is turned either in the same direction or in the opposite direction a definite number of holes. The resultant movement of the crank is either the sum or difference of the two separate simple indexing operations.

It is required to find the movement of the index crank of a Brown and Sharpe spiral head for 96 divisions.

Proceed as follows:

1. From the circles available on the plates provided with the spiral head, select two circles on the same plate. The circles selected are the 18- and 20-hole circles.

2. Find two factors of 96, the required number of divisions.

Factors of 96 are 8 × 12.

3. Factor the difference between the number of holes in the two circles selected.

The difference is 20 − 18 = 2.

The factors of 2 are 2 × 1.

4. Factor the constant 40, the number of turns of the index crank required to make one revolution of the spindle.

The factors of 40 are 2 × 2 × 2 × 5.

5. Find the factors of the two circles selected.

The factors of the larger circle are $2 \times 2 \times 5$.

The factors of the smaller circle are $2 \times 3 \times 3$.

6. Make a fraction in which the numerator is the indicated product of the factors of the number of divisions required and the difference between the number of holes in the two circles selected, and the denominator is the indicated product of the factors of 40 and the factors of the larger circle and the factors of the smaller circle.

Thus

$$\frac{(2 \times 2 \times 2)(2 \times 2 \times 3)(2 \times 1)}{(2 \times 2 \times 2 \times 5)(2 \times 2 \times 5)(2 \times 3 \times 3)}$$

7. Now cancel the factors above the line into those below the line.

$$\frac{(\not{2} \times \not{2} \times \not{2})(\not{2} \times \not{2} \times \not{3})(\not{2} \times \not{1})}{(\not{2} \times \not{2} \times \not{2} \times 5)(\not{2} \times \not{2} \times 5)(\not{2} \times \not{3} \times \underset{3}{\not{3}})}$$

Check: All factors in the numerator must cancel into the factors in the denominator if the circles selected can be used.

8. The product of the uncanceled factors in the denominator is taken as the numerator of each of two fractions, their denominators being the circles selected.

The fractions are $\frac{75}{18}$ and $\frac{75}{20}$.

If the index crank is moved clockwise 75 holes in the 18-hole circle and the stop pin is then disengaged and the plate and the index crank moved counterclockwise 75 holes in the 20-hole circle, the difference between the two movements will equal the movement necessary to index 96 divisions.

Check:

$$\frac{75}{18} - \frac{75}{20} = \frac{1,500}{18 \times 20} - \frac{1,350}{18 \times 20} = \frac{150}{360} = \frac{5}{12}$$

The spindle movement is $\frac{1}{40}$ of the crank movement.

$$\frac{1}{40} \times \frac{5}{12} = \frac{5}{480} = \frac{1}{96}$$

In the above indexing movement there is always the possibility of some error due to movement of the crank in opposite directions. If any whole number is subtracted algebraically from each of the two fractions, the result is not affected. Subtracting 4 from each fraction, the result is

$$\tfrac{75}{18} = 4\tfrac{3}{18}; \qquad \tfrac{75}{20} = 3\tfrac{15}{20}$$

$$\begin{array}{rcl} 4\tfrac{3}{18} & - & 3\tfrac{15}{20} \\ -4 & & +4 \\ \hline \tfrac{3}{18} & + & \tfrac{5}{20} \end{array}$$

Check:

$$\frac{3}{18} + \frac{5}{20} = \frac{60}{18 \times 20} + \frac{90}{18 \times 20} = \frac{150}{360} = \frac{5}{12}$$

and

$$\tfrac{1}{40} \times \tfrac{5}{12} = \tfrac{5}{480} = \tfrac{1}{96}$$

Subtracting 4 from each fraction results in a smaller number of holes required for each movement, and the direction of the moves are clockwise.

Differential Indexing.—Only a limited number of divisions can be obtained by using the three plates regularly supplied with the spiral head for plain indexing. Many divisions not obtainable by plain indexing can be indexed by the differential indexing method. Differential indexing is similar to compound indexing in that both the crank and the plate are revolved, but in differential indexing the movement of the crank

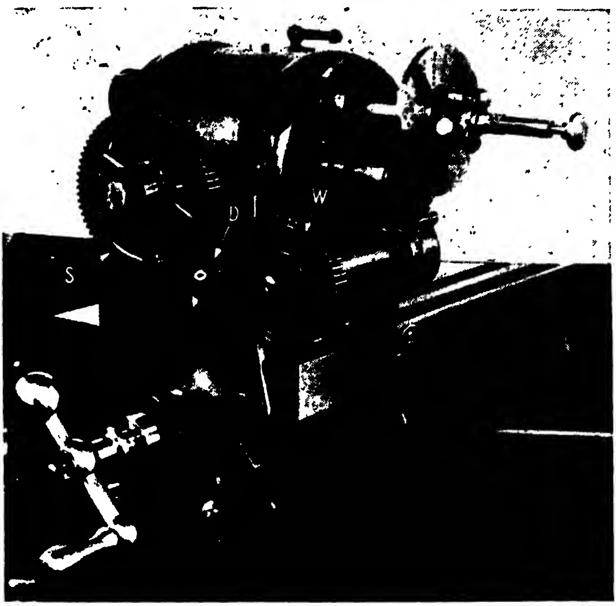


FIG. 44. (*From Brown & Sharpe Manufacturing Company, "Practical Treatise on Milling."*)

and plate occur at the same time, resulting in less chance for error than in compound indexing. The movement of the crank and plate is produced by a train of gears interposed between the spindle of the spiral head and the worm shaft that imparts motion to the plate. By this arrangement, the index crank is moved in the same circle of holes and the operation is like plain indexing.

If gears are placed on the spindle *S* and the worm shaft *W* of the spiral head (Fig. 44) and one intermediate gear is used to complete the train, the plate will rotate in the same direction (clockwise) as the crank. With two intermediate gears (simple gearing), the plate will rotate in a direction opposite (counterclockwise) to the crank.

If the index plate is stationary, the index crank will pass the point *P* forty times (Fig. 45) for one revolution of the spindle. With gears having the same numbers of teeth placed on both the spindle *S* and the worm *W* (Fig. 44) and one intermediate gear, the plate will rotate in the same direction as the crank. The crank will pass the point *P* only thirty-nine times for one turn of the spindle, because the plate, in making one complete turn, makes up for the fortieth turn or one division is subtracted from the regular indexing. Likewise, with two intermediate gears used to complete the train, the plate will rotate in a direction opposite to the crank, and the crank will pass the point *P* forty-one times for one turn of the spindle, in order to make up for the one complete turn of the plate in the opposite direction, and one division is added to the regular indexing. Thus the total movement of the index crank at every indexing is equal to its movement relative to the plate plus the movement of the plate when the plate rotates in the same direction as the crank and minus the move-

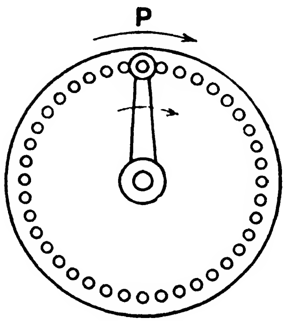


FIG. 45.

ment of the plate when the plate rotates in the direction opposite to the crank. The desired movement of the plate for each indexing can be obtained by the use of proper change gears.

The change gears regularly supplied with the Brown and Sharpe spiral head are 24 teeth (two gears), 28, 32, 40, 44, 48, 56, 64, 72, 86, and 100 teeth.

Change Gears for Differential Indexing.—It is required to cut a gear with 119 teeth. To obtain the change gears (simple gearing) for the spindle S and the worm W, proceed as follows.

1. Select the index circle. As the index plates regularly supplied with the spiral head are to be used, select a number of divisions that can be obtained by plain indexing and find the required movement of the index crank. The number of divisions selected can be either larger or smaller than the number to be indexed.

Thus,

119 = number of divisions required.

120 = number of divisions selected to find index circle.

$$40 \div 120 = \frac{40}{120} = \frac{1}{3} \quad \text{and} \quad \frac{1 \times 6}{3 \times 6} = \frac{6}{18}$$

Use plate 1, and set the sector arms for an index crank movement of six holes in the 18-hole circle.

2. Find the change gears to give the required movement of the plate.

The required movement of the spindle is $\frac{1}{119}$ revolution for each tooth cut. The required movement of the index crank to rotate the spindle $\frac{1}{119}$ revolution is $40 \times \frac{1}{119}$, or $\frac{40}{119}$ revolution.

The movement of the plate is the difference between the movement for indexing the number of divisions required (119) and the number of divisions selected (120).

$$\frac{40}{119} - \frac{40}{120} = \frac{40}{119} - \frac{1}{3}$$
$$= \frac{120}{3 \times 119} - \frac{119}{3 \times 119} = \frac{1}{3 \times 119}$$

The movement of the plate is $1/(3 \times 119)$ revolution while the spindle makes $\frac{1}{119}$ revolution.

As the spindle imparts motion to the plate, the change gears necessary to revolve the plate $1/(3 \times 119)$ revolution while the spindle revolves $\frac{1}{119}$ revolution can be found.

Multiply both fractions by the same quantity to change them to whole numbers.

$$\frac{1}{3 \times 119} \times (3 \times 119) = 1 \text{ revolution of plate}$$
$$\frac{1}{119} \times (3 \times 119) = 3 \text{ revolutions of spindle}$$

The number of teeth in the gears is inversely proportional to the number of revolutions the gears make; thus the teeth in the gears are found by inverting the number of revolutions they make.

$$\frac{\text{1 revolution of plate}}{\text{3 revolutions of spindle}} = \frac{\text{3 teeth in worm gear (plate gear)}}{\text{1 tooth in spindle gear}}$$

and

$$\frac{3}{1} = \frac{3 \times 24}{1 \times 24} = \frac{72}{24} \quad \frac{\text{worm gear}}{\text{spindle gear}}$$

Place the 24-tooth gear on the spindle *S* and the 72-tooth gear on the worm *W* and use one intermediate gear to complete the train, then the spindle will be indexed $\frac{1}{119}$ revolution when the index crank is moved six holes in the 18-circle.

The same results can be obtained by the formula:

S = number of teeth in spindle gear.
W = number of teeth in the worm gear.
G_1 = first intermediate gear on the stud.
G_2 = second intermediate gear on the stud.
N = number of divisions required.
N' = number of divisions selected that can be obtained by plain indexing.

$$Q = \frac{40}{N'}$$

The formula

$$\frac{S}{W} = 40 - (Q \times N) \qquad (50)$$

requires one intermediate gear and is used when N is less than N'.

The formula

$$\frac{S}{W} = (Q \times N) - 40 \qquad (51)$$

requires two intermediate gears and is used when N is greater than N'.

To find the change gears for 119 divisions by formula, we have

$$\frac{S}{W} = 40 - (Q \times N)$$

$$Q = \frac{40}{N'} = \frac{40}{120} = \frac{1}{3}$$

$$\frac{S}{W} = 40 - \left(\frac{1}{3} \times 119\right)$$

$$\frac{S}{W} = \left(\frac{120}{3} - \frac{119}{3}\right) = \frac{1}{3}$$

Multiplying both the numerator and denominator by 24 to get gears with the numbers of teeth that are available, we get

$$\frac{S}{W} = \frac{1 \times 24}{3 \times 24} = \frac{24}{72} \quad \frac{\text{spindle gear}}{\text{worm gear}}$$

When the required movement of the plate relative to the spindle cannot be obtained by simple gearing, compound gearing is used.

Formulas for compound gearing are given below:

When N is less than N' and requires no intermediate gear:

$$\frac{S \times G_1}{W \times G_2} = 40 - (Q \times N) \qquad (52)$$

When N is greater than N' and requires one intermediate gear:

$$\frac{S \times G_1}{W \times G_2} = (Q \times N) - 40 \qquad (53)$$

The number of divisions required is 241.

$N = 241, \qquad N' = 240 \qquad \text{and} \qquad Q = \frac{40}{240}$

$$\frac{S \times G_1}{W \times G_2} = (Q \times N) - 40$$

$$\frac{S \times G_1}{W \times G_2} = \left(\frac{40}{240} \times 241\right) - 40$$

$$\frac{S \times G_1}{W \times G_2} = \left(\frac{1}{6} \times 241\right) - 40$$

$$\frac{S \times G_1}{W \times G_2} = \frac{241}{6} - 40$$

$$\frac{S \times G_1}{W \times G_2} = \frac{241}{6} - \frac{240}{6} = \frac{1}{6}$$

$$\frac{S \times G_1}{W \times G_2} = \frac{\frac{1}{2} \times 2}{1 \times 6} = \frac{(\frac{1}{2} \times 64)\ (2 \times 12)}{(1 \times 64)\ (6 \times 12)} = \frac{32 \times 24}{64 \times 72}$$

$$\frac{S \times G_1}{W \times G_2} = \frac{32 \times 24}{64 \times 72}$$

$$\frac{\text{product of teeth in driving gears}}{\text{product of teeth in driven gears}}$$

$S = 32$ teeth, $W = 64$ teeth, $G_1 = 24$ teeth, $G_2 = 72$ teeth

The four gears above properly arranged give the plate the required movement to index 241 divisions. If 239 divisions were required instead

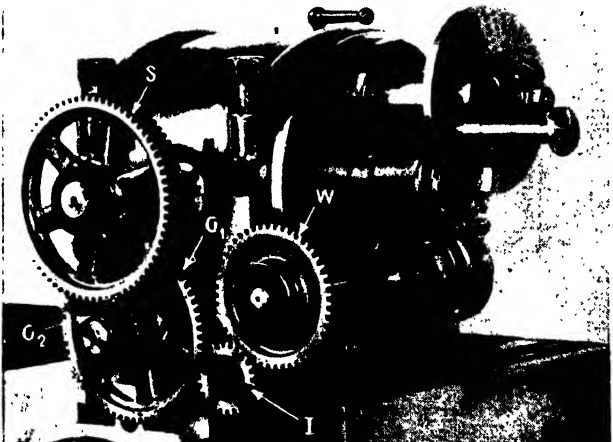


FIG. 46.—(*From Brown & Sharpe Manufacturing Company, "Practical Treatise on Milling."*)

of 241, the same gears could be used without the intermediate gear I interposed between the spindle and the plate as shown in Fig. 46.

Spiral Milling.—Cutting a spiral in the milling machine (Fig. 47) is similar to cutting a thread in the lathe. Motion is transmitted from the

Fig. 47.—Cutting spiral teeth in milling cutter. (*From Brown & Sharpe Manufacturing Company, "Practical Treatise on Milling."*)

feed screw to the spindle of the spiral head by means of a set of change gears.

In Fig. 48 is shown an end view of a spiral head mounted on top of the milling-machine table. The gear *A* is placed on the end of the feed screw and is commonly called the "feed-screw gear" or "gear on screw." Gear *A* imparts motion through the intermediate gear *B* to the gear *C* which is placed on the stud *D* of the spiral head; from the stud, in turn, motion is imparted to the worm of the spiral head and from the worm to the worm gear mounted on the spiral-head spindle. Thus, when connected by gearing in this manner, the spiral-head spindle may be rotated from the feed screw of the milling

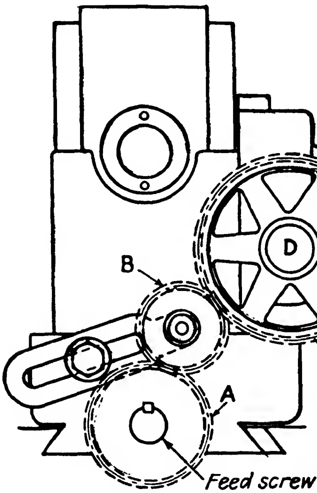


Fig. 48.

machine. The gear *C* on the stud is called the "gear on worm" and should not be confused with the worm gear which is permanently attached to the spiral-head spindle. The calculation of change gears for cutting spirals on the milling machine is the same, in principle, as the method for determining the change gears for the engine lathe.

The Lead of a Milling Machine.—The lead of a milling machine corresponds to the lathe constant of an engine lathe. Place gears with equal numbers of teeth on the feed screw and the worm shaft and use any intermediate gear *B* to complete the train (see Fig. 48). Then the lead of the milling machine is the distance the table will travel while the spiral-head spindle makes one complete revolution. This distance is a constant used in calculating the change gears. It may vary for different milling machines.

The lead of a spiral (helix) is the distance, measured along the axis of the work, in which the spiral makes one whole turn around the work. Thus, the lead of the milling machine may also be expressed as the lead of a spiral that will be cut when gears with an equal number of teeth are placed on the feed screw and worm shaft and a suitable intermediate gear is interposed between them.

To find the lead of a milling machine, place equal gears on the feed screw and worm shaft (Fig. 48) and find the number of revolutions of the feed screw to produce one complete turn of the spiral-head spindle. The number of revolutions of the feed screw multiplied by the lead of the thread of the feed screw equals the lead of the milling machine.

If L = lead of milling machine, N = number of revolutions made by the feed screw to produce one revolution of the spiral-head spindle, and P = lead of the feed screw, then we can write the formula as follows:

$$L = N \times P \qquad (54)$$

The lead of the feed screw of a milling machine is $\frac{1}{4}$ in., and 40 revolutions of the feed screw are required to turn the spiral-head spindle one complete revolution when gears A and C (Fig. 48) are equal.

$$L = N \times P$$
$$L = 40 \times \tfrac{1}{4} = 10 \text{ in.}$$

The Purpose of Change Gears When Milling Spirals.—The lead of the milling machine is the distance the table of the milling machine moves forward in order to turn the work placed on the spiral-head spindle one whole revolution when change gears with equal numbers of teeth are used. If it is required to cut a spiral with a lead twice that of the machine, change gears of such a ratio must be used that the spiral-head spindle will turn only one-half revolution while the table moves forward a distance equal to the lead of the machine.

It is required to find the change gears to cut a spiral having a lead of 20 in. on a milling machine that has a lead of 10 in. The ratio between the speeds of the feed screw and the worm shaft must be 2 to 1, which means that the feed screw is required to turn twice while the worm shaft turns once. Therefore the feed-screw gear must have one-half the number of teeth of the gear placed on the worm shaft. If the lead of the machine is 10 and the lead of the spiral required to be cut is 25 in., then the ratio between the speed of the gears would be 2.5 to 1, which is the same as the ratio between the lead of the spiral to be cut and the lead of the machine.

Cutting Spirals with Simple Gearing.—To find the change gears, form a fraction, the numerator of the fraction is the same as the lead of the spiral to be cut, and its denominator is the same as the lead of the milling machine. Multiply the numerator and the denominator of this fraction by the same number (any number) to get a new fraction. In this new fraction, the numerator gives the number of teeth in the gear on the worm shaft and the denominator the number of teeth in the gear on the feed screw. This can be written as an equation as follows:

$$\frac{\text{Lead of spiral to be cut}}{\text{Lead of milling machine}} = \frac{\text{teeth in gear on worm shaft}}{\text{teeth in gear on feed screw}} \qquad (55)$$

It is required to find the change gears to cut a spiral with a lead of 6 in. in a milling machine with a feed screw that has four threads per inch.

1. Find the lead of the milling machine. As the feed screw is single-threaded and has four threads per inch, the lead of the screw thread is $\frac{1}{4}$ in. Using Eq. (54) to find the lead of the milling, we have

$$L = N \times P \qquad (54)$$
$$L = 40 \times \tfrac{1}{4} = 10 \text{ in.}$$

2. Find the change gears using Eq. (55)

$$\frac{\text{Lead of spiral to be cut}}{\text{Lead of milling machine}} = \frac{6 \times 4}{10 \times 4} = \frac{24}{40} \; \frac{\text{gear on worm}}{\text{gear on screw}} \qquad (55)$$

The 24-tooth gear (gear on worm) is placed on the worm shaft, and the 40-tooth gear (gear on screw) is placed on the feed screw. An intermediate gear (any number of teeth) is placed between two gears to complete the train.

The number of teeth in the intermediate gear has no influence on the speed ratio of the feed screw and worm shaft, but simply serves to transmit motion from one gear to the other. The above setup is for cutting a right-hand spiral. If a left-hand spiral is required, it is only necessary to add a second intermediate gear to the train.

Cutting Spirals with Compound Gearing.—It is not always possible to find a set of two change gears that will transmit the required motion, nor is it always desirable, and it is then necessary to use compound gearing. The manner in which the gears are found is exactly the same as the method used for compound gearing in the engine lathe. The numerator and the denominator of the fraction are divided into two factors each, and then each pair of factors (one factor in the numerator and one in the denominator making one pair) is multiplied by the same number (any number) until numbers of teeth are obtained that are available in the change gears supplied. The formula for compound gearing is as follows:

$$\frac{\text{Lead of spiral to be cut}}{\text{Lead of milling machine}} = \frac{\text{product of driven gears}}{\text{product of driving gears}} \quad (56)$$

It is required to find the change gears to cut a spiral having a 36-in. lead in a milling machine with a lead of 10 in.

$$\frac{36}{10} = \frac{4 \times 9}{2 \times 5} = \frac{(4 \times 16)(9 \times 8)}{(2 \times 16)(5 \times 8)} = \frac{64 \times 72}{32 \times 40}$$

The 72-tooth gear is the gear on worm and meshes with the 32-tooth gear called the "first gear on stud." The 64-tooth gear called the "second gear on stud" is driven by the 40-tooth gear called the "gear on screw." This makes the gears having 72 and 64 teeth the driven gears and the gears having 32 and 40 teeth the driving gears, the train of gears being driven

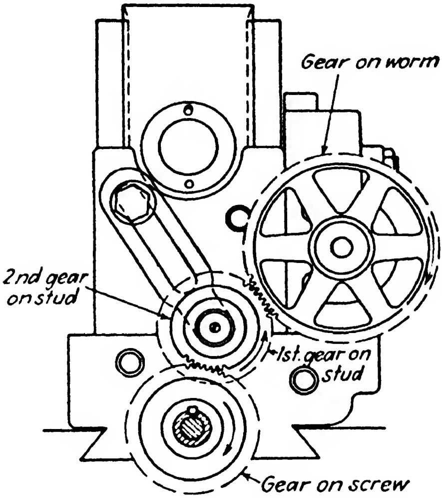


Fig. 49.

from the feed screw of the table. The driven gears may be interchanged, or the driving gears may be interchanged, but never interchange a driving gear for a driven gear. Figure 49 shows the change gears (compound gearing) arranged to cut a right-hand spiral, and Fig. 50 shows the change gears (compound gearing) arranged to cut a left-hand spiral.

Cam Cutting with the Spiral Head.—Face, peripheral, and cylindrical cams of all ordinary sizes can be cut upon a milling machine, and a far more satisfactory job can be obtained than is possible by drilling around the outline of a cam blank, breaking it off, and then filing to a line.

Screw-machine cams are required in great variety, each differing from the other by only a few thousandths of an inch and with practically no duplications, making the use of master cams out of the question.

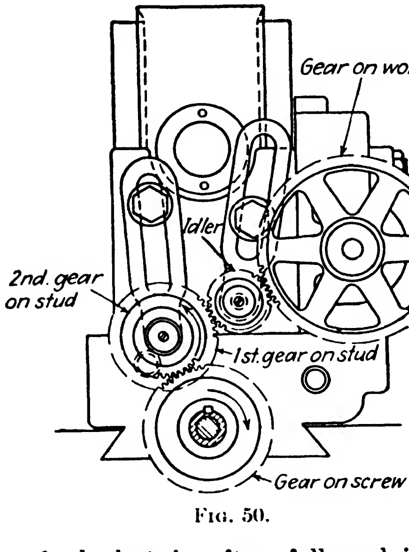


FIG. 50.

A method that is often followed in cutting peripheral cams, especially those for use on automatic screw machines, is that of using the spiral head and a vertical-spindle milling attachment (see Fig. 51). The spiral head is geared to the table feed screw, the same as in cutting ordinary spirals, and the cam blank is fastened to the end of the index spindle. An end mill is used in the vertical-spindle milling attachment, which is set in each case to mill the periphery of the cam at right angles to its sides. In other words, the axes of the spiral-head spindle and attachment spindle must always be parallel to mill cams according to this method. The cutting is done by the teeth on the periphery of the end mill.

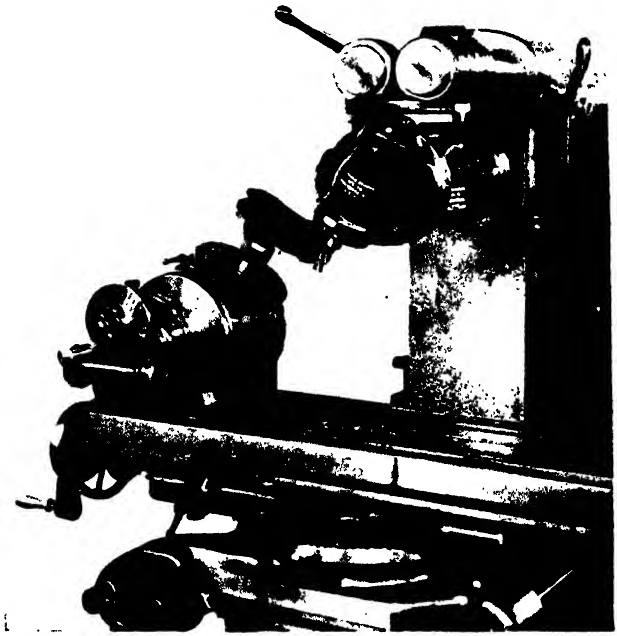


FIG. 51.—Milling screw machine cam, showing use of extension for spiral head. *(From Brown & Sharpe Manufacturing Company, "Practical Treatise on Milling.")*

The principle of this method is as follows: Suppose the spiral head is elevated to 90, or at exact right angles to the surface of the table, and is geared for any given lead (see Fig. 52). It is then apparent that, as the table advances and the blank is turned, the distance between the axes of the index spindle and attachment spindle

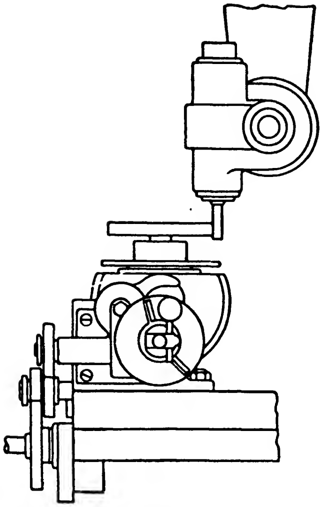


FIG. 52.

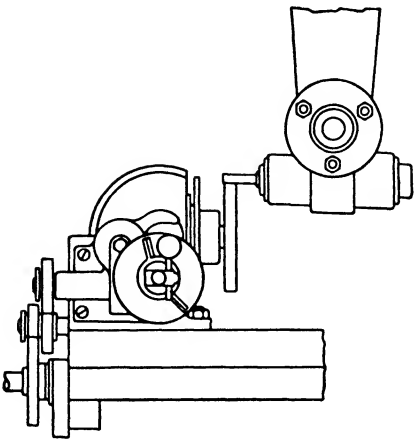


FIG. 53.

becomes less. In other words, the cut becomes deeper and the radius of the cam is shortened, producing a spiral lobe, the lead of which is the same as that for which the machine is geared.

Now suppose the same gearing is retained and the spiral head is set at zero, or parallel to the surface of the table (see Fig. 53). It is apparent,

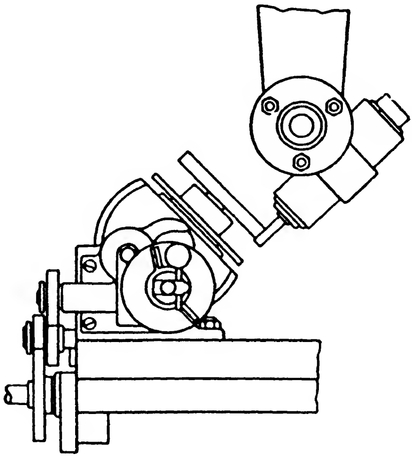


FIG. 54.

also, that the axes of the index spindle and attachment spindle are parallel to one another. Therefore, as the table advances and the blank is turned, the distance between the axes of the index spindle and attachment spindle remains the same. As a result, the periphery of the blank, if milled, is concentric or the lead is 0.

If, then, the spiral head is elevated to any angle between zero and 90 (see Fig. 54) the amount of lead given to the cam will be between that for which the machine is geared and 0. Hence it is clear that cams with a very large range of different leads can be obtained with one set of change gears, and the problem of milling the lobes of a cam is reduced to the question of finding the angle at which to set the head to obtain any given lead.

In order to illustrate the method of obtaining the correct angle, a drawing of a cam to be milled

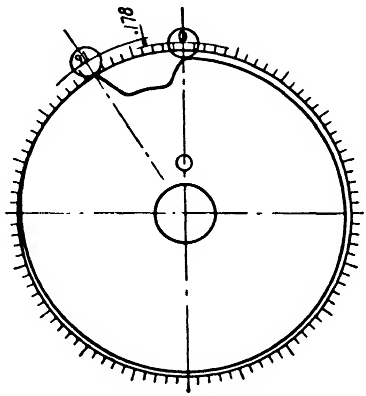


Fig. 55.

and data connected with same are given in Fig. 55.

It is first necessary to know the lead of the lobes of a cam, that is, the amount of rise of each lobe if continued the full circumference of the cam. This can be obtained from the drawing as follows: For cams where the face is divided into hundredths, as that shown, multiply 100 by the rise of the lobe in inches and divide by the number of hundredths of circumference occupied by the lobe. For cams that are figured in degrees of circumference, multiply 360 by the rise of the lobe in inches and divide by the number of degrees of circumference occupied by the lobe. Take Fig. 55 for example, we have a cam of one lobe that extends through $\frac{91}{100}$ of the circumference and has a rise of 0.178 in. Then $(100 \times 0.178)/91 = 0.1956$-in. lead of lobe, or 0.196 in., which is near enough for all practical purposes. As a 0.196-in. lead is much less than 0.67 in., which is the shortest lead regularly obtainable on the milling machine, the change gears that will give a lead of 0.67 in. may be used, and then the angle of the head can be adjusted so that a lead of 0.196 in. will be obtained on the cam lobe with these change gears. The rule for this is as follows:

Divide the given lead of the cam lobe by a lead obtainable on the machine, and the result is the sine of the angle at which to set the head.

Continuing the calculation for the lobe of the cam in Fig. 55, we therefore have

$$\frac{0.196 \text{ in.}}{0.67} = 0.29253$$

Hence, 0.29253 is the sine of the correct angle. Turning to a table of sines and cosines, we find that 0.29253 is very near 0.29265, which is the sine of an angle of 17 deg. and 1 minute. As the spiral head is not graduated closer than quarter degrees, it will be satisfactory to elevate the head just a hair over 17. Then, with the gearing for a lead of 0.67 in., a cam with a lead of 0.196 in. will be obtained.

The minute errors between the actual lead 0.1956 and 0.196 in. and in the sines and angles of this calculation can safely be ignored, as it is not possible in practice to work very much closer than outlined.

The portion of the periphery of the cam from $\frac{81}{100}$ to 0 represents a clearance of the cutting tool prior to the beginning of the throw. It is usually milled to a line or drilled, broken out, and filed.

Whenever possible, the job should be set up so that the end mill will cut on the lower side of the blank, as this brings the mill and table nearer together and makes the job more rigid. It also prevents chips from accumulating and enables the operator to see better any lines that may be laid out on the face of the cam.

When the lead of the machine is over 2 in., the automatic feed can be used, but when the lead is less than 2 in., the job should be fed by hand, with the index crank.

Problems

1. It is required to cut a gear with 55 teeth. Find the movement of the index crank.

2. A reamer is to have nine flutes regularly spaced. Find the movement of the index crank.

3. A ratchet wheel is to have 45 teeth. Find the movement of the index crank.

4. A 10-pitch gear of 132 teeth is to be cut. Find the plate, circle, and movement of the index crank.

5. A 14-pitch 18-tooth gear is to be cut in the milling machine. Find the plate, circle, and movement of the index crank.

6. It is required to have an indexing movement of $13\frac{1}{2}$ deg. Find the plate, circle, and number of turns of the crank.

7. Find the plate, circle, and number of turns of the crank to have an indexing movement of 37°20′.

8. An indexing movement of $10\frac{1}{2}$ deg. is required. Find the plate, circle, and number of turns of the index crank.

9. Find the plate, circle, and number of turns of the crank to have an indexing movement of 38°40′.

10. Find the plate, circles, and number of turns of the crank for indexing a 77-tooth gear. Use compound indexing.

11. Find the plate, circles, and number of turns of the crank for indexing a 69-tooth ratchet wheel. Use compound indexing.

12. A 10-pitch 91-tooth gear is to be cut in the milling machine. Find the plate, circles, and number of turns of the crank for indexing by compound indexing.

13. Find the indexing movement for 93 divisions, using compound indexing.

14. Select the plate and circle and find the change gears for indexing a 119-tooth gear. Use differential indexing.

15. Select the plate and circle, and find the change gears for indexing 106 divisions. Use differential indexing.

16. It is required to cut a gear with 250 teeth. Select the plate and circle, and find the change gears for indexing. Use differential indexing.

17. Find the change gears, plate, and circle for indexing for 239 divisions. Use differential indexing.

18. The lead of a spiral is 12 in. Find the change gears if the lead of the machine is 10 in.

19. Find the change gears for milling a spiral with a lead of 10.5 in. in a milling machine with a lead of 10 in.

20. The lead of a milling machine is 10 in. Find the change gears for cutting a spiral with a lead of 48 in.

21. Find the change gears for cutting a spiral with a lead of 8 in. in a milling machine with a lead of 10 in.

22. The lead of a spiral is 40 in. Find the change gears for cutting the spiral in a milling machine with a lead of 10 in.

CHAPTER VII

TOLERANCES AND ALLOWANCES FOR MACHINE PARTS[1]

In manufacturing machine parts according to modern methods, certain maximum and minimum dimensions are established, particularly for the more important members of whatever machine or mechanism is to be constructed. These limiting dimensions serve two purposes: They prevent unnecessary accuracy and also excessive inaccuracies. A certain degree of accuracy is essential to the proper functioning of the assembled parts of a mechanism, but it is useless and wasteful to make parts more precise than needed to meet practical requirements; hence, the use of proper limiting dimensions promotes efficiency in manufacturing and also ensures standards of accuracy and quality that are consistent with the functions of the different parts of a mechanical device.

Parts made to specified limits usually are considered interchangeable, or capable of use without selection, but there are several degrees of interchangeability in machinery manufacture. Strictly speaking, interchangeability consists in making the different parts of a mechanism so uni-

[1] Reprinted from John M. Amiss and Franklin D. Jones, "The Use of Handbook Tables and Formulas," by permission from the Industrial Press.

form in size and contour that each part of a certain model will fit any mating part of the same model, regardless of the lot to which it belongs or when it was made. However, as often defined, interchangeability consists in making each part fit any mating part in a certain series; that is, the interchangeability exists only in the same series. Selective assembly is sometimes termed interchangeability, but it involves a selection or sorting of parts as explained later. It will be noted that the strict definition of interchangeability does not imply that the parts must always be assembled without handwork, although that is usually considered desirable. It does mean, however, that when the mating parts are finished, by whatever process, they must assemble and function properly, without fitting individual parts one to the other.

When a machine having interchangeable parts has been installed possibly at some distant point, a broken part can readily be replaced by a new one sent by the manufacturer, but this feature is secondary as compared with the increased efficiency in manufacturing on a interchangeable basis. In order to make parts interchangeable, it is necessary to use gauges and measuring tools, to provide some system of inspection, and to adopt suitable tolerances. Whether absolute interchangeability is practicable or not may depend upon the tolerances adopted, the relation between the different parts and their form.

Meanings of the Terms Limit, Tolerance, and Allowance.—The terms "limit" and "tolerance" and also "tolerance" and "allowance" are often used interchangeably, but each of these terms has a distinct meaning and refers to different dimensions. As shown by the diagram (Fig. 56) the limits of a hole or shaft are its diameters. Tolerance is the difference between two limits or limiting dimensions of a given part, and the term means that a certain amount of error is tolerated for practical reasons. Allowance is the difference between limiting dimensions on mating parts that are to be assembled either loosely or tightly, depending upon the amount allowed for the fit.

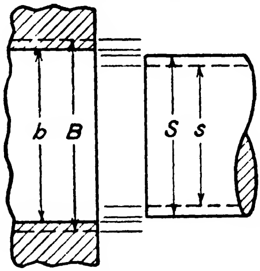


FIG. 56.—Difference between limit, tolerance, and allowance. Limits: B, maximum limit of bore; b, minimum limit of bore; S, maximum limit of shaft, s, minimum limit of shaft. Tolerances: B-b, maximum tolerance of bore; S-s, maximum tolerance of shaft. Allowances: B-s, maximum allowance, or if s is greater than B (as for tight or forced fits) then s-B is minimum allowance for fit; b-S, minimum allowance, or if S is greater than b (as for tight or forced fits) then S-b is maximum allowance for fit.

Example 1. The American Standard for medium fits is found in Table VII, page 218. From this table, explain how the tolerances and allowances are established for a diameter of 2 in.

Maximum hole = 2 + 0.0010 and minimum hole = 2 − 0.000; hence,

Hole tolerance = 2.0010 − 2 = 0.0010 in.

Maximum shaft = 2 − 0.0014 = 1.9986 in.; minimum shaft = 2 − 0.0024 = 1.9976 in.; hence,

Minimum allowance = minimum hole − maximum shaft = 2 − 1.9986 = 0.0014 in.

Maximum allowance = maximum hole − minimum shaft = 2.0010 − 1.9976 = 0.0034 in.

Example 2. Tables VIII and IX, pages 219–220, give the dimensions for the American Standard coarse-thread series—medium fit. Determine the pitch-diameter tolerance of both screw and nut and also the minimum and maximum allowance between screw and nut at the pitch diameter, assuming that the nominal diameter is 1 in. and the pitch $\frac{1}{8}$ in.

The maximum pitch diameter or limit of the screw = 0.9188, and the minimum pitch diameter, 0.9134; hence, the tolerance = 0.9188 − 0.9134 = 0.0054 in.

The nut tolerance = 0.9242 − 0.9188 = 0.0054 in.

The maximum allowance for medium fit = maximum pitch diameter of nut − minimum pitch diameter of screw = 0.9242 − 0.9134 = 0.0108 in.

The minimum allowance = minimum pitch diameter of nut − maximum pitch diameter of screw = 0.9188 − 0.9188 = 0.0000.

Relation of Tolerances to Limiting Dimensions and How Basic Size Is Determined.—The absolute limits of the various dimensions and surfaces indicate danger points, inasmuch as parts made beyond these limits are unserviceable. A careful analysis of a mechanism shows that one of these danger points is more sharply defined than the other. For example, a certain stud must always assemble into a certain hole. If the stud is made beyond its maximum limit, it may be too loose or too weak to function. The absolute maximum limit in this case may cover a range of 0.001 in., whereas the absolute minimum limit may have a range of at least 0.004 in. In this case, the maximum limit is the more sharply defined.

The basic size expressed on the component drawing is that limit which defines the more vital of the two danger points, while the tolerance defines the other. In general, the basic dimension of a male part, such as a shaft, is the maximum limit that requires a minus tolerance. Similarly, the basic dimension of a female part is the minimum limit requiring a plus tolerance, as shown in Fig. 57. There are, however, dimensions that define neither a male nor a female surface, such for example as dimensions for the location of holes. In a few cases of this kind, a variation in one direction is less dangerous than

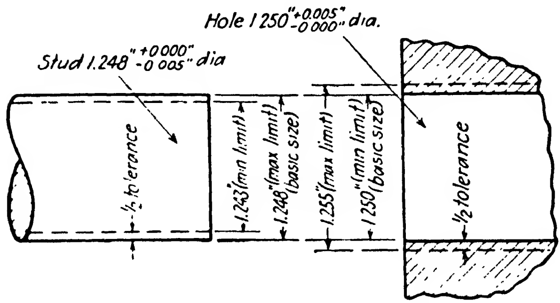


FIG. 57.—Basic size or dimension.

a variation in the other. Under these conditions, the basic dimension represents the danger point, and the unilateral tolerance permits a variation only in the less dangerous direction. At other times, the conditions are such that any variation from a fixed point in either direction is equally dangerous. In such a case, the basic size represents this fixed point, and tolerances on the drawing are bilateral and extend equally in both directions.[1]

When Allowance Provides Clearance between Mating Parts.—When one part must fit freely into another part like a shaft in its bearing, the

[1] For an explanation of unilateral and bilateral tolerances see "Machinery's Handbook" or Colvin, "Gages and Their Uses in Inspection," p. 110.

allowance between the shaft and bearing represents a clear space. It is evident that the amount of clearance varies widely for different classes of work. The minimum clearance should be as small as will permit the ready assembly and operation of the parts, while the maximum clearance should be as great as the functioning

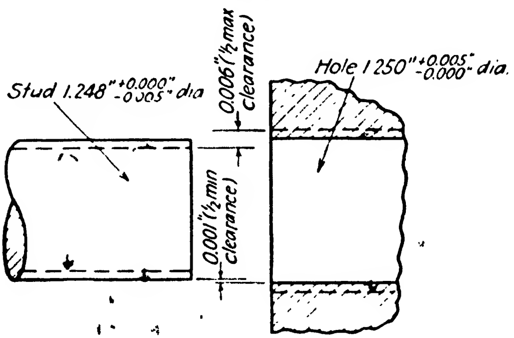


FIG. 58.—Maximum and minimum clearance.

of the mechanism will allow. The difference between the maximum and minimum clearances defines the extent of the tolerances. In general, the difference between the basic sizes of companion parts equals the minimum clearance (see Fig. 58), and the term "allowance" if not defined as maximum or minimum is commonly applied to the minimum clearance.

When Interference of Metal Is the Result of Allowance.—If a shaft or pin is larger in diameter than the hole into which it is forced, there is, of course, interference between the two parts. The metal surrounding the hole is expanded and compressed as the shaft or other part is forced into place. Engine crankpins, car axles, and various other parts are assembled in this way. Tables of selected fits are given in various handbooks. If interchangeable parts are to be forced together, the minimum interference establishes the danger point. This means that for force fits the basic dimension of the shaft or pin is the minimum limit requiring a plus tolerance, while the basic dimension of the hole is the maximum limit requiring a minus tolerance (see Fig. 59).

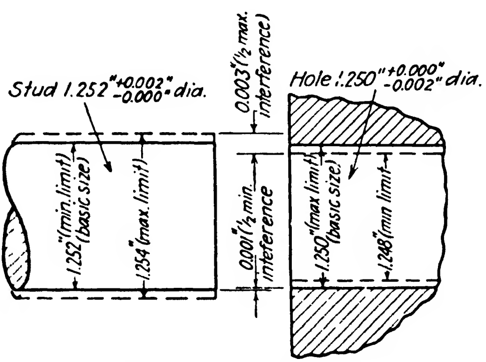
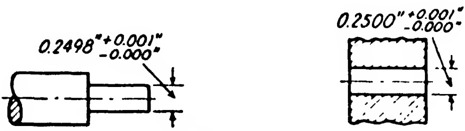


FIG. 59.—Maximum and minimum interference.

Obtaining Allowance by Selection of Mating Parts.—The term "selective assembly" is applied to a method of manufacturing that is similar in many of its details to interchangeable manufacturing. In selective assembly, the mating parts are sorted according to size and assembled or interchanged with little or no machining. The chief purpose of manufacturing by selective assembly is the production of large quantities of duplicate parts as economically as possible. As a general rule, the smaller the tolerances, the more exacting and expensive will be the manufacturing processes; but it is possible to use comparatively large tolerances and then reduce them, in effect, by selective assembly, provided the quantity of parts is large enough to make such selective fitting possible. To illustrate the procedure, Fig. 60 shows a plug or stud that has a plus tolerance of 0.001 in. Assume stud that has a plus tolerance of 0.001 in. and a hole that also has a plus tolerance of 0.001 in. Assume that this tolerance of 0.001 in. represents the



GROUP A 0.2498"	+ 0.0002" − 0.0000"	GROUP A 0.2500"	+ 0.0002" − 0.0000"
GROUP B 0.2500"	+ 0.0002" − 0.0000"	GROUP B 0.2502"	+ 0.0002" − 0.0000"
GROUP C 0.2502"	+ 0.0002" − 0.0000"	GROUP C 0.2504"	+ 0.0002" − 0.0000"
GROUP D 0.2504"	+ 0.0002" − 0.0000"	GROUP D 0.2506"	+ 0.0002" − 0.0000"
GROUP E 0.2506"	+ 0.0002" − 0.0000"	GROUP E 0.2508"	+ 0.0002" − 0.0000"

FIG. 60.—Information placed on drawings used in selective assembly manufacturing to facilitate grading of parts.

normal size variation on each part when manufactured efficiently. With this tolerance, a minimum plug in a maximum hole would have a clearance of 0.2510 − 0.2498 = 0.0012 in.; and a maximum plug in a minimum hole would have a metal interference of 0.2508 − 0.2500 = 0.0008 in. But suppose the clearance required for these parts must range from zero to 0.0004 in. This reduction can be obtained by dividing both plugs and holes into five groups (see Fig. 60). Any studs in group *A*, for example, will assemble in any hole in group *A*, but the studs in one group will not assemble properly in the holes in another group. When the largest stud in group A is assembled in the smallest hole in group A, the clearance equals zero. When the smallest stud in group A is assembled in the largest hole in group A, the clearance equals 0.0004 in. Thus, in selective assembly manufacturing, there is a double set of limits, the first being the manufacturing limits, and the second the assembling limits. In many cases, two separate drawings are made of a part that is to be graded before assembly. One shows the manufacturing tolerances only, so as not to confuse the operator, while the other gives the proper grading information.

Example 3. According to the American Standard for heavy force- and shrink-fit tolerances and allowances, the maximum allowance for a 2-in. diameter is 0.0028 in. The loosest fit is 0.0012 in., and the selective fit, 0.0020 in. Explain how these figures are determined.

The maximum allowance for a force fit = maximum shaft − minimum hole = 2.0028 − 2 = 0.0028 in.

Loosest fit = minimum shaft − maximum hole = 2.0020 − 2.0008 = 0.0012 in.

The desired allowance or selected fit for this diameter is midway between the minimum allowance (loosest fit) and the maximum allowance and equals 0.0020 in. In order to obtain shafts that are 0.0020 in. larger than the mating holes, it is necessary to select the sizes, assuming that the parts are made in large enough quantities to permit such selection.

Dimensioning Drawings to Ensure Obtaining Required Tolerances.—In dimensioning the drawings of parts requiring tolerances, there are certain fundamental rules that should be applied.

Rule 1. In interchangeable manufacturing, there is only one dimension (or group of dimensions) in the same straight line which can be controlled within fixed tolerances. This is the distance between the tool and the locating, or registering, surface of the part being machined. Therefore, it is incorrect to locate any point or

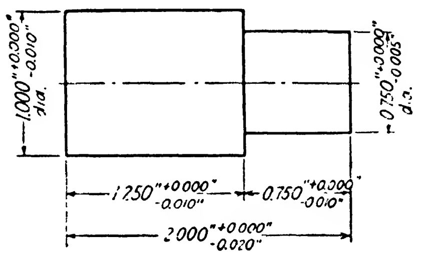


FIG. 61.—Common but incorrect method of dimensioning.

surface with tolerances from more than one point in the same straight line.

Rule 2. Dimensions should be given between those points which it is essential to hold in a specific relation to each other. The majority of dimensions, however, are relatively unimportant in this respect. It is good practice to establish common location points in each plane and to give, as far as possible, all such dimensions from these points.

Rule 3. The basic dimensions given on component drawings for interchangeable parts should be, except for force fits and other unusual conditions, the maximum metal sizes (maximum shaft or plug and minimum hole).

The direct comparison of the basic sizes should check the danger zone, which is the minimum clearance condition in the majority of cases. It

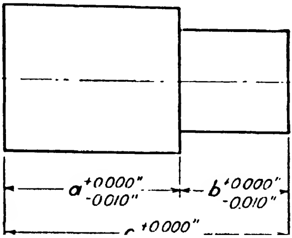


FIG. 62.—One interpretation of dimensioning in Fig. 61.

is evident that these sizes are the most important ones, as they control the interchangeability, and they should be the first determined. Once established, they should remain fixed if the mechanism functions properly and the design is unchanged.

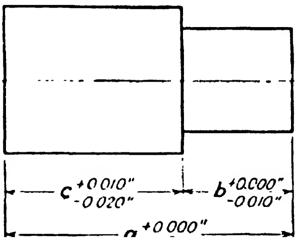


FIG. 63.—Second interpretation of dimensioning in Fig. 61.

The direction of the tolerances, then, would be such as to recede from the danger zone. In the majority of cases, this means that the direction of the tolerances is such as will increase the clearance. For force fits, the basic dimensions determine the minimum interference, while the tolerances limit the maximum interference.

Rule 4. Dimensions must not be duplicated between the same points. The duplication of dimensions causes much needless trouble, due to changes being made in one place and not in the others. It causes less trouble to search a drawing to find a dimension than it does to have them duplicated and more readily found but inconsistent.

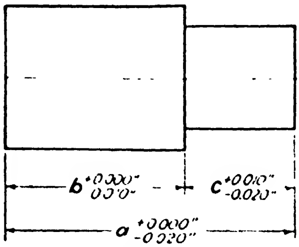


FIG. 64.—A third interpretation of dimensioning in Fig. 61.

Rule 5. As far as possible, the dimensions on companion parts should be given from the same relative locations. Such a procedure assists in detecting interferences and other improper conditions.

In attempting to work in accordance with the general laws or principles, one other elementary rule should always be kept in mind. Special cases require special consideration. The following detailed examples are given to illustrate the application of these five laws and to indicate results of their violation.

Violations of Rules for Dimensioning.—Figure 61 shows a very common method of dimensioning a part as the stud shown, but one that is bad practice. It violates the first and second rules. As the dimensions given for the diameters are correct, they are eliminated from the discussion. The dimensions given for the various lengths are wrong, (1) because they give no indication as to the essential lengths; (2) because of several pos-

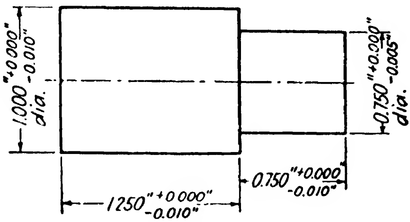


FIG. 65.—Correct dimensioning if length of body and length of stem are most important.

sible sequences of operations, some of which would not maintain the specified conditions.

Figure 62 shows one possible sequence of operations indicated alphabetically. If we first finish the dimension *a* and then finish *b*, the dimension

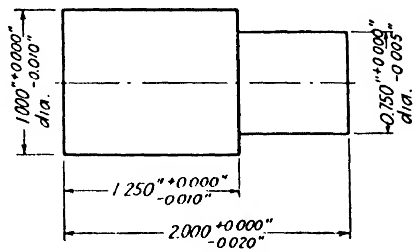


FIG. 66.—Correct dimensioning if length of body and over-all length are most important.

c will be within the specified limits. In this case, however, the dimension *c* is superfluous. Figure 63 gives another possible sequence of operations. If we first establish *a* and then *b*, the dimension *c* may vary 0.030 in. instead of 0.010 in. as is specified in Fig. 61. Figure 64 gives a third possible sequence of operations. If we first finish the over-all length *a* and then the length of the body *b*, the stem *c* may vary 0.030 in. instead of 0.010 in. as specified in Fig. 61.

If three different plants were manufacturing this part, each one using a different sequence of operations, it is evident from the foregoing that

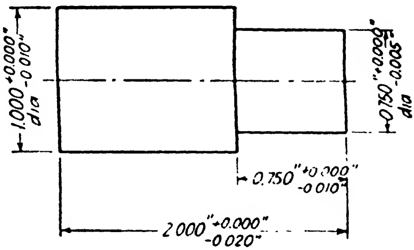


FIG. 67.—Correct dimensioning if over-all length and length of stem are most important.

a different product would be received from each plant. The example given is the simplest one possible. As the parts become more complex and the number of dimensions increases, the number of different combinations possible and the extent of the variations in size that develop also increase.

Figure 65 shows the correct way to dimension this part if the length of the body and the length of the stem are essential dimensions. Figure 66 is the correct way if the length of the body and the length over-all are the most important. Figure 67 is correct if the length of the stem and the length over-all are the most important. If the part is dimensioned in accordance with either Fig. 66, 67, or 68, the product from any number of factories should be alike.

CHAPTER VIII

THE USE OF COMMON PRECISION MEASURING DEVICES

HOW TO READ MICROMETERS

Micrometers Graduated to Thousandths of an Inch.—The customary pitch of the screw is $\frac{1}{40}$ in. (40 threads to the inch). Thus, the distance traversed by the screw or spindle during one complete revolution is $\frac{1}{40}$ in. or 0.025 in. As the graduations on the barrel conform to the pitch of the screw (40 to the inch), each division equals 0.025 in. and every four divisions represent 0.100 in., thus reading 0, 0.100 in., 0.200 in., 0.300 in., etc. (tenths of an inch); each tenth of an inch is numbered 0, 1, 2, etc. The beveled edge of the thimble is graduated into 25 parts and figured every fifth division 0, 5, 10, 15, and 20. When 25 of these graduations have passed the horizontal line on the barrel, the spindle, having made one revolution, has moved 0.025 in. Thus when the spindle moves only far enough to cause one graduation to pass the horizontal line on the barrel, it will have moved $\frac{1}{25}$ of 0.025 in., or 0.001 in. The distance between the graduations on the thimble is great enough to permit half and quarter thousandths of an inch to be readily estimated.

To read: First note the last figure visible on the scale on the barrel, representing the tenths of an inch. Multiply the number of divisions visible beyond this figure by 25, and add the number of the division on the scale on the thimble that coincides with the line of graduations on the barrel. Then this sum expressed in thousandths, added to the tenths shown, is the reading.

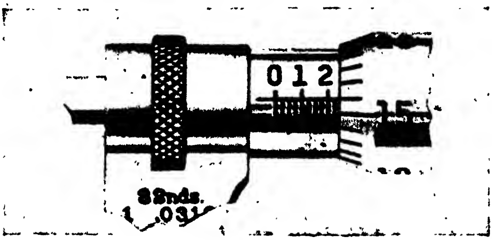


FIG. 68.

Example. In Fig. 68, 0.200 in. ($\frac{2}{10}$ in.) is shown by the figures on the scale on the barrel and one graduation beyond a tenth graduation is also visible, while on the bevel of the thimble the graduation shows 16 divisions from the zero to the line coincident with the horizontal line on the barrel. Then the reading = 0.200 in. + 0.025 in. + 0.016 in. = 0.241 in.

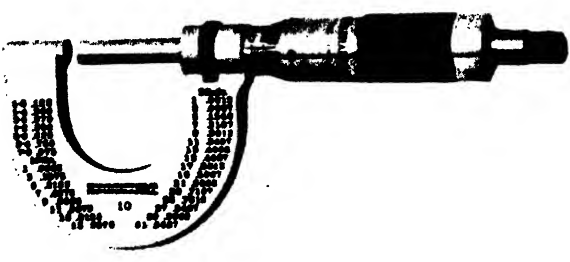


FIG. 69.—(*Courtesy of Brown & Sharpe Manufacturing Company, Providence, R.I.*)

Figure 69 shows a micrometer with a reading of 0.126 in.

Micrometers Graduated to Ten-thousandths of an Inch.—To obtain readings in ten-thousandths of an inch, a vernier is employed on the barrel of the micrometer caliper. The vernier used consists of 10 divisions, which equal, in over-all space, 9 divisions on the thimble.

Thus, one division on the vernier

$$= \frac{1}{10} \times \frac{9}{1{,}000} \text{ in.} = \frac{9}{10{,}000} \text{ in.}$$

Since each graduation on the thimble = 1/1,000 in. or 10/10,000 in., the difference in space

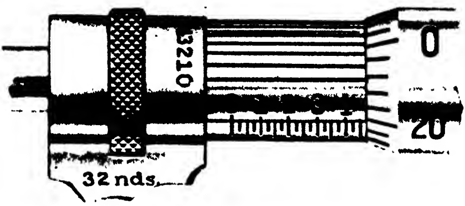


FIG. 70.

between a division on the thimble and a division on the barrel

$$= \frac{10}{10{,}000} \text{ in.} - \frac{9}{10{,}000} \text{ in.} = \frac{1}{10{,}000} \text{ in.}$$

Since the two zero lines on the vernier coincide with lines on the thimble when the reading is exact with respect to the number of thousandths, the difference between the lines on the thimble and the lines on vernier at numbers 1, 2, 3, etc., equals 0.0001 in., 0.0002 in., 0.0003 in., etc. Thus, when the first, second, or third, etc., lines coincide, the thimble has moved past the exact zero setting 0.0001 in., 0.0002 in., or 0.0003 in., etc., to bring these lines together.

To read: First obtain the readings for the thousandths in the manner described in the preceding section and then add the ten-thousandths, the number of which is indicated by the line on

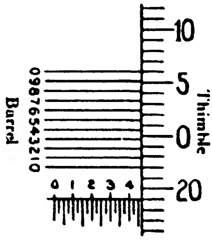


FIG. 71.

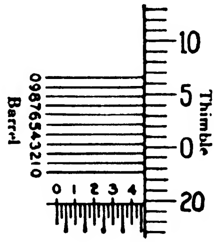


FIG. 72.

the vernier that coincides with a line on the thimble.

Example. As shown in Fig. 71, there are no ten-thousandths to be added, for the two zeros on the vernier coincide with lines on the thimble; the reading = 0.4690 in. In Fig. 72 the seventh graduation on the vernier coincides with a line on the thimble, indicating that 0.0007 should be added to the thousandths reading; the reading = 0.4690 in. + 0.0007 in. = 0.4697 in.

HOW TO READ VERNIERS

Principle of the Vernier.—The vernier was invented by Pierre Vernier in 1631 and from him it derived its name. It consists of a small scale on which is a certain number of graduations that equal in combined length a certain number of graduations, usually one more or one less, on the scale of the tool. It is evident that if, in the same extreme length, the vernier has divisions greater or less in number by one than the scale, there is a small difference between a division on the vernier and a division on the scale. Upon the difference between the vernier and the scale

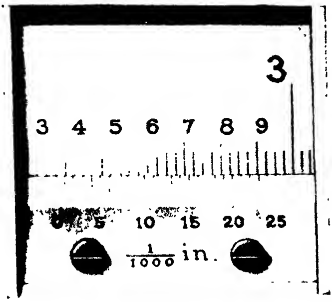


FIG. 73.

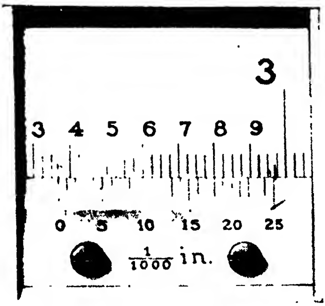


FIG. 74.

divisions depend the readings, for the vernier is attached to one of the measuring parts and the scale to the other.

Reading the Vernier. Figures 73 and 74 show the vernier used with a scale that is graduated into $\frac{1}{40}$ or 0.025 in. The vernier has 25 divisions that are numbered every fifth division and that equal, in extreme length, 24 divisions on the scale, or $24 \times \frac{1}{40} = 24 \times 0.025$ in. $= 0.600$ in.

Thus, one division on the vernier equals $\frac{1}{25}$ of 0.600 in. = 0.024 in. Therefore, the difference between a division on the scale and a division on the vernier = 0.025 in. − 0.024 in. = 0.001 in.

When the reading is exact, with respect to the number of fortieths of an inch, the zero on the vernier coincides with a graduation on the scale—either inch, tenth, or fortieth, as the case may be. This leaves a space between lines on the scale and the 1, 2, 3, 4, 5, 6, etc., lines on the vernier of 0.001 in., 0.002 in., 0.003 in., 0.004 in., 0.005 in., 0.006 in., etc., respectively, the difference increasing 0.001 in. at each vernier division in numerical order until, at the twenty-fifth graduation, the lines again coincide (see Fig. 73)

Thus, when the first, second, third, or fourth, etc., line on the vernier coincides with a line on the scale, the zero on the vernier has moved 0.001 in., 0.002 in., or 0.003 in., etc., past the previous fortieth graduation to bring these lines together.

To read: Note the inches, tenths, and fortieths of an inch that the zero on the vernier has moved from the 0 on the scale, and to this reading add the number of thousandths indicated by the line on the vernier that coincides with a line on the scale.

Example. Figure 73 shows the zero graduation on the vernier coinciding with a fortieth graduation on the scale (the second fortieth beyond an even tenth graduation). This indicates that the reading is exact with respect to the fortieths of an inch. The reading therefore equals 2.000 in. + 0.300 in. + 0.050 in. = 2.350 in. Figure 74, however, shows the eighteenth vernier graduation coinciding with a line on the scale. This indicates that 0.018 in. should be added to the scale reading. The reading, then, equals 2.000 in. + 0.300 in. + 0.050 in. + 0.018 in. = 2.368 in.

Verniers with 25 divisions are used, for English measure, on most verniers.

Reading the Vernier on Universal Bevel Protractor. The vernier indicates every 5 minutes (5′), or $\frac{1}{12}$ deg. Each space on the vernier is 5 minutes shorter than two spaces on the true scale.

When the zero on the vernier exactly coincides with a graduation on the scale, the reading is in exact degrees. When the zero graduation of the vernier does not exactly coincide with a graduation on the scale, the graduation on the vernier that does coincide with a graduation on the scale indicates the number of twelfths of a degree or units of 5 minutes to be added to the whole degree reading.

To Read the Protractor Setting.—Read off directly from the true scale the number of whole degrees between 0 and the 0 of the vernier scale. Then count, in the same direction, the number of spaces from the 0 of the vernier scale to a line that coincides with a line on the true scale; multiplying this number by 5, the product will be the number of minutes to be added to the whole number of degrees.

Example. As the vernier is shown in Fig. 75 it has moved 12 whole degrees to the right of the 0 upon the true scale, and the eighth line on the vernier coincides with a line upon the true scale as indicated by *. Multiplying 8 by 5, the product 40 is the number of minutes to be added to the whole number of degrees, thus indicating a setting of 12°40′.

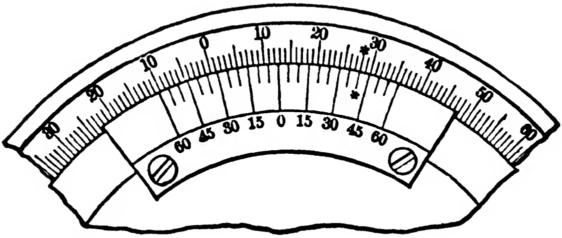


FIG. 75.

In Fig. 76 the reading is 12°50′, and in Fig. 77 the reading is 17°0′.

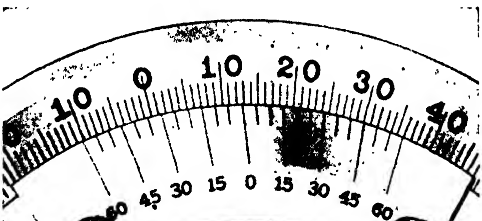


FIG. 76.

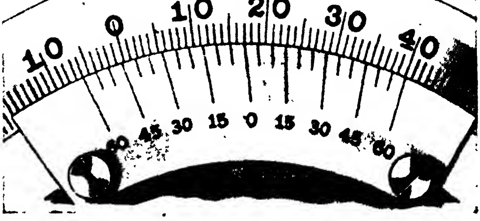


FIG. 77.

THE SINE BAR

The sine bar is made of high-grade alloy steel, heat-treated and hardened for stability and durability. Its edges are ground parallel within 0.0001 in. to the center line of the measuring studs. These studs are of identical diameters

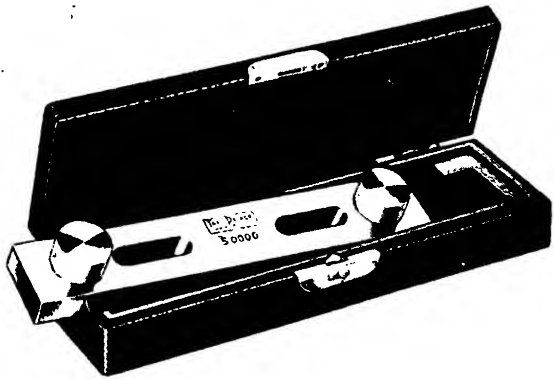


FIG. 78.

located 5.0000 in. ± 0.0002 in. from center to center on the 5-in. sine bar, and

10.0000 in. ± 0.00025 in.

from center to center on the 10 in. size.

The sine bar is used either for measuring angles accurately or for locating work to a given angle within very close limits. It consists of an accurate straightedge in which are set two hardened and ground plugs. These plugs must be of the same diameter, and their center distance should be an even dimension, to facilitate calculations. The edges of the straightedge must be parallel with the line of the plug centers. The sine bar is always used in conjunction with a true surface, such as a surface plate, from which measurements are taken. An explanation of the principles underlying the use of the sine bar is given in detail below.

The 10-in. sine bar should be used where extreme accuracy is sought, since it reduces by

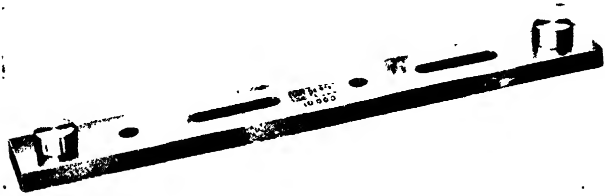


FIG. 79.

nearly one-half the possible error encountered in the smaller size.

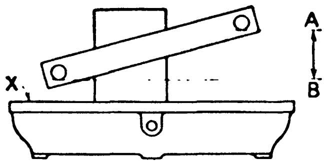


FIG. 80.

In machine-shop practice the sine bar is set up to represent the hypotenuse of the required angle, as shown in Fig. 80. Then the difference in the height of the measuring plugs $A - B$ measured from the base or surface plate X, divided by a constant of 5 or 10, depending on whether the 5- or 10-in. sine bar is used, is the natural sine of the angle, and the value of the angle itself is readily and accurately determined by reference to any table of natural sines.

For example, in setting the 5-in. sine bar shown in Fig. 78, distance *AB* was determined by means of a vernier height gauge to be 2.68525 in. This figure divided by 5 (because the 5-in. sine bar was used) gave a result of 0.53705 in.—the sine of the angle required. Reference to a table of

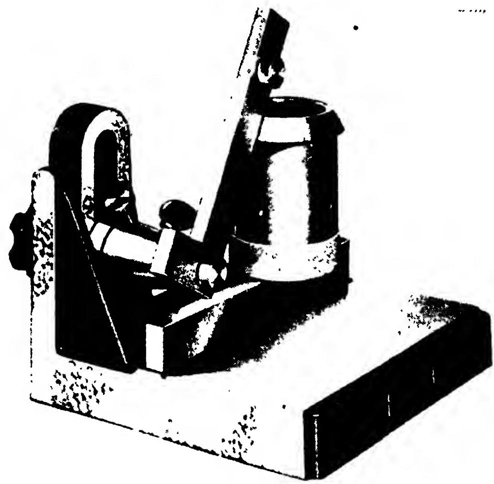


Fig. 81.

natural sines shows the value of the angle in question to be 32°29″.

If a 10-in. sine bar were used, distance *AB* would have been found to be 5.3705 in. or ten times the natural sine of an angle of 32°29″. It will be noted that an error in setting the 10-in. sine bar would cause but half the inaccuracy in measurement of the angle that would be caused by the same error in setting the 5-in. bar.

In setting up for a given angle, the method outlined above is reversed. By referring to a table of natural sines, the sine of an angle of

32°29″ is found to be 0.53705. Then by setting the 5-in. sine bar in a manner such that the

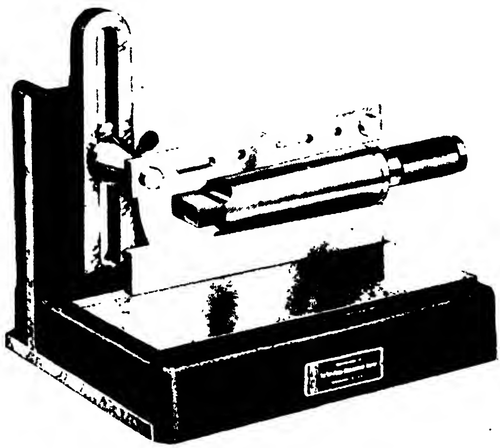


Fig. 82.

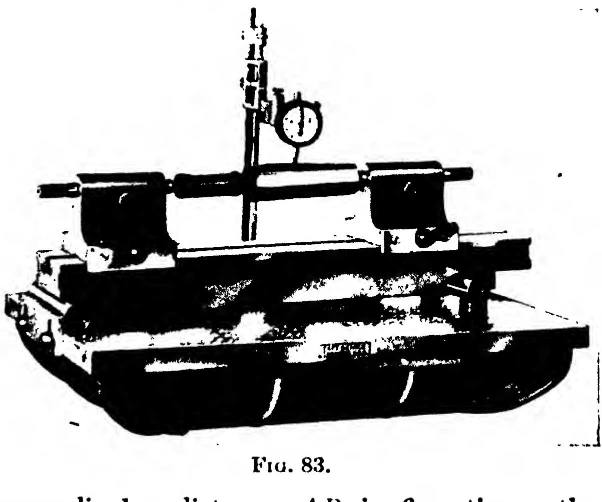


Fig. 83.

perpendicular distance AB is five times the natural sine, or 2.68525 in., the correct angle is at once established between the edge of the sine bar and the surface plate on which it is mounted.

THREAD MEASUREMENT BY THE THREE-WIRE PROCESS

The fit of a screw thread depends largely upon pitch-diameter limits and tolerances, the mainte-

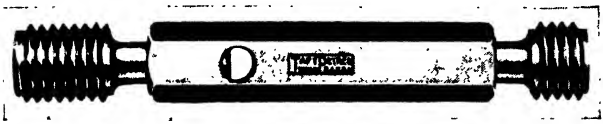


FIG. 84.

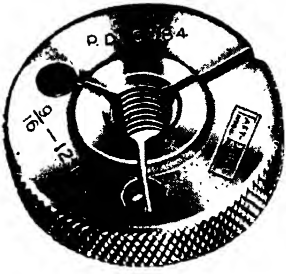


FIG. 85.

nance of which requires the use of limit thread gauges. These, in turn, depend upon the absolute measurements of master gauges. The adoption of a uniform practice for measuring pitch diameter is, therefore, highly desirable. The three-wire method has been found to be the most accurate and satisfactory when properly carried out and is recommended for universal use in the direct measurement of thread plug gauges by the Bureau of Standards and the National Screw Thread Commission.

This method consists in measuring the pitch diameter of the thread over the diameters of three small hardened steel cylinders, or wires,

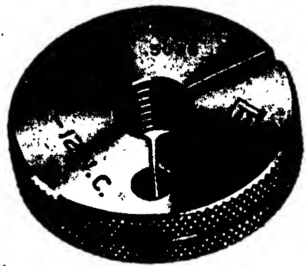


FIG. 86.

two of which are inserted between the threads and measuring anvil on one side, and one on the opposite side and midway between the two wires. The size of wire that touches exactly at the midslope of a perfect thread of a given pitch is termed the "best size" wire, or "best wire" for that pitch. A table of best wire diameters for pitches from 80 threads per inch to 4 threads per inch is shown on page 134.

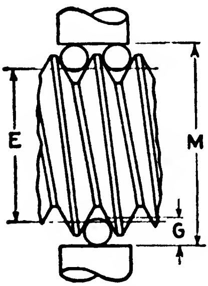


FIG. 87.

A method of calculating the angle diameter which does not take into account the effect of the helix angle is given below, but unless this angle is large the error will seldom amount to 0.0001.

$$E = M + \frac{\cot a}{2N} - G(1 + \operatorname{cosec} a)$$

For a 60-deg. thread of correct angle this equation is

$$E = M + \frac{0.86603}{N} - 3G$$

where E = pitch diameter, M = measurement

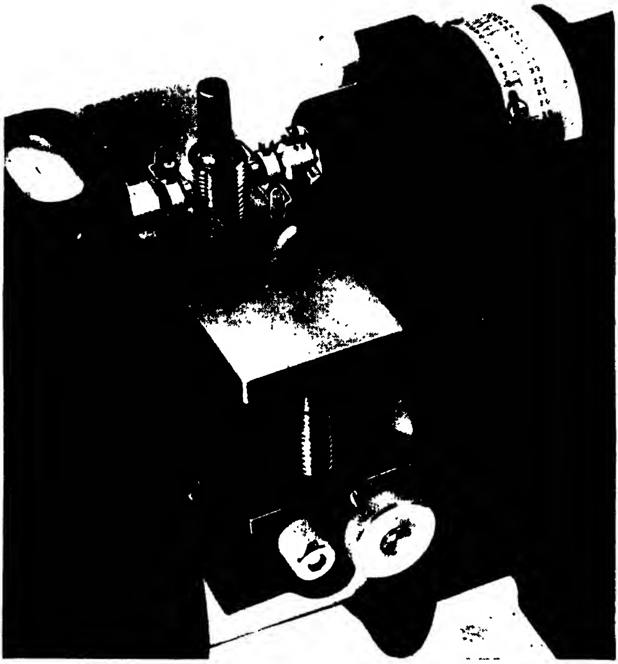


FIG. 88.—(*From Pratt & Whitney Company.*)

over wire, a = one-half included angle of thread, N = number of threads, G = diameter of wires.

Measurment over wires $M = E - \frac{0.86603}{N} + 3G$

Too much pressure should not be applied when measuring over wires. For pitches finer than

20 threads per inch a pressure of 1 lb. is recommended and $2\frac{1}{2}$ lb. for pitches of 20 threads per inch and coarser.

Measuring wires to Bureau of Standards specifications—glass-hard, highly polished, and accurate within 0.000025 in.—can be furnished in the best wire sizes listed below.

N = No. threads per inch	$G = \frac{0.57735}{N}$ diameter best wire	N = No. threads per inch	$G = \frac{0.57735}{N}$ diameter best wire	N = No. threads per inch	$G = \frac{0.57735}{N}$ diameter best wire
80	0.00722	28	0.02062	$11\frac{1}{2}$	0.05020
72	0.00802	26	0.02221	11	0.05249
64	0.00902	24	0.02406	10	0.05774
56	0.01031	22	0.02624	9	0.06415
50	0.01155	20	0.02887	8	0.07217
48	0.01203	19	0.03039	7	0.08248
44	0.01312	18	0.03207	6	0.09622
40	0.01443	16	0.03608	$5\frac{1}{2}$	0.10497
36	0.01604	14	0.04124	5	0.11547
32	0.01804	13	0.04441	$4\frac{1}{2}$	0.12830
30	0.01924	12	0.04811	4	0.14434

Figure 88 shows a thread gauge being checked by the three-wire method for pitch diameter. The measuring is done with a Pratt and Whitney Supermicrometer.

CHAPTER IX

PRESS TOOLS

The class of shop equipment called "punches and dies," or "press tools," has been extended into so many lines of manufacture that no one responsible for factory output or connected with the design, manufacture, or tool-department activities can afford to remain uninformed about the details of construction and application of these tools. Today there are not many lines of metal manufacture where such tools cannot be used to advantage, although in many instances their complete possibilities may not be recognized and appreciated.

At one time it was only the highly specialized shop that made use of press tools, but today every well-organized plant has adopted this method for the production of small and medium parts. It has become common practice to produce with press tools parts that were formerly manufactured from bar stock, castings, or forgings. Even in those industries where large articles are produced from sheet metals, a large percentage of the work and often the whole of it is accomplished in the press department by blanking, drawing, forming, or some other operation.

General Class of Dies.—When we speak of dies or press tools, we mean both punches and dies. Often we say a set of punches and dies, meaning a punch and die. While the die is usually fastened to the bed of the press and the punch carried above it in the traveling ram, this arrangement is not universal, there being many cases where marked operating advantages are secured by inverting the usual order and placing the die above and the punch below. In compound dies, we may have a die proper that carries a punch and a punch that has in it a die so that neither tool in the set as a whole can be distinguished as a die or a punch. Therefore, no matter what the relative positions of the punch or die may be, we may define the die as the tool that establishes the external form of the work and the punch as the corresponding internal member.

In considering press tools it is natural to think of sheet metal such as aluminum, brass, copper, iron, silver, steel, tin, and zinc. Other materials that are widely used are cardboard, fabrics, fiber, leather, mica, paper, plastics, and wood.

In grouping the various forms of press tools according to their action upon materials, we find that as a general rule they may be classified under the following form heads.

1. Tools that operate by cutting or "shearing" the metal. Blanking dies, shearing or cutoff dies, and notching dies are examples of this class. Included in this class also may be the following tools: piercing, shaving, and trimming dies.

2. Tools that shape the article by drawing the material, causing it to flow under tension. Cupping dies, drawing dies, bulging dies, and reducing dies are examples of this class.

3. Tools that by compressing, forcing, or squeezing the material cause it to flow into the desired form. Coining dies, extruding dies, and swaging dies are examples of this class.

4. Tools that manipulate the stock or blank already cut out, by some form of bending process. Simple and compound bending dies, forming dies, curling and wiring dies are of this class.

Blanking-die Action.—Of the many kinds of press tool in service, blanking dies are perhaps

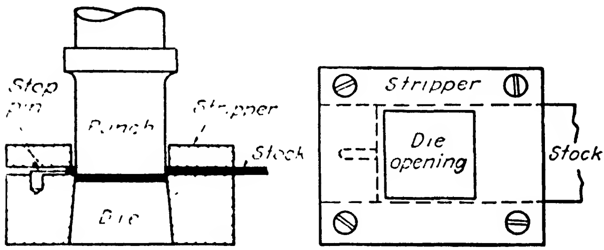


Fig. 89.

the most generally used of all, unless it be the piercing die, usually called in elementary form a punching die. The blanking die and the piercing die have characteristics that are almost identical in respect to their action on the metal. We have only to consider a piercing tool of fair diameter as compared with the outside diameter of the blank to recognize the fact that the piercing punch is merely an internal blanking tool whose purpose is to form an accurate opening in the stock by cutting out the blank, which is discarded as scrap from the punch just as the material around the outside of the blank is scrap from the blanking dies. The scrap from the piercing punch is called a "slug." A typical example of a simple blanking die is shown in Fig. 89.

The Cutting Edges.—Consider the metal to be blanked as very thin stock. It is then easily seen that the action of the edges of the punch in respect to the die is that of a pair of shear blades cutting a piece of paper or very thin metal. As the gauge of the stock becomes heavier (thicker), it is not quite so easy to recognize the cutting character of the press-tool cut-

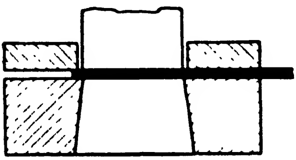


FIG. 90.

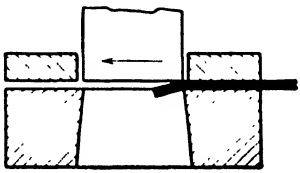


FIG. 91.

ting edges. If, instead of passing a strip of sheet metal across the face of the die as shown in Fig. 90, the end is merely extended over the die opening as shown in Fig. 91, then if the material is fairly heavy as compared with the size of the dies, the punch, when it strikes the surface of the strip at one side, will have a tendency to crowd away from the work in the direction indicated by the arrow and bend the end of the strip down as shown in Fig. 91 before the cut can start. If, however, the face of the punch is beveled as shown in Fig. 92, giving to its cutting edge a decided degree of shear, we can see at once that the sheet metal will be readily severed by the shearing action of the punch past the die edges. The action in Fig. 91 is referred to merely to show the effect of an ordinary flat end punch striking at one edge only on heavy-gauge material. In practice the end of the punch strikes the material squarely as shown in Fig. 90, cutting out the blank clean and sharp as the

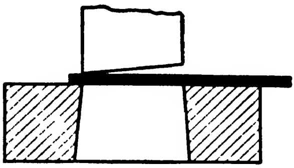


FIG. 92.

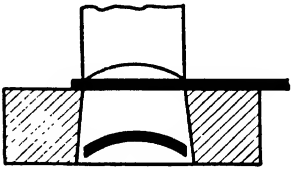


FIG. 93.

punch edges move down to the die. When heavy stock is used, the diemaker sometimes makes use of this shearing action to facilitate matters by providing the punch or the die with shear. In Fig. 93 is shown a punch the end of which is made slightly concave and produces a

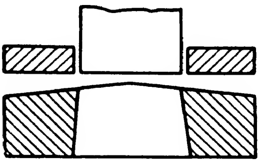


FIG. 94.

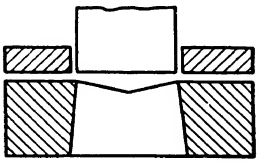


FIG. 95.

curved blank as shown. In Figs. 94 and 95 are shown dies provided with shear.

Piercing Tools.—In Fig. 96 is shown a type of die usually called a "punch die"—sometimes called a "button die." Such tools are often made of such small diameter that the punches must have their cutting ends supported by passing through a guide plate provided with holes in which the punches are a close fit. Radial or side-closing punches also are made for piercing work through the side. While it is somewhat natural to think of piercing dies as used chiefly

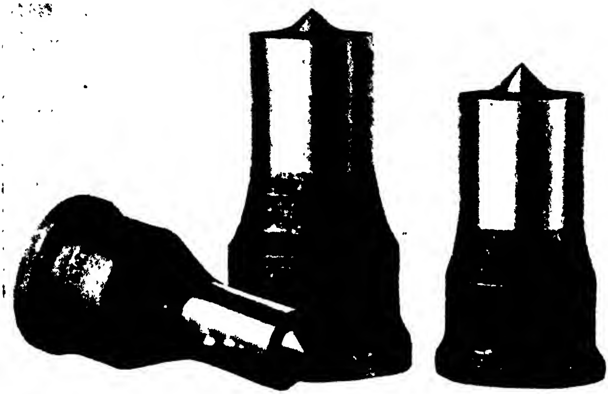


FIG. 96a.—(*From Pratt & Whitney Company.*)

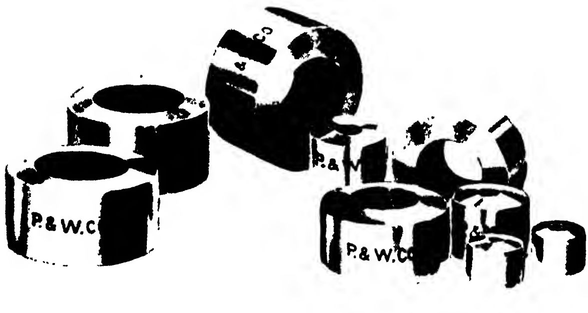


FIG. 96b.—(*From Pratt & Whitney Company.*)

for round holes, they are employed almost as frequently for making openings of other shapes, that is, square, oblong, curved, slotted, irregular, etc.

Shaving Tools.—Shaving dies are employed on press work that requires dimensional accuracy and sharp clean edges. These may be in the form of external shaving tools or internal shaving tools. External shaving tools are used for finishing around the outer edges of a blank produced in an earlier operation. Internal shaving tools are used with equally good results for internal cuts as in finishing the edges of a slot or opening of any shape produced by piercing tools.

They form one of the most important tools for assisting the diemaker to secure the best results as to accuracy of product. Usually they are made to remove a very thin shaving around the edge of the blank. The amount removed in shaving is only a few thousandths at the most, and sometimes re-shaving tools are used to follow the first shaving dies to ensure perfectly smooth and accurate edges on the blank. It is not always possible to produce a perfectly clean smooth contour, if in blanking too small an allowance is left for shaving. A reasonable allowance tends to increase the life of the dies by permitting them to take a clean cut around the blank without the possibility of very thin chips wedging in between the working faces of the punch and die.

Shaving operations are usually performed by placing a blanked piece into a locating device or "nest" on top of the shaving die. However, there are occasions where only a small portion of a piece requires shaving, as in some important working surface on a piece that is otherwise of only an ordinary degree of accuracy. The shaving operation in such cases may be accomplished before blanking by piercing out an opening in the stock around the surface to be shaved, then shaving the accurate portion, and following up by blanking out the piece. This method involves the use of a progressive die in which the three operations are carried on at once for as many pieces after the stock has been advanced through the dies.

Cutting-off or Parting Dies.– Cutting-off tools are constructed in a variety of ways and for

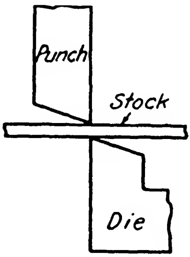


FIG. 97.

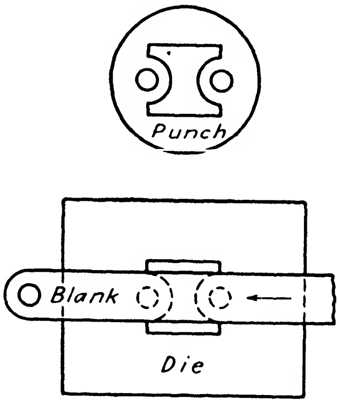


FIG. 98.

various classes of work. In their simplest form they are employed for severing flat, round, and other stock. They are often fitted with other tools into the same set of dies for cutting off a piece from the strip stock after certain other operations have been performed.

When made solely for cutting across a piece of metal, it is sometimes desirable to give their edges considerable shear to enable them to cut more freely, as shown in Fig. 97. Oftentimes they are shaped to give some special shape or curve to the end of the piece cut off and a similar shape to the leading end of the next piece. For this reason the tools are made symmetrical, and the same stroke that cuts off the last end of the one piece also shapes the first end of the next, which will in turn be cut off to shape when the stock is fed forward and the punch makes its next downward stroke. This is illustrated by Fig. 98.

This design balances the cut on opposite sides of the flat-ended punch and allows a single tool to operate on the ends of two pieces at the same time.

Trimming Dies.—Trimming dies are tools for producing sharp clean edges on certain articles

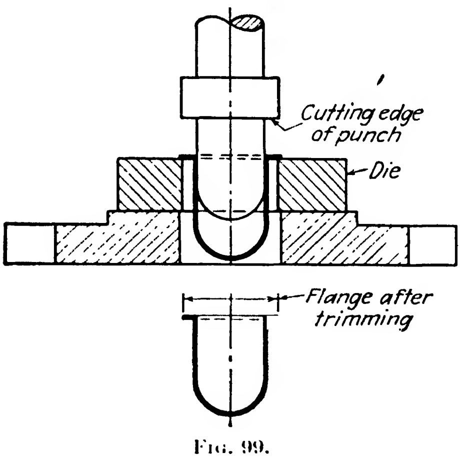


FIG. 99.

that have been produced by the blanking, forming, or drawing process where an irregular edge or a small amount of metal in the nature of a fin has been left which must be trimmed off to bring the work to the finished condition. In Fig. 99

is shown an example of such a tool applied to a drawn shell with the work ready for trimming and also with the edge of the flange finished by passing through the die. The punch is so formed as to enter the drawn shell and carry it through the die with the result that the edge of the flange is trimmed smoothly all the way around. The cutting action of the trimming tools is like that of an ordinary set of blanking dies (see Fig. 89) except that the punch is piloted to enter the drawn shell and support it during the movement past the cutting face of the die. Trimming dies are used extensively on flat blanks and for cutting out certain portions of the edges on some classes of work.

Hollow Cutting Tools.—"Drinking dies" is the shop term applied to a certain type of blanking

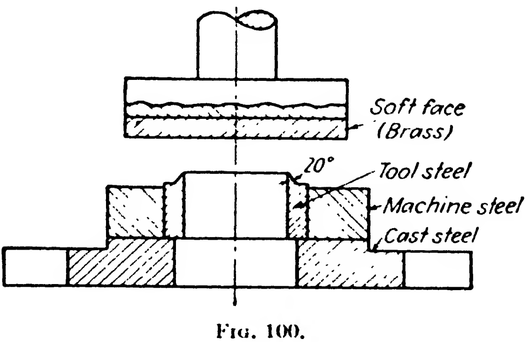


FIG. 100.

die that is used for materials softer than sheet metal. While in principle they are similar to some forms of blanking tools for metal work, they differ in having, usually, a rather keenly beveled cutting edge which cuts freely through the material, leather, paper, rubber, etc., in much the same way as a biscuit cutter passes through a sheet of dough.

In Fig. 100 is shown such a tool for cutting cardboard or leather. Their cutting edges, according to the material to be cut, are commonly sloped to an angle of 10 to 20 deg. Sometimes the interior is fitted with an ejecting device similar to that employed in metal-working dies for parts of the same general form.

Drawing Dies.—When a piece of flat stock, brass, steel, or other metal is pushed through a round die by means of a dull punch that cannot cut through the stock, we "draw" the blank into a cylindrical shell. If our punch and die have been of such a size as to allow for the full thickness of the metal between the punch and the interior of the die, the area of the drawn shell will be practically that of the blank from which it started. Thus, the operation accomplished is one of simple drawing without appreciable "stretching," of the original area, although the form of the work has passed through a decided change. In this change certain elements of the metal have been compressed locally to a degree, and others, directly opposite, stretched correspondingly.

Consider the disk *A* in Fig. 101. Here we have a thin metal blank of a given area, which we can force down through a drawing die *B* of the simplest form and produce the cup *C*. With certain proportions between the punch and die, the cup will have practically the same total area as the blank *A*. By a second drawing operation with smaller dies and punches, we can reduce the diameter of the cup and extend the length into a short shell *D*, and by a third drawing operation into a longer, smaller shell *E*. In performing the drawing operations above, it is usually possible if necessary to hold the original area of surface, and while the metal has been bent, stretched, and compressed, at certain portions, the general body of metal has not been stretched as a whole.

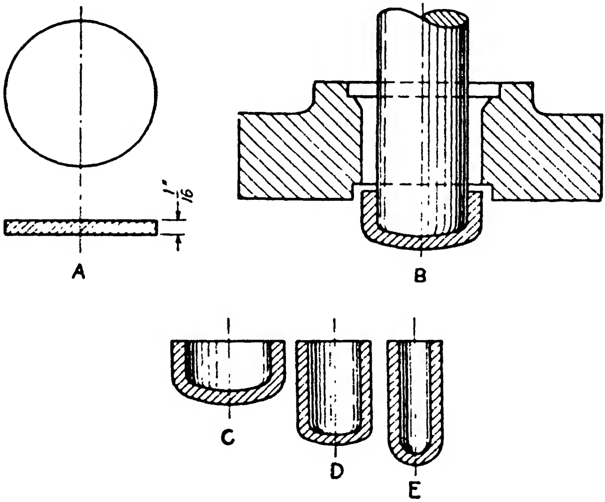


FIG. 101.

In the drawing process it is usually desired to stretch the original metal out as the operations proceed, gradually reducing the thickness proportionately to the length of the "draw" and generally bringing the complete shell to a very thin section as compared with the original blank. Now and then a metal is encountered that will not permit of marked if any reduction in thickness by the drawing method, and in such cases the stock is selected of the same thickness as is desired in the finished product. The flowing under tension of the metal as it is thinned down by being elongated in the drawing dies tends to harden the material. As the majority of the work requires a number of drawing operations to complete the piece, it is essential to anneal the shell between successive draws. Otherwise the stock will tear apart, or the bottom of the shell be punched out, an indication of the degree of tension that the material is subjected to under usual drawing conditions.

Action of Drawing Tools.—To prevent undue wrinkling, the draw should not be too deep. In a shallow draw the action of the punch and the opposing die surface tend to "iron" out the surface of the metal as it is drawn into the cup form and prevent wrinkling. As the cup is redrawn into longer shells, the reduction in each stage is limited, and with properly made tools undesirable wrinkling is avoided.

Where deep draws are required, and especially where blanks of the larger sizes are drawn, the simple tools shown in Fig. 101 will not suffice. They are replaced by double-action dies that operate in conjunction with a pressure pad, and the metal is ironed out as it is pulled under pressure down into the drawing die. Such tools are used in presses that have an outer slide for carrying the pressure pad, while the inner slide carries the regular drawing punch and die.

Combination dies and compound dies are used for work requiring shallow drawing. The work may be blanked, drawn to a depth, and pierced if so desired at one stroke of the press, the tools

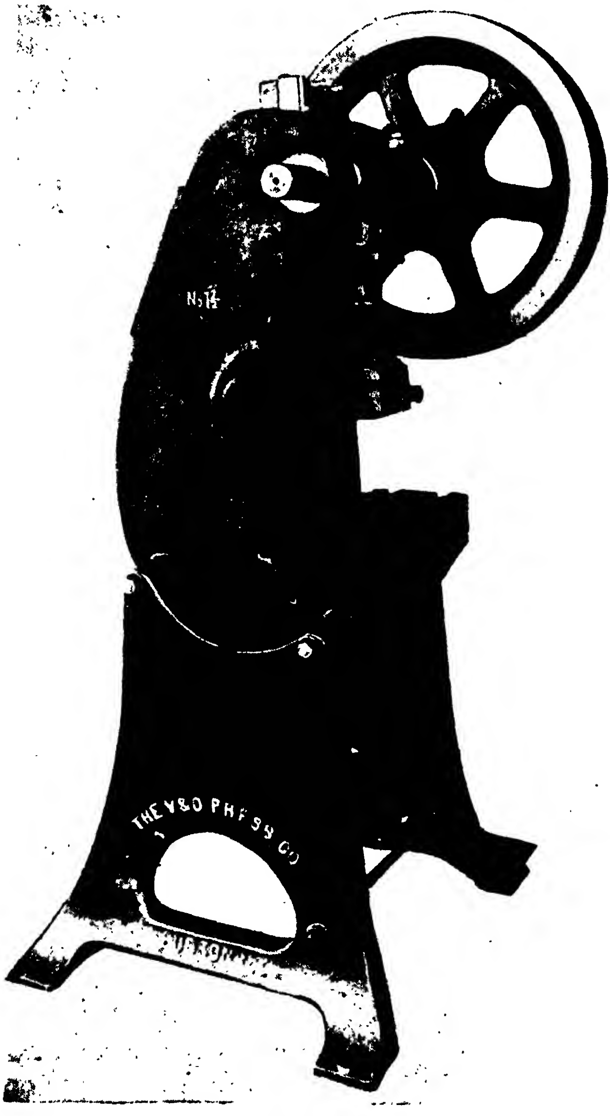


Fig. 102.

carrying their own pressure pads and being used in simple presses as shown in Fig. 102.

Curling and Wiring Dies.—Figure 103 represents a simple curling and wiring die for cylindrical work, where a curled edge with a wire stiffener is required on a shell, pail, or other utensil. The construction is practically the same whether or not the wire is required, except

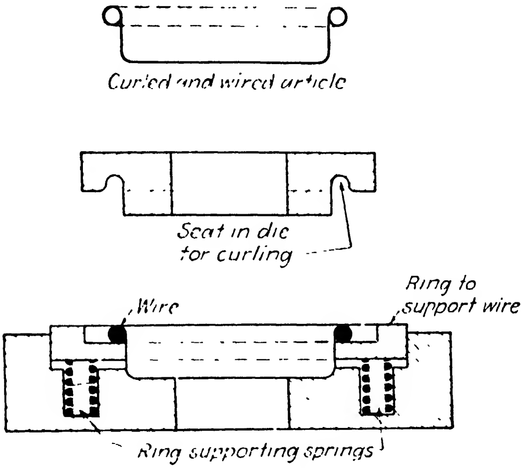


FIG. 103.

that the mouth of the die for the wire may be modified to suit the diameter of the wire.

Extruding Dies.—These tools are employed to force out from a flat surface of the blank a pin or other projection that may be needed for pivoting or riveting to some other member. The extruding punch action upon the metal is to cause it to flow ahead into the die. With properly designed tools a clean accurate projection pin is formed which has many advantages over the common rivet as generally employed. Figure 104 represents a simple extruding die.

Size Factor.—In Fig. 96 is shown a blanking die and punch for use on heavy stock such as boiler plates. In a simple tool of this kind it is immaterial which part is made first. Should this same type of tool be used for piercing a small hole in heavy stock, the tendency would be to spring the slender punch, and in such cases it is the

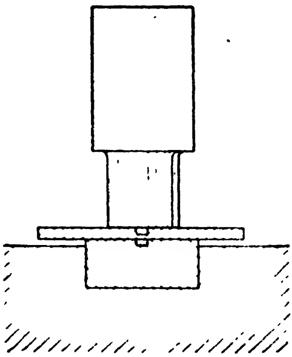


FIG. 104.

practice to support and guide the punch by the stripper. Assuming that $\frac{1}{8}$-in.-thick hard-rolled stock is to be pierced with a $\frac{5}{16}$-in.-diameter punch, the first step is to find the difference in diameters between the punch and die or the clearance.

Clearance.—One rule for clearance is to multiply the thickness of the stock in thousandths of an inch by 0.06; the result obtained is the difference in thousandths of an inch between the punch and die.

Whether to increase the size of the die or to decrease the size of the punch depends upon the nature of the stock and whether the blank must be of a certain diameter or whether the hole size must be maintained.

If the blank diameter must be 0.3125 in., then the die is made 0.3125 in.; whereas if the hole pierced is required to be 0.3125 in., then the punch is made 0.3125 in., and the clearance is obtained by increasing the size of the die. This is essential only where the diameter of the hole or of the blank is to be maintained in thousandths of an inch. Applying the rule given above, we find the clearance to be $0.06 \times 0.125 = 0.0075$ in.

CHAPTER X

TYPICAL STANDARD DIE SETS[1]

Classification Sheet for Class A Die.—Dies made under this heading (Fig. 105) shall be of high-grade workmanship and material suitable for average requirements calculated to give long service for large production and shall have the following important features: leader pins and bushings to be hardened and ground; automatic stop, first and second stop when necessary; dies to be of proper thickness best suited for the work, preferably not less than $\frac{7}{8}$ in.; cutting parts of the die to be within ± 0.002 in. limits where required; die to have $\frac{1}{2}$-deg. taper for entire thickness and to have bushings for all small perforating holes whenever possible; stripper plate to be machine steel fitted to punch and perforator; punch to be solid with base or whenever possible set in machine steel plate; perforators to be set in punch base or whenever possible in plate; punches and perforators to have spring shedder pins for light material; small punches and perforators to have steel backing plate or disks behind each perforator; clearance between punch and die and line up of all cutting edges shall be such as to produce blanks free from burrs. These specifications are to be followed for the average first-class die for large production.

[1] Abstracted from specifications of National Die & Special Tool Builders Association.

Classification Sheet for Class AA Die.—Dies made under this heading (Fig. 106) shall be of the highest grade in workmanship and material, calculated to give maximum service for large production requirements, and shall have the following important features: leader pins and bushings hardened, ground, and lapped; auto-

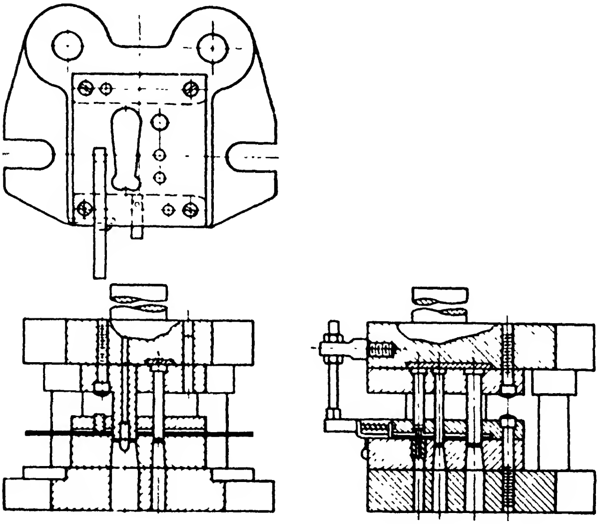


Fig. 105.

matic stop, first and second stop when necessary; die to be proper thickness best suited for the work, preferably not less than $\frac{7}{8}$ in.; cutting parts of die to be within ± 0.001-in. limits where required; die to have $\frac{1}{4}$ deg. taper for entire thickness or to be nearly straight for about $\frac{3}{8}$ to $\frac{1}{2}$ in. from top, followed by about 2 deg. taper for balance of thickness; die to have bushings for all small perforating holes whenever it is possible to use bushings; stripper plate to be machine steel, fitted closely to punch and to have hardened steel bushings for all small perforators whenever possible; punch to be solid with base; perforators to be set in base of punch when possible; perforators if smaller than $\frac{1}{4}$ in. diameter, shall be in hardened quills when possible; punches and perforators to have spring shedder

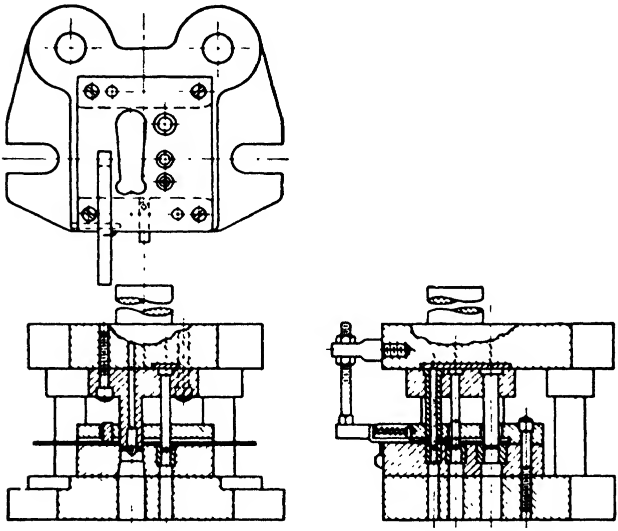


FIG. 106.

pins when possible; punch to have, if possible, hardened steel backing plate to take the thrust of perforators, or hardened round disks of at least three times the diameter of perforators behind each perforator; dowel pins and holes to be ground and lapped; clearance and line-up of cutting edges shall be such as to produce blanks smooth and free from burrs. These specifications are for extreme accuracy and large production requirements.

Classification Sheet for Class B Die.—Dies made under this heading (Fig. 107) shall be of first-class workmanship and material and shall have the following features: plain punch and die holders; plain flat or pin stop; die to be of proper thickness best suited for the work, preferably not less than $\frac{11}{16}$ in.; cutting parts of the die to be

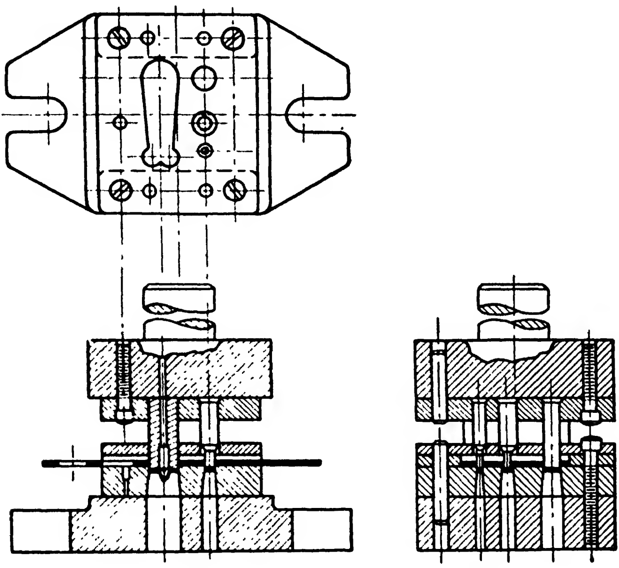


FIG. 107.

within ±0.003-in.-limits; die to have $\frac{3}{4}$ deg. taper for entire thickness; stripper plate to be machine steel and not contact with punch or perforator; punch to be set in machine steel plate or screwed to punch holder; perforator bushings to be set in die shoe casting whenever possible; clearance between punch and die to be such as to produce fairly smooth blanks, but small burrs are permissible. General appearance of the die is not important; refinements are not necessary. These specifications are to be followed for a die for medium production requirements or when dimensions are not important.

Classification Sheet for Class C Die.—Dies made under this heading (Fig. 108) shall be of first-class workmanship and material, but because of only small production requirements they shall

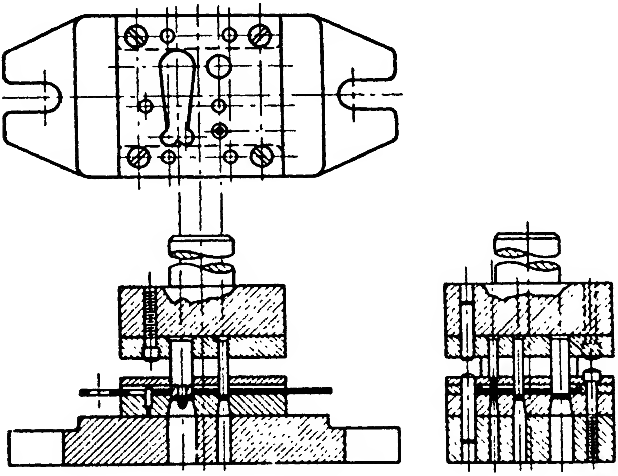


FIG. 108.

be known as temporary dies and shall have the following features: plain punch and die holders; plain flat or pin stop; die to be ½ in., or less, in thickness, according to requirements; cutting parts to be within ±0.005 in. or more limits; die to have ¾ to 1 deg. taper; stripper plate to be thin machine steel plate with strips underneath to form guide for stock and not to contact with punch or perforator; punch and perforators to be short and set in machine steel plate or held with screws.

CHAPTER XI

MACHINE TOOL ANALYSIS

VALUES OF K*

Material	Constant, p.s.i.
H.C. steel	[illegible]
Mild steel	270,000
Cast iron	132,000
Wrought iron	198,000
Bronze	110,000

VALUES OF y†

No. teeth	12	13	14	15	16	17	18	19
y	0.067	0.070	0.072	0.075	0.077	0.080	0.083	0.087
No. teeth	20	21	23	25	27	30	34	38
y	0.090	0.092	0.094	0.097	0.100	0.102	0.104	0.107
No. teeth	43	50	60	75	100	150	300	Rack
y	0.110	0.112	0.114	0.116	0.118	0.120	0.122	0.124

VALUES OF s†

Vel., ft. per min.	100 or less	200	300	600	900	1,200	1,800	2,400
Cast iron...	8,000	6,000	4,800	4,000	3,000	2,400	2,000	1,700
Steel.......	20,000	15,000	12,000	10,000	7,500	6,000	5,000	4,300

* Average values taken from Kent's "Mechanical Engineers' Handbook."

† Values of y and s for use in finding strength of gear teeth, using the Lewis formula, Kent's "Mechanical Engineers' Handbook."

POWER REQUIRED FOR LATHE WORK

Lathe-cutting Tools.—When a lathe tool removes metal from the parent material, there is a downward pressure exerted on the tool. This pressure (chip pressure) depends on several things, such as material being cut, shape and sharpness of tool, and size and shape of chip being removed. A simple and sufficiently accurate formula for pressure on a lathe tool is

$$P = KA \tag{57}$$

where P = chip pressure, lb.
A = sectional area of chip, sq. in.; it is product of depth of cut and feed per spindle revolution.
K = a constant depending on material being cut.

See the table on page 157 for values of K.

$$\text{Horsepower} = \frac{\text{load in pounds} \times \text{distance in feet per minute}}{33{,}000} \tag{58}$$

If the cutting speed and chip pressure are known, the formula for power required at the cutting edge is

$$HP = \frac{PS}{33{,}000} \tag{59}$$

where P = chip pressure, lb.
S = cutting speed, ft. per min.
HP = horsepower required at cutting edge.

By substituting in Eq. (59) from Eq. (57), we get a formula that is often more convenient:

$$HP = \frac{KAS}{33{,}000} \tag{60}$$

Cutting speed is the rate at which the cutting edge of the tool moves past the work or the work moves past the cutting edge of the tool. The formula for cutting speed is

$$S = \frac{\pi DN}{12} \tag{61}$$

where S = cutting speed, ft. per min.
D = diameter of work, in.
N = speed of spindle, r.p.m.

By solving Eq. (61) for N, the maximum spindle speed is obtained.

$$N = \frac{12S}{\pi D} \tag{62}$$

The horsepower transmitted by a leather belt as given in Marks' "Mechanical Engineers' Handbook" is

$$HP = \frac{(T_1 - T_2)V}{33{,}000} \text{ for belt 1 in. wide} \tag{63}$$

where $T_1 - T_2$ = difference in belt tensions, lb.
V = belt velocity, ft. per min.

Values of $T_1 - T_2$ as given have a wide variation. An average value of $T_1 - T_2$ as given in handbooks and by belting manufacturers is 42 lb. per in. of width for single belts and 70 lb. per in. of width for double belts.

$$HP = \frac{\pi \mathbf{D} NWP}{12 \times 33{,}000} \tag{64}$$

where **D** = diameter of pulley, in.
N = r.p.m. of same pulley.
P = effective pull, lb. per in. of width of belt = $T_1 - T_2$.

W = width of belt, in.
but $V = \pi DN$.
and WP = effective pull, lb. per width of belt.

then
$$HP = \frac{3\frac{1}{7} \times D \times N \times W \times 42}{12 \times 33{,}000} \qquad (65)$$
$$= \frac{DNW}{3{,}000} \quad \text{for single belting}$$
$$HP = \frac{3\frac{1}{7} \times D \times N \times W \times 70}{12 \times 33{,}000} \qquad (66)$$
$$= \frac{DNW}{1{,}800} \quad \text{for double belting}$$

The power determined in Eqs. (59) and (60) is the power required at the cutting tool. This must be divided by mechanical efficiency of the machine to obtain the power requirements of the machine.

For the spindle speeds with the back gears, the power that may be transmitted depends also on the strength of the back gearing. Power that may be transmitted through gearing may be determined by the Lewis formula as given in Kent's "Mechanical Engineers' Handbook."

$$W = spfy \qquad (67)$$

where W = safe working load on tooth, lb.
s = safe working stress of material, lb. per sq. in. at pitch circle speed.
p = circular pitch = 3.1416 divided by P.
f = face of gear, in.
y = strength factor, depending on the form and number of teeth.
P = diametral pitch.

A more convenient formula is

$$W = \frac{s\pi f y}{P} \tag{68}$$

Values for s and y are given in the table on page 157.

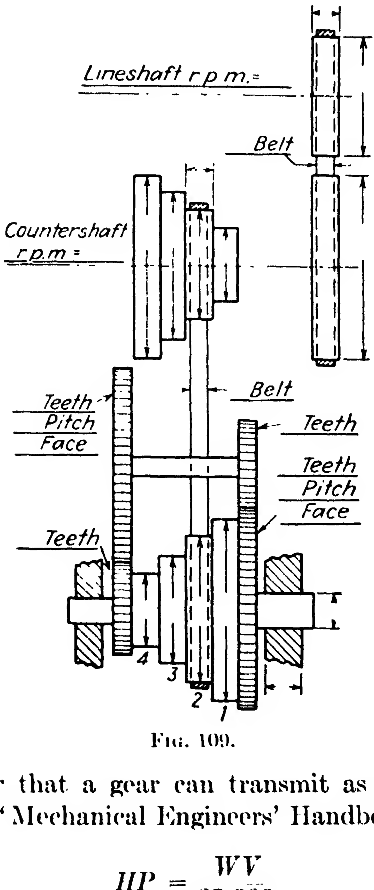


FIG. 109.

Power that a gear can transmit as given in Kent's "Mechanical Engineers' Handbook" is

$$HP = \frac{WV}{33{,}000} \tag{69}$$

POWER REQUIRED FOR LATHE WORK

Engine lathe No. M____

Swing over bed:____diameter

Maximum length:____ft.

Max. chip area for:
- Mild steel:____
- High-carbon steel:____
- Wrought iron:____
- Cast iron:____
- Bronze:____

	Direct					Back gears				
Spindle-speeds symbol	1	2	3	4	5	1	2	3	4	5
Spindle speeds, r.p.m..										
Horsepower at cutting tool.......										
Max. AS for mild steel, sq.in. × ft.per min....										
Max. AS for wrought iron, sq.in. × ft.per min......										
Max. AS for H.C. steel, sq.in. × ft.per min..........										
Max. AS for cast iron, sq.in. × ft.per min										
Max. AS for bronze sq.in. × ft.per min..										
Longitudinal and cross-feeds available, inches per spindle revolution										

where W = safe working load on gear tooth, lb.
V = pitch circle speed, ft. per min.

The pressure allowable on the front spindle bearing is equal to the allowable bearing pressure in pounds per square inch times the projected area of the spindle bearing.

The maximum chip area will depend on the feed mechanism's strength, and the machine must be checked to find its weakest part.

Actual time required to take a cut may be determined from the formula

$$T = \frac{L}{F \times N} \tag{70}$$

where T = time, min.
L = length of cut, in.
N = spindle speed, r.p.m.
F = feed, in. per revolution of spindle.

POWER REQUIRED FOR MILLING

Milling Machine.—The method of analysis of a milling machine is somewhat different from that of a lathe. It does not seem apparent from the many investigations that have been made to determine the power required for milling that any definite relationship exists between power and amount of metal removed for any given material when such factors as kind of cutter, cutting speed and feed, depth and width of cut are varied. From results given in Kent's "Mechanical Engineers' Handbook," it requires roughly 1 hp. to remove 1 cu. in. of cast iron and 2 hp. per cu. in. of mild steel. These are average values, and if an over-all efficiency for the milling machine is assumed as 75 per cent the power at the cutter would be

$$HP = 0.75AF \quad \text{for cast iron} \tag{71}$$
$$HP = 1.5AF \quad \text{for mild steel} \tag{72}$$
$$HP = 1.88AF \quad \text{for high-carbon steel} \tag{73}$$
$$HP = 2.23AF \quad \text{for alloy steel} \tag{74}$$
$$HP = 0.98AF \quad \text{for malleable} \tag{75}$$
$$HP = 0.38AF \quad \text{for brass and bronze} \tag{76}$$
$$HP = 0.24AF \quad \text{for aluminum} \tag{77}$$

where A = area of cut, sq. in.
F = feed of table, in. per min.

From these formulas the maximum value of AF may be found. It should be noted that in the case of the lathe S is in feet per minute, while in the milling machine F is feed in inches per minute.

Formulas for power that a leather belt may safely transmit as derived previously are

$$HP = \frac{\mathbf{D}NW}{3{,}000} \quad \text{for single belting} \tag{65}$$
$$HP = \frac{\mathbf{D}NW}{1{,}800} \quad \text{for double belting} \tag{66}$$

where W = width of belt, in.
$\mathbf{D}$ = diameter of either pulley, in.
N = r.p.m. of same pulley.

The power that is transmitted by the belt between the cone pulleys is found for the various spindle speeds by using Eqs. (65) and (66).

The cutting speed in milling is the peripheral speed of the cutter and is found by the formula

$$S = \frac{\pi DN}{12} \tag{61}$$

where S = cutting speed, ft. per min.
D = diameter of cutter, in.
N = speed of cutter, r.p.m.

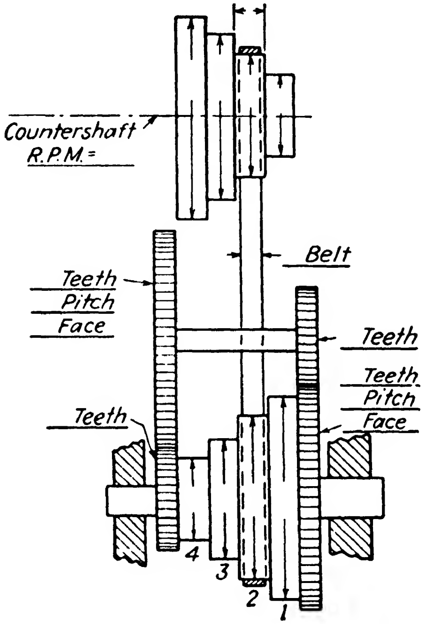


FIG. 110.

Solving Eq. (61) for N, the maximum spindle speed is obtained:

$$N = \frac{12S}{\pi D} \tag{62}$$

The time required to make a cut may be found from the formula:

$$T = \frac{L + A}{F \times N} \tag{78}$$

where T = time, min.
L = length of cut, in.

F = table feed, in. per revolution of cutter.
A = approach of the cutter to the work, in.

POWER REQUIRED FOR MILLING

Milling machine No. M____

Table working surface:____in. × ____in.

Face of column to harness brace in position:____in.

Longitudinal table feed:____in.

Traverse feed:____in.

Vertical feed:____in.

	Direct				Back gears			
Spindle-speeds symbol	1	2	3	4	1	2	3	4
Spindle speeds, r.p.m.								
Horsepower at milling cutter. . . .								
Max. AF for mild steel, cu. in. per min.								
Max. AF for wrought iron, cu. in. per min								
Max. AF for H.C. steel, cu. in. per min.								
Max. AF for cast iron, cu. in. per min.								
Max. AF for bronze, cu. in. per min.								

Longitudinal feeds available, inches per spindle revolution

POWER REQUIRED FOR DRILLING

Drilling.—The power available, strength of feed mechanism, etc., may be analyzed in the same way as other machine tools. The complicated relationships between drill diameter, torque, end thrust, power, speed, and feed are shown by the fractional exponents of the variables in the formulas to follow. The general relationships for torque and end thrust for medium-hard cast iron and steel as found by Dempster Smith and A. Poliakoff and given in Kent's "Mechanical Engineers' Handbook" are

$$T = 740D^{1.8}F^{0.7} \quad \text{for cast iron} \quad (79)$$
$$T = 1{,}640D^{1.8}F^{0.7} \quad \text{for steel} \quad (80)$$
$$P = 35{,}500D^{0.7}F^{0.75} \quad \text{for cast iron} \quad (81)$$
$$P = 35{,}500D^{0.7}F^{0.6} \quad \text{for steel} \quad (82)$$

where T = torque, ft.-lb. at drill.
P = end thrust, lb.
D = diameter of drill, in.
F = feed, in. per revolution of drill.

Horsepower required at the drill in terms of torque in foot-pounds is given by the formula

$$HP = \frac{2\pi TN}{33{,}000} \quad (83)$$

where T = torque, ft.-lb. at drill.
N = speed of drill, r.p.m.

Substituting the values of T from Eqs. (79) and (80), the following formulas are obtained:

$$HP = 0.141D^{1.8}F^{0.7}N \quad \text{for cast iron} \quad (84)$$
$$HP = 0.312D^{1.8}F^{0.7}N \quad \text{for steel} \quad (85)$$

The cutting speed of a drill is its peripheral speed in feet per minute. The cutting speed of a drill is found by the following formula:

$$S = \frac{\pi DN}{12} \qquad (61)$$

where S = cutting speed, ft. per min.
D = diameter of drill, in.
N = spindle speed, r.p.m.

Solving Eq. (61) for N we get the formula:

$$N = \frac{12S}{\pi D} \qquad (62)$$

Formulas (84) and (85) give the horsepower required to turn the drill. Power is required to feed the drill, but it is such a small part of the whole that it can safely be neglected in determining the power required in drilling.

Formulas (84) and (85) may be still further simplified by substituting the value of N from Eq. (62):

$$HP = 0.54SD^{0.8}F^{0.7} \quad \text{for cast iron} \qquad (86)$$

$$HP = 1.2SD^{0.8}F^{0.7} \quad \text{for steel} \qquad (87)$$

For a given drill press, power available (motor drive) is a constant, and for a given material, cutting speed is a constant, so that in any actual case maximum feed may be expressed in terms of drill diameter by solving Eqs. (86) and (87) for F:

$$\text{Max. } F = \left(\frac{HP}{0.54S}\right)^{\frac{10}{7}} \times \frac{1}{D^{\frac{8}{7}}} \quad \text{for cast iron} \qquad (88)$$

$$\text{Max. } F = \left(\frac{HP}{1.2S}\right)^{\frac{10}{7}} \times \frac{1}{D^{\frac{8}{7}}} \quad \text{for steel} \qquad (89)$$

POWER REQUIRED FOR DRILLING

Sliding head drill press

No. M____

Actual swing:____in.
Diameter of table:____in.
Vertical traverse of spindle:____in.
Maximum distance of spindle to table:____in.
Maximum distance of spindle to base:____in.
Traverse of table on column:____in.

For cast iron

D	1/4	3/8	1/2	5/8	3/4	7/8	1	1 1/8	1 1/4	1 3/8	1 1/2	1 5/8	1 3/4	2
F														
N														
NF														
h	0.075	0.113	0.150	0.188	0.225	0.263	0.301	0.338	0.376	0.414	0.451	0.488	0.526	0.601

For steel

D	1/4	3/8	1/2	5/8	3/4	7/8	1	1 1/8	1 1/4	1 3/8	1 1/2	1 5/8	1 3/4	2
F														
N														
NF														
h	0.075	0.113	0.150	0.188	0.225	0.263	0.301	0.338	0.376	0.414	0.451	0.488	0.526	0.601

D = drill diameter, in.
N = spindle speed, r.p.m.
F = spindle feed, in. per spindle revolution.
NF = spindle feed, in. per min.
h = cone height of drill, in.

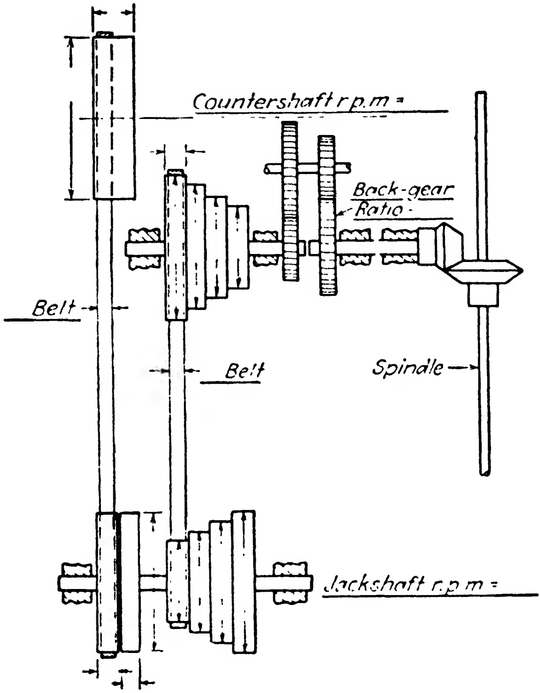


FIG. 111.

In the same way Eqs. (81) and (82) can be solved for F, to obtain maximum allowable feed, based on strength of feed mechanism, as follows:

$$\text{Max. } F = \left(\frac{P}{35,500}\right)^{\frac{4}{3}} \times \frac{1}{D^{1\frac{1}{5}}} \quad \text{for cast iron} \qquad (90)$$

$$\text{Max. } F = \left(\frac{P}{35,500}\right)^{\frac{5}{8}} \times \frac{1}{D^{\frac{7}{8}}} \quad \text{for steel} \qquad (91)$$

Time required to drill through any thickness of material is given by the formula:

$$T = \frac{L + h}{F \times N} \qquad (92)$$

where T = time, min.
L = thickness of work, in.
h = cone height of drill, in.
N = speed of drill, r.p.m.
F = feed of drill, in. per revolution.

POWER REQUIRED FOR PLANING

Planer Cutting Tools.—When a tool removes metal, there is a pressure between the tool and the work. This so-called "chip pressure" depends on many things, such as material being cut, shape and sharpness of tool, and shape and size of chip that is being removed. For the planer tool a simple and sufficiently accurate formula is

$$P = KA \qquad (57)$$

where P = chip pressure, lb.
A = sectional area of chip, sq. in.; it is product of depth of cut and feed per stroke.
K = a constant depending on material being cut.

Values for K are given in the table on page 157.

If the chip pressure and cutting speed are known, the power required at the cutting edge is

$$HP = \frac{PS}{33{,}000} \qquad (59)$$

where P = chip pressure, lb.
S = cutting speed, ft. per min.
HP = horsepower required at cutting edge.

POWER REQUIRED FOR PLANING

Planer No. M____ Table working surface: ____ in. × ____ in. Maximum height under crossrail: ____ in.	
Driving pulley speed, r.p.m.	
Horsepower available at cutting tool	
Cutting speed	
Max. A for H.C. steel	
Max. A for mild steel	
Max. A for wrought iron	
Max. A for cast iron	
Max. A for bronze	
Feeds available, in. per stroke	

Depth of cut, in.	Maximum feed per stroke				
	H.C. steel	Mild steel	Cast steel	Cast iron	Bronze
1/32					
1/16					
3/32					
1/8					
5/32					
3/16					
7/32					
1/4					

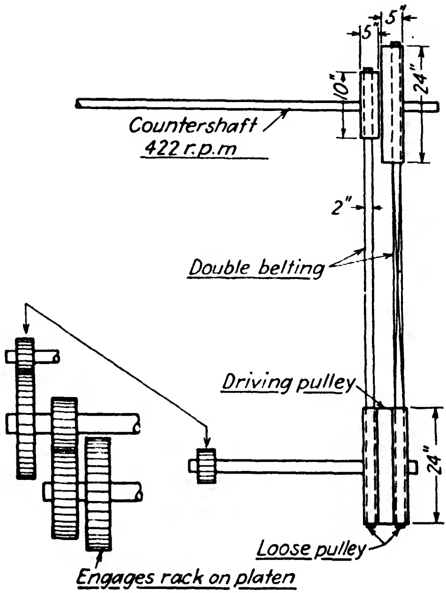


FIG. 112.

It is often more convenient to use the value of chip pressure from Eq. (57), from which

$$HP = \frac{KAS}{33{,}000} \qquad (60)$$

Cutting speed is the rate at which the cutting edge moves past the work or the work moves past the cutting edge.

$$S = N \times \left(L + \frac{L}{P}\right) \qquad (93)$$

where S = cutting speed, ft. per min.
N = number of strokes per minute.
L = length of the stroke, ft.

P = ratio of the return speed to the cutting speed.

Formulas for power that a leather belt may safely transmit, as derived previously, are

$$HP = \frac{\mathbf{D}NW}{3,000} \quad \text{or single belting} \qquad (65)$$

$$HP = \frac{\mathbf{D}NW}{1,800} \quad \text{for double belting} \qquad (66)$$

where W = belt width, in.
$\mathbf{D}$ = diameter of either pulley, in.
N = r.p.m. of same pulley.

EXPERIMENTS

1. HAND TOOLS AND THEIR USE

Object: To study the hand tools in general use in the machine shop and to chip, file, and scrape a flat surface.

Equipment:

Set of hammers.
Set of files (*for study purposes only*).
Cold chisel.
Cape chisel.
Ball-peen hammer.
10-in. hand bastard file.
Flat scraper.
Surface plate.
Prussian blue.
Object to be chipped, filed, and scraped.

"Machine Tool Operation," Vol. I, by Burghardt, for reference purposes.

Procedure: *First 1½ Hour.* Study the tools, paying particular attention to their characteristics: name, shape, size, cutting edges, cutting angles, teeth, etc. Answer all the questions given below.

Second 1½ Hour. Chip, file, and scrape one surface of the object supplied.

Questions

1. Make a sketch of each of the three kinds of machinist's hammers, and name their parts.

2. How is the size of a hammer designated?

3. What do you understand when you see the term "soft hammer"? When is it used?

4. Open-end wrenches are usually made with the handle at an angle with the jaws. What is the angle commonly used for open-end wrenches for square-head bolts? For hexagon-head bolts? Why?

5. Make sketches of the various kinds of chisels, and tell for what kind of work they are used.

6. What precautions should be taken in grinding a chisel?

7. What is meant by "choking the hammer"?

8. What precautions should be taken when chipping cast metals?

9. Should the workman look at the cutting edge or the head of the chisel when chipping? Why?

10. Make a sketch showing the difference between a single-cut and a double-cut file.

11. Make a sketch showing the difference between a hand bastard file and a flat bastard file.

12. What is a safe-edge file?

13. What is meant by cross-filing? Drawfiling?

14. What is a Vixen file?

15. Give a list of don't's in filing.

16. Give the reasons for scraping.

17. What tools are used in scraping a flat surface?

2. THE LATHE

Object: To study the lathe and its parts, and to become familiar with their functions. The selection of the proper feeds and speeds and their use in finding the time required for turning.

Equipment.

14-in. by 5-ft. lathe.
14-in. by 6-ft. lathe.
16-in. by 6-ft. lathe.

Revolution counter.

Lathe kit for lathe.

Right-hand side tool.

Center gauge.

Cast-iron cylinder $1\frac{3}{8}$ in. diameter by $8\frac{1}{2}$ in. long. "Machine Tool Operation," Vol. I, by Burghardt, for reference purposes.

Procedure: *First* $1\frac{1}{2}$ *Hour.* Answer questions 1 to 5.

Second $1\frac{1}{2}$ *Hour.* Study the lathes and their parts, giving special attention to the headstock, tailstock, carriage, and feed mechanism.

Questions

1. Make a table giving the spindle speeds for all steps of the cone pulley with and without the back gears engaged.

2. What is meant by the feed of a lathe tool? (See page 24.)

3. What is meant by cutting speed in regard to a lathe? (See page 10.)

4. Given a cast-iron cylinder $8\frac{1}{2}$ in. long by $1\frac{3}{8}$ in. diameter and using a cutting speed of 50 ft. per min. (cutter high-speed steel), calculate the time required to take one cut over the length of the cylinder. Use an actual feed that can be obtained on the lathe used.

5. True the live center, align the lathe centers, mount the work between centers and fasten the driving dog securely, adjust the toolholder, and find the time required to take one cut. The time found in 4 is the calculated time and does not take into consideration the time necessary to load and unload the machine.

Clean the Machine.

6. Make a sketch of the headstock of the lathe, showing the back gears. Give the use of the back gears.

7. Where is the live center located? Where is the dead center located?

8. Why are centers called "live" and "dead"?

9. Which is hard? Which is soft? Why?

10. How is the live center removed? How is the dead center removed?

11. What is meant by the swing of a lathe? The length?

12. How is the tailstock adjusted sideways? Why is it necessary first to loosen the clamping bolts?

13. What part of the carriage is called the saddle? The apron? The tool rest?

14. Can you move the carriage by hand when the split nut is closed? When the feed-control knob is tightened? When the carriage clamping screw is tightened? Give reasons.

15. Why does a machinist, before starting to work on a lathe, always try the carriage to make sure it runs freely?

16. How is motion transmitted from the main spindle to the feed rod? To the lead screw?

17. Describe the action of the split nut. Why is it called a split nut?

18. Make a sketch of the gear train from the main spindle to the feed rod and to the lead screw.

19. What is meant by change gears? By screw gear? By idler gear?

20. Show the difference between a simple train and a compound train of gears. (See pages 42 and 44.)

3. DRILLING MACHINES

Object: To study and to become familiar with the drilling machines in use in the machine shop. The selection of the proper feeds and speeds for drilling and reaming and their use in finding the time required for drilling and reaming. Practice in drilling and reaming.

Equipment.

Sensitive drill.
Two-spindle drill.
Vertical drill.
Radial drill.
Three-jaw work chuck.
No. 2 Titan quick-change drill chuck and three sleeves.
$\frac{63}{64}$-in. taper-shank twist drill.
1-in. taper-shank chucking reamer.
Revolution counter.
Material to be drilled.

Procedure: First Hour. Make a table showing the spindle speeds (r.p.m.) for all the steps of the cone pulley on a vertical drill with and without the back gears engaged. Select the proper cutting speed for the cutter and the material given, and find the r.p.m. of the spindle for a $\frac{63}{64}$ in. drill (see pages 10 to 14).

Use the spindle speed of the machine nearest to the speed determined above, and with a feed per revolution selected from the table of feeds on the machine, find the time required for drilling. (Speeds for reaming are the same as for drilling.) Estimate the time required to load and unload the machine. Actual drilling time plus time for loading and unloading the machine is the floor-to-floor time. Find the time required to complete one piece (see page 25).

Second Hour. Set up the machine. Drill and ream the material furnished, and make a note of the time required to complete one piece (see Appendix for grinding twist drills, pages 226 to 228).

Third Hour. Study the drilling machines, chucks, drills, reamers, etc. Answer all the questions given below.

Clean the Machine.

Questions

1. What is a sensitive drill press? Why is it so named?

2. What type of drilling machine is usually called simply a "drill press"? Why?

3. What is a radial drill?

4. How are drilling machines classified as to size?

5. In operation, the drill-press spindle revolves in the spindle sleeve. Does the sleeve revolve?

6. How does the sleeve move in its bearing?

7. How is the spindle supported at its upper end? How is it made to revolve? Why is the long keyway or "spline" cut in the spindle?

8. How is the drill press started and stopped? What is meant by loose pulley? Tight pulley? How is the loose pulley oiled?

9. How many direct speeds has the vertical drill press? How are they obtained? Has it any back gears?

10. How many speeds has this drill press?

11. When are the back gears used? Why?

12. How are the back gears engaged?

13. Is it safe to engage the back gears when the machine is running?

14. What is the use of the rack fastened to the spindle sleeve? What engages it?

15. How is the spindle caused to move in a downward direction by hand?

16. How many methods of feeding by hand are provided? Which is the most sensitive? Why? Which requires the least effort? Why?

17. Examine the mechanism that transmits motion from the spindle to the feed-rack pinion. How do you change the speed of the feed-rack pinion without changing the spindle speed?

18. How many changes may be made? How many power feeds has this machine?

19. How is the power feed engaged?

20. How does the worm and worm wheel operate to give slow motion to the feed-rack pinion?

21. How is the power feed released? Has it an automatic release?

22. How is the automatic release set for required depth?

23. What is the minimum feed? Maximum feed?

24. What kind of hole is in the spindle? Why?

25. How should a drill be removed from the spindle?

4. THE MILLING MACHINE

Object: To study the milling machine and its parts and to become familiar with their functions. The selection of the proper feeds and speeds and their use in finding the time required for milling. To practice milling the sides of a rectangular cast-iron block.

Equipment.

No. 2 plain milling machine.
Milling-machine vise.
Parallel.
6-in. piece $\frac{1}{2}$ in. diameter drill rod.
Lead hammer.
4-in. try square.
Ideal indicator.
Revolution counter.
$2\frac{1}{2}$ in. diameter milling cutter and arbor.

Procedure: First hour. Place the arbor and cutter in the milling machine, and determine the spindle speeds. Make a table showing the spindle speeds.

Select from the table on page 201 in the Appendix the proper cutting speed for the cutter and material given, and find the r.p.m. for cutter (see pages 10 to 15).

Using the r.p.m. of the machine nearest to that found above and assuming a feed of 3 in. per min., determine the feed per revolution of the cutter and the feed per tooth.

Find the time required to mill the sides of the cast-iron block. Estimate the time for loading and unloading the machine. The floor-to-floor time equals the machining time plus the loading and unloading time.

Second hour. Place the vise on the milling-machine table, and clamp securely with the bolts furnished. Use an indicator to check the vise setting. Clamp the block in the vise, making certain that it rests on the parallel. Make all other adjustments necessary, and mill all sides of the block. Check the time with that found above.

Third hour. Study the milling machine and its parts, paying particular attention to the spindle, back gears, method of driving the cutter, hand and power feeds, stops for releasing power feeds.

Answer all the questions given below.

Questions

1. Name five types of milling machines.

2. What is meant by a plain milling machine? A universal milling machine?

3. What is meant by the column of a milling machine? The knee? The saddle? The table? Table feeding mechanism?

4. Loosen the knee clamp, and lower the table. What is the advantage of the telescopic elevating screw?

5. How is the knee clamped to the column? What kind of gib is used? *Always see that the knee clamps are loosened before lowering the knee.* Why?

6. What is meant by longitudinal feed? By traverse feed? Vertical feed? How are they operated?

7. How many spindle speeds has this machine? Why are there so many? How is the reverse direction of the spindle obtained?

8. How is the saddle clamped to the knee?

9. How do you reverse the power feeds?

10. How are the back gears engaged? For what are they used?

11. Set the feed mechanism for the slowest feed and throw in the longitudinal feed. How far does the table move in 1 min.?

12. Arrange the mechanism for fastest feed. How far does the table move in 1 min.?

13. Is the feed of this milling machine independent of the spindle speed?

14. What is the purpose of having a telescopic feed shaft with universal joints?

15. Outline the proper procedure for milling all sides of a square or rectangular block.

Clean the Machine.

5. THE SHAPER

Object: To study the shaper, its construction, the functions of its several parts, and its value as a machine tool. The selection of proper feeds and speeds and their use in finding the time required for shaping. Practice shaping the sides of a rectangular cast-iron block.

Equipment.

16-in. shaper.
Tool holder with high-speed steel tool bit.
One 6-in. scale.
Lead hammer.
Pair of parallels.
4-in. try square.
Piece $\frac{1}{2}$ in. diameter drill rod.
Cast-iron block, $1\frac{1}{4}$ by 2 by $4\frac{1}{2}$ in.

Procedure: First hour. Find the cutting speed and time required for shaping the sides of a cast-iron block as outlined under Time Required for Shaping.

Second $1\frac{1}{2}$ hr. Study the shaper, its parts, and their functions, giving special attention to adjustment for length of stroke, placing the ram, feed mechanism, and back gears.

Answer all the questions given below.

Questions

1. What is a crank shaper?

2. In a crank shaper how does the bull wheel drive the vibrating arm?

3. How is the vibrating arm anchored to the base of the machine?

4. What is the use of the slot in the vibrating arm?

5. Where and how is the crankpin held?

6. How is the screw that moves the crankpin moved?

7. Why are bevel gears used?

8. Where is the check nut? What purpose does it serve?

9. How much movement has the vibrating arm when the crankpin is on center? Why? Away off center?

10. What is meant by the stroke of a shaper? Cutting stroke? Return stroke?

11. What is the reason for the slot in the top of the ram?

12. How is the vibrating arm connected to the ram?

13. How is the position of this connection changed?

14. How does it change the position of the stroke?

15. Why are bevel gears used between the handle and the screw?

16. Explain the quick return of the shaper.

17. What clamping bolts are loosened before adjusting the table vertically? How much should they be loosened?

18. What is the value of the table brace support?

19. How is the table moved horizontally? What is the movement called?

20. How are the different speeds obtained in the shaper? What is the need of having several speeds?

21. Are back gears provided in a shaper? Why?

22. What is the feed rocker arm?

23. How is motion transmitted from the rocker arm to the pawl?

24. Describe the operation of a ratchet and pawl.

25. How is the amount of feed changed?

26. How is the power feed reversed?

27. Have the feed arranged to cause the table to move from right to left, and note if the feed operates on the forward or return stroke. Reverse the feed and note if it operates on the same stroke as before.

28. What do you change to make the feed operate on the return stroke when the direction of the feed is reversed?

29. Why should the feed always operate on the return stroke?

30. Examine the head of the shaper. How is it fastened to the ram? How may it be swiveled?

31. What is meant by down feed in a shaper?

32. How many thousandths does one revolution of the down-feed screw handle move the slide?

33. How many graduations are there on the graduated collar? If you move the handle one graduation, how far will the slide move?

6. THE PLANER

Object: To study the planer, its construction, the functions of its parts, and its value as a machine tool. The selection of the proper feeds and speeds for planing and their use in finding the time required for planing.

Equipment.

Gray planer.
Work-holding fixture.
6-ft. rule.
Lead hammer.
12-in. monkey wrench.
Work to be planed.

Procedure: First 1½ Hour. Find the cutting speed and time for planing the work assigned. Read pages 15 to 19 and 30 to 32.

Second 1½ Hour. Study the planer, its parts and their functions, giving special attention to the driving, reversing, and feed mechanism, also the stops for regulating the length of stroke.

Answer the questions given below.

Questions

1. Make a sketch of the driving mechanism.

2. The bull wheel revolves much slower than the driving pulley. How do you explain this? Why is it necessary?

3. What is a rack? How is the platen moved?

4. Make a sketch of the belt-shifting mechanism.

5. What is the purpose of having a quick return of the platen?

6. Why are the holes in the platen reamed?

7. How is the length of the stroke changed?

8. Does the planer reverse at exactly the same point each stroke?

9. What are the features of the housings that give them strength and rigidity?

10. How are the front faces of the housings finished? Why?

11. How is the crossrail clamped to the housings? When is it clamped? When is it loosened? Why?

12. How is the crossrail raised or lowered? How is it adjusted to cut parallel to the platen?

13. What is the purpose of the saddle? Why is a gib placed between the saddle and the crossrail bearing surfaces?

14. When is the saddle binding screw used?

15. What is the difference between the swivel graduations and the graduations on the down-feed screw?

16. The down feed may be accomplished by turning either one of two handles. Where are these handles?

17. In the planer the feed operates at the beginning of the stroke and not during the stroke. How do you explain this? How would the feed operate if there were no friction stop?

18. Make a sketch of the feed mechanism.

19. How is the amount of feed controlled?

7. SMALL TOOLS AND PRECISION MEASURING INSTRUMENTS

Object: To become familiar with the small tools and precision measuring instruments in common use in the machine shop. Methods of adjusting, reading, and their use.

Equipment.

Steel scale.
Spring outside caliper.
Spring inside caliper.
Spring divider.
Firm-joint outside caliper.
Firm-joint hermaphrodite caliper.
Try square.
Two one-thousandth micrometers.
One ten-thousandths micrometer.
Micrometer depth gauge.
Set of inside micrometers.
Vernier caliper.
Vernier height gauge.
Vernier bevel protractor.
Small Tools Catalog.

Procedure: Study the various tools, size, shape, construction, and method of adjusting, paying particular attention to the proper method of using for making measurements.

Answer all the questions given below.

Questions

1. Make a sketch of the micrometer, and name its several parts.

2. Make a sketch showing the correct way to set an outside caliper for a scale measurement of $1\frac{1}{2}$ in. Inside caliper. Divider. Hermaphrodite caliper.

3. What is meant when a scale is said to have a No. 7 graduation?

4. What is the difference between a scale and a straightedge?

5. Why is the use of the folding wood rule objected to in the machine shop?

6. How should a caliper be held in relation to the work when making a measurement?

7. Should the work be calipered while it is running?

8. What accuracy can be expected with caliper measurements?

9. Are calipers efficient for measuring?

10. What is the method of adjusting a firm-joint caliper?

11. What part of an inch does one revolution of the thimble of a micrometer move the spindle? Why?

12. Into how many divisions is the sleeve or hub graduated? How are these divisions numbered?

13. Into how many divisions is the beveled edge of the thimble graduated? How numbered? Why?

14. What is the use of the ratchet or friction stop on the micrometer thimble?

15. Explain the difference in using a micrometer and a C clamp.

16. The main scale on the vernier has how many divisions per inch? How are they numbered? Why?

17. Into how many divisions is the vernier scale graduated? How are they numbered?

18. How many divisions on the scale correspond to the divisions on the vernier?

19. Each division on the scale equals what fractional part of an inch? How many thousandths of an inch?

20. What is the difference in length between a division on the scale and a division on the vernier?

21. Are verniers ever used with more or less than 25 divisions?

22. Make a sketch of a vernier showing the main scale having 50 divisions and the vernier scale having the correct number of divisions to make measurements to thousandths of an inch.

8. THE SPIRAL HEAD

Object: To study the construction of the universal spiral head, its parts and their functions; also its use for simple and differential indexing, and its use in conjunction with the milling machine for cutting spirals.

Theory, calculations, problems, and practice.

Equipment.

10-in. Universal spiral head and tailstock.
No. 2 Universal milling machine.
Cutter arbor.
No. 6, 10-pitch involute gear cutter.
$\frac{3}{4}$-mandrel.
Cast-iron gear blank.

Procedure: First hour. Study the spiral head giving special attention to its construction, its parts, and their functions and adjustments (see Fig. 39).

Answer the questions given below.

Questions

1. How is the spiral-head spindle rotated?

2. How many turns must the crank *E* make in order that the spindle make one complete turn?

3. How many teeth must worm gear *A* have if the worm *C* is single-threaded?

4. Can the worm *C* and worm gear *A* be thrown out of mesh? If so how?

5. What are the two small levers for on the left side of the spiral head?

6. How many index plates are furnished with the spiral head? Give the number of holes in the index circles of the various plates.

7. How is the crank *E* adjusted so that the pin *F* fits the holes in the different index circles? How is the index plate mounted on the spiral head?

8. What is meant by the sector? What is its use?

9. What is the spring washer, and for what is it used?

10. What bolts must be loosened in order that the spindle of the spiral head can be set at an angle?

11. Through how large an arc can the spindle be moved?

12. Where is the stop pin?

13. It is necessary to cut squares on some reamer shanks. How many turns of the crank are needed for each cut or side of the square?

14. A gear having 18 teeth is required. How many turns of the crank are required for each tooth?

15. A ratchet wheel having 65 teeth is required. How many turns of the crank are required for each tooth?

Procedure: Second hour, differential indexing.

1. Place an extension center in the spindle of the spiral head, and place equal gears on the spindle *S* and the worm shaft *W*. Complete the gear train by using one idler gear *D*.

2. Turn the crank forty times in a clockwise direction. In what direction does the plate move? How many complete turns does the plate make?

3. Add a second idler gear to the gear train, and again turn the crank forty times in a clockwise direction. In what direction does the plate move? How many complete turns does the plate make?

4. When the crank and plate turn in the same direction, the movement of the spindle is equal to what? When they move in opposite directions, the spindle movement is equal to what?

5. Read pages 82 and 90 of the text.

6. Find the plate to use and the change gears necessary to index for 121 teeth.

7. Make a sketch showing the arrangement of the gears, and give the number of teeth in each gear.

8. Make a sketch showing how motion is transmitted from the crank to the plate.

Procedure: Third hour, spiral cutting.

1. Mount the spiral head on the milling-machine table, and fasten it securely with the bolts supplied.

2. Place equal gears on the milling-machine feed screw and the worm shaft of the spiral head.

3. Remove the stop pin from the index plate, and turn the feed screw by hand.

4. When the feed screw is revolved, does the spiral-head spindle revolve?

5. Make a sketch showing how motion is transmitted from the feed screw to the spiral-head spindle.

6. How many turns of the feed screw are required to make the spindle revolve once?

7. How far does the table travel while the spindle makes one revolution?

8. What is meant by the lead of a milling machine? (See text—pages 90–94.)

9. What is the lead of the thread on the feed screw?

10. Calculate the lead on the machine.

11. What is meant by the lead of a spiral?

12. The lead of a spiral, the spiral angle, and the circumference of the work can be represented by a right triangle. If the diameter of the work is 1.800 in. and the spiral angle is 20 deg. 4 minutes, find the lead and the change gears necessary to cut this lead.

Make a sketch showing setup of gears.

Clean the Machine.

9. GAUGES AND GAUGING

Object: To study the types of gauges and their use in the production of interchangeable parts; also the use of the supermicrometer and precision gauge blocks for checking manufacturing and inspection gauges, intricate jigs and fixtures, and the accurate measurement of angles.

Equipment.

- ½-in. diameter plug gauge.
- 1.125 in. diameter plug gauge.
- 1.374 in. diameter plug gauge.
- 1.375 in. diameter plug gauge.
- ⅜-in. diameter by 16-thread plug thread gauge.
- 1.125 in. diameter ring gauge.
- 1.375 in. diameter ring gauge.
- Two taper ring gauges.
- 2¾-in. snap gauge.
- ¾-in. limit snap gauge.
- 1½ to 2 in. diameter adjustable-limit snap gauge.
- 1 in. diameter by eight-thread adjustable-limit snap thread gauge.
- Cast-iron plate with 30-deg. angle.
- 8- by 10-in. surface plate.
- Pair ⅝- by 1½-in. parallels.
- *V* block.
- One angle plate.
- Ground test cylinder.
- One 4-in. try square.
- Dial indicator.
- Set of wires for three-wire thread measurement.

Taper test gauge and standard.
Lead tester.
Pair of toolmaker's clamps.
Supermicrometer.
Set of Hoke precision gauge blocks.
One 5-in. sine bar.

Procedure: Read pages 105 to 119 and pages 127 to 131.

Answer all the questions given below.

Questions

1. Give three reasons why interchangeability is desirable in manufacturing.

2. In Fig. 58 show how the maximum and minimum clearances are determined.

3. Name eight classes of fits used in manufacturing.

4. In what class of fit do the mating parts offer interference?

5. What is the chief purpose of manufacturing by selective assembly?

6. How many sets of limits are there in selective-assembly manufacturing? Name them.

7. In Example 3 show how the desired allowance for selective fits is arrived at.

8. Add the intermediate length dimensions in Fig. 61. Do they equal the over-all dimension?

9. Add the intermediate dimensions, taking in account the minus tolerances. Do they equal the over-all dimension with the minus tolerance?

10. From the dimensions given, which is the most important?

11. In Fig. 63 show that *C* may vary from 0.030 in. instead of 0.010 in.

12. In Fig. 64 show that *C* may vary from 0.030 in. instead of 0.010 in.

13. What is a master gauge? What is the use of a master gauge? What is a work gauge? What is an inspection gauge?

14. Should the work gauge or the inspection gauge be the most accurate? Why?

15. Study the supermicrometer, and measure the diameters of the two large plug gauges. What is the difference in their diameters?

16. Is the hole in the ring gauge (1.375 in. diameter) the same size as the plug gauge?

17. If you were making a plug gauge and a ring gauge of the same size, which would you make first? Why?

18. Make setup and measure the effective diameter of the $\frac{3}{8}$ in. diameter by 16-thread plug thread gauge.

19. Is the effective diameter the same as that marked on the gauge? If not, how much does it vary?

20. How accurate is this method of measuring screw threads? Are the wires you used the best diameter wires? Could wires of another size be used?

21. What is the lead of a screw? (See page 35.)

22. Calculate the lead of the thread gauge.

23. Using the lead tester, check the variation in the lead of the plug thread gauge. What is the maximum variation in the lead? Is the variation all in the same direction?

24. Referring to Table X, page 221, select the diameters of the plug thread gauges that would be used to set the 1 in. by eight-thread adjustable snap thread gauge for a Class 4 fit.

25. Study the set of Hoke precision gauge blocks and note how the combinations are figured, how they are wrung together, and their care.

26. Using the precision gauge blocks, check the dimensions of the $\frac{3}{4}$-in. limit snap gauge. Is it oversize or undersize and how much?

27. Using the gauge blocks find the go and not-go dimensions of the Johansson adjustable limit snap gauge. What is the difference of the go and not-go dimensions? What is this called?

28. Using the caliper jaws furnished with the gauge blocks, measure the diameters of the $1\frac{1}{8}$- and $1\frac{3}{8}$-in. ring gauges. Are they oversize or undersize, and how much?

29. Referring to page 127, select the gauge blocks necessary to be used with the 5-in. sine bar for an angle of 30 deg. Check the angle of the cast-iron plate.

30. How accurate is the angle? Have we any other tool that we could use to check this angle? Would we be able to measure as accurately with it?

31. Determine the dimension to which the plugs on the master bar, furnished with the taper test gauge, are set so that the taper test gauge will have the same rate of taper as that specified on the blueprint furnished.

32. Can this gauge be set to duplicate other tapers?

33. What are its advantages over the ring type of taper gauge?

10. PRECISION METHODS OF LOCATING HOLES

Object: To become familiar with the methods in most general use for the precise location of holes in jigs, fixtures, and machine parts.

Equipment.

8- by 10-in. surface plate.
4- by 5-in. angle plate.
Set knife-edge straightedges.
Center tester.
One Universal test indicator.
Set toolmakers' buttons, 0.300 in. diameter.

Set toolmakers' buttons, 0.400 in. diameter.
Three 2 in. diameter disk gauges.
0.650 in. diameter disk gauge.
0.790 in. diameter disk gauge.
1.070 in. diameter disk gauge.
Plate $\frac{1}{4}$ by 2 by 5 in.
Plate $\frac{5}{8}$ by $2\frac{3}{4}$ by $4\frac{3}{8}$ in.
Plate $\frac{5}{16}$ by 5 by 6 in.

Procedure: Read Chap. I, "Accurate Tool Work" by Goodrich and Stanley giving particular attention to the Size Block Method, The Button Scheme, and The Disc Method.

Answer all the questions that follow, and perform all operations and setups in the order given.

Questions

1. How accurate can the center distance between holes in a jig located by the size block or the vernier method and drilled and reamed in a drill press be held? What must be the relation of the spindle and the table of the drill press when doing work of this class?

2. If the work were strapped to the face plate of a lathe and carefully "indicated up" with a center tester, would greater accuracy be assured? Why?

3. In the size-block method the accuracy of the location of the holes is dependent on the fulfillment of what four conditions?

4. What accuracy is obtainable by this method?

5. If much work is to be done by this method, what kind of commercial size blocks would you suggest be used?

6. When the work is too large to swing in the lathe, what other machine may be used?

7. Why are both the milling-machine table and knee secured?

8. When is the button scheme recommended in preference to the size-block method for work done in the lathe?

9. When is the button scheme recommended in preference to the size-block method for work done in the milling machine?

10. When should the disk method be used?

11. What accuracy can be obtained by this method?

12. What is its chief advantage over the size block or button scheme?

13. Make a sketch of the $\frac{1}{4}$- by 2- by 5-in. (approximate dimensions) plate, and show the exact length and width. Call it sketch 1.

Make a second sketch of the same plate showing vertical and horizontal center lines (sketch 2).

This plate is to have five holes, each 0.375 in. diameter, and the central hole is to be in the exact center of the plate. The other holes are to be in line with the center hole, and the distance between the holes is to be 1.000 in. Show the location of the holes in sketch 2, and give all dimensions necessary to three decimal places. Using a set of toolmakers' buttons, 0.300 in. diameter, locate the button for the central hole (see Figs. 7 and 9 on page 7 of "Accurate Tool Work"). What must the measurement across the buttons be if the center distance of the holes is 1.000 in.? Locate the buttons for the five holes. Check the measurement across each pair of buttons and the measurement across the center button and each of the other buttons. Check the buttons for alignment with a straightedge (see Fig. 70 on page 69 of "Accurate Tool Work"). Make sketch 3 showing the buttons in place. Why are the precautions stated above necessary? The plate now is ready to be strapped to the lathe face plate and the central button carefully "indicated up." The button is then removed, and the hole is first drilled and then bored to size (see Fig. 8 on page 7 of "Accurate Tool Work").

Why is the hole drilled first and then bored? Why is the central hole finished first? How can the center distances be checked for accuracy after all the holes have been bored?

14. Make three full-size sketches of Plate 2. The first sketch should show the location of the holes and have all necessary dimensions. The second sketch is to be similar to Fig. 22 on page 16 of "Accurate Tool Work." The third sketch should show the plate with the disks located. Find the diameter of the disks required by the method given on pages 15 and 17 of "Accurate Tool Work." Show all calculations. Check the diameters of the disks furnished, and locate the central disk first. Carefully locate the other two disks.

With the three disks carefully located, the plate is ready to be strapped to the lathe face plate. Make a sketch of Fig. 18 showing method of "indicating up" the disk prior to its removal for drilling and boring.

APPENDIX

TABLE I.—CUTTING SPEEDS

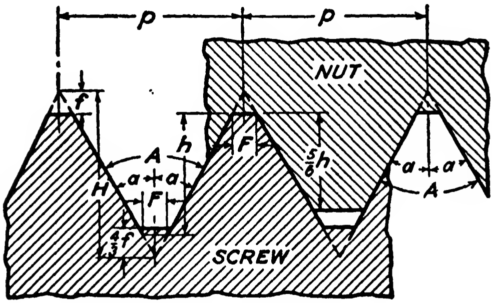
Material in cutter	Cutting speeds, in feet per minute, for material to be cut										
	Hard steel	Medium steel	Soft steel	Malleable iron	Chilled cast iron	Hard cast iron	Soft cast iron	Hard bronze	Bronze	Brass	Aluminum
Carbon steel	....	30– 40	30– 45	35– 50	....	....	30– 40	....	30– 60	40– 80	250– 500
High-speed steel.....	30– 50	50– 80	60– 90	70–100	....	30– 50	50– 80	30– 50	65–130	70–175	500–1,000
Super-High-speed steel	40– 70	60– 90	70–100	80–125	30– 50	40– 70	60–115	50– 70			
Stellite.....	....	....	....	115–150	40– 60	60– 90	90–130	....	100–160	150–250	800–1,500
Cemented carbide..	100–200	125–300	150–400	250–400	100–250	150–300	250–350	125–250	200–500	350–700	1,000–2,000

TABLE II.—DECIMAL EQUIVALENTS

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

TABLE III.—CONE HEIGHT OF DRILLS

D	$\frac{1}{4}''$	$\frac{3}{8}''$	$\frac{1}{2}''$	$\frac{5}{8}''$	$\frac{3}{4}''$	$\frac{7}{8}''$	$1''$
h	0.075	0.113	0.150	0.188	0.225	0.263	0.301
D	$1\frac{1}{8}''$	$1\frac{1}{4}''$	$1\frac{3}{8}''$	$1\frac{1}{2}''$	$1\frac{5}{8}''$	$1\frac{3}{4}''$	$2''$
h	0.338	0.376	0.414	0.451	0.488	0.526	0.601



National form of thread

n = Number of threads per inch

H = 0.866025p depth of 60° sharp V thread

h = 0.649519p depth of national form thread

$\frac{5}{6}h$ = 0.541266p

F = 0.125000p width of flat at crest and root of national form

f = 0.108253p, $\frac{1}{8}H$, $\frac{1}{6}h$ } depth of truncation

FIG. 113.—Standard thread systems.

TABLE IV.—NATIONAL COARSE-THREAD SERIES

Identification		Basic diameters			Thread data					
Sizes	Threads inch n	Major diameter D, in.	Pitch diameter E, in.	Minor diameter K, in.	Metric equivalent of major diameter, mm.	Pitch p, in.	Depth of thread h, in.	Basic width of flat $p/8$, in.	Minimum width of flat at major diameter of nut $p/24$, in.	Helix angle at basic pitch diameter s
1.........	64	0.073	0.0629	0.0527	1.854	0.01562	0.01015	0.00195	0.00065	4°31′
2.........	56	0.086	0.0744	0.0628	2.184	0.01786	0.01160	0.00223	0.00074	4 22
3........	48	0.099	0.0855	0.0719	2.515	0.02083	0.01353	0.00260	0.00087	4 26
4.........	40	0.112	0.0958	0.0795	2.845	0.02500	0.01624	0.00312	0.00104	4 45
5.........	40	0.125	0.1088	0.0925	3.175	0.02500	0.01624	0.00312	0.00104	4 11
6.........	32	0.138	0.1177	0.0974	3.505	0.03125	0.02030	0.00391	0.00130	4 50
8.........	32	0.164	0.1437	0.1234	4.166	0.03125	0.02030	0.00391	0.00130	3 58
10	24	0.190	0.1629	0.1359	4.826	0.04167	0.02706	0.00521	0.00174	4 39
12	24	0.216	0.1889	0.1619	5.486	0.04167	0.02706	0.00521	0.00174	4 1
¼	20	0.2500	0.2175	0.1850	6.350	0.05000	0.03248	0.00625	0.00208	4 11

$\frac{5}{16}$	18	0.3125	0.2764	0.2403	7.938	0.05556	0.03608	0.00694	0.00231	3 40
$\frac{3}{8}$	16	0.3750	0.3344	0.2938	9.525	0.06250	0.04059	0.00781	0.00260	3 24
$\frac{7}{16}$	14	0.4375	0.3911	0.3447	11.113	0.07143	0.04639	0.00893	0.00298	3 20
$\frac{1}{2}$	13	0.5000	0.4500	0.4001	12.700	0.07692	0.04996	0.00962	0.00321	3 7
$\frac{9}{16}$	12	0.5625	0.5084	0.4542	14.288	0.08333	0.05413	0.01042	0.00347	2 59
$\frac{5}{8}$	11	0.6250	0.5660	0.5069	15.875	0.09091	0.05905	0.01136	0.00379	2 56
$\frac{3}{4}$	10	0.7500	0.6850	0.6201	19.050	0.10000	0.06495	0.01250	0.00417	2 40
$\frac{7}{8}$	9	0.8750	0.8028	0.7307	22.225	0.11111	0.07217	0.01389	0.00463	2 31
1	8	1.0000	0.9188	0.8376	25.400	0.12500	0.08119	0.01562	0.00521	2 29
$1\frac{1}{8}$	7	1.1250	1.0322	0.9394	28.575	0.14286	0.09279	0.01786	0.00595	2 31
$1\frac{1}{4}$	7	1.2500	1.1572	1.0644	31.750	0.14286	0.09279	0.01786	0.00595	2 15
$1\frac{1}{2}$	6	1.5000	1.3917	1.2835	38.100	0.16667	0.10825	0.02083	0.00694	2 11
$1\frac{3}{4}$	5	1.7500	1.6201	1.4902	44.450	0.20000	0.12990	0.02500	0.00833	2 15
2	$4\frac{1}{2}$	2.0000	1.8557	1.7113	50.800	0.22222	0.14434	0.02778	0.00926	2 11
$2\frac{1}{4}$	$4\frac{1}{2}$	2.2500	2.1057	1.9613	57.150	0.22222	0.14434	0.02778	0.00926	1 55
$2\frac{1}{2}$	4	2.5000	2.3376	2.1752	63.500	0.25000	0.16238	0.03125	0.01042	1 57
$2\frac{3}{4}$	4	2.7500	2.5876	2.4252	69.850	0.25000	0.16238	0.03125	0.01042	1 46
3	4	3.0000	2.8376	2.6752	76.200	0.25000	0.16238	0.03125	0.01042	1 36

The National course threads are recommended for general use in engineering work, in machine construction where conditions are favorable to the use of bolts, screws, and other threaded components, and where quick and easy assembly of parts is desired.

TABLE V.—NATIONAL FINE-THREAD SERIES

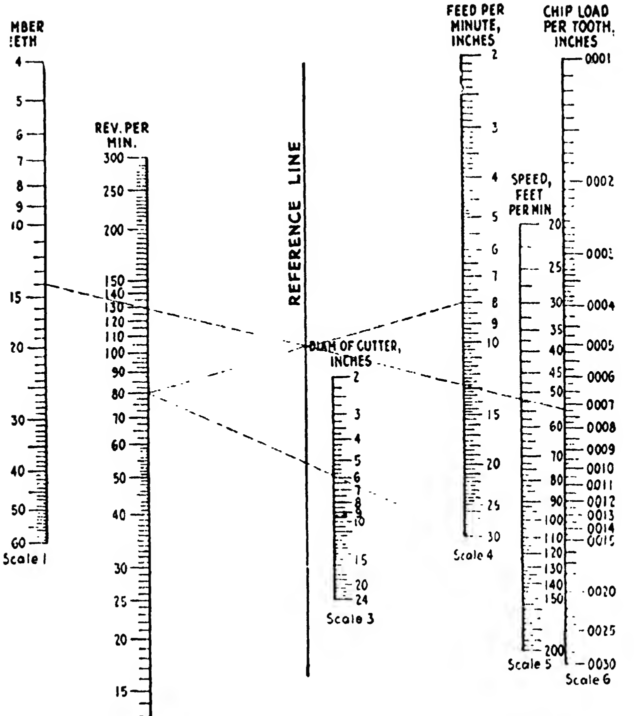
Identification		Basic diameters			Thread data					
Sizes	Threads per inch n	Major diameter D, in.	Pitch diameter E, in.	Minor diameter K, in.	Metric equivalent of major diameter, mm.	Pitch p, in.	Depth of thread, h, in.	Basic width of flat $p/8$, in.	Minimum width of flat at major diameter of nut $p/24$, in.	Helix angle at basic pitch diameter s
0	80	0.060	0.0519	0.0438	1.524	0.01250	0.00812	0.00156	0.00052	4°23′
1	72	0.073	0.0640	0.0550	1.854	0.01389	0.00902	0.00174	0.00058	3 57
2	64	0.086	0 0759	0 0657	2.184	0 01562	0.01015	0.00195	0.00065	3 45
3	56	0.099	0.0874	0.0758	2.515	0.01786	0.01160	0.00223	0.00074	3 43
4	48	0.112	0.0985	0.0849	2.845	0.02083	0.01353	0.00260	0.00087	3 51
5	44	0.125	0 1102	0 0955	3 175	0.02273	0.01476	0.00284	0.00095	3 45
6	40	0.138	0 1218	0 1055	3 506	0.02500	0.01624	0.00312	0.00104	3 44
8	36	0.164	0.1460	0 1279	4 166	0 02778	0.01804	0.00347	0.00116	3 28
10	32	0 190	0 1697	0.1494	4 826	0 03125	0.02030	0.00391	0.00130	3 21
12	28	0 216	0 1928	0 1696	5 486	0 03571	0.02320	0.00446	0.00149	3 22
1/4	28	0 2500	0.2268	0 2036	6 350	0 03571	0 02320	0.00446	0.00149	2 52
5/16	24	0 3125	0.2854	0 2584	7 938	0.04167	0.02706	0.00521	0.00174	2 40

TABLE V.—NATIONAL FINE-THREAD SERIES.—(*Continued*)

Identification		Basic diameters			Thread data					
Sizes	Threads per inch n	Major diameter D, in.	Pitch diameter E, in.	Minor diameter K, in.	Metric equivalent of major diameter, mm.	Pitch p, in.	Depth of thread, h, in.	Basic width of flat $p/8$, in.	Minimum width of flat at major diameter of nut $p/24$, in.	Helix angle at basic pitch diameter ψ
3/8	24	0.3750	0.3479	0.3209	9.525	0.04167	0.02706	0.00521	0.00174	2 11
7/16	20	0.4375	0.4050	0.3725	11.113	0.05000	0.03248	0.00625	0.00208	2 15
1/2	20	0.5000	0.4675	0.4350	12.700	0.05000	0.03248	0.00625	0.00208	1 57
9/16	18	0.5625	0.5264	0.4903	14.288	0.05556	0.03608	0.00694	0.00231	1 55
5/8	18	0.6250	0.5889	0.5528	15.875	0.05556	0.03608	0.00694	0.00231	1 43
3/4	16	0.7500	0.7094	0.6688	19.050	0.06250	0.04059	0.00781	0.00260	1 36
7/8	14	0.8750	0.8286	0.7822	22.225	0.07143	0.04639	0.00893	0.00298	1 34
1	14	1.0000	0.9536	0.9072	25.400	0.07143	0.04639	0.00893	0.00298	1 22
1 1/8	12	1.1250	1.0709	1.0167	28.575	0.08333	0.05413	0.01042	0.00347	1 25
1 1/4	12	1.2500	1.1959	1.1417	31.750	0.08333	0.05413	0.01042	0.00347	1 16
1 1/2	12	1.5000	1.4459	1.3917	38.100	0.08333	0.05413	0.01042	0.00347	1 3

The National fine threads are recommended for general use in automotive and aircraft work, for use where the design requires both strength and reduction in weight, and where special conditions require a fine thread.

CHART FOR CALCULATING SPEEDS AND FEEDS OF SOLID AND INSERTED-BLADE • MILLING CUTTERS •



To obtain table travel (feed per minute), connect load per tooth on Scale No. 6 and number of teeth per cutter on Scale No. 1 with a straight line. Then connect R.P.M. spindle on Scale No. 2 through the point where the first line crossed the refer- ce line and carry this over to Scale No. 4, which gives the feed in inches per inute.

o obtain chip load per tooth, connect feed per minute Scale No. 4 with R.P.M., :ale No. 2. Then connect number of teeth Scale No. 1 through point where first ıe crossed the reference line and read chip load per tooth on Scale No. 6.

o obtain speed in feet per minute connect R.P.M. Scale No. 2 with diameter of ıtter Scale No. 3 and read speed on Scale No. 5.

o obtain R.P.M. connect speed Scale No. 5 with diameter of cutter Scale No. 3 ıd read R.P.M. on Scale No. 2.

Fig. 114.

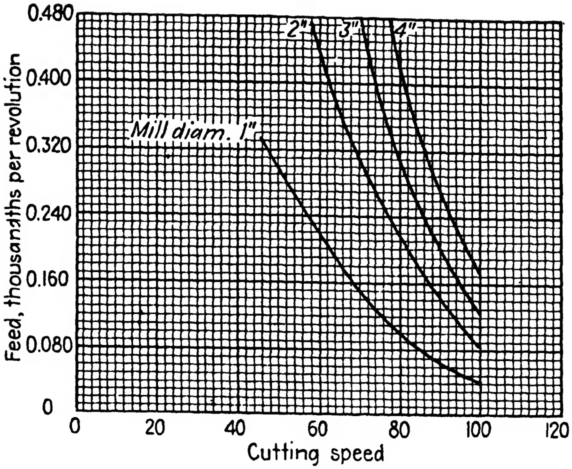


FIG. 115.—Spiral mills. Roughing $\frac{1}{16}$ in. deep, cast iron.

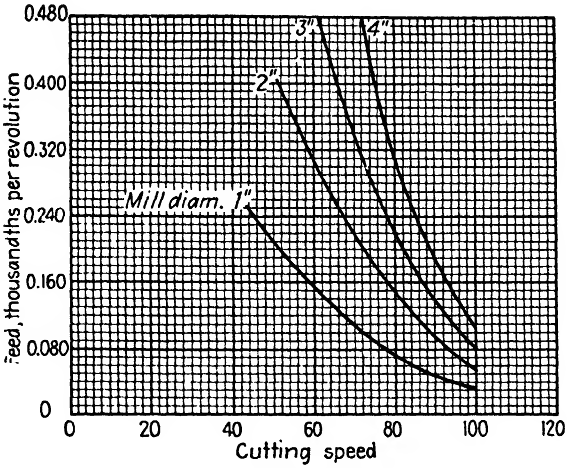


FIG. 116.—Spiral mills. Roughing $\frac{1}{8}$ in. deep, cast iron.

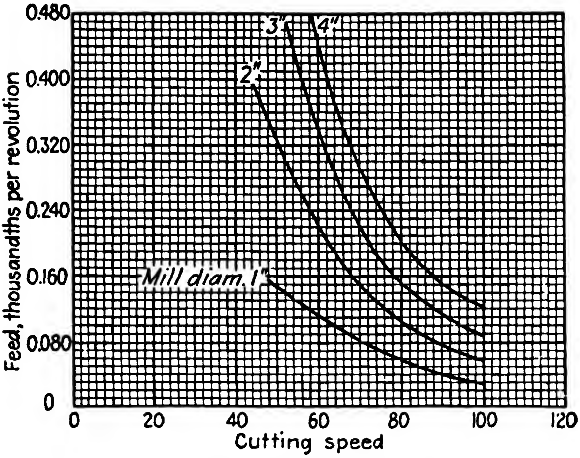


Fig. 117.—Spiral mills. Roughing $\frac{3}{16}$ in. deep, cast iron.

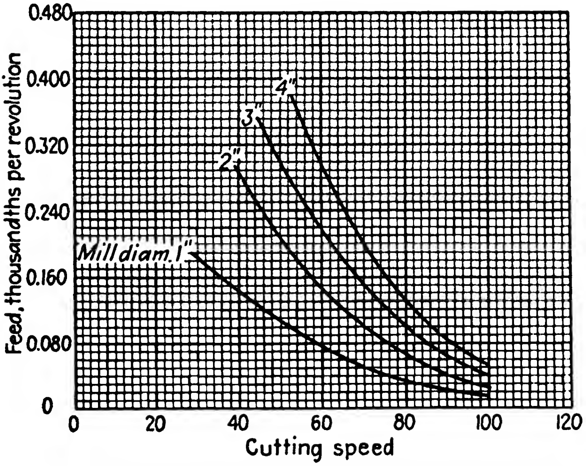


Fig. 118.—Spiral mills. Roughing $\frac{1}{4}$ in. deep, cast iron.

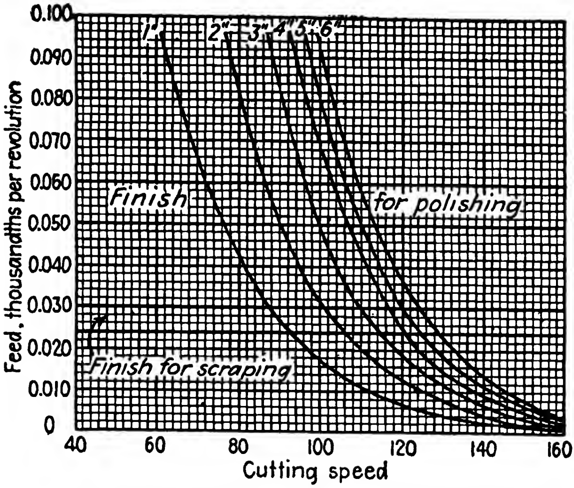


FIG. 119.—Spiral mills. Finishing $\frac{1}{64}$ in. deep, cast iron.

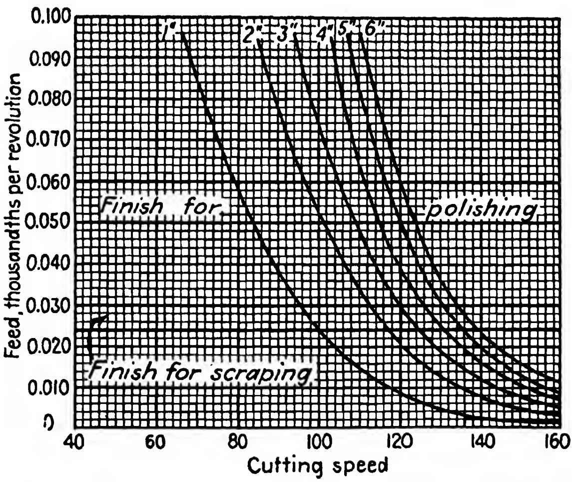


FIG. 120.—Spiral mills. Finishing $\frac{1}{32}$ in. deep, cast iron.

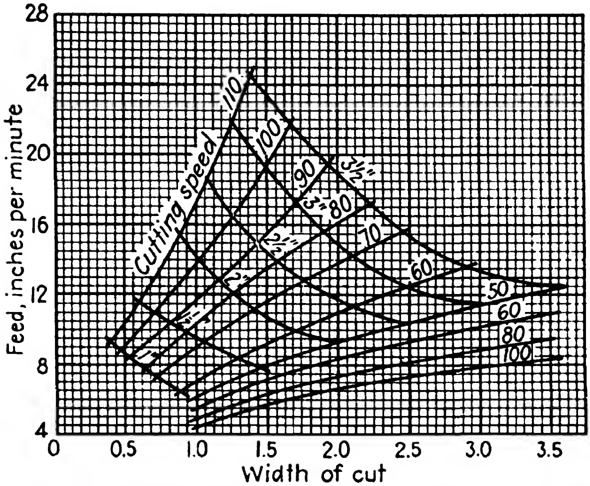


FIG. 121.—Shell end mills. Broken corners. Roughing, cast iron $\frac{1}{8}$ to $\frac{3}{16}$ in. deep.

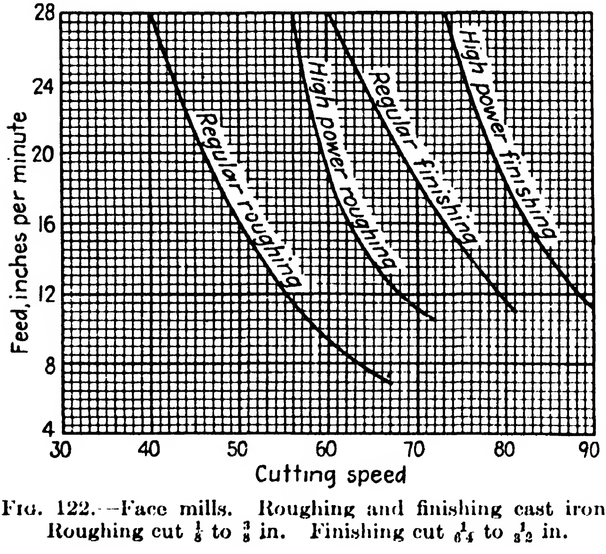


FIG. 122.—Face mills. Roughing and finishing cast iron. Roughing cut $\frac{1}{8}$ to $\frac{3}{8}$ in. Finishing cut $\frac{1}{64}$ to $\frac{1}{32}$ in.

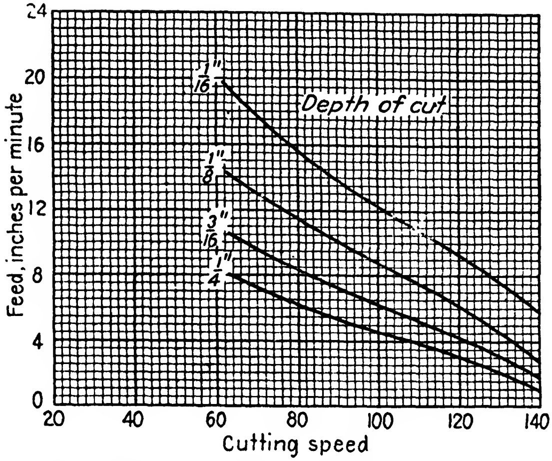


FIG. 123.—Spiral mills. Machinery steel. Stream lubrication.

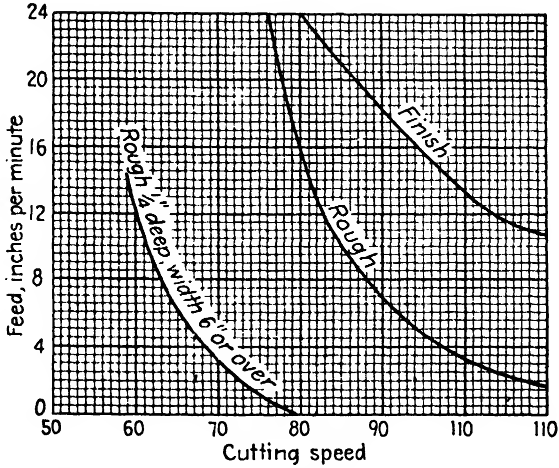


FIG. 124.—Special mills. Steel castings and machinery steel Width of cut, ½ to ¾ the diameter of cutter.

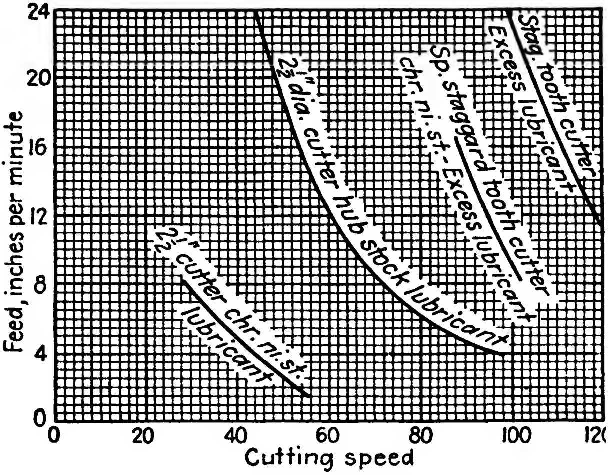


FIG. 125.—Keywaying. Machinery steel. Stream lubrication.

A.S.M.E. STANDARDS FOR METAL FITS

Allowance is the minimum clearance space intended between mating parts. It represents the condition of the tightest permissible fit.

Tolerance is the amount of variation permitted in the size of a part, or the total variation permissible in a given dimension.

Limit means the extreme permissible dimension of a part, or the variation allowable under or over the given dimension.

Fits are classified as loose, free, medium, snug, wringing, tight, medium force, heavy force, shrink.

Loose Fit.—This fit provides for a large allowance giving considerable freedom and is used where accuracy is not essential. The factor for calculating for a loose fit is 0.0025.

Free Fit.—This is for running fits with speeds of 600 r.p.m. or over and journal pressures of 600 lb. per sq. in. or over. There is a closer allowance here than for a loose fit. The factor for calculating the allowance is 0.0014.

Medium Fit.—This is used for parts revolving easily, for speeds under 600 r.p.m. and with journal pressures less than 600 lb. per sq. in. This is also applied to sliding parts and is the largest allowance for freedom consistent with accuracy. The factor for calculating the allowance is 0.0009.

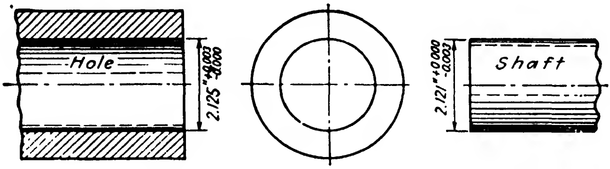


FIG. 126.—Example of loose fit.

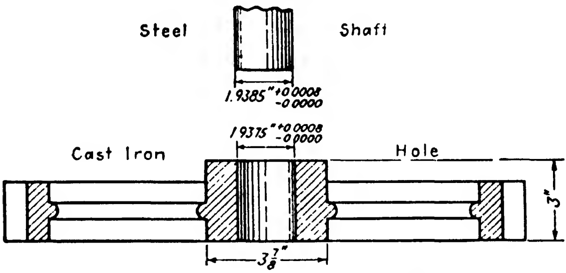


FIG. 127.—Cast-iron gear and steel shaft. Example of medium force fit.

Snug Fit.—This calls for zero allowance and is the closest fit that can be assembled by hand without appreciable pressure. It will not rotate easily, and no shake is permissible. A snug fit is not intended to move freely under a load.

Wringing Fit.—This is a metal-to-metal contact with no negative allowance. It allows for no movement and is assembled with slight pressure. Wringing fits are not usually interchangeable.

Tight Fit.—This is a wringing fit with a slight negative allowance. This fit is for parts permanently assembled or subject to pressure. It is much used in ordnance work. The factor for calculation of interference of metal is 0.00025*d*.

Medium Force Fit.—This is for permanently assembled parts, but subject to disassembly without severe pressure. The fit is the tightest possible for cast iron or parts where internal stress will be detrimental. The formula for calculating the interference of metal is 0.0005*d*.

Heavy Force Fit.—This is used for steel holes where the metal can be highly stressed without exceeding its elastic limit. Parts united by force fit form one unit without other means of holding. The interference of metal is calculated as 0.001*d*.

Shrink Fit.—This is the heavy force fit applied to larger parts where a force fit is impractical, such as for locomotive wheel tires. A definite negative allowance is given, and the outer part is expanded by heat before assembly.

TABLE VIII.—COARSE-THREAD SERIES—MEDIUM FIT
Screw Sizes, Inches

Size	Threads per inch	Major diameter		Pitch diameter*		Maximum minor diameter
		Maximum*	Minimum	Maximum†	Minimum	
1	64	0.0730	0.0692	0.0692	0.0615	0.0538
2	56	0.0860	0.0820	0.0744	0.0729	0.0641
3	48	0.0990	0.0946	0.0855	0.0839	0.0734
4	40	0.1120	0.1072	0.0958	0.0941	0.0813
5	40	0.1250	0.1202	0.1088	0.1071	0.0943
6	32	0.1380	0.1326	0.1177	0.1158	0.0997
8	32	0.1640	0.1586	0.1437	0.1418	0.1257
10	24	0.1900	0.1834	0.1629	0.1605	0.1389
12	24	0.2160	0.2094	0.1889	0.1865	0.1649
1/4	20	0.2500	0.2428	0.2175	0.2149	0.1887
5/16	18	0.3125	0.3043	0.2764	0.2734	0.2443
3/8	16	0.3750	0.3660	0.3344	0.3312	0.2983
7/16	14	0.4375	0.4277	0.3911	0.3875	0.3499
1/2	13	0.5000	0.4896	0.4500	0.4463	0.4056
9/16	12	0.5625	0.5513	0.5084	0.5044	0.4603
5/8	11	0.6250	0.6132	0.5660	0.5618	0.5135
3/4	10	0.7500	0.7372	0.6850	0.6805	0.6273
7/8	9	0.8750	0.8610	0.8028	0.7979	0.7387
1	8	1.0000	0.9484	0.9188	0.9134	0.8466
1 1/8	7	1.1250	1.1080	1.0322	1.0263	0.9497
1 1/4	7	1.2500	1.2330	1.1572	1.1513	1.0747
1 1/2	6	1.5000	1.4798	1.3917	1.3846	1.2955
1 3/4	5	1.7500	1.7268	1.6201	1.6119	1.5046
2	4 1/2	2.0000	1.9746	1.8557	1.8468	1.7274
2 1/4	4 1/2	2.2500	2.2246	2.1057	2.0968	1.9774
2 1/2	4	2.5000	2.4720	2.3376	2.3279	2.1933
2 3/4	4	2.7500	2.7220	2.5876	2.5779	2.4433
3	4	3.0000	2.9720	2.8376	2.8279	2.6933

* Basic diameter.

† The tolerances are cumulative and include errors of lead and angle.

TABLE IX.—COARSE-THREAD SERIES—MEDIUM FIT
Nut Sizes, Inches

Size	Threads per inch	Min.* major diam.†	Pitch diameter		Minor diameter	
			Min.*	Max.	Min.	Max.
1	64	0.0730	0.0629	0.0643	0.0561	0.0578
2	56	0.0860	0.0744	0.0759	0.0667	0.0686
3	48	0.0990	0.0855	0.0871	0.0764	0.0787
4	40	0.1120	0.0958	0.0975	0.0849	0.0876
5	40	0.1250	0.1088	0.1105	0.0979	0.1006
6	32	0.1380	0.1177	0.1196	0.1042	0.1076
8	32	0.1640	0.1437	0.1456	0.1302	0.1336
10	24	0.1900	0.1629	0.1653	0.1449	0.1494
12	24	0.2160	0.1889	0.1913	0.1709	0.1754
1/4	20	0.2500	0.2175	0.2201	0.1959	0.2013
5/16	18	0.3125	0.2764	0.2794	0.2524	0.2584
3/8	16	0.3750	0.3344	0.3376	0.3073	0.3141
7/16	14	0.4375	0.3911	0.3947	0.3602	0.3679
1/2	13	0.5000	0.4500	0.4537	0.4167	0.4251
9/16	12	0.5625	0.5084	0.5124	0.4723	0.4813
5/8	11	0.6250	0.5660	0.5702	0.5266	0.5364
3/4	10	0.7500	0.6850	0.6895	0.6417	0.6526
7/8	9	0.8750	0.8028	0.8077	0.7547	0.7667
1	8	1.0000	0.9188	0.9242	0.8647	0.8782
1 1/8	7	1.1250	1.0322	1.0381	0.9704	0.9858
1 1/4	7	1.2500	1.1572	1.1631	1.0954	1.1108
1 1/2	6	1.5000	1.3917	1.3988	1.3196	1.3376
1 3/4	5	1.7500	1.6201	1.6283	1.5335	1.5551
2	4 1/2	2.0000	1.8557	1.8646	1.7594	1.7835
2 1/4	4 1/2	2.2500	2.1057	2.1146	2.0094	2.0335
2 1/2	4	2.5000	2.3376	2.3473	2.2294	2.2564
2 3/4	4	2.7500	2.5876	2.5973	2.4794	2.5064
3	4	3.0000	2.8376	2.8473	2.7294	2.7564

* Basic diameter.

† Allowable only with tap having sharp corners.

Table X.—Coarse-thread Series—Close Fit (Class 4)

Screw Sizes, Inches

Size	Threads per inch	Major diameter		Pitch diameter†		Maximum minor diameter†
		Max.*	Min.	Max.	Min.	
1	64	0.0730	0.0692	0.0630	0.0623	0.0538
2	56	0.0860	0.0820	0.0746	0.0739	0.0641
3	48	0.0990	0.0946	0.0857	0.0849	0.0734
4	40	0.1120	0.1072	0.0960	0.0951	0.0813
5	40	0.1250	0.1202	0.1090	0.1081	0.0943
6	32	0.1380	0.1326	0.1179	0.1169	0.0997
8	32	0.1640	0.1586	0.1439	0.1429	0.1257
10	24	0.1900	0.1834	0.1632	0.1620	0.1389
12	24	0.2160	0.2094	0.1892	0.1880	0.1649
1/4	20	0.2500	0.2423	0.2178	0.2165	0.1887
5/16	18	0.3125	0.3043	0.2767	0.2752	0.2443
3/8	16	0.3750	0.3660	0.3348	0.3332	0.2983
7/16	14	0.4375	0.4277	0.3915	0.3897	0.3499
1/2	13	0.5000	0.4896	0.4504	0.4485	0.4056
9/16	12	0.5625	0.5513	0.5089	0.5069	0.4603
5/8	11	0.6250	0.6132	0.5665	0.5644	0.5135
3/4	10	0.7500	0.7372	0.6856	0.6833	0.6273
7/8	9	0.8750	0.8610	0.8034	0.8010	0.7387
1	8	1.0000	0.9848	0.9195	0.9168	0.8466
1 1/8	7	1.1250	1.1080	1.0330	1.0300	0.9497
1 1/4	7	1.2500	1.2330	1.1580	1.1550	1.0747
1 1/2	6	1.5000	1.4798	1.3926	1.3890	1.2955
1 3/4	5	1.7500	1.7268	1.6211	1.6170	1.5046
2	4 1/2	2.0000	1.9746	1.8568	1.8524	1.7274
2 1/4	4 1/2	2.2500	2.2246	2.1068	2.1024	1.9774
2 1/2	4	2.5000	2.4720	2.3389	2.3341	2.1933
2 3/4	4	2.7500	2.7220	2.5889	2.5841	2.4433
3	4	3.0000	2.9720	2.8389	2.8341	2.6933

* Basic diameter.

† The tolerances are cumulative and include errors of lead and angle

TYPES OF MANUFACTURING GAUGES

Gauging is the process of measuring manufactured materials to assure the specified uniformity of size and contour required by the industries.

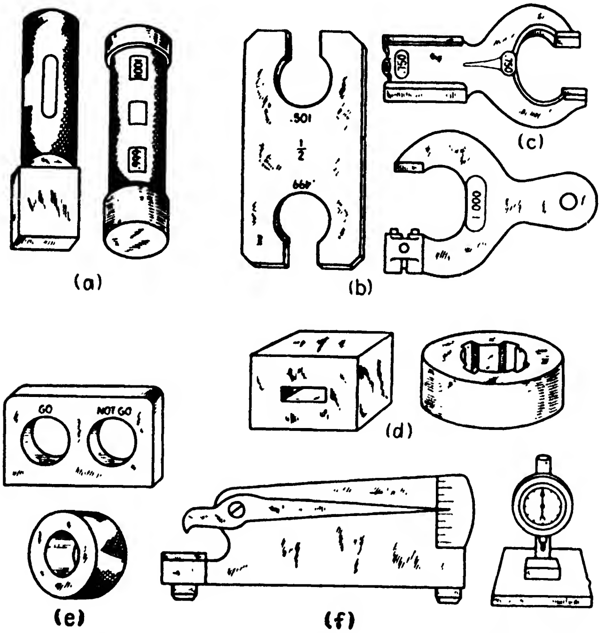


FIG. 128.—(*a*) Types of plug gauges. (*b*) Solid and adjustable snap gauges. (*c*) Caliper gauge. (*d*) Receiving gauges. (*e*) Ring gauges. (*f*) Indicating gauges.

Standard.—The standard for gauging is a physical representation of a form, dimension, or size established by law or by general usage and consent.

Standard sizes are recognized or accepted sizes corresponding to various subdivisions of a recognized unit of length. These are usually expressed in inches or in millimeters, but sometimes by arbitrary numbers or letters.

Basic size is the exact theoretical size from which all limiting variations are made.

Nominal size is a designation given to the subdivision of the unit of length having no specified limits of accuracy, but indicating a close approximation to a standard size.

Fig. 129.—Cylindrical gauge, internal.

Gauges are devices for determining whether or not one or more of the dimensions of a manufactured part are within specified limits. The types of common manufacturing gauges employed for measuring the accuracy, or variation from standard, of a manufactured article are: ring, plug, receiving, snap, caliper, indicating, and fixture.

Ring Gauge.—One whose inside measuring surfaces are circular in form.

Plug Gauge.—One whose outside measuring surfaces are arranged to verify the specified uniformity of holes. A plug gauge may be straight or tapered and of any cross-sectional shape.

Receiving Gauge.—One whose inside measuring surfaces are arranged to verify the specified uniformity of size and contour of material.

Fig. 130.—Cylindrical gauge, external.

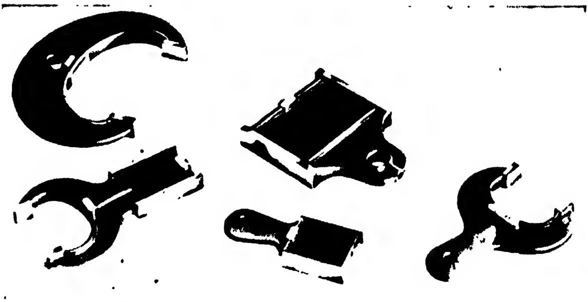


Fig. 131.—Standard caliper gauges.

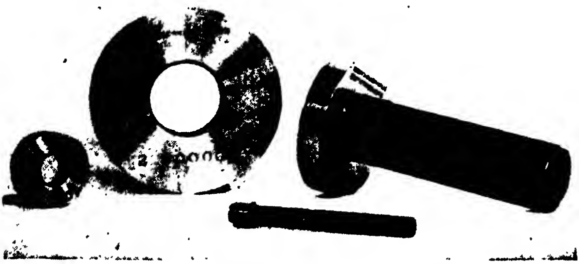


Fig. 132.—Standard reference disks.

Snap Gauge.—A fixed gauge arranged with inside measuring surfaces for calipering.

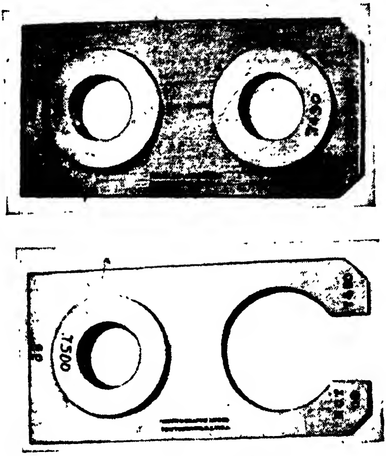


Fig. 133.—Limit gauges.

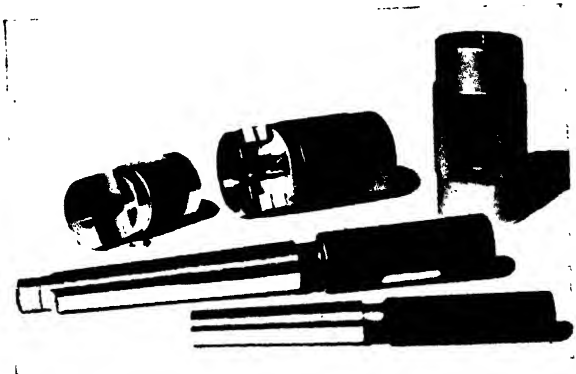


Fig. 134.—Standard taper cylindrical gauges.

Caliper Gauge.—One which, for external use, is similar to a snag gauge and, for internal use, is similar to a plug gauge.

Indicating Gauge.—One that exhibits visually the variations in the uniformity of dimensions or contour, the amount of the variations being usually indicated by a lever on a graduated scale or dial.

Fixture Gauge.—The name given to a combination of any or all of the above types. It is employed for the purpose of measuring more than one point on an irregular piece at one setting, or for determining the accuracy of one point in relation to another.

Master Gauge.—One whose gauging dimensions represent as exactly as possible the physical dimensions of the component. It is the gauge to which all other gauges and all dimensions are checked.

Working Gauge.—One used by the workman to check the work as it is produced. The working gauge should not accept any product that the inspection gauge will reject.

Inspection Gauge.—A gauge used by the manufacturer or purchaser in accepting the finished product. These gauges are checked against the master gauge or against the drawing of the part.

INSTRUCTIONS FOR GRINDING TWIST DRILLS

Efficiency in drilling operations is largely dependent on a correctly ground drill. A modern drilling machine and the proper applications of speeds and feeds are important factors, but if the twist drill is not ground correctly it is natural to expect unsatisfactory results. There are no standard instructions that can be applied to the drilling of different materials, as each requires a characteristic grind.

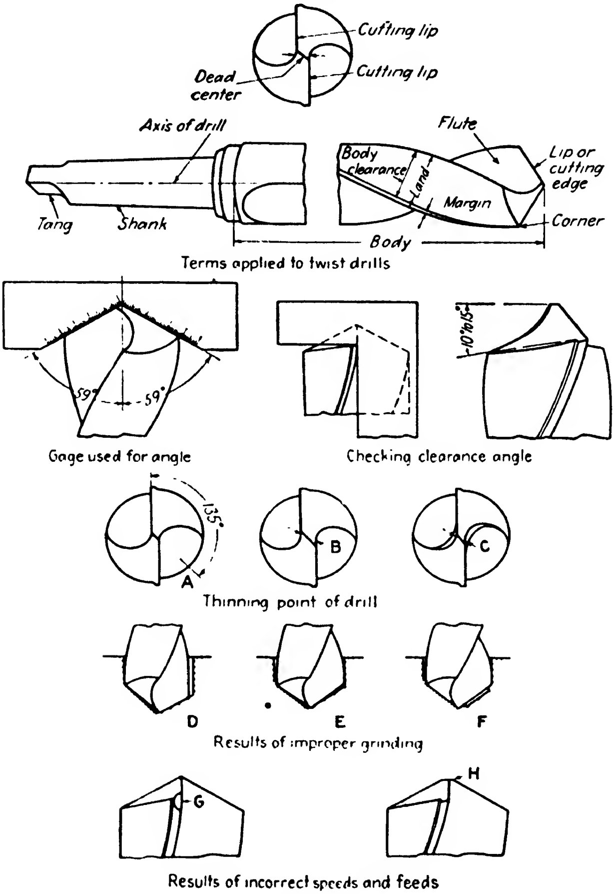


FIG. 135.

Twist drills should be ground so that both lips have the same angle and are of equal length. In Fig. 135 is shown a gauge often used for checking the point, angle, and length of lip. An angle of

59 deg. is recommended for general work. If one lip is ground to 59 deg. and the other to 61 deg., the latter will have to do all the cutting.

Lip clearance, or the relief given the cutting edges, is important to ensure proper penetration without interference. Approximate lip clearance may be checked by use of a small square as illustrated in Fig. 135. The angle of lip clearance should be increased gradually as the center of the drill is approached, until the line across the center of the drill stands at an angle with the cutting edges of about 135 deg. as shown in Fig. 135 at *A*.

The web separating the flutes is the supporting section of the drill. It increases in thickness slightly toward the shank, giving the drill additional rigidity. The thickening of the web decreases the flute area; therefore, as the drill is used it is necessary to compensate for the increased thickness by "thinning the point." At *B* is shown the shape of the web near the shank. To thin the point is a delicate operation. A round-face abrasive wheel should be used, forming the point as shown at *C*.

When both of the lips are ground to an equal angle but vary in their lengths, the hole produced will be oversized such as shown in Fig. 135 at *D*. If the angles of the lips are different, the result will be that one lip will do most of the work, producing a hole as shown at *E*. A combination of these conditions, unequal lips and different angles, produces the hole shown at *F*.

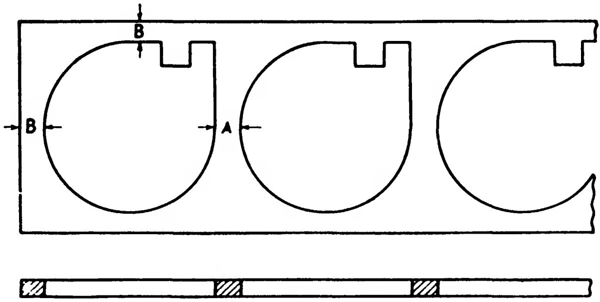
Drills should be operated at correct speeds and feeds to produce the best results. When the point is ground correctly and the drill burns at the outer edges, as shown in Fig. 135 at *G*, or if the point "mushrooms," or turns over, as at *H*, the cause may generally be traced to excessive speed.

BLANKING DIE DATA

When laying out work for a plain or a progressive blanking die, it should be remembered that one side of the blank produced will have sharp edges and the edges of the other side will be rounded. When the blank is assembled to other units, it is advisable to have the side with the sharp edges located next to the assembled part. The part drawing should specify which side is to have sharp edges so the die layout can be made accordingly. In cases where either side can be used, the layout can be made for the best run to suit the shape of the blank. The side with sharp edges lies next to the punch, and the side with rounded edges next to the die. Exception is made when the blank is to be formed, in which case the side of the blank with the rounded edges should be next to the forming die.

In a layout for blanking from strip stock, it is good practice to allow a distance between blanks, and between the blanks and the edges of stock, equal to the thickness of the stock as shown in Fig. 136 at *A* and *B*.

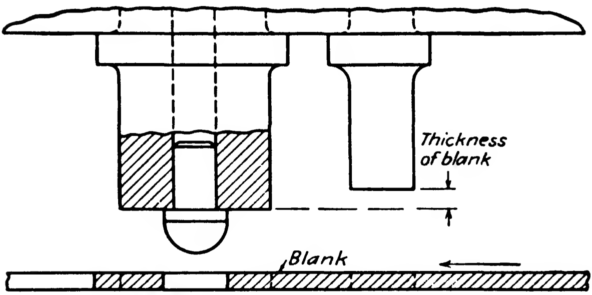
The amount of material to allow around the contour of the blank for shaving depends largely on the nature of the metal being blanked. A general rule is as follows: For a single shaving operation, allow approximately 0.003 in. for each $\frac{1}{32}$ in. of blank thickness. For two shaving operations, allow 0.002 in. for each $\frac{1}{32}$ in. of blank thickness for the first shave, and 0.001 to 0.0015 in. for the second, keeping the ratio about 2 to 1.



Amount of metal to be left between punchings

FIG. 136.

When making a progressive blanking die, as shown in Fig. 137, a distance equal to the thickness of the blank should be allowed between the

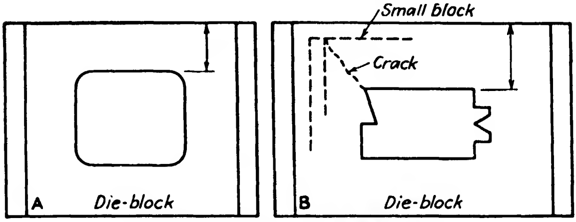


Arrangement of punches for progressive work

FIG. 137.

various punches. This method, called step punching, distributes the pressure over a greater length of the press stroke rather than applying all the pressure at once.

Die blocks should be made large enough so that spreading or cracking will not occur. Figure 138, at *A*, shows a die having rounded corners in which the distance from the die opening to the outer edge is 1½ in., minimum. When

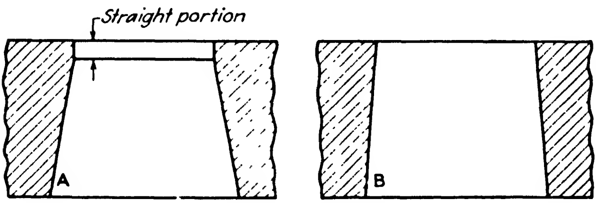


Thickness of die-blocks around openings

FIG. 138.

the die has sharp corners, as at *B*, a greater distance should be allowed.

The openings of die blocks are generally made taper, as shown in Fig. 139. At *A* is shown a die having a straight portion next to the face, beyond which it has the usual taper. At *B* the tapered portion is started from the die face.



Die-block openings

FIG. 139.

The angle of taper is usually ½ deg., although for some work 1 deg. may be used. After the die face has been ground down, making the opening too large, the die can be reconditioned by annealing, peening, and rehardening.

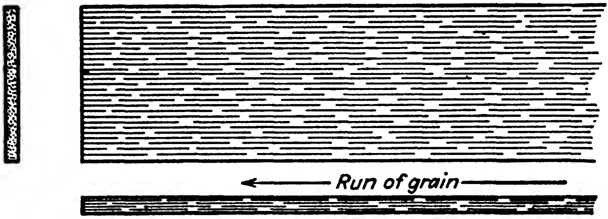


Fig. 140.

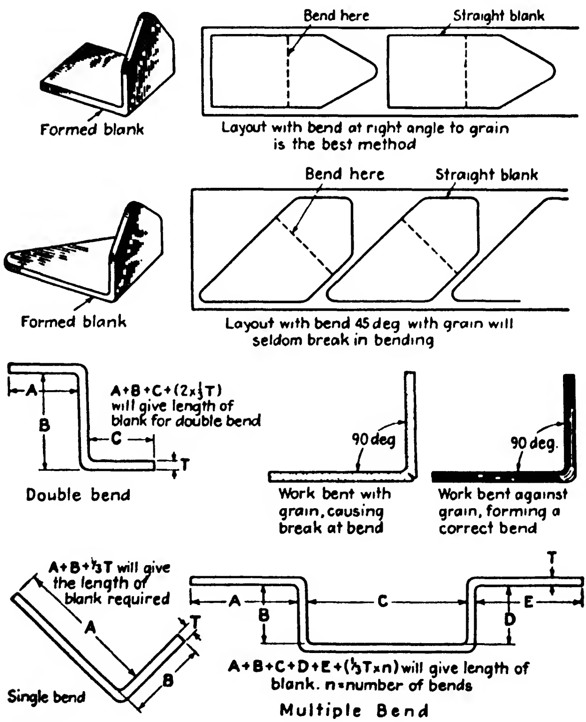


Fig. 141.

For double-run die work, including a single run along one edge of the stock, after which the stock is turned end for end and a single run made on the opposite side, the procedure is similar to single-run dies. When double blanking dies are required, the distance between the blanking punches should be made to suit the work. If the distance is too small, the die block will break down between the openings. A safe minimum distance is $\frac{13}{16}$ in. This rule applies to all multiple-punching operations.

All blanking dies should be provided with stripper plates and guide plates. If two guide plates are to be used, one on each side of the stock, be sure to make allowances for variations in the width of the stock, also for the spreading of the stock due to punching. The application of a flat spring on one stripper plate is good practice. A liberal clearance can be made between the stock and the stripper plate.

Table XI.—Constants for Angle Bends

Angle	Constant = S	Angle	Constant = S
0 10 min.	0.00074	35	0.1554
0 30 min.	0.00222	40	0.1776
1	0.00444	45	0.1998
2	0.00888	50	0.2220
3	0.01332	55	0.2442
4	0.01776	60	0.2664
5	0.02220	65	0.2886
10	0.0444	70	0.3108
15	0.0666	75	0.3300
20	0.0888	80	0.3552
25	0.1110	85	0.3774
30	0.1332	90	0.4000

RULES FOR LAYING OUT BENDING DIES

Blanks that are to be formed with a bend of 90 deg. should be laid out so that the bend will come as near as possible against the grain of the metal to ensure against breaking while bending. These rules apply for sharp bends only.

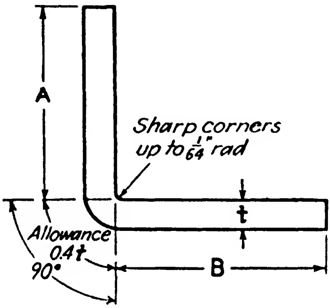


FIG. 142.

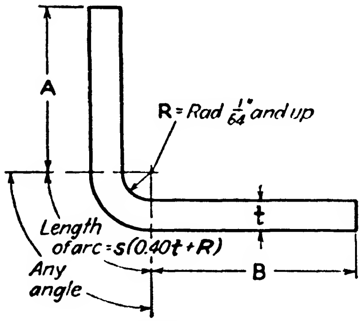


FIG. 143.

The grain of sheet metal always runs lengthwise (see Figs. 140 and 141).

NATIONAL CASH REGISTER CO. METHOD

For right-angle bends as in Fig. 142, where the radius of bend is less than $\frac{1}{64}$ in., the custom is to multiply the thickness of stock by the constant 0.40 and add to the inside dimensions. Thus the length of blank L equals $A + B + 0.40t$.

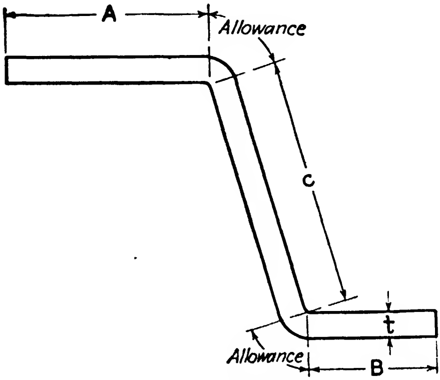


FIG. 144.

For bends having a radius greater than $\frac{1}{64}$ in., (Fig. 143) and of any angle, the formula is

$$L = A + B + S(0.40t + R),$$

where S is a constant for the angle taken from

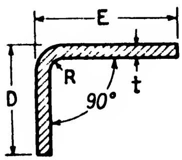


FIG. 145.

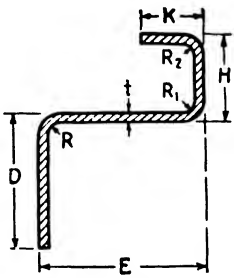


FIG. 146.

Table XI, page 233. The developed length of a blank can be computed by considering each bend separately.

The formula for the length of blank in Fig. 144 is $L = A + B + C + 2S(0.40t - R)$.

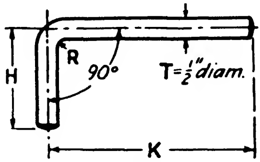


FIG. 147.

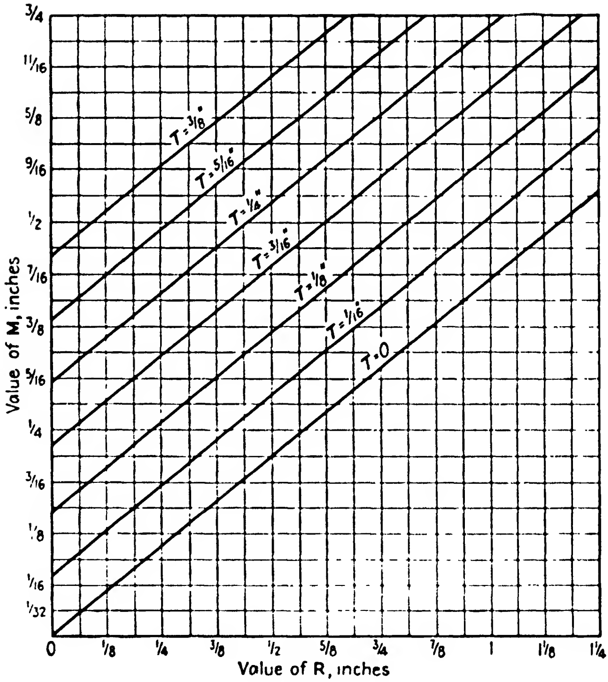


FIG. 148.

For intricately formed blanks it has been found to be more satisfactory to cut out trial blanks for the forming dies.

QUICK METHOD FOR RIGHT-ANGLE BENDS

For computing the developed length of a metal blank with a 90-deg. bend, as in Fig. 145, the outside lengths are added and a quantity *m* is subtracted, which is derived from the size of the inner radius and thickness of stock. In Fig. 148 is given a method of obtaining the value of *m* for radii from 0 to 1¼ in. and thicknesses of stock from 0 to ⅜ in. To find *m* for a given value of *R*, follow the vertical line until it intersects the line corresponding to the thickness of stock *t*. The value of *m* is then read directly from the left-hand column. The formula for the developed length of the blank in Fig. 145 is $L = D + E - m$. The formula for the case in Fig. 146 is

$$L = D + E + H + K - (m + m_1 + m_2)$$

the three values of *m* corresponding to the different radii. The developed length of a rod, as in Fig. 147, can be calculated in the same manner as for sheet stock, but the center line should be taken for the dimensions, and half the diameter of the rod should be used for *t* in figuring the values.

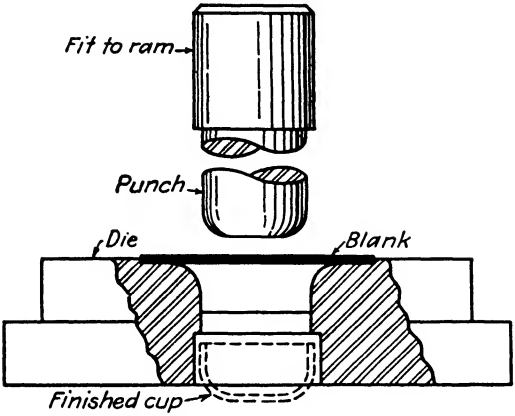


FIG. 149.—Plain form of drawing die to be used when the part has been previously blanked. The counterbore in the die aids in stripping the shell from the punch.

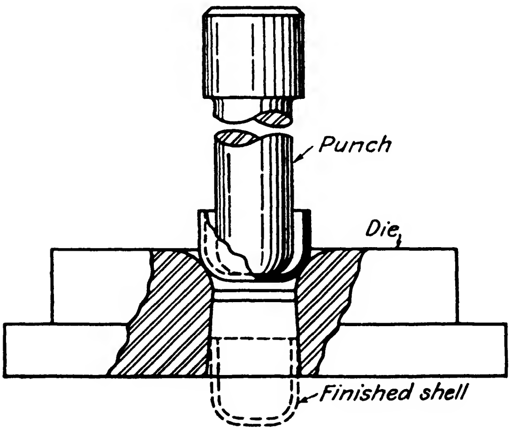


FIG. 150.—A redrawing die of plain design for the second operation on shells which cannot be completed in one draw. Usually it is necessary to anneal the work prior to making the redraw.

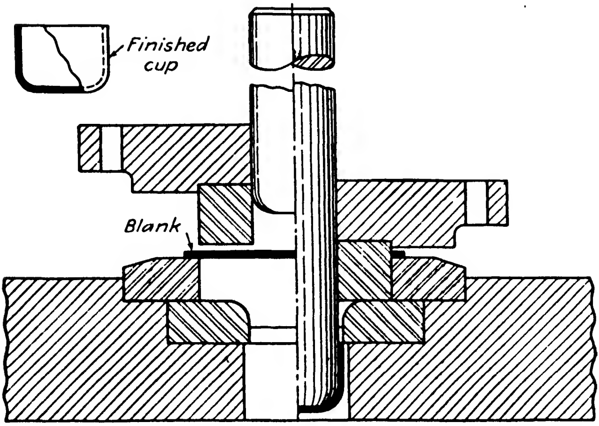


FIG. 151.—Shells of a simple nature can be formed in a combination blanking and drawing die such as shown here. Dies of this type are mounted in double-action presses and arranged so that the cutting and drawing sections can be replaced.

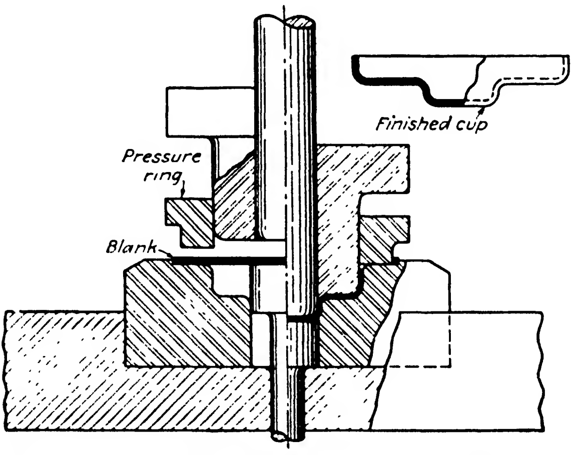


FIG. 152.—A double drawing die for an irregular part. A pressure ring holds the work firmly in order to avoid wrinkles; then the outer draw is completed, after which the central punch draws the smaller form. The plunger in the die aids in keeping the work in place while being drawn and also ejects the finished cup.

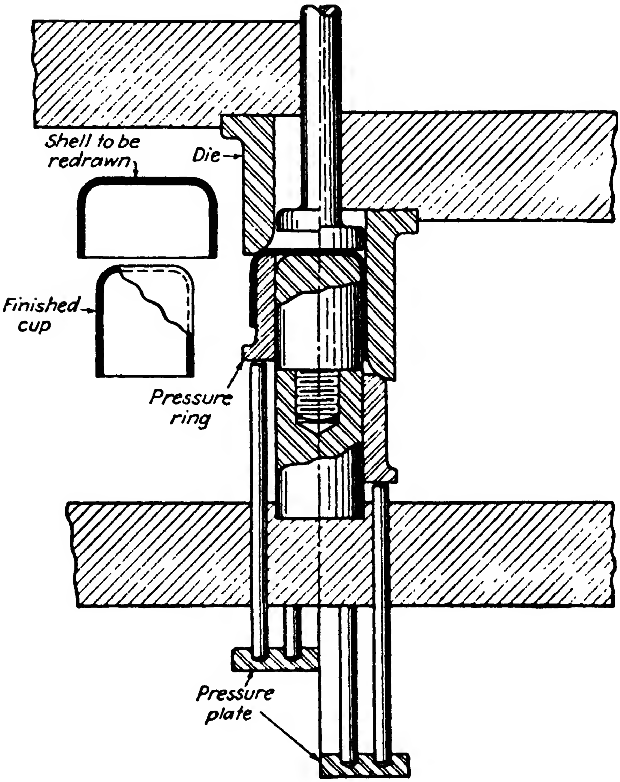


FIG. 153.—A redrawing die for forming shells made of metal that is likely to wrinkle and that cannot be drawn in a plain-type die. Pressure exerted between the die and the pressure ring irons out the work as it is drawn. Ejection from the punch is made by the pressure ring.

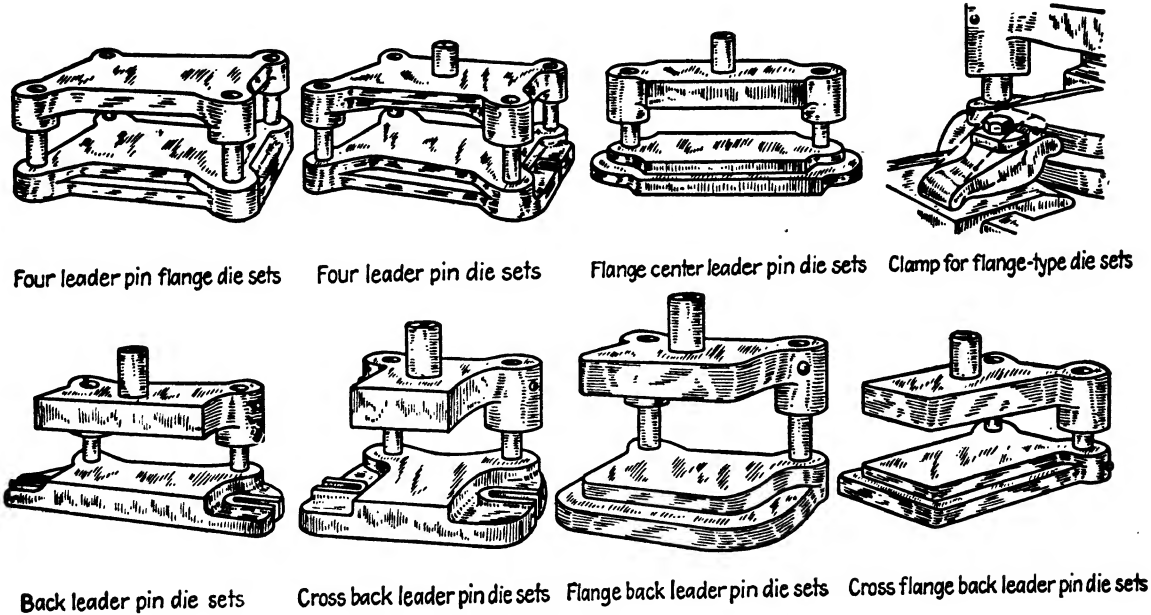

Four leader pin flange die sets
Four leader pin die sets
Flange center leader pin die sets
Clamp for flange-type die sets
Back leader pin die sets
Cross back leader pin die sets
Flange back leader pin die sets
Cross flange back leader pin die sets

FIG. 154.

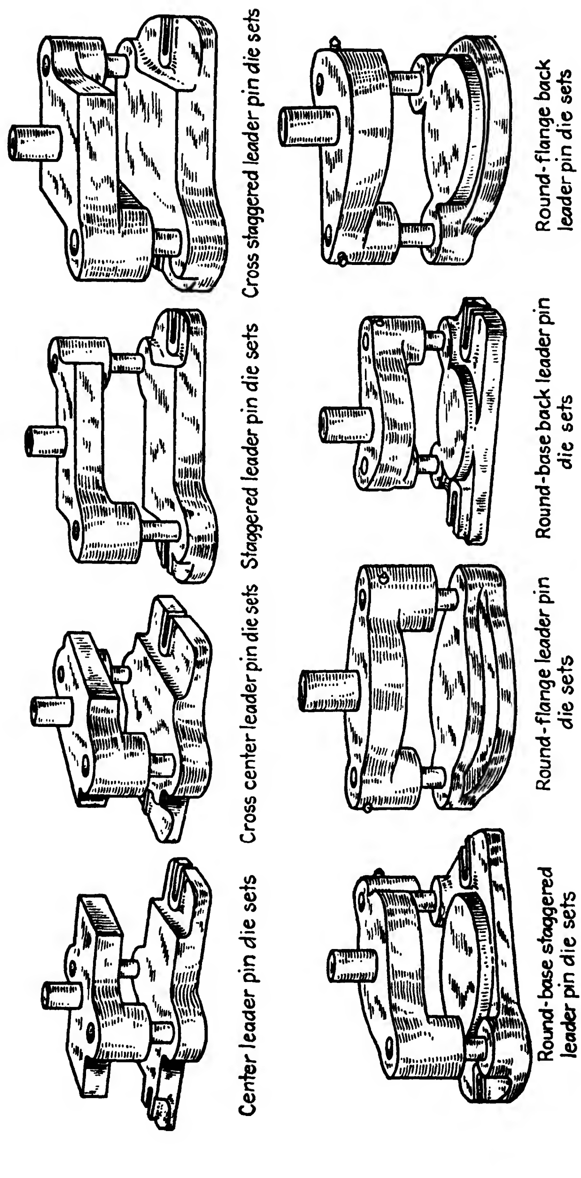
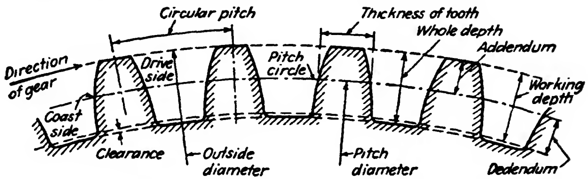

Center leader pin die sets
Cross center leader pin die sets
Staggered leader pin die sets
Cross staggered leader pin die sets
Round-base staggered leader pin die sets
Round-flange leader pin die sets
Round-base back leader pin die sets
Round-flange back leader pin die sets

FIG. 155.

TABLE XII.—RULES FOR CALCULATING SPUR GEARS



Having	To get	Rule	Formula
Diametral pitch	Circular pitch	Divide 3.1416 by the diametral pitch	$P' = \frac{3.1416}{P}$
Outside diameter and the number of teeth.	Circular pitch	Divide outside diameter by the product of 0.3183 and number of teeth plus 2	$P' = \frac{D}{0.3183\ (N+2)}$
Number of teeth and the circular pitch	Pitch diameter	Continued product of the number of teeth, the circular pitch and 0.3183	$D' = NP'\ 0.3183$
Number of teeth and the outside diameter.	Pitch diameter	Divide the product of number of teeth and outside diameter by number of teeth plus 2	$D' = \frac{ND}{N+2}$
Number of teeth and the circular pitch	Outside diameter	Continued product of the number of teeth plus 2, the circular pitch and 0.3183	$D = (N+2)P'\ 0.3183$
Circular pitch	Thickness of tooth	One-half the circular pitch	$t = \frac{P'}{2}$
Circular pitch	Addendum	Multiply the circular pitch by 0.3183 or $s = D'/N$	$s = P'\ 0.3183$
Circular pitch	Working depth	Multiply the circular pitch by 0.6366	$D'' = P'\ 0.6366$
Thickness of tooth	Clearance	One-tenth the thickness of tooth at pitch line	$f = \frac{t}{10}$

TABLE XII.—RULES FOR CALCULATING SPUR GEARS.—(*Continued*)

Having	To get	Rule	Formula
Circular pitch	Diametral pitch	Divide 3.1416 by the circular pitch	$P = \frac{3.1416}{P'}$
Outside diameter and the number of teeth.	Diametral pitch	Divide number of teeth plus 2 by outside diameter	$P = \frac{N + 2}{D}$
Number of teeth and the outside diameter.	Pitch diameter	Divide the product of outside diameter and number of teeth by number of teeth plus 2	$D' = \frac{DN}{N + 2}$
Outside diameter and the diametral pitch	Pitch diameter	Subtract from the outside diameter the quotient of 2 divided by the diametral pitch	$D' = D - \frac{2}{P}$
Number of teeth and the diametral pitch	Outside diameter	Divide number of teeth plus 2 by the diametral pitch	$D = \frac{N + 2}{P}$
Pitch diameter and the number of teeth	Outside diameter	Divide the number of teeth plus 2 by the quotient of number of teeth divided by the pitch diameter	$D = \frac{N + 2}{\frac{N}{D'}}$
Pitch diameter and the diametral pitch	Number of teeth	Multiply pitch diameter by the diametral pitch	$N = D'P$
Outside diameter and the diametral pitch.	Number of teeth	Multiply outside diameter by the diametral pitch and subtract 2	$N = DP - 2$
Diametral pitch	Thickness of tooth	Divide 1.5708 by the diametral pitch	$t = \frac{1.5708}{P}$
Diametral pitch	Working depth	Divide 2 by the diametral pitch	$D'' = \frac{2}{P}$
Diametral pitch	Whole depth	Divide 2.157 by the diametral pitch	$D'' + f = \frac{2.157}{P}$
Diametral pitch	Clearance	Divide 0.157 by the diametral pitch	$f = \frac{0.157}{P}$

The stub form of tooth is designated by two figures, such as 4/5, which represents that the tooth is four pitch but cut to a depth of a 5-pitch tooth. These gear calculations are figured accordingly.

INDEX