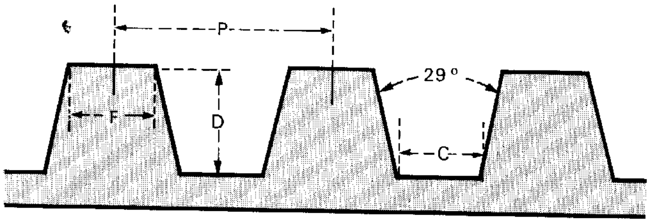

Figure 55-7 *American National Acme thread.*

The *American National Acme thread* (Fig. 55-7) is replacing the square thread in many cases. It has a 29° angle and is used for feed screws, jacks, and vises.

$$D = \text{minimum } 0.500P$$
$$= \text{maximum } 0.500P + 0.010$$
$$F = 0.3707P$$
$$C = 0.3707P - 0.0052$$
(for maximum depth)

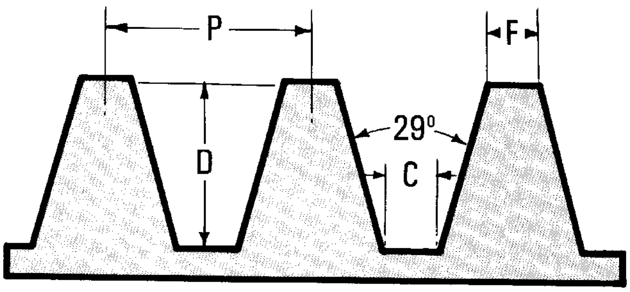

Figure 55-8 *Brown and Sharpe worm thread.*

The *Brown and Sharp worm thread* (Fig. 55-8) has a 29° included angle as does the Acme thread; however, the depth is greater and the widths of the crest and root are different. This thread is used to mesh with worm gears and transmit motion between two shafts at right angles to each other but not in the same plane. The self-locking feature makes it adaptable to winches and steering mechanisms.

$$D = 0.6866P$$
$$F = 0.335P$$
$$C = 0.310P$$

The *square thread* (Fig. 55-9) is being replaced by the Acme thread due to the difficulty of cutting it, particularly with taps and dies. Square threads were often found on vises and jack screws.

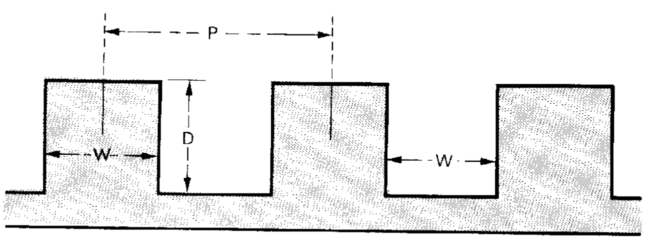

Figure 55-9 *Square thread.*

$$D = 0.500P$$
$$F = 0.500P$$
$$C = 0.500P + 0.002$$

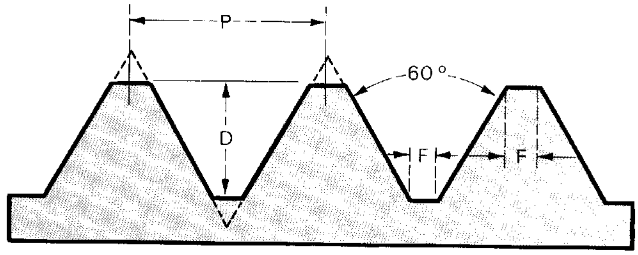

Figure 55-10 *International metric thread.*

The *international metric thread* (Fig. 55-10) is a standardized thread used in Europe. This thread has a 60° included angle with a crest and root truncated to one-eighth the depth. Although this thread is used extensively throughout Europe, its use in North America has been confined mainly to spark plugs and the manufacture of instruments.

$$D = 0.7035P \text{ (maximum)}$$
$$= 0.6855P \text{ (minimum)}$$
$$F = 0.125P$$
$$R = 0.0633P \text{ (maximum)}$$
$$0.054P \text{ (minimum)}$$

THREAD FITS AND CLASSIFICATIONS

Certain terminology is used when referring to thread classifications and fits. To understand any thread system properly, the terminology relating to thread fits should be understood.

Fit is the relationship between two mating parts, which is determined by the amount of clearance or interference when they are assembled.

Allowance is the intentional difference in size of the mating parts or the minimum clearance between mating parts (Fig. 55-11). With threads, the allowance is the permissible difference between the largest external thread and the smallest internal thread. This produces the tightest fit acceptable for any given classification.

The allowance for a 1 in.—8 UNC (Unified National Coarse) Class 2A and 2B fit is:

Minimum pitch diameter of the internal thread (2B)	= 0.9188 in.
Maximum pitch diameter of the external thread (2A)	= 0.9168 in.
Allowance or intentional difference	= 0.002 in.

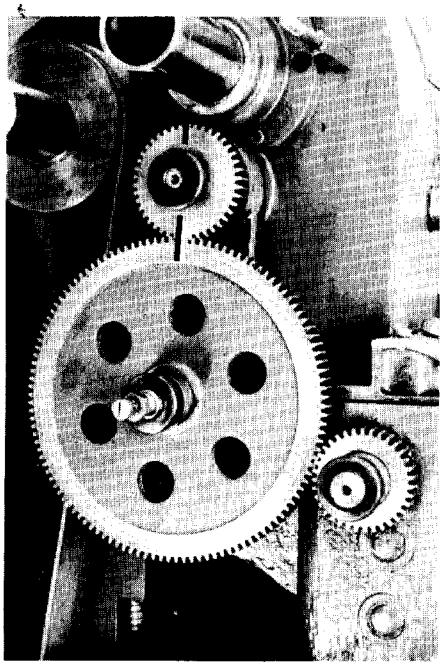

Figure 55-27 *Marking the spindle and intermediate gear before indexing the workpiece exactly one-half turn for cutting a double thread.*

spindle gear has 24 teeth, count 12 teeth and mark this one with chalk.

(5) Revolve the lathe spindle by hand one-half turn to bring the marked tooth in line with the chalk mark on the intermediate gear.

(6) Reengage the intermediate gear.

7. Reset the crossfeed handle to the same position as when cutting the first thread.

8. Cut the second thread, feeding the compound rest handle until the graduated collar is at the same setting as for the first thread.

The Thread-Chasing Dial Method of Cutting Multiple Threads

Double-start threads with an odd-numbered lead (for example, 1/5, 1/7, etc.) may be cut using the thread-chasing dial.

1. Take one cut on the thread by engaging the split nut at a *numbered* line on the chasing dial.
2. Without changing the depth of cut, take another cut at an *unnumbered* line on the chasing dial. The second thread will be exactly in the middle of the first thread.
3. Continue cutting the thread to depth, taking two passes (one on a numbered line, the other on an unnumbered line) for every depth-of-cut setting.

SQUARE THREADS

Square threads were often found in vise screws, jacks, and other devices where maximum power transmission was required. Because of the difficulty of cutting this thread with taps and dies, it is being replaced by Acme thread. With care, square threads can be readily cut on a lathe.

The Shape of a Square Threading Tool

The square threading tool looks like a short cutting-off tool. It differs from it in that both sides of the square threading tool must be ground at an angle to conform to the helix angle of the thread (Fig. 55-28).

The helix angle of a thread, and therefore the angle of the square threading tool, depends on two factors:

1. The helix angle changes for each *different lead* on a

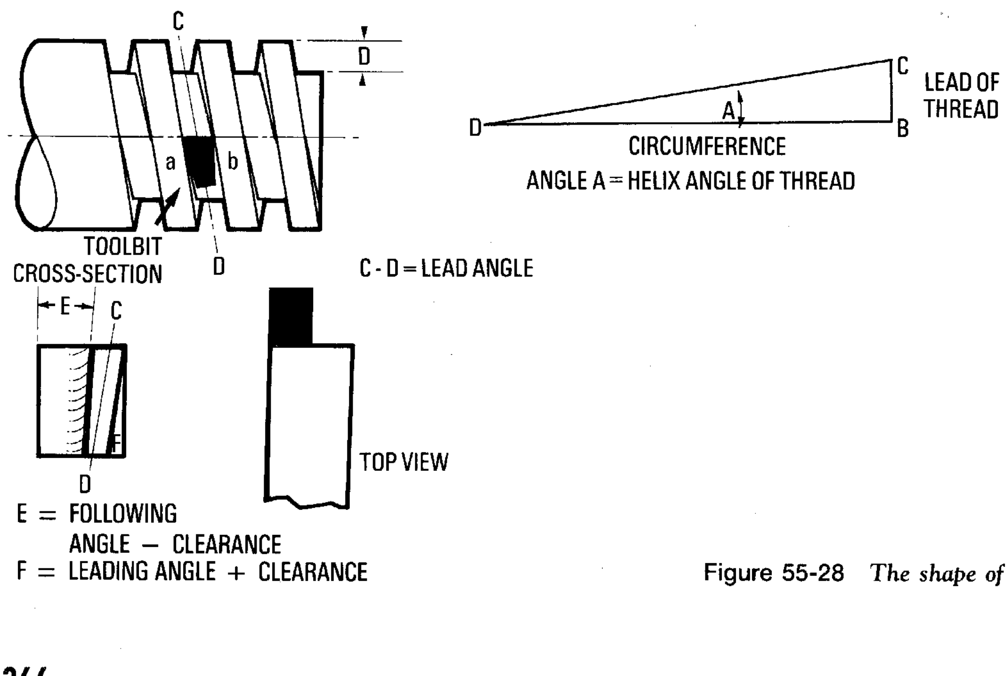

Figure 55-28 *The shape of a square threading tool.*

given diameter. The greater the lead of the thread, the greater will be the helix angle.

2. The helix angle changes for each *different diameter* of thread for a given lead. The larger the diameter, the smaller will be the helix angle.

The helix angle of either the leading or following side of a square thread can be represented by a right-angle triangle (Fig. 55-28). The side opposite equals the *lead* of the thread, and the side adjacent equals the circumference of either the major or minor diameter of the thread. The angle between the hypotenuse and the side adjacent represents the helix angle of the thread.

To Calculate the Helix Angles of the Leading and Following Sides of a Square Thread

$$\text{Tan leading angle} = \frac{\text{lead of thread}}{\text{circumference of minor diameter}}$$

$$\text{Tan following angle} = \frac{\text{lead of thread}}{\text{circumference of major diameter}}$$

Clearance

If a square toolbit is ground to the same helix angles as the leading and following sides of the thread, it would have no clearance and the sides would rub. To prevent the tool from rubbing, it must be provided with approximately 1° clearance on each side, making it thinner at the bottom (Fig. 55-29). For the leading side of the tool, *add* 1° to the calculated helix angle. On the following side, *subtract* 1° from the calculated angle.

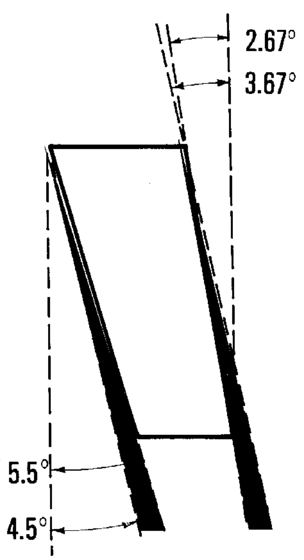

Figure 55-29 *Helix angles of the thread and clearance angles necessary for a square threading tool.*

EXAMPLE

To find the leading and following angles of a threading tool to cut a 1¼ in.—4 square thread.

SOLUTION

$$\text{Lead} = 0.250 \text{ in.}$$

$$\text{Single depth} = \frac{0.500}{4} = 0.125 \text{ in.}$$

$$\text{Double depth} = 2 \times 0.125 = 0.250 \text{ in.}$$

$$\text{Minor diameter} = 1.250 - 0.250 = 1.000 \text{ in.}$$

$$\text{Tan leading angle} = \frac{\text{lead}}{\text{minor dia. circumference}} = \frac{0.250}{1.000 \times \pi} = \frac{0.250}{3.1416} = 0.0795 \text{ in.}$$

$$\therefore \text{the angle of the thread} = 4°33'$$

$$\text{The toolbit angle} = 4°33' \text{ plus } 1° \text{ clearance} = 5°33'$$

$$\text{Tan following angle} = \frac{\text{lead}}{\text{major dia. circumference}} = \frac{0.250}{1.250\pi} = \frac{0.250}{3.927} = 0.0636 \text{ in.}$$

$$\therefore \text{the angle of the thread} = 3°38'$$

$$\text{The toolbit angle} = 3°38' \text{ minus } 1° \text{ clearance} = 2°38'$$

To Cut a Square Thread

1. Grind a threading tool to the proper leading and following angles. The width of the tool should be approximately 0.002 in. (0.05 mm) wider than the thread groove. This will allow the completed screw to fit the nut readily. Depending on the size of the thread, it may be wise to grind two tools; a roughing tool 0.015 in. (0.38 mm) undersize, and a finishing tool 0.002 in. (0.05 mm) oversize.
2. Align the lathe centers and mount the work.
3. Set the quick-change gearbox for the required number of *tpi*.
4. Set the compound rest at 30° to the right. This will

provide side movement if it becomes necessary to reset the cutting tool.

5. Set the threading tool square with the work and on center.

6. Cut the right-hand end of the work to the minor diameter for approximately 1/16 in. (1.58 mm) long. This will indicate when the thread is cut to the full depth.

7. If the work permits, cut a recess at the end of the thread to the minor diameter. This will provide room for the cutting tool to "run out" at the end of the thread.

8. Calculate the single depth of the thread as

$$\left(\frac{0.500}{N}\right)$$

9. Start the lathe and just touch the tool to the work diameter.

10. Set the *crossfeed graduated collar* to zero (0).

11. Set a 0.003-in. (0.08-mm) depth of cut with the *crossfeed screw* and take a trial cut.

12. Check the thread with a thread pitch gage.

13. Apply cutting fluid and cut the thread to depth, moving the *crossfeed* in from 0.002 to 0.010 in. (0.05 to 0.25 mm) for each cut. The depth of the cut will depend on the thread size and the nature of the workpiece.

NOTE: Since the thread sides are square, *all cuts* must be set using the *crossfeed screw.*

ACME THREAD

The *Acme thread* is gradually replacing the square thread because it is stronger and easier to cut with taps and dies. It is used extensively for lead screws because the 29° angle formed by its sides allows the split nut to be engaged readily during thread cutting.

The Acme thread is provided with 0.010-in. clearance for both the crest and root on all sizes of threads. The hole for an internal Acme thread is cut 0.020 in. larger than the minor diameter of the screw, and the major diameter of a tap or internal thread is 0.020 in. larger than the major diameter of the screw. This provides 0.010-in. clearance between the screw and nut on both the top and bottom.

To Cut an Acme Thread

1. Grind a toolbit to fit the end of the Acme thread gage (Fig. 55-30). Be sure to provide sufficient side clearance so that the tool will not rub while cutting the thread.

2. Grind the point of the tool flat until it fits into the slots of the gage indicating the number of threads per inch to be cut.

NOTE: If a gage is not available, the width of the toolbit point may be calculated as follows:

$$\text{Width of point} = \frac{0.3707}{N} - 0.0052 \text{ in.}$$

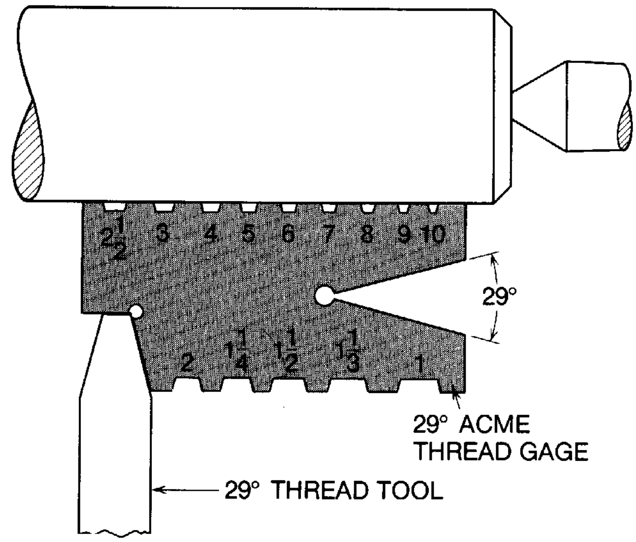

Figure 55-30 *The toolbit is squared with the workpiece by an Acme thread gage.*

3. Set the quick-change gearbox to the required number of threads per inch.

4. Set the compound rest 14½° to the right (half the included thread angle).

5. Set the Acme threading tool on center and square it with the work using the gage shown in Fig. 55-30.

6. At the right-hand end of the work, cut a section 1/16 in. long to the minor diameter. This will indicate when the thread is to the full depth.

7. Cut the thread to the proper depth by feeding the cutting tool, using the *compound rest.*

Measuring Acme Threads

For most purposes, the *one-wire method* of measuring Acme threads is accurate enough. A single wire or pin of the correct diameter is placed in the thread groove (Fig. 55-31) and measured with a micrometer. The thread is the correct size when the micrometer reading over the wire is the same as the major diameter of the thread and the *wire is tight in the thread.*

NOTE: It is important that the burrs be removed from the diameter before using the one-wire method.

The diameter of the wire to be used can be calculated as follows:

$$\text{Wire diameter} = 0.4872 \times \text{pitch}$$

For example, if 6 threads per inch are being cut, the wire diameter should be:

$$\text{Wire size} = 0.4872 \times \frac{1}{6}$$
$$= 0.081 \text{ in.}$$