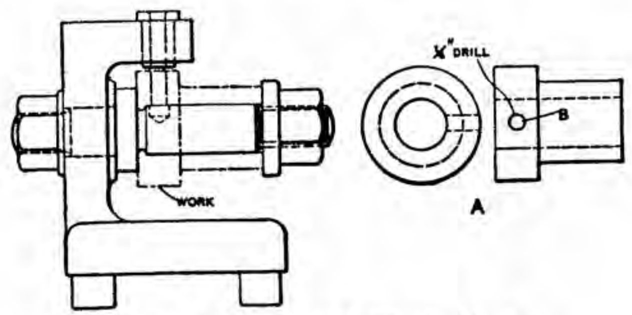


Fig. 5-21. **Drill jig for drilling bushing.**

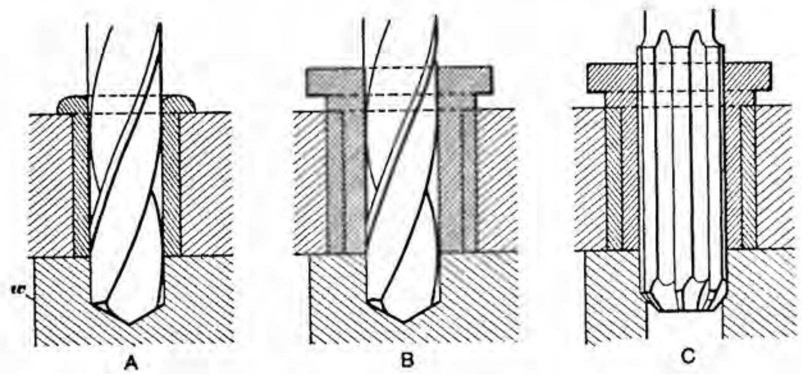


Fig. 5-22. Fixed and removable guide bushings for drill jigs.

it in proper position. Some jigs have fixed guide bushings and others have removable guide bushings. A fixed bushing is shown by the sectional view at A, Fig. 5-22, which also indicates how the drill is guided while it is drilling the work, *w*. Jigs are equipped with removable bushings should drills of a different size need to be used or if drilled holes are to be finished by reaming. For example, if a hole is to be drilled and reamed, a removable bushing is used that fits the drill, as shown at B, and this is replaced by a bushing that fits the reamer, as shown at C.

Drill fixtures are similar to the drill jigs shown except that they do not have drill bushings. For this reason they are generally more open in construction than drill jigs. They are always fastened to the table of the machine so that their relationship to the spindle remains fixed. Drill jigs, on the other hand, may or may not be fastened to the table.

CHAPTER **6**

Engine Lathe Construction

The origin of the term "engine lathe" undoubtedly can be traced to the manner in which early metal-cutting lathes were powered. These early lathes were driven by a reciprocating steam engine through a system of pulleys, belts, and clutches. Usually, a single steam engine provided the power for several lathes as well as for the other machine tools in the machine shop. Later, a single electric motor replaced the steam engine

Courtesy of The R. K. LeBlond Machine Tool Company

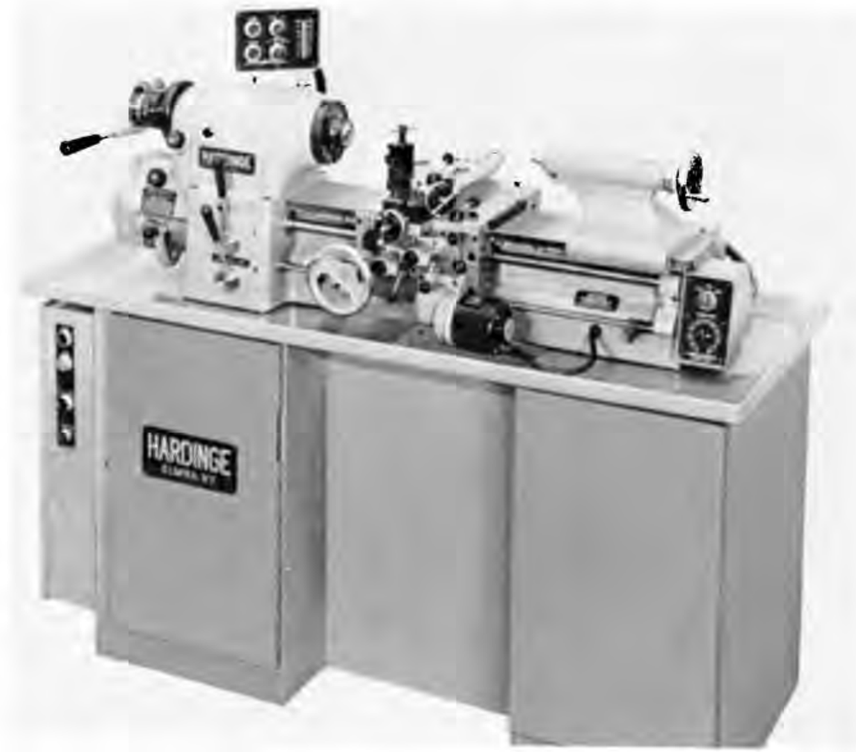
Fig. 6-1. Large lathes at work.

Courtesy of the Atlas Press Company

Fig. 6-2. Modern 12" bench lathe.

to drive a group of machine tools. All modern engine lathes, as well as other machine tools, are driven by an individual electric motor. The individual motor drive eliminates the need for overhead shafting, reduces power losses, provides a more direct drive, and provides greater flexibility in the placement of the lathe on the machine shop floor.

Engine lathes differ from other lathes by their greater simplicity of construction and by their greater versatility in performing many different machining operations. Turning cylindrical surfaces, facing flat surfaces, cutting threads, drilling and boring holes, and cutting internal threads are some of the typical operations that are performed on engine lathes. Of equal importance in considering the versatility of the engine lathe is the ease with which the workpiece can be set up and held, and the ease with which the lathe can be manipulated in performing different operations. Although production lathes and numerically controlled lathes are used, to a large extent, where a variety of parts must be made, the engine lathe is still the basic machine tool in the machine shop. It is used extensively in jobbing shops and in tool and die shops. When operated with skill and with an understanding of its capabilities, it can produce parts to exacting standards of accuracy and finish. When the appropriate size of machine is used, the engine lathe can produce both large and small parts. A group of large engine lathes at work is shown in Fig. 6-1. As a contrast in size, a small bench lathe is shown in Fig. 6-2. A small high-precision toolroom lathe is shown in Fig. 6-3. The lathe shown in Fig. 6-4 is a sliding bed gap lathe which is also called a gap bed lathe. The bed of this lathe can be moved over the base to form an opening, as shown, which provides extra swing to allow the lathe to handle larger and odd shaped parts. Sliding the bed open also provides a greater bed length to allow it to handle longer parts. When the bed is closed, it functions as a



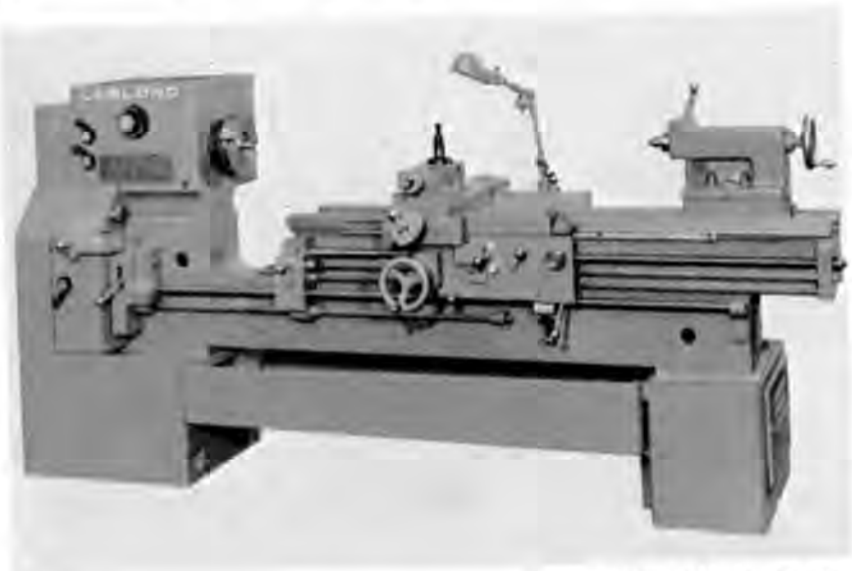
Courtesy of Hardinge Brothers Inc.

Fig. 6-3. Precision toolroom lathe.

regular engine lathe. A hollow spindle lathe is shown in Fig. 6-5. Parts can be turned on this lathe that are considerably longer than the normal center distance by positioning the workpiece through the hollow spindle. The workpiece can be held by chucks on both ends of the spindle. Hollow spindle lathes are used extensively in oil fields for turning and threading the ends of oil pipes.

Principal Features

The principal parts of an engine lathe are shown in Fig. 6-6. The bed of the lathe, Fig. 6-7, is a heavy rigid casting made from gray cast iron. Gray cast-iron is used as a material of construction for machine tool beds and frames because it has excellent vibration dampening qualities and it is a good bearing material for sliding bearings. The bed rests on two legs with the exception of small bench lathes which have the bed



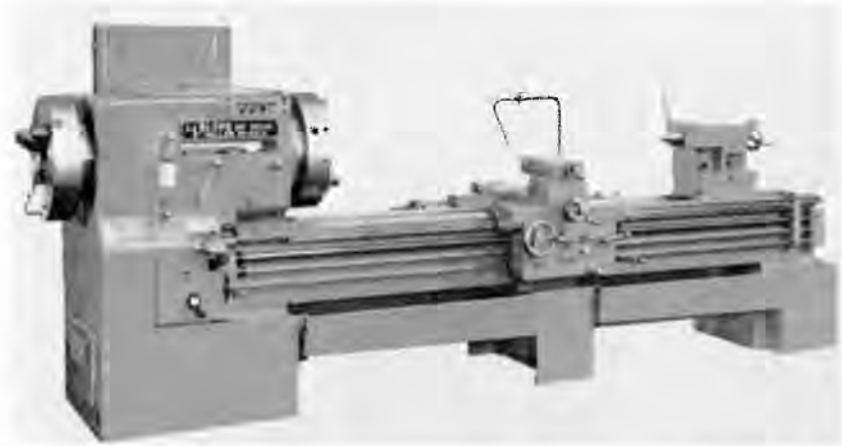
Courtesy of LeBlond Machine Tool Co.

Fig. 6-4. Sliding bedgap lathe.

attached directly to the bench. The top surfaces of the bed are accurately machined to provide a sliding surface for the carriage and to provide for the precise alignment of the headstock and the tailstock. This alignment is achieved by providing "ways" which are raised surfaces machined on the top of the bed. The ways are machined parallel to each other. One of the two ways is used to align the headstock and the tailstock. The other way is used to guide the movement of the carriage as it slides over the bed causing it to be parallel to the axis of the lathe. The direction of this carriage movement is called longitudinal. Some lathes, that shown in Fig. 6-2, for example, have flat beds that do not have ways. The surfaces on the sides of the bed that are perpendicular and adjacent to the top of the bed are carefully machined parallel to each other. These surfaces perform the function of the ways on flat bed lathes.

Methods of Drive

The lathe is driven by an electric motor that is usually located inside the cabinet leg at the headstock end. A multiple V-belt drive transmits the power from the motor to the headstock. The principal function of the headstock is to support and to align the headstock spindle. On many lathes it also contains a selective gear transmission that provides for different speeds of the lathe spindle. A typical geared headstock is shown in Fig. 6-8. The spindle speeds on geared head lathes are changed by placing the spindle speed change levers, located on the side of the headstock, in the position prescribed by the instruction plate located behind



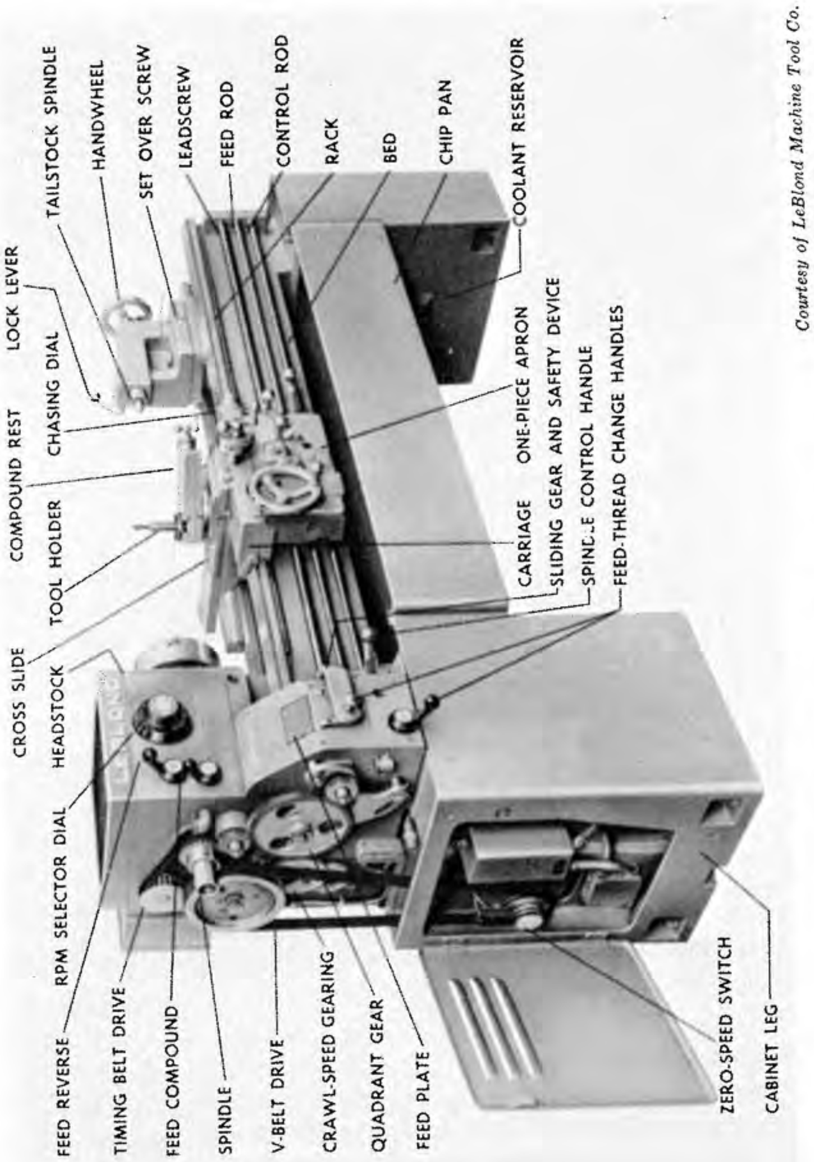
Courtesy of LeBlond Machine Tool Co.

Fig. 6-5. Hollow spindle lathe.

these levers. (See Fig. 6-9.) The speed change mechanism on many smaller lathes consists of stepped pulleys of different diameters as shown in Fig. 6-10. The belt can be on any of the four different pairs of pulleys that provide four different speeds to the lathe spindle. Four additional spindle speeds are obtained through a two-step pulley arrangement on the drive from the electric motor to the countershaft, which contains one of the stepped pulleys, or sheaves, that drives the spindle. Eight additional spindle speeds are obtained by engaging the back gears, thus providing the lathe with a total of 16 spindle speeds. The back gears consist of four gears located inside the headstock that reduce the speed of the stepped pulley in the headstock. When the back gears are engaged this pulley is disconnected from the headstock spindle; when the back gears are disengaged this pulley is connected to the headstock and drives it directly. Another type of headstock construction is shown in Fig. 6-11. A spindle-speed mechanism, located outside the headstock (not shown in Fig. 6-11), may be a stepped pulley arrangement, a gear transmission, or a variable-speed drive. The back-gear drive can be seen in this illustration. It provides the slow spindle speeds.

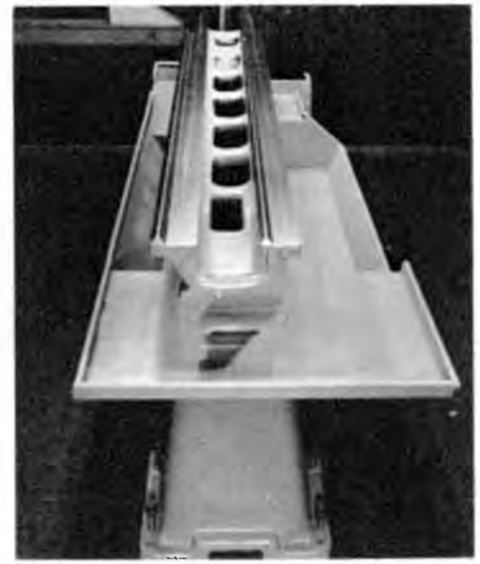
The Headstock Spindle

The lathe headstock spindle is mounted on machine-tool grade anti-friction bearings which can be seen in Fig. 6-11. These bearings are manufactured to closer dimensional tolerances than ordinary anti-friction bearings so that the runout, or eccentricity, of the spindle as it rotates is held to a minimum. The spindle is hollow to allow bar stock and lathe attachments to pass completely through, if desired. The spindle nose is very accurately machined to provide locating surfaces for lathe centers,



Courtesy of LeBlond Machine Tool Co.

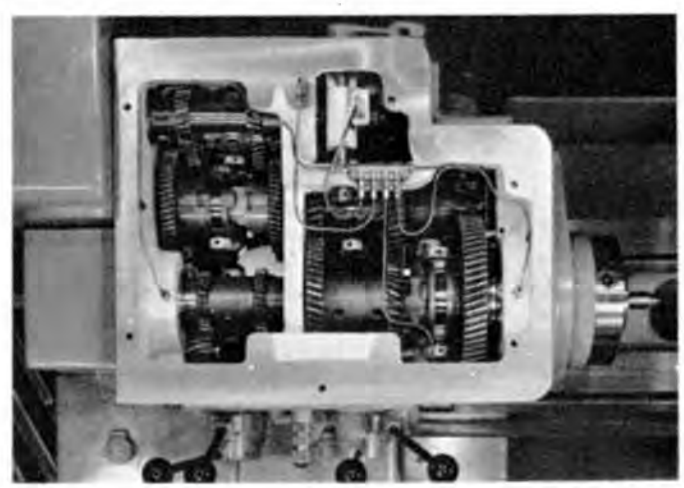
Fig. 6-6. The principal parts of an engine lathe.



Courtesy of The Monarch Machine Tool Company

Fig. 6-7. Lathe bed showing the ways of the lathe.

faceplates, and chucks. The inside of the spindle nose (Fig. 6-12) has a taper which is ground to a high degree of accuracy and surface finish. This taper provides a seat for an adapter which is used to reduce the size of the taper required by the headstock center. The headstock center



Courtesy of The Monarch Machine Tool Company

Fig. 6-8. Geared type headstock.

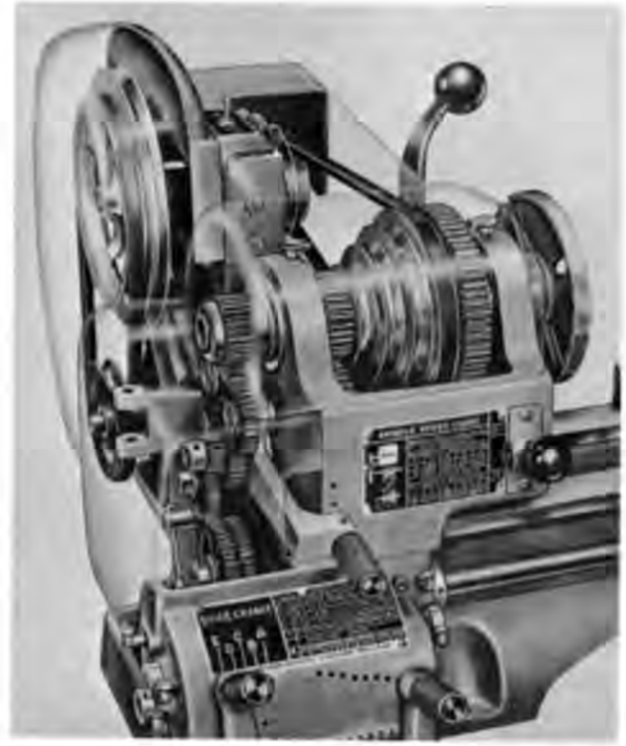
Courtesy of The Monarch Machine Tool Company

Fig. 6-9. Geared type headstock with quick-change gearbox.

is held by the taper in the adapter. The construction of the inside of the spindle nose for all engine lathes is the same, except for size. There are three different designs for the outside of the spindle noses, and these are shown in Fig. 6-13. At the left is the long, tapered type of spindle nose. The spindle nose mountings (chucks and faceplates) are located by the taper and are driven by the key. The threaded ring screws onto a matching thread on the chuck and faceplate, and it holds them securely on the taper. The center view (Fig. 6-13) shows a cam-lock type of spindle nose. It has a short, tapered surface which locates the spindle mountings. The chucks and faceplates have a cam stud (see Fig. 6-12) which fits into each of the holes located above the taper. The cam is turned by a chuck key, to hold the spindle mounting firmly, on the short taper. The cam studs also act as drivers for the mounting. A threaded spindle nose is shown in the view to the right, in Fig. 6-13. The spindle nose mounting is held in place and is driven by the threads. The shoulder behind the threads is used to help locate the mounting on the spindle nose. The spindle nose should be kept free of nicks and scratches, and it should be carefully cleaned before a chuck or a face is mounted on it. Also, the corresponding surfaces of the chuck or faceplate must be clean and kept free from any damage.

The Quick-Change Gearbox

The quick-change gearbox is located at the left side of the lathe below the headstock (see Figs. 6-9 and 13-11 in Chapter 13). It is connected to the spindle by a gear train as shown in Fig. 6-14. The gears in this train are called pick-off gears because they can be interchanged rapidly. Some of these gears are used to cut threads that cannot be cut with the standard pick-off gears and the settings of the quick-change gearbox.

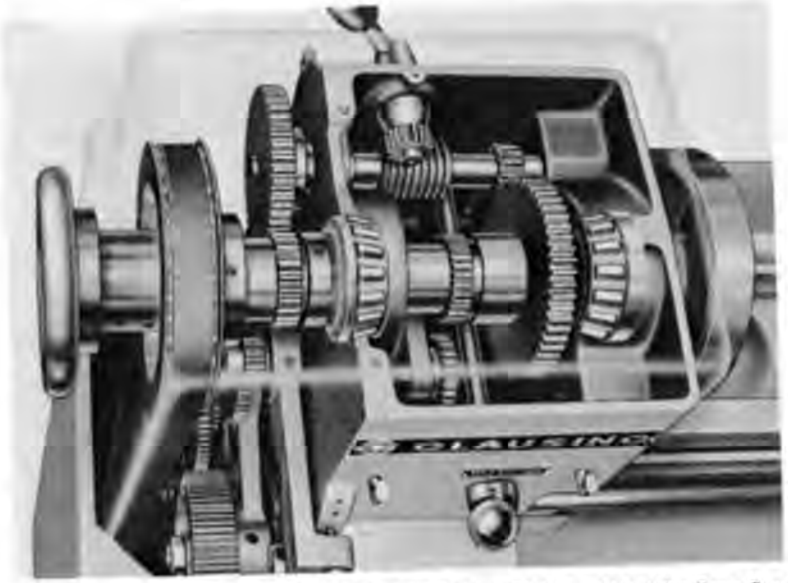


Courtesy of the Atlas Press Company

Fig. 6-10. Stepped pulley type headstock.

The function of the quick-change gearbox is to provide the lathe with the capability of cutting different screw threads and operating at different feed rates. Invented by the American, Frederick B. Miles, and later improved by another American, Wendell P. Norton, it was for many years called the "*Norton Quick-Change Gearbox*." It consists of a tumbler gear which can be positioned to engage any one of a cluster of different size gears to provide a variety of speed ratios. The tumbler gear is engaged by a lever located in front of the gearbox. Often, to provide additional speed ratios, a second set of gears is used and still another, third set can be used to reverse the direction of rotation of the leadscrew and the feed rod. The required number of threads per inch—or thread pitch—and the required feed rate are obtained by positioning the handles according to instructions provided on the quick-change gearbox.

The output shafts of the feed-thread gearbox are the feed rod and the lead screw. The function of the feed rod is to actuate the automatic-power feed mechanism located in the lathe apron. The lead screw is used to feed the carriage in the longitudinal (parallel to the lathe axis) direction when cutting threads. It should be used only to cut threads in order not to impair its accuracy as the result of wear and the imposition of



Courtesy of the Clausing Division of the Atlas Press Company

Fig. 6-11. Modern headstock showing back gears and antifriction spindle bearings.

heavy feeding forces. Some lathes (Fig. 6-6) have a control rod located below the lead screw and feed rod which is used to start and stop the headstock spindle. This rod is actuated by the spindle control handle.

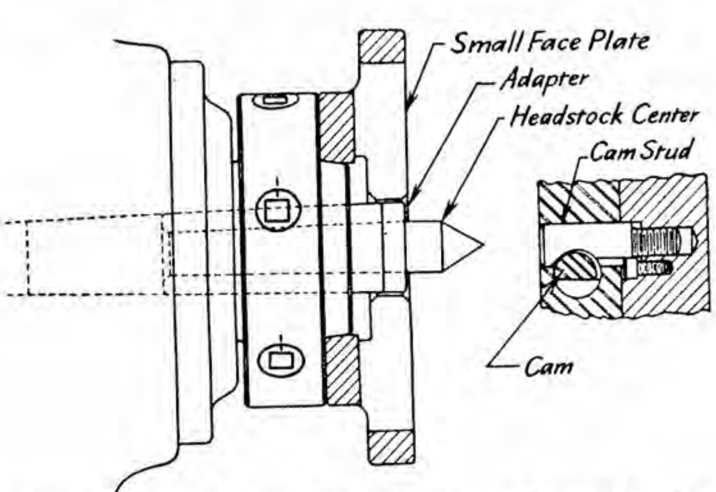


Fig. 6-12. Cam-lock type spindle nose showing adapter and headstock center in place.

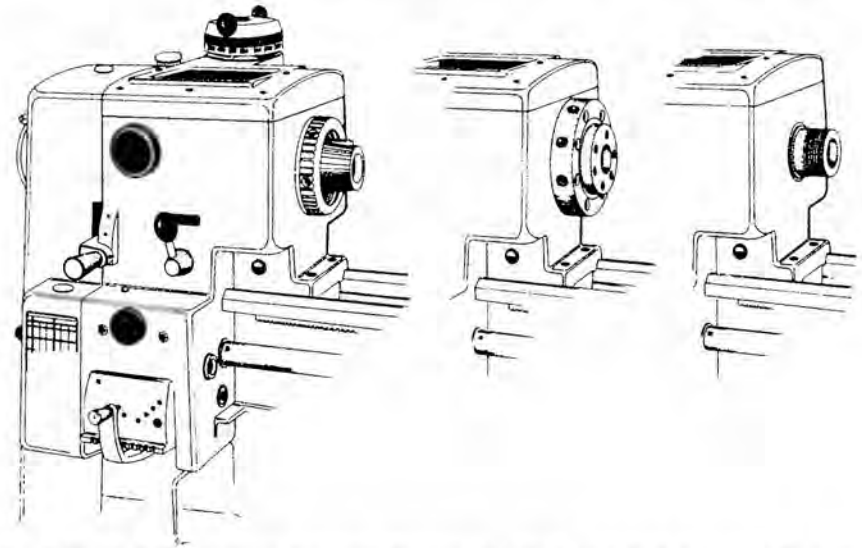


Fig. 6-13. (Left) Long taper type spindle nose. (Center) Cam-lock type spindle nose. (Right) Threaded spindle nose.

The Carriage

The carriage, Fig. 6-15, consists of the saddle, cross slide, compound rest, and apron. The saddle, Fig. 6-16, is an H-shaped casting that slides on the top of the lathe bed. It enables the cutting tool to be moved parallel or perpendicular to the axis of the lathe. The saddle rides on one of the ways on the lathe bed which guides the longitudinal movement of the carriage parallel to the lathe axis.

The Cross Slide

The cross slide moves over the dovetail slide on top of the saddle. The movement of the cross slide is perpendicular to the lathe axis. This movement is actuated by a feed screw, shown in Fig. 6-17. The feed screw can be turned manually by means of the cross-feed handle. The automatic cross feed is actuated by a gear in the apron.

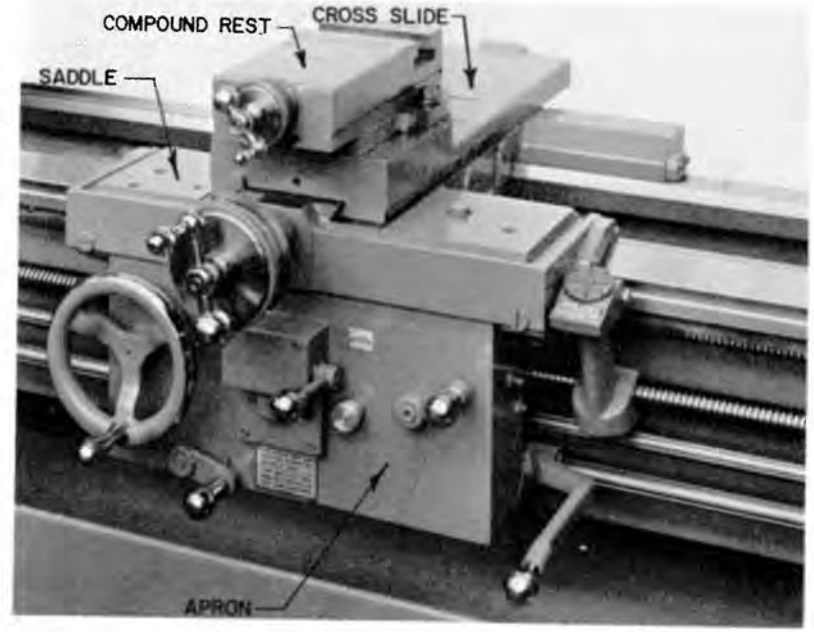
Mounted behind the cross-feed handle is a micrometer cross-feed dial that has accurate graduations cut on its circumference which enables the lathe operator to determine the distance that the cross slide is moved. On many lathes each graduation corresponds to a movement of .0005 inch of the carriage which causes .001 inch to be removed from the diameter of the workpiece that is being turned, because a cut is taken from both sides of the workpiece as it revolves. Cross-feed dials graduated in this manner are said to be "direct reading" dials. For example, if .050 inch is to be removed from the diameter of the workpiece the cross-feed dial is moved 50 graduations. On some lathes the graduations correspond

Courtesy of LeBlond Machine Tool Co.

Fig. 6-14. Front end of engine lathe showing gear train from headstock spindle to quick-change gearbox. The pick-off gears in the train can be quickly removed and replaced by other gears.

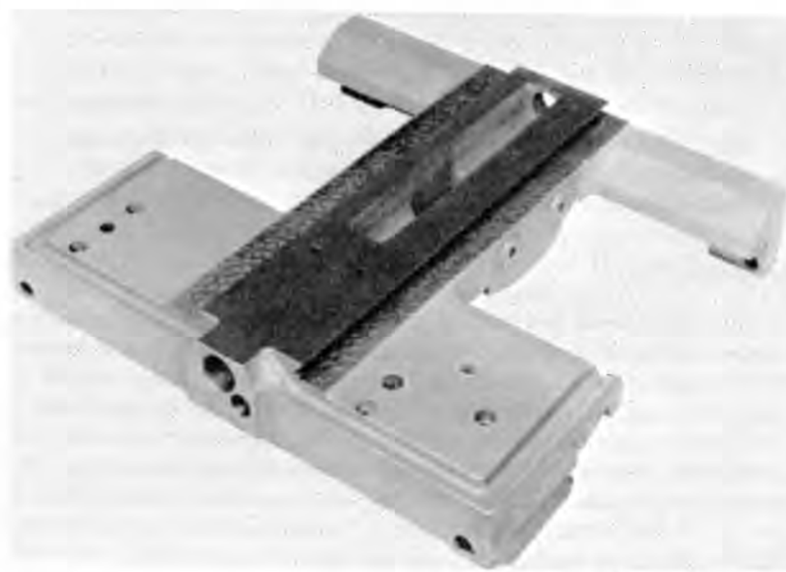
to a movement of .001 inch, thereby removing .002 inch from the diameter of the workpiece. In this case the amount to be removed from the diameter of the workpiece must be divided by two in order to determine the number of graduations through which the cross-feed dial must be moved. For example, if .050 inch is to be removed from the diameter of the workpiece, the cross-feed dial is moved 25 graduations.

Clearance must be provided between the cross-feed screw and the cross-feed nut (Fig. 6-17) in order to allow the screw to rotate without binding. This clearance is intentional although it results in a certain amount of lost motion between the nut and the screw. Thus, when the cross-feed screw is turned in one direction and then in the opposite direction, it will not move the carriage until the lost motion is taken up. In order to accommodate this lost motion, the cutting tool must be positioned by always moving the cross slide toward the workpiece. If the



Courtesy of LeBlond Machine Tool Co.

Fig. 6-15. Principal parts of the lathe carriage.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-16. Lathe saddle.

Courtesy of The Monarch Machine Tool Company

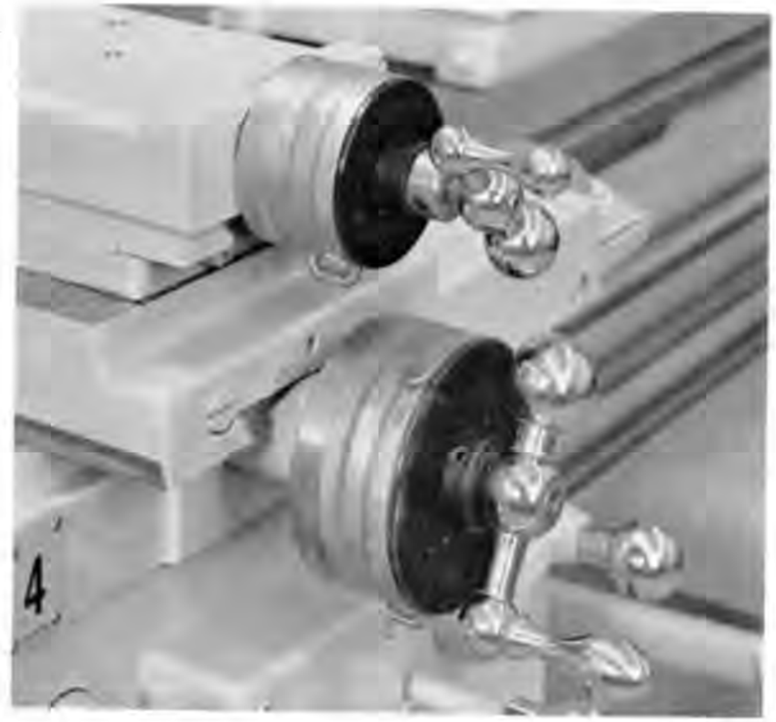
Fig. 6-17. Cross feed screw.

cutting tool must be positioned an exact distance in a direction away from the workpiece axis, the cross slide should be first moved back beyond the desired tool position and then moved forward toward the workpiece until the desired position is reached as determined by the cross-feed dial.

Many lathes are equipped with dual reading micrometer dials on the cross feed and on the compound rest screws as shown in Fig. 6-18. Dual reading dials have two scales, one of which is graduated in inches and the other in millimeters. The dials shown in Fig. 6-18 have one scale reading in .001-inch increments while the other reads 0.02 millimeter per division. Both scales are direct reading. The scales are located on separate dials which do not rotate at the same speed; one dial drives the other through an internal gear train located inside the housing. If the feed screw has an inch thread, the inch dial will rotate at the same speed as the feed screw; otherwise it is the metric dial that will rotate at the same speed as the metric feed screw.

The Compound Rest

The compound rest is clamped to the cross slide and its base can turn through an angle of 360 degrees when the clamp nuts are loosened. The base has graduations enabling it to be clamped at any angle with respect to the lathe axis. The compound-rest slide is moved over the base by a feed screw that is similar in design to the cross-feed screw. The compound-rest feed can only be actuated manually, by turning the compound-rest feed handle. A micrometer dial permits making accurate movements of the compound-rest slide. The compound-rest slide has a large T slot which is used to clamp a tool post or tool block. The cutting tool, or the cutting tool holder, is clamped in the tool post, or tool block. The function of the compound rest is to provide a movement of the cutting tool at any desired angle, with respect to the axis of the work.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-18. Dual inch-metric micrometer dials on cross feed and compound-rest screws.

The Apron

The apron of the lathe, Fig. 6-19, is fastened to the underside of the saddle. It is essentially a housing that contains the gears and clutches required to actuate the automatic power, longitudinal and croos-feeds. It also supports a large handwheel that actuates the manual longitudinal feed and a lever, shown at the far right, called the "split-nut" lever, which actuates a split nut. The split nut engages the lead screw and drives the carriage when cutting threads. It is disengaged when the thread has been cut to the desired length in order to stop the motion of the carriage. The split-nut lever should never be used to feed the carriage to take any cut other than cutting screw threads. The lever in the center of the apron is used to selectively engage either the cross feed or the longitudinal feed. The longitudinal feed is actuated by a small pinion at the rear of the apron that engages a rack (see Fig. 6-15) which is located on the side of the bed above the lead screw.

The Tailstock

The tailstock is used to support long workpieces and also to hold drills, reamers, and other cutting tools. A drawing of a tailstock showing

Courtesy of LeBlond Machine Tool Co.

Fig. 6-19. Lathe apron.

all of its parts is illustrated in Fig. 6-20. It consists of a base that rests on the bed of the lathe and a body that is clamped to the base. The base, which is aligned with the axis of the lathe by one of the ways of the lathe, has an accurately machined guide that fits the body. The body and the base will slide over the lathe bed until it is in the desired position, then it is clamped to the bed by tightening the clamp bolt nut. The tailstock spindle is prevented from turning, by a key. When the tailstock handwheel is turned, the screw causes the tailstock spindle to slide in or out of the body. The spindle may be clamped by turning the binder handle. The inside of the spindle nose has a taper which is used to hold the tailstock center, drill chucks, or cutting tools such as drills or reamers. This taper must be absolutely clean before a center or other tool is inserted in it. It serves, primarily, to locate; its ability to resist torsion is very limited. It should not be used to hold large drills because the cutting torque of the drill may cause the taper to slip and possibly score the surface. When this occurs, the taper will no longer locate the center or the tool accurately, and it will no longer be able to hold. The repair of a tailstock spindle is a very expensive procedure. Scored tailstock spindle tapers cannot always be repaired, thus requiring the replacement of the entire spindle. The centers, drill chucks, and cutting tools are removed from the tailstock spindle by backing the spindle into the body until the object that is held in the spindle bumps against the plug end of the screw which forces it out. The body of the tailstock can be moved a limited distance perpendicular to the axis of the lathe by unclamping the clamp bolt nut and turning two adjusting screws located on each side of the base. This movement is used to accurately align the headstock and the tailstock spindles. It can also be used to turn tapers. An

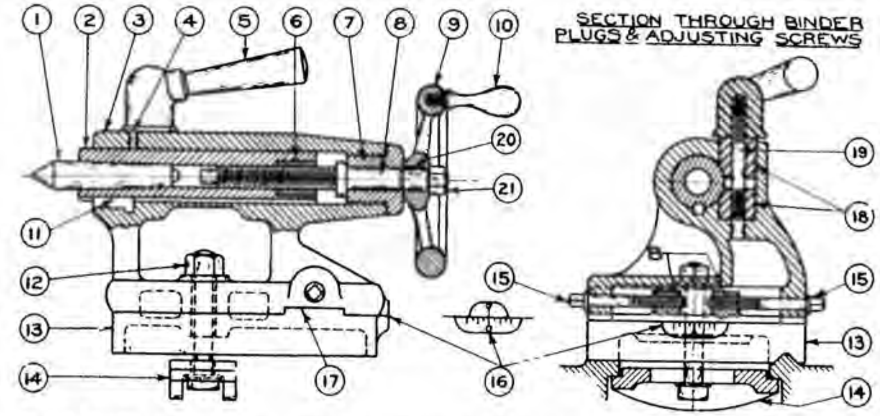


Fig. 6-20. Details of engine lathe tailstock: 1. Tailstock center; 2. Spindle; 3. Body of tailstock (top); 4. Oil hole; 5. Binder handle; 6. Spindle nut; 7. Screw bushing; 8. Screw; 9. Handwheel; 10. Machine handle; 11. Key for spindle; 12. Clamp bolt; 13. Base; 14. Clamp; 15. Adjusting screws; 16. Alignment scale; 17. Guide; 18. Binder plugs; 19. Binder stud; 20. Woodruff key; and 21. Handwheel nut.

alignment scale on the back of the tailstock gives a visual indication of the amount of offset or indicates when the spindle is properly aligned. This scale should not, however, be depended upon for great accuracy.

Engine Lathe Attachments

Many attachments have been developed to increase the usefulness and the accuracy of the engine lathe. Some of these attachments will be discussed in greater detail in subsequent chapters. A group of frequently used engine lathe attachments is shown in Fig. 6-21. A collet chuck and a group of collets are shown at the top of this illustration. A three-jaw universal chuck is shown in the lower left side. Separate jaws for holding smaller outside diameters, and for holding on inside diameters, are shown. The jaws of this chuck are actuated by the chuck key. They move in and out together and hold the round workpiece in a central location without the need for further adjustment. A four-jaw independent chuck is shown in the lower right-hand view. Each jaw is adjusted independently of the other jaws by the chuck key which is the T-shaped key adjacent to the chuck. The jaws can be reversed in the chuck body in order to grip onto inside or outside surfaces. A drill chuck is shown between the three- and four-jaw chucks. The drill chuck is used to hold straight-shank twist drills, reamers, and other cutting tools. It is usually held in the tailstock spindle when used in a lathe. A large faceplate is shown in Fig. 6-22. Workpieces are bolted directly to faceplates to perform various turning and facing operations. Lathe fixtures are also bolted to faceplates. A lathe fixture is a special device to hold and to locate workpieces, in a lathe, that cannot be held by other methods.

Courtesy of The Monarch Machine Tool Company

Fig. 6-21. Engine lathe attachments.

Courtesy of LeBlond Machine Tool Co.

Fig. 6-22. Large faceplate for holding workpieces that cannot conveniently be held by other methods.

A micrometer carriage stop is shown in Fig. 6-23. It is used to obtain accurate positions of the carriage and to move it exact distances, longitudinally. A length rod is extended from the headstock. It can be accu-

Courtesy of The Monarch Machine Tool Company

Fig. 6-23. A micrometer carriage stop.

rately moved in one-inch increments. A micrometer head at the end of the length rod is used to make fine adjustments of less than one inch. The dial test indicator, which is mounted on the carriage, is used to position the carriage so that it will always be at the same distance from the end of the length rod.

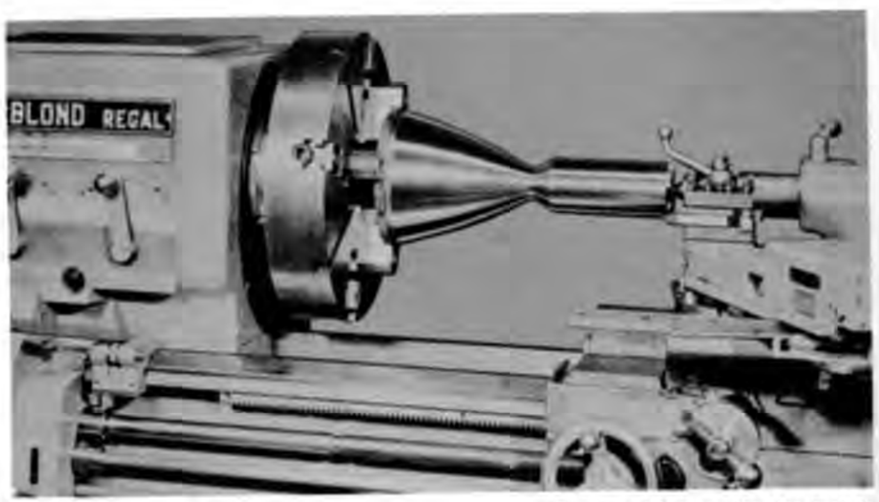
A large steady rest is shown in Fig. 6-24. Steady rests are used to support long and slender parts. The steady rest also permits an unobstructed access to the ends of parts so that facing and boring operations can be performed on these ends. The steady rest is composed of a frame containing three jaws. These jaws form a three-point bearing surface and they can be adjusted in or out, radially, by turning the adjusting screws. The frame is hinged on one side allowing the upper half to swing open for inserting or removing the work. The base of the frame has V-grooves in it that fit the ways of the lathe bed. When the steady rest is in use, it is clamped to the bed and the jaws are set against the work thus supporting, or steadying it, during the operation.

Tracer attachments can be used to turn and to bore irregular contours, as shown in Fig. 6-25. They can also be used to cut stepped shafts with shoulders and to turn tapers. Internal tapers, internal shoulders, and counterbores can be readily cut with this attachment. The tracer attachment is mounted on the carriage in place of the compound rest, as can be seen in Fig. 6-26. A template with the profile of the part to be turned is mounted on the carriage but is prevented from moving with the carriage. The stylus of the tracer attachment follows the profile of the template which causes the cutting tool to turn this profile on the part in a manner to be described.

The regular cross slide is removed from the lathe and replaced by one having a top slide (similar to the compound rest) mounted on it. This slide is actuated by a hydraulic cylinder which has its movement controlled by a stylus. The nose of the stylus must have the same shape as

Courtesy of LeBlond Machine Tool Co.

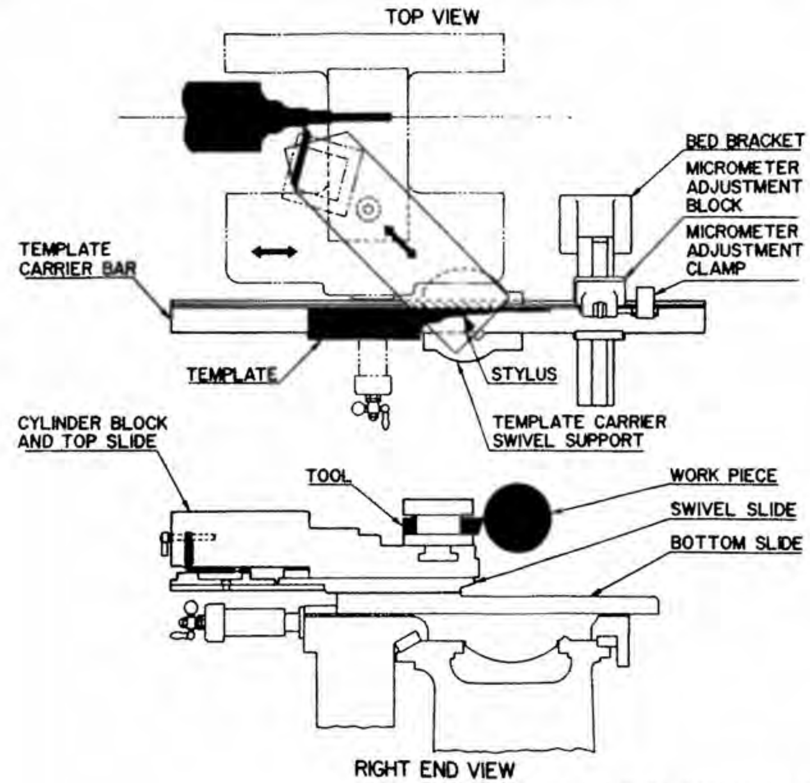
Fig. 6-24. **Boring a large workpiece in a lathe with the end held by a steady rest.**



Courtesy of LeBlond Machine Tool Co.

Fig. 6-25. Part turned on engine lathe equipped with tracer attachment.

the nose of the cutting tool. The stylus follows the outline of the template which is mounted on the cross slide and fastened to the bed of the lathe. The position of the template can be adjusted by the micrometer adjustment clamp.



Courtesy of LeBlond Machine Tool Co.

Fig. 6-26. Principle of operation of tracer control attachment.

In operation, the first diameter is cut to the required size by taking a trial cut, using the cross slide to set the tool to the required depth of cut; then the stylus is brought into contact with the template and the automatic power, longitudinal feed is engaged. (The trial cut can usually be eliminated on subsequent parts.) The stylus will then pilot the cutting tool causing it to reproduce the template outline on the workpiece in the following manner: the longitudinal feed moves the carriage toward the tailstock at a steady rate; when the stylus detects a change in the diameter of the part, or a shoulder, the top slide will move in or out at an angle, with respect to the axis of the workpiece. This angular movement of the top slide may be considered as being made up of two component motions: one parallel to the lathe axis and one perpendicular to it. For example, when a square shoulder is encountered, the top slide will move at an angle away from the work axis. The rate of movement of that portion of the angular motion of the top slide that is parallel to the lathe axis will be equal and opposite to the longitudinal feed rate. Since these opposite movements are equal and parallel to the lathe axis the tool will, in effect, remain stationary with respect to this direction of

Courtesy of LeBlond Machine Tool Co.

Fig. 6-27. Tool post attachment grinding a lathe center.

movement. However, it will be carried away from the axis of the work by that portion of the top slide motion that is moving away from the work axis. Thus, the tool is caused to feed out just as if the cross feed were being used. Contours are cut by making that portion of the angular movement that is parallel to the lathe axis, operate at a slower or faster rate than the longitudinal feed rate, while the tool is moving toward or away from the work axis.

A tool post grinding attachment is shown grinding a hardened lathe center in Fig. 6-27. Workpieces which have been heat treated to have a high hardness can only be machined by grinding. While most cylindrical and internal grinding is done on machines designed for that purpose, there are occasions when external and internal cylindrical and tapered surfaces are ground on an engine lathe.

CHAPTER 7

Single-Point Cutting Tools and Their Performance

Single-point cutting tools are used on engine lathes, shapers, and planers. Boring tools, which are also single-point cutting tools, are used on a large variety of power machine tools to locate and enlarge holes. The performance of these tools has a direct relation to the quality of the work done and the rate at which it is produced. Treatment of this subject will include discussions of: cutting speeds, single-point cutting tool nomenclature, tool shapes, chip formation, chip control, and tool wear. An understanding of these topics will form a necessary foundation for the successful application of these tools in the shop. While the machinist and the toolmaker can only use those machine tools available to them in their respective shops, they will frequently have the responsibility of selecting the best cutting tool and cutting-tool shape available; and, they must know the correct cutting speed and feed with which these tools are to be used.

Hardness and Hardness Testing

Three factors have a very decisive influence on the cutting speed that can be used:

1. The cutting tool material.
2. The material that is to be cut.
3. The metallurgical condition of the material when it is being cut.

One indication of the metallurgical condition of a metal is its hardness; technically this is usually measured by determining its resistance to penetration. The most common hardness-testing machines used in the United States are the Brinell hardness tester and the Rockwell hardness tester. The diameter of the impression made by a steel ball is measured and this measurement is converted to a Brinell hardness number, abbreviated HB, which defines its hardness. Harder metals have a higher HB. There are several Rockwell hardness scales. One which is frequently encountered in measuring the hardness of hardened ferrous metals is the Rockwell C Scale, that is abbreviated HRC. It, too, has the higher numbers indicating higher hardnesses. In general, the Brinell hardness reading is given for softer ferrous metals and the Rockwell reading is given for hardened ferrous metals, although there is a range in which they overlap.

The strength of a metal increases or decreases roughly in proportion to its hardness. It is not surprising then to discover that harder metals must be cut more slowly than softer metals. The hardness of any piece of soft metal can be increased by *cold drawing* or *cold rolling*. Annealing and normalizing are heat treating processes used to decrease the hardness of steel and cast iron although normalizing may be used to increase the hardness of a piece of steel that has been previously annealed. Most steels, when heated above a specific temperature and then quenched and tempered, will have their hardness increased. *Solution treating and aging* is another heat treatment process which is used to increase the hardness of some metals, particularly aluminum. The details of these processes are a subject in the study of metallurgy; however, the machinist and toolmaker must be aware that the condition of the metals that they are cutting is affected by these processes and that any particular type of metal can have different hardnesses. In fact, it is quite common to have a variation in hardness in a *single* piece of metal. It should now be clear that a cutting speed cannot be given for a specific type of metal without also specifying the hardness. The cutting speeds in Tables 7-1 through 7-5 are, therefore, listed not only for a given metal but also for the different hardnesses at which the metal is commonly cut.

Steels used in the construction of machinery are commonly designated by AISI (American Iron and Steel Institute) numbers. Tool steels are specified by a letter to identify the type, which is followed by a number, such as shown in Table 7-3. Other metals have similar methods of specification. These specifications are in common use in industry and they are used in this book.

Cutting Speed

The cutting speed is given in terms of feet per minute (fpm) in the customary inch system of units. Sometimes this is called surface feet per minute (sfpm). In the metric system of units the cutting speed is specified in meters per minute (m/min). Tables 7-1 through 7-5 give the recommended cutting speeds for cutting many materials in terms of feet per minute. These cutting speeds must be considered to be a starting point which may have to be modified as determined by actual experience on the job. When only a few parts are to be machined these speeds may be used without change, but when machining many parts that are alike and when maximum production must be obtained, a modification may be desirable. Many factors affect the cutting speed which may combine to warrant an increase or sometimes a decrease in the cutting speed. These factors will now be considered.

Tool Material. The type of material used to make the cutting tool is of prime importance with respect to the cutting speed used to cut any material. Cutting tool materials are treated in greater detail elsewhere

Table 7-1. Recommended Cutting Speeds in Feet per Minute for Turning Plain Carbon and Alloy Steels

Material AISI and SAE Steels)	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Free Machining Plain Carbon Steels (Resulphurized), 1212, 1213, 1215	100-150	HR, A	150	600
	150-200	CD	160	625
1108, 1109, 1115, 1117, 1118, 1120 1126, 1211	100-150	HR, A	130	500
	150-200	CD	120	525
1132, 1137, 1139, 1140, 1144, 1146, 1151	175-225	HR, A, N, CD	120	400
	275-325	Q and T	75	300
	325-375	Q and T	50	225
	375-425	Q and T	40	200
Free Machining Plain Carbon Steels (Leaded), 11L17, 11L18, 12L13, 12L14	100-150	HR, A, N, CD	140	550
	150-200	HR, A, N, CD	145	560
	200-250	N, CD	110	400
Plain Carbon Steels, 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	100-125	HR, A, N, CD	120	450
	125-175	HR, A, N, CD	110	400
	175-225	HR, N, CD	90	350
	225-275	CD	70	300
1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	125-175	HR, A, N, CD	100	375
	175-225	HR, A, N, CD	85	325
	225-275	N, CD, Q and T	70	225
	275-325	Q and T	60	200
	325-375	Q and T	40	160
	375-425	Q and T	30	140
1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125-175	HR, A, N, CD	100	370
	175-225	HR, A, N, CD	80	320
	225-275	N, CD, Q and T	65	220
	275-325	Q and T	50	180
	325-375	Q and T	35	150
	375-425	Q and T	30	130
Free Machining Alloy Steels (Resulphurized), 4140, 4150	175-200	HR, A, N, CD	110	400
	200-250	HR, N, CD	90	350
	250-300	Q and T	65	300
	300-375	Q and T	50	225
	375-425	Q and T	40	165
Free Machining Alloy Steels (Leaded), 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40 *	150-200	HR, A, N, CD	120	430
	200-250	HR, N, CD	100	380
	250-300	Q and T	75	275
	300-375	Q and T	55	220
	375-425	Q and T	50	200
Alloy Steels, 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125-175	HR, A, N, CD	100	400
	175-225	HR, N, CD	90	350
	225-275	CD, N, Q and T	70	300
	275-325	Q and T	60	250
	325-375	Q and T	50	200
	375-425	Q and T	35	175

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.
* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; and HB, Brinell hardness number.

Table 7-1 (Concluded). Recommended Cutting Speeds in Feet per Minute for Turning Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Alloy Steels, 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30	175-225 225-275 275-325 325-375 375-425	HR, A, N, CD N, CD, Q and T N, Q and T N, Q and T Q and T	85 70 60 40 30	325 275 230 200 150
Alloy Steels, E51100, E52100	175-225 225-275 275-325 325-375 375-425	HR, A, CD N, CD, Q and T N, Q and T N, Q and T Q and T	70 65 50 30 20	310 260 220 180 140
Ultra High Strength Steels (Not AISI) AMS 6421 (98B37 Mod.), AMS 6422 (98BV40), AMS 6424, AMS 6427, AMS 6428, AMS 6430, AMS 6432, AMS 6433, AMS 6434, AMS 6436, AMS 6442, 300M, D6ac	220-300 300-350 350-400 43-48 HRC 48-52 HRC	A N N Q and T Q and T	65 50 35 25 10	270 200 150 120 80
Maraging Steels (Not AISI) 18% Ni Grade 200 18% Ni Grade 250 18% Ni Grade 300 18% Ni Grade 350	250-325 50-52 HRC	A Maraged	60 10	300 80
Nitriding Steels (Not AISI) Nitralloy 125 Nitralloy 135 Nitralloy 135 Mod. Nitralloy 225 Nitralloy 230 Nitralloy N Nitralloy EZ Nitrex 1	200-250 300-350	A N, Q and T	70 30	300 225

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.
* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

in this chapter; therefore, only a few words will suffice here. The tables list the cutting speeds for use with high-speed steel and cemented carbide cutting tools. Although there are slight differences in the cutting speed that can be used with tools made of different types of high-speed steel, these differences are generally small. A greater difference exists between different grades of cemented carbides and it is essential that the correct grade be used. The tables are based on the assumption that the correct grade of uncoated carbide is used. Coated carbides cannot be used to cut

Table 7-2. Recommended Cutting Speeds in Feet per Minute for Turning Tool Steels

Material Tool Steels (AISI Types)	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Water Hardening W1, W2, W5	150-200	A	100	325
Shock Resisting S1, S2, S5, S6, S7	175-225	A	70	300
Cold Work, Oil Hardening O1, O2, O6, O7	175-225	A	70	250
Cold Work, High Carbon High Chromium D2, D3, D4, D5, D7	200-250	A	45	175
Cold Work, Air Hardening A2, A3, A8, A9, A10	200-250	A	70	250
A4, A6	200-250	A	55	200
A7	225-275	A	45	175
Hot Work, Chromium Type H10, H11, H12, H13, H14, H19	150-200 200-250 325-375 48-50 HRC 50-52 HRC 52-54 HRC 54-56 HRC	A A Q and T Q and T Q and T Q and T Q and T	80 65 50 20 10 — —	300 225 175 95 80 60 40
Hot Work, Tungsten Type H21, H22, H23, H24, H25, H26	150-200 200-250	A A	60 50	250 200
Hot Work, Molybdenum Type H41, H42, H43	150-200 200-250	A A	55 45	225 175
Special Purpose, Low Alloy L2, L3, L6	150-200	A	75	325
Mold P2, P3, P4, P5, P6	100-150	A	90	400
P20, P21	150-200	A	80	350
High Speed Steel M1, M2, M6, M10, T1, T2, T6	200-250	A	65	225
M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8	225-275	A	55	200
T15, M3-2	225-275	A	45	170

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.
* Abbreviations designate: A, annealed; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

all materials; however, where they can be used, they can usually be operated at a 20 to 40 percent faster cutting speed than given in the tables and sometimes by as much as 50 percent. Since there are so many grades of cemented carbides, the selection of a specific grade should be made by obtaining the recommendations of a carbide producer.

Work Material. Of equal importance in selecting the cutting speed is the

Table 7-3. Recommended Cutting Speeds in Feet per Minute for Turning Stainless Steels

Material Stainless Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm HSS	Cutting Speed, fpm Carbide
Free Machining Stainless Steels				
(Ferritic), 430F, 430F Se	135-185	A	110	400
(Austenitic), 203EZ, 303, 303Se, 303MA, 303Pb, 303Cu, 303 Plus X	135-185	A	100	350
	225-275	CD	80	325
(Martensitic); 416, 416Se, 416 Plus X, 420F, 420F Se, 440F, 440F Se	135-185	A	110	400
	185-240	A, CD	100	350
	275-325	Q and T	60	250
	375-425	Q and T	30	125
Stainless Steels				
(Ferritic), 405, 409, 429, 430, 434, 436, 442, 446, 502	135-185	A	90	300
(Austenitic), 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135-185	A	75	225
	225-275	CD	65	200
(Austenitic), 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	135-185	A	70	225
(Martensitic), 403, 410, 420, 501	135-175	A	95	350
	175-225	A	85	300
	275-325	Q and T	55	200
	375-425	Q and T	35	125
(Martensitic), 414, 431, Greek Ascoloy	225-275	A	60	250
	275-325	Q and T	50	200
	375-425	Q and T	30	125
(Martensitic), 440A, 440B, 440C	225-275	A	55	200
	275-325	Q and T	45	150
	375-425	Q and T	30	125
(Precipitation Hardening) 15-5PH, 17-4PH, 17-7PH, AF-71, 17-14CuMo, AFC-77, AM-350, AM-355, AM-362, Custom 455, HNM, PH13-8, PH14-8Mo, PH15-7Mo, Stainless W	150-200	A	60	225
	275-325	H	50	200
	325-375	H	40	130
	375-450	H	25	90

Based on a feed rate of .012 in. per rev. and a depth of cut of .125 in.
* Abbreviations designate: A, annealed; CD, cold drawn; Q and T, quenched and tempered; H, precipitation hardened; and HB, Brinell hardness number.

work material. Each work material will have a range of cutting speeds at which it can be cut. The cutting speeds recommended in this chapter are based on the expectancy that the material can be cut at the given speed with a reasonable tool life. It has already been explained that most metals can have a different hardness, depending on how the metal has been heat treated or cold worked. For most materials, the cutting speeds in the tables are given for several different hardness levels. Since the hardness is not always known by the machine operator, the material condition associated with the hardness is provided in a separate column. In this way the machinist might estimate the hardness. When in doubt, start at the lower cutting speed and then see if the workpiece can be cut

Table 7-4. Recommended Cutting Speeds in Feet per Minute for Turning Ferrous Cast Metals

Material Ferrous Cast Metals	Hardness, HB*	Material Condition*	Cutting Speed, fpm HSS	Cutting Speed, fpm Carbide
Gray Cast Iron				
ASTM Class 20	120-150	A	120	450
ASTM Class 25	160-200	AC	90	350
ASTM Class 30, 35, and 40	190-220	AC	80	275
ASTM Class 45 and 50	220-260	AC	60	200
ASTM Class 55 and 60	250-320	AC, HT	35	125
ASTM Type 1, 1b, 5 (Ni Resist)	100-215	AC	70	225
ASTM Type 2, 3, 6 (Ni Resist)	120-175	AC	65	210
ASTM Type 2b, 4 (Ni Resist)	150-250	AC	50	200
Malleable Iron				
(Ferritic), 32510, 35018	110-160	MHT	130	500
(Pearlitic), 40010, 43010, 45006, 45008, 48005, 50005	160-200	MHT	95	400
	200-240	MHT	75	275
(Martensitic), 53004, 60003, 60004	200-255	MHT	70	250
(Martensitic), 70002, 70003	220-260	MHT	60	225
(Martensitic), 80002	240-280	MHT	50	140
(Martensitic), 90001	250-320	MHT	30	125
Nodular (Ductile) Iron				
(Ferritic), 60-40-18, 65-45-12	140-190	A	100	450
(Ferritic-Pearlitic), 80-55-06	190-225	AC	80	350
	225-260	AC	65	210
(Pearlitic-Martensitic), 100-70-03	240-300	HT	45	175
(Martensitic), 120-90-02	270-330	HT	30	100
	330-400	HT	15	50
Cast Steels				
(Low Carbon), 1010, 1020	100-150	AC, A, N	110	400
(Medium Carbon), 1030, 1040, 1050	125-175	AC, A, N	100	400
	175-225	AC, A, N	90	350
	225-300	AC, HT	70	300
(Low Carbon Alloy), 1320, 2315, 2320 4110, 4120, 4320, 8020, 8620	150-200	AC, A, N	90	350
	200-250	AC, A, N	80	325
	250-300	AC, HT	60	250
(Medium Carbon Alloy), 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175-225	AC, A, N	80	325
	225-250	AC, A, N	70	280
	250-300	AC, HT	55	250
	300-350	AC, HT	45	220
	350-400	HT	30	150

Based on a feed rate of .012 inch per revolution and a depth of cut of .125 inch.
* Abbreviations designate: A, annealed; AC, as cast; N, normalized; HT, heat treated; MHT, malleablizing heat treatment; and HB, Brinell hardness number.

Table 7-5. Recommended Cutting Speed, in Feet Per Minute, for Turning Nonferrous Metals and Alloys

Material	Material Condition	Cutting Speed, fpm HSS	Cutting Speed, fpm Carbide
All Wrought Aluminum Alloys	CD	600	1200
	ST and A	500	1100
Aluminum Sand and Permanent Mold Casting Alloys	AC	750	1400
	ST and A	600	1200
All Aluminum Die Casting Alloys*	AC	125	550
	ST and A	100	450
*Except Alloys 390.0 and 392.0	AC	80	500
	ST and A	60	425
All Magnesium Alloys	A, CD, AC, ST and A	800	2000
Leaded Commercial Bronze, Leaded Brass, Free-Cutting Brass, Free Cutting Muntz Metal, Forging Brass, Architectural Bronze, Leaded Naval Brass, Free-Cutting Phosphor Bronze	A	300	650
	CD	350	600
Red Brass, Low Brass, Jewelry Bronze, Cartridge Brass, Muntz Metal, Admiralty Brass, Low-Silicon Bronze, High-Silicon Bronze, Manganese Bronze, Aluminum Brass, Leaded Nickel Silver	A	200	500
	CD	250	550
Oxygen-Free Copper, Beryllium Copper, Electrolytic Tough Pitch Copper, Phosphorus Deoxidized Copper, Commercial Bronze, Phosphor Bronze, Aluminum Bronze, Copper Nickel, Nickel Silver	A	100	200
	CD	110	225
Commercially Pure Titanium	110-150 HB	110	400
	180-240 HB	90	300
	250-275 HB	70	250

* Abbreviations designate: A, annealed; AC, as cast; CD, cold drawn; ST and A, solution treated and aged; HB, Brinell hardess number.

faster. Other work material factors that influence the cutting speed are surface scale, hard spots in castings and weldments, and hard work-hardened layers resulting from previous machining or cold working operations. Some variations in the hardness and microstructure may occur within a single bar of a wrought material, such as steel. Noticeable variations in the abilty to be cut by machining may occur between bars of the same type of steel made from different heats, as a result of steel mill practices.

Tool Life. Tool life is the length of time that a cutting tool will cut before it becomes dull or requires replacement. When cutting most metals, cutting tools will operate successfully over a rather wide range of cutting speeds, up to a cutting speed where the tool life is too short to be acceptable. Below this speed the tools will behave in the following manner when normal flank wear occurs and an abnormal type of tool failure does not occur.

A reduction in the cutting speed will cause a much larger increase in the tool life. Conversely, increasing the cutting speed will cause a much larger reduction in the tool life.

This is a very important performance, characteristic of all metal cutting tools, that should be remembered. For example, when the cutting speed is reduced from 350 fpm to 300 fpm while turning AISI 1040 steel with a cemented carbide tool, the tool life will be increased from approximately 30 minutes to 50 minutes. This is a 67 percent increase in tool life obtained by a 14 percent reduction in the cutting speed. When, in this case, the cutting speed is reduced about 29 percent, from 350 fpm to 250 fpm, the tool life will be increased 200 percent to approximately 90 minutes. Even larger percentage increases will occur when using high-speed steel cutting tools.

Cutting tools can perform successfully over a rather wide range of cutting speeds, except when cutting very hard materials and certain other difficult-to-machine materials. The cutting speed selected should be based on a desired tool life. If the cutting speed used is found to result in a tool life that is too short, the first remedial step should be to reduce the cutting speed. While a slow cutting speed will result in a very long tool life, this is always at the expense of an unnecessarily long cutting time and an increased part cost.

Feed and Depth of Cut. The feed rate and the depth of cut also have an effect on the tool life and, therefore, on the cutting speed. The cutting speeds in Tables 7-1 through 7-5 are based on using a feed rate of .012 in./rev (0.30 mm/rev) and a depth of cut of .125 in. (3.18 mm). When a different feed or depth of cut is to be used, the cutting speed should be changed accordingly in order to have approximately the same tool life as before. The amount of change in the cutting speed can be calculated by applying the factors for feed and depth of cut in Table 7-6. These factors should not, however, be used with Table 7-7 because their validity for many nonferrous metals and alloys is uncertain. Applying the feed and depth of cut factors will show that when light cuts are taken the cutting speed can be increased and when heavy roughing cuts are taken it should be decreased.

Cutting Tool Geometry. This topic is treated at length elsewhere in this

Table 7-6. Cutting Speed Feed and Depth of Cut Factors for Turning *

Feed, in./rev.	Feed Factor, F_f	Depth of Cut, in.	Depth of Cut Factor, F_d
.002	1.50	.005	1.50
.003	1.50	.010	1.42
.004	1.50	.016	1.33
.005	1.44	.031	1.21
.006	1.34	.047	1.15
.007	1.25	.062	1.10
.008	1.18	.078	1.07
.009	1.12	.094	1.04
.010	1.08	.100	1.03
.011	1.04	.125	1.00
.012	1.00	.150	.97
.013	.97	.188	.94
.014	.94	.200	.93
.015	.91	.250	.91
.016	.88	.312	.88
.018	.84	.375	.86
.020	.80	.438	.84
.022	.77	.500	.82
.025	.73	.625	.80
.028	.70	.688	.78
.030	.68	.750	.77
.032	.66	.812	.76
.035	.64	.938	.75
.040	.60	1.000	.74
.045	.57	1.125	.73
.050	.55	1.250	.72
.060	.50	1.375	.71

* For use in conjunction with Table 7-1, 7-2, 7-3, and 7-4 only.

chapter. It is, therefore sufficient to say here that the shape of the cutting tool has some effect on the cutting speed.

Cutting Fluids. Coolant type cutting fluids, when correctly applied, will lower the temperature of the cutting tool and tend to increase the tool life. As an alternative, the coolant will cool the tool enough so that a higher cutting speed can be used without overheating. The degree of improvement that can be obtained depends on the type of coolant used.

Selecting the Cutting Conditions.

The tool life of a cutting tool is most affected by the cutting speed, secondly by the feed rate, and least by the depth of cut. This important principle of metal cutting should be memorized. Stated in another way, reducing the cutting speed will increase the tool life more than a reduction in the feed rate; and a reduction in the feed rate will increase the tool life more than a reduction in the depth of cut. Conversely, increasing the depth of cut will reduce the tool life less than an increase in the feed rate; and, increasing the feed rate will reduce the tool life less than an increase in the cutting speed. The latter principle leads to the logical sequence of selecting the cutting conditions, which are given below:

1. Select the depth of cut. Use the largest depth of cut possible, as determined by the amount of metal to be removed from the part and by the available power on the machine.
2. Select the feed rate. Use the heaviest feed rate possible consistent with the surface finish requirement on the workpiece, the rigidity of the machine and workpiece, and the available power on the machine.
3. Select the cutting speed. Use an appropriate table of cutting speeds as a guide, such as provided in this book.
4. Find the appropriate feed and depth of cut factors, when available. Use Table 7-6 in this book, but only with the cutting speed tables provided in this book and in *Machinery's Handbook.* Do not use this table for nonferrous metals and alloys.
5. Calculate the cutting speed and machine spindle speed. How this is done will be shown in the next section of this chapter.
6. Check to see if the machine has sufficent power. This step is optional. In actual practice, this is not done by many persons. When taking only a medium or light cut it is unnecessary. When taking a very heavy cut it is a useful step to determine if the planned cut is within the capacity of the machine. Details on how this is done are given further on in this chapter.

Calculating the Cutting Speed

The formulas used to calculate the cutting speed and the spindle speed of the lathe are given below for both customary inch and metric units. Either inch or metric units can be used in Equation 7-1. The answer will be in terms of the units used for the cutting speed selected from the table.

$$V = V_o F_f F_d \tag{7-1}$$

$$N = \frac{12\,V}{\pi D} \quad \text{(Inch units only)} \tag{7-2}$$

$$N = \frac{1000\,V}{\pi D} \quad \text{(Metric units only)} \tag{7-3}$$

Where:

V = Cutting speed as modified for feed and depth of cut, or from Table 7-5, fpm or m/min
V_o = Cutting speed from Tables 7-1 through 7-4, fpm or m/min
F_f = Feed factor, from Table 7-6
F_d = Depth of cut factor, from Table 7-6
N = Spindle speed of lathe, rpm

D = Diameter of workpiece, in. or mm. (This is the diameter before turning, not the finish turned diameter.)
π = 3.14 (pi)

Example 7-1:

Calculate the cutting speed and the spindle speed for turning a 1.500 in. diameter bar of AISI 4340 steel having a hardness of 220 HB using a cemented carbide (uncoated) cutting tool. The depth of cut selected is .200 in. and the feed selected is .020 in./rev.

$V_o = 275$ fpm (from Table 7-1)
$F_f = .80$ (from Table 7-6)
$F_d = .93$ (from Table 7-6)

$$V = V_o F_f F_d = 275 \times .80 \times .93$$
$$= 200 \text{ fpm}$$
$$N = \frac{12\,V}{\pi D} = \frac{12 \times 200}{\pi \times 1.500}$$
$$= 500 \text{ rpm}$$

In calculations of this type the answers are always rounded-off; e.g., the calculated cutting speed is 204.6 fpm, which is rounded-off to 200 fpm. The actual spindle speed that would be used is the closet available on the lathe. Sometimes, but not always, when the closest available speed is much faster than the calculated spindle speed, the closest lower spindle speed is used.

Example 7-2:

Calculate the cutting speed and the spindle speed for turning a 200 mm diameter Class 30 gray iron casting with a high-speed steel cutting tool. The depth of cut selected is 3.80 mm (.150 in.) and the feed selected is 0.38 mm/rev. (.015 in./rev).

$V_o = 80$ fpm (From Table 7-4)
$F_f = .91$ (From Table 7-6)
$F_d = .97$ (From Table 7-6)

$$V_o = 80 \times .3048 = 24 \text{ m/min}$$
$$V = V_o F_f F_d = 24 \times .91 \times .97$$
$$= 21 \text{ m/min}$$
$$N = \frac{1000\,V}{\pi D} = \frac{1000 \times 21}{\pi \times 200}$$
$$N = 33 \text{ rpm}$$

Cutting Tool Materials

Cutting tool materials must have the following properties at the high temperatures encountered in metal cutting: 1. high hardness; 2. good abrasion resistance; and 3. resistance to chemical-metallurgical interaction with the work material. Cutting tools must also be able to withstand mechanical and thermal shock. The room temperature properties of cutting tool materials serve only as a rough guide to their ability to perform as a cutting tool; the only reliable guide is their actual performance when cutting. While high-speed steel can be broadly applied as a cutting tool material, where higher cutting speeds are required it is necessary to be able to select from other cutting tool materials the material to suit a particular job.

Table 7-7. Chemical Composition of Frequently Used High-Speed Steels *

AISI Designation	Nominal Chemical Composition, Percent					
	C	W	Mo	Cr	V	Co
M1	0.80	1.50	8.00	4.00	1.00	....
M2	0.85	6.00	5.00	4.00	2.00	....
M6	0.80	4.00	5.00	4.00	1.50	12.00
M42	1.10	1.50	9.50	3.75	1.15	8.00
T1	0.70	18.00	...	4.00	1.00	....
T6	0.80	20.00	...	4.50	1.50	12.00
T15	1.50	12.00	...	4.00	5.00	5.00

* C-Carbon, W-Tungsten, Mo-Molybdenum, Cr-Chromium, V-Vanadium, Co-Cobalt

High-Speed Steel

High-speed steel is the name given to a group of very similar tool steels by virtue of their excellent red hardness. Red hardness is the ability of a steel to retain its hardnes at a high temperature. High-speed steels will retain sufficient hardness at temperatures up to 1100 F to 1200 F which will enable them to cut other materials while at these temperatures. When cooled, the high-speed steels will return to their room temperature hardness, which is 63 to 65 HRC for cutting tools made of this material. High-speed steels are very deep hardening; this enables them to be ground to a tool shape from solid stock and to be resharpened, when required, without a loss of hardness. They can be softened by annealing and then machined into complex cutting tool shapes, such as twist drills, reamers, and milling cutters.

While the types of high-speed steels that are available are designed to meet some special requirement or to provide a special advantage, as a group they are almost a universal cutting tool material inasmuch as they are not too sensitive to the characteristic of the material being cut. Most work materials can be cut successfully with any type of high-speed steel.

It is not necessary to be overly concerned about the type of high-speed steel used. Exceptions are the high-temperature alloys and other exotic materials. The limitations of high-speed steel relative to other cutting tool materials are its lower hardness and the slower cutting speeds that must be used.

The composition of a few typical high-speed steels is given in Table 7-7. The alloying elements in high-speed steels are tungsten, molybdenum, chromium, vanadium, and carbon. There are two classes of high-speed steel. Those designated by the letter T have tungsten as their principal alloying element. To conserve tungsten, the class designated by the letter M has some of the tungsten replaced by molybdenum. The original type of high-speed steel was T1, which is still used to make cutting tools. The most widely used type for making cutting tools is M2.

The addition of cobalt to high-speed steels increases its red hardness; i.e., it will have a higher hardness and will be more wear resistant than other types at the metal cutting temperatures. Cobalt high-speed steels, such as T6 and M6, are, therefore, recommended for taking roughing cuts in abrasive materials and through abrasive surface scale. They are, however, more brittle and susceptible to cutting edge chipping. They are thus not recommended for fine finish cutting operations.

The M40 series and T15 high-speed steels (see M42 in Table 7-8) are a group of high-hardness, or so-called super-high-speed steels. They can be hardened to 70 HRC, although for cutting tool applications they are usually hardened to 67 to 68 HRC to reduce their brittleness and tendency to chip. They are designed specifically to cut high temperature alloys and other high-strength alloys. Some high-speed steels are made by the particle metallurgy process. Tools made from these steels can be about 1 HRC harder without sacrificing toughness. For high-speed steel cutting tools, 1 HRC is a significant increase in hardness. These high-speed steels are particularly useful on applications where the tool life is limited by chipping of the cutting edge.

Cemented Carbides

Cemented carbides, also called sintered carbides or just carbides, are much harder and more wear resistant than high-speed steel. They can retain their hardness at a higher temperature (1400 F and higher) than high-speed steel which enables them to cut at much faster cutting speeds. They are produced by the powdered metallurgy process. They are, however, as compact and dense as other metals. Like most very hard materials, carbides are somewhat brittle. Cemented carbides are expensive; for this reason carbide cutting tools are made in the form of small tips brazed onto a steel shank, or small inserts that are mechanically clamped in place. Although not common, some solid cemented carbide cutting tools, such as small twist drills, are made.

Many different grades of cemented carbides are available and the importance of selecting the correct grade cannot be overemphasized. The cutting speeds for carbides given in the tables of this book are based on the assumption that the correct grade of carbide is being used. To select a carbide grade, it is best to seek the recommendation of a carbide producer. As a guideline in selecting a grade, it is first necessary to find the grades having the correct chemical and metallurgical specification for cutting the workpiece material. Then within this group, a grade having a suitable hardness is selected. Harder grades are selected for light finishing cuts, and for roughing cuts less hard but tougher grades are selected. In general, select the carbide grade having the highest hardness with sufficient strength to prevent the cutting edge from chipping.

The principal types of cemented carbides are straight tungsten carbides, crater resistant carbides, titanium carbides, and coated carbides. Straight tungsten carbide grades contain from 94 to 97 percent tungsten carbide with the remainder being cobalt. The higher cobalt bearing grades are tougher and more shock resistant; they are recommended for heavy roughing cuts. The higher tungsten bearing carbides are harder and more wear resistant; they are recommended for taking lighter, finishing cuts. As a class, straight tungsten carbides are recommended for machining gray cast iron, ferritic malleable iron, austenitic stainless steels, high temperature alloys, and plastics. They should not be used to cut plain carbon and alloy steels because these materials will form a crater on the face of the tool which will result in a very short tool life.

Plain carbon and alloy steels should be cut with a crater resistant grade of carbide. Crater resistance is imparted in a carbide by the addition of titanum carbide. In addition, tantalum carbide is usually added to these grades to prevent the cutting edge from deforming at the temperature and pressure encountered when taking a heavier cut. Crater resistant grades of carbides are used to cut all of the plain carbon and alloy steels, alloy cast irons, pearlitic malleable irons, nodular iron, Monel metal, martensitic stainless steel, ferritic stainless steel, and all of the tool steels.

Titanium carbide grades contain no tungsten carbide; they are made from titanium carbide and a small quantity of nickel or molybdenum. These grades have excellent crater and heat resistance which enables them to be used at higher cutting speeds. They will produce an excellent surface finish on the workpiece. Their use is limited by their low resistance to shock, thermal cracking, and to oxide films and surface scale. Titanium carbides are used to take finish and semi-finishing cuts on low carbon-low alloy steel, wrought aluminum, and cast iron. They should not be used to machine abrasive materials because of their low overall resistance to abrasion.

Coated cemented carbides consist of a cemented carbide insert, called the substrate, on which a thin coating of titanium carbide (TiC), aluminum oxide (Al_2O_3), or titanium nitride has been deposited. The thickness of the coating is only .0002 to .0004 inch (0.005 to 0.010 mm), which is firmly bonded to the substrate. Usually a single coating is used although some inserts are double and triple coated. Since grinding would remove the coating, coated carbides are available only in the form of disposable inserts. The advantage of coated carbides lies in the fact that a wear resistant coating can be applied to a selected substrate to obtain a combination of favorable performance characteristics such as wear resistance, toughness, strength, and resistance to thermal shock, crater formation, and deformation. Where they can be used, coated carbides will generally operate at a higher cutting speed without a sacrifice of tool life; conversely, at the same cutting speed as an uncoated grade, the coated insert will give a longer tool life. In some cases the cutting speed can be increased 20 to 40 percent and sometimes up to 50 percent when using a coated carbide. Coated carbides have been developed to take interrupted cuts and to perform milling operations, although they are slightly less shock resistant than the tougher uncoated grades of carbides.

There are different grades of coated carbides and the correct grade must be selected for each operation. In general, coated carbides are recommended for taking medium and heavy-duty roughing cuts and high-speed finishing cuts on carbon, alloy, and tool steels, free machining steels, alloy cast iron, and the 400-500 series stainless steels. They are also recommended for medium roughing and high-speed finishing cuts on gray cast iron and the 200-300 series stainless steels. They are not recommended for the machining of cobalt, nickel, or iron base high-temperature alloys, titanium base alloys, refractory metals, free machining brass, bronze, aluminum-bronze, manganese bronze, monel, cupro-nickel, silicon-aluminum, nickel, copper, magnesium alloys, zinc base alloys, and nonmetals such as carbon, fiberglass, and plastics.

Ceramics and Cermets

Ceramic cutting tools are made by sintering fine grains of aluminum

oxide into a dense structure. Cermets are a mixture of approximately 70 percent aluminum oxide and 30 percent titanium carbide. Ceramic cutting tools are quite brittle and sensitive to shock. They can be used to cut at extremely high cutting speeds. As an example, ceramic cutting tools have been used to cut AISI 1040 steel at 18,000 fpm (5500 m/min) on a lathe designed to take high-speed cuts. At this speed a very good tool life was obtained. Ordinary machine tools cannot, however, cut at these speeds. Ceramic inserts should be honed or ground along the cutting edge to prevent chipping. It is customary to grind a "K" land on ceramic tools, which is a narrow land .002 in. to .006 in. (0.05 mm to 0.15 mm) wide and ground at an angle of 30 degrees with respect to the face of the insert. Cermets are stronger than ceramics and can be used for interrupted cuts. They have been used successfully in milling. To be effective, however, they should be used at very high cutting speeds. Ceramic and cermet cutting tools căn be used to machine plain carbon and alloy steels, gray cast iron, malleable iron, nodular iron, and stainless steel. They should not be used to cut aluminum or aluminum alloys because the aluminum oxide in these tools will react with these metals.

Other Cutting Tool Materials

Several other cutting tool materials are firmly established by virtue of their superior performance on special classes of work.

Diamonds. Single crystal diamond cutting tools are extremely hard and very wear resistant. They produce an outstanding surface finish on all materials that they are capable of cutting because diamonds will not react with other materials at the cutting temperatures (see Fig. 9-32). They can operate at fast cutting speeds; however, the diamond may disintegrate if the temperature at the cutting edge becomes too high. For this reason they should not be used to cut steel, cast iron, or other iron-base maerials. The diamond is also susceptible to chipping and interrupted cuts are not normally recommended. Diamond cutting tools are able to cut some very hard materials and are especially useful for cutting abrasive materials. Their excellent wear resistance enables them to produce a fine surface finish and a uniformly precise dimension when taking long cuts. One outstanding characteristic of these tools is their ability to produce excellent surface finishes on materials that are difficult to finish by other means, such as aluminum, magnesium, copper, and babbit. They are also used to finish brass, bronze, stainless steel, titanium, and the precious metals—gold, silver, and platinum. The finish obtained can be comparable or better than that obtained by grinding, fine honing, or lapping.

Poly-Crystalline Diamonds. These cutting tools are made in the form of inserts from diamond powders that have been compacted and sintered together. Another type consists of a layer of poly-crystalline powders supported on a cemented carbide base. They are more resistant to chipping and can be resharpened by grinding. Otherwise, their properties and field of application are similar to single-crystal diamonds.

Cubic Boron Nitride. The trade name of this material is Borozon, by which it is better known. It is used as an abrasive in grinding wheels and is compacted to make edge-type cutting tools in the form of inserts. Borozon is very hard and can operate for long periods of time at a temperature of 2000 F. It will cut the difficult-to-machine high temperature alloys at a cutting speed that is four to six times faster than cemented carbide tools. The temperature developed at the cutting edge is high enough to produce a small ball of fire and to reduce the strength of the work material, making it easier to cut. A water base cutting fluid should be used to prevent the chips from welding onto the surface of the workpiece.

Cast Nonferrous Alloys

Cast nonferrous alloys can be used for cutting at speeds about 20 to 50 per cent higher than high-speed steels. They are not as tough, but are more wear resistant than high-speed steel. They cannot cut as fast as cemented carbides and their application is limited. The composition of these alloys is approximately 40 to 53 per cent cobalt, 30 to 33 per cent chromium, and 14 to 22 per cent tungsten.

Single-Point Cutting Tool Nomenclature

The nomenclature of a single-point cutting tool is shown in Fig. 7-1. Most cutting tools have two cutting edges; a side-cutting edge and end-cutting edge (see View A, Fig. 7-1). When doing ordinary turning the cutting is done with the side-cutting edge. The nose is that portion of the cutting edge which serves to connect the two cutting edges. It is a very critical part of the cutting edge because it produces the finished surface in turning. The flank of the tool is the surface on the side of the tool below the cutting edges. It must be relieved in order to permit the cutting edges to penetrate into the metal being cut. If the flanks below the side-cutting edge, nose, and end-cutting edge were not relieved, the tool would rub and could not penetrate into the workpiece. The face of the cutting tool provides a surface over which the chip that is formed by the cut slides. It also has a great influence in determining the angle of the shear plane on which the grains of the metal are deformed in order to form the chip. This will be described later in this chapter. The metal remaining in back of the cutting edge is the shank. It supports the cutting edge and provides a surface upon which the cutting tool can rest and be held.

The cutting tool in Fig. 7-1 is a right-hand tool. A right-hand cutting tool can be identified by having the side-cutting edge on the left when viewed from the shank. A left-hand cutting tool has the side-cutting edge on the right side when viewed from the shank. The tool shown is also classified as a side-cutting-edge tool because it is provided with a side-cutting edge and because it is intended that most of the cutting will be done with the side-cutting edge. End-cutting-edge tools (see H, J, and K, Fig. 7-14) are intended to cut exclusively with their end-cutting edges and are made without a side-cutting edge.

When specifying a single-point cutting tool, it is necessary to specify the width and height of the shank. In addition, it is necessary to specify the radius of the nose and the six principal tool angles which are: 1. end relief angle; 2. side relief angle; 3. back rake angle; 4. side rake angle; 5. end-cutting-edge angle; and 6. side-cutting-edge angle. These angles are all shown in View B, Fig. 7-1.

Relief Angles

The side relief angle provides the relief for the side-cutting edge in order to allow it to penetrate into the workpiece so that it can form the chip. The end relief angle allows the end-cutting edge to penetrate into the workpiece. The relief angle below the nose is a blend of both the end relief and the side relief angles. Usually, both relief angles are equal although this is not always the case. The size of the relief angles has a very pronounced effect on the performance of the cutting tool; if they are too small they will cause a decided decrease in the tool life; if too large, the support below the cutting edge will be weakened to the extent that the cutting edge may break or chip. For average work, a relief angle of 10 degrees is recommended. For high-speed-steel cutting tools the relief angle should be in the range of 8 to 16 degrees. Harder materials may require a smaller relief angle while softer materials are more successfully cut when a larger relief angle is used. The relief angle for cemented-carbide cutting tools should be in the range of 5 to 12 degrees. Since carbides are harder and more brittle than high-speed steel, a better support under the cutting edge is required.

Rake Angles

The slope of the face of the cutting tool is determined by the back-rake and side-rake angles. The side-rake angle is measured perpendicular to the side-cutting edge and the back-rake angle is measured parallel to the side-cutting edge. The rake angles may be positive, as shown at B in Fig. 7-1, or may be negative, as shown at C. Note that the positive rake angles cause the face of the cutting tool to slant downward when moving away from the cutting edges, and that negative rake angles cause this surface to slant upward when moving away from the cutting edges. Positive side-rake angles are generally preferred because less cutting force is required in order to take a given size cut as compared to a tool with a negative side-rake angle.

The cutting force required to take a given cut will increase approxi-

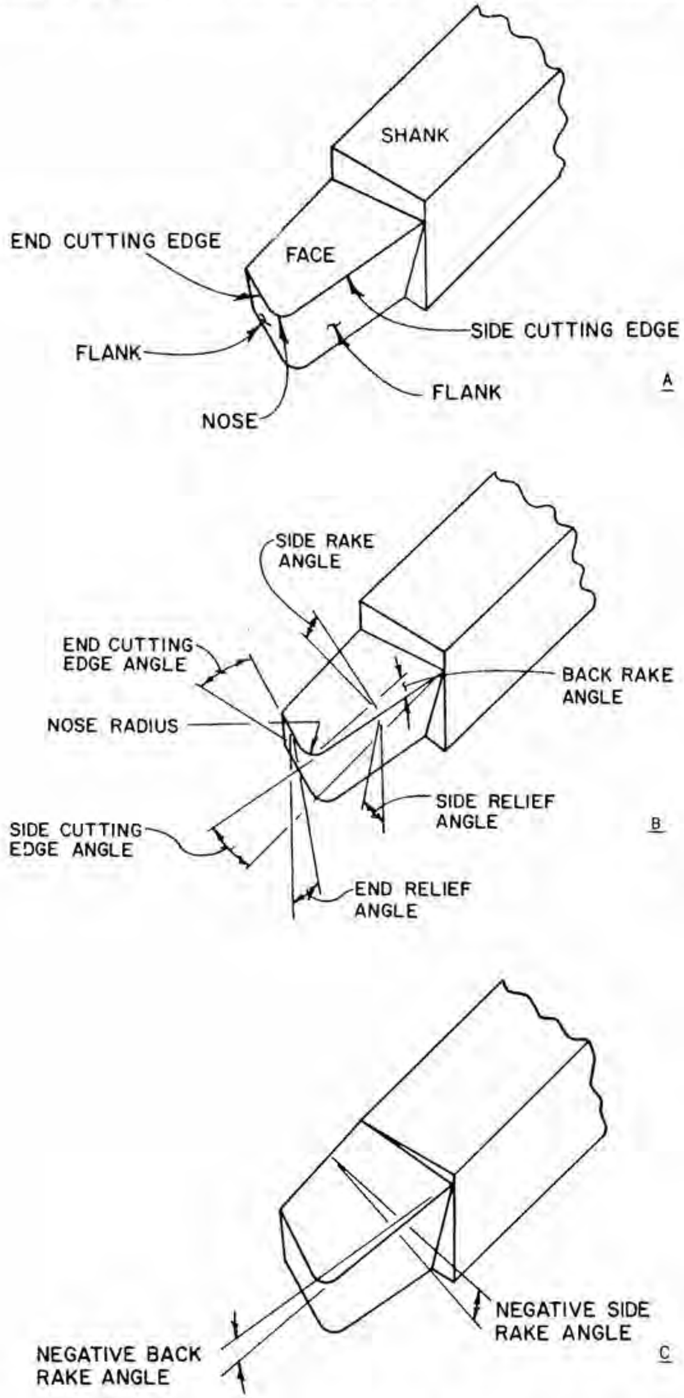


Fig. 7-1. The nomenclature of the single-point cutting tool.

mately one per cent per degree that the size of the positive rake angle is decreased, or that the size of the negative side-rake angle is increased. For example, if a tool with a 5-degree, positive side-rake angle is replaced by one with a 5-degree negative side-rake angle, the force required to take the cut will be increased approximately ten per cent. The back-rake angle does not have as much effect on the cutting force. The positive side-rake angle tool will generate less heat and will have a longer tool life than the negative side-rake angle cutting tool. The side-rake angle on most cutting tools should be made with as large a positive rake-angle as possible, up to a certain limit, for if it is made too large, the cutting edge will be weakened and will readily fail. The advantage of having a negative side-rake angle, and a negative back-rake angle, is that the cutting edge is strengthened; thereby enabling it to withstand more severe cutting loads. This is an advantage when cutting very hard and tough materials and when taking interrupted cuts. Negative rake angles are often preferred on disposable or throw-away carbide cutting tools (see Fig. 7-6) because relief angles do not have to be ground on the inserts; also, the edges on both the top and the bottom of the inserts can be used as cutting edges. This provides twice as many cutting edges as a comparable, positive rake cutting tool which has cutting edges avail-

Table 7-8. Recommended Rake Angles

Material	Hardness, HB	High-Speed Steel		Carbide	
		Back Rake Angle, deg.	Side Rake Angle, deg.	Back Rake Angle, deg.	Side Rake Angle, deg.
Plain Carbon Steel	100 to 200	5 to 10	10 to 20	0 to 5	7 to 15
	200 to 300	5 to 7	8 to 12	0 to 5	5 to 8
	300 to 400	0 to 5	5 to 10	−5 to 0	3 to 5
	400 to 500	−5 to 0	−5 to 0	−8 to 0	−6 to 0
Alloy Steel	100 to 200	5 to 10	10 to 16	0 to 5	5 to 15
	200 to 300	5 to 7	8 to 12	0 to 5	5 to 8
	300 to 400	0 to 5	5 to 10	−5 to 0	3 to 5
	400 to 500	−5 to 0	−5 to 0	−6 to 0	−6 to 0
Aluminum					
Non Heat-Treated	...	10 to 20	30 to 35	0 to 15	15 to 30
Heat Treated	...	5 to 12	15 to 20	0 to 5	8 to 15
Magnesium	...	5 to 10	10 to 20	0 to 5	10 to 20
Stainless Steel					
Ferritic	130 to 190	5 to 7	8 to 10	0 to 5	5 to 7
Austenitic	130 to 190	5 to 7	8 to 12	0 to 5	5 to 7
Martensitic	130 to 220	0 to 5	5 to 8	−5 to 5	6 to 15
Gray Cast-Iron	100 to 200	5 to 10	10 to 15	0 to 5	6 to 15
	200 to 300	5 to 7	5 to 10	0 to 5	5 to 7
	300 to 400	−5 to 0	−5 to 0	−6 to 0	−6 to 0
Malleable Iron					
Ferritic	110 to 160	5 to 15	12 to 20	0 to 10	7 to 15
Pearlitic	160 to 200	5 to 8	10 to 12	0 to 5	5 to 8
	200 to 280	0 to 5	−5 to 8	−5 to 5	−5 to 8
Brass-Free Cutting	....	−5 to 5	0 to 10	...	...
Brass					
Red, Yellow, Naval, Manganese Bronze, Nickel Silver	....	−5 to 5	−5 to 5	−5 to 5	−5 to 5
Hard Phosphor Bronze	....	−5 to 0	−6 to 3	−5 to 0	−5 to 5

able on only one face of the insert. The rake angles recommended for cutting different work materials are given in Table 7-8.

End-Cutting-Edge Angle

The end-cutting-edge angle is the amount that the end-cutting edge slopes away from the nose of the tool, so that it will clear the finished surface on the workpiece, when cutting with the side-cutting edge. The size of this angle is very important, particularly when cutting materials that tend to form a large crater on the face of the tool. This crater will then tend to enlarge toward the end-cutting edge where it will eventually break through and cause the tool to fail. This is shown at D, Fig. 7-24. When severe cratering occurs, the size of the end-cutting edge should be limited to 8 to 15 degrees; otherwise, it can be made as large as 45 degrees without an adverse effect on the performance of the tool.

Side-Cutting Edge and Lead Angles

Since the chip will flow approximately perpendicular to the side-cutting edge, the side-cutting-edge angle, thereby, influences the direction of the chip flow. It also affects the thickness of the chip and the direction of the longitudinal component of the cutting force. These, however, are more affected by the lead angle. An understanding of the relationship of the lead angle to the side-cutting-edge angle is necessary in order to understand how single-point cutting tools perform.

The lead angle is defined as the angle at which the side-cutting edge is positioned, with respect to a plane that is perpendicular to the axis of the workpiece. (See Fig. 7-2.) When the cutting tool is positioned perpendicular to the workpiece, the lead angle is equal to the side-cutting-edge angle as shown at B. In Fig. 7-3, the tool at A, has a zero degree lead angle and the tool at B, has a larger lead angle. The feed, or distance that the cutting tool advances per revolution, and the depth of cut are

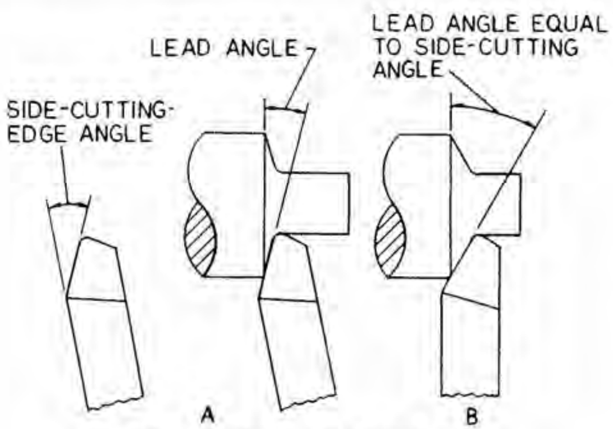


Fig. 7-2. Definition of the lead angle.

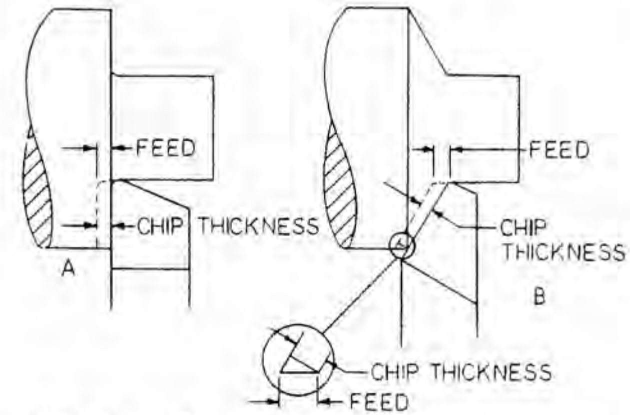


Fig. 7-3. The effect of the lead angle on the chip thickness.

equal, in both cases. The chip thickness is measured perpendicular to the side-cutting edge of the tool. At A, the chip thickness is equal to the feed; while at B, the chip thickness is less than the feed. In each case the same amount of metal is removed—in the form of a chip—for each revolution of the work; however, the chip at B, is longer and thinner. Thus, the effect of the lead angles is to produce a longer and thinner chip, which can be considered the equivalent of increasing the depth of cut and decreasing the feed rate. A study of the effect of feed on the cutting speed, in Table 7-6, will show that the thinner chip, or lower feed rate, will permit the cutting speed to be increased; or, if the cutting speed remains the same, the feed can be increased. In either case, an increase in the lead angle can bring about an increase in the production rate that can be attained. The effect of the depth of cut is relatively small, and can usually be ignored.

There is, however, an adverse effect caused by a large lead angle. This is shown in Fig. 7-4. A force is required to feed the cutting tool into the workpiece; this force is called the "longitudinal" force. It is perpendicular to the side-cutting edge. Although shown in Fig. 7-4, as reacting against the work, it reacts with an equal magnitude against the cutting tool. At A, this force is parallel to the axis of the workpiece and there is very little tendency to cause it to bend the work. At B, this force is directed so that it will cross the workpiece axis which will cause the workpiece to bend away from the tool. The elasticity in the workpiece material will cause it to spring back into its original position. When this cycle is repeated at certain frequencies a very noticeable vibration, called chatter, will set in. Chatter impairs the surface finish on the work and causes a rapid breakdown of the cutting edge as well. In all instances it must be eliminated. It can be caused by many factors, such as: loose headstock-spindle bearings, a loose fit on the cross slide, too much overhang of the cutting tool, or a very dull cutting tool. However, the most frequent cause

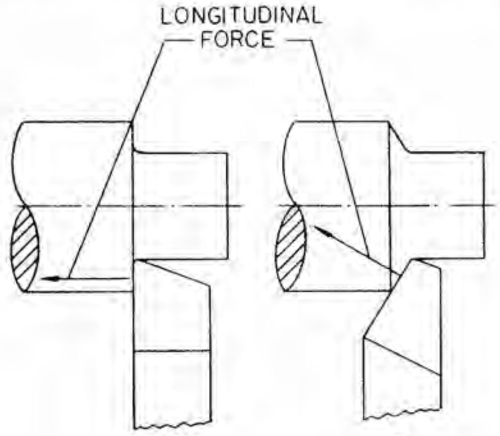


Fig. 7-4. The direction of the reaction of the longitudinal cutting force is perpendicular to the side-cutting edge. An equal and opposite force that is not shown reacts against the cutting edge.

is a lead angle that is too large; especially when turning a long and slender workpiece. The first step to take is to examine the lead angle used and to reduce it if it is too large. In a great number of instances, this will eliminate the chatter completely. When boring with a cutting tool having a large lead angle, the reaction of the longitudinal force against the cutting edge will deflect the boring bar, thus causing chatter or some difficulty in boring the hole to a desired dimension. For this reason the lead angle on boring tools should be kept small. When boring either very deep or small diameter holes, it is best to use a zero degree lead angle.

Nose Radius

The nose is an extension of the cutting edges. Its point must be formed into a radius in order to prevent the formation of a sharp threadlike groove on the surface of the work. When taking a cut, the chip thickness of the nose is equal to the chip thickness of the side-cutting edge at the point where they join. At the point where the nose is tangent to the workpiece, the chip thickness becomes zero in magnitude. Thus, the chip becomes increasingly thinner as the point of tangency between the nose and the work is approached. If the nose radius is large, there will be a large area where a very thin chip is formed. When this occurs the cutting edge may fail to penetrate the work in this region, thus causing it to rub. This is very apt to occur if the cutting edge is dull. Should the cutting edge fail to penetrate the work, chatter may be set up or the tool will get dull rapidly, in this region. Thus, we have another cause of chatter, a nose radius that is too large. Hence, reducing the size of the nose radius will reduce the tendency to chatter. When finish turning, the nose

radius must be made large enough to prevent any formation of threadlike grooves that have the form of the nose. If the size of the nose radius cannot be increased, the feed must be reduced to obtain a good surface finish. The size of the nose radius is then dependent upon the feed rate, the surface finish required on the work, and the requirement to prevent the occurrence of chatter. It should be pointed out that the life of the cutting tool can be adversely affected by a nose radius that is either too large or too small. For average conditions when turning, a nose radius of $\frac{1}{64}$ to $\frac{1}{8}$ inch is recommended, although larger nose radii can sometimes be used successfully.

Tool Holders

In most cases the cutting tool is clamped in a tool holder; however, single-point cutting tools are sometimes clamped directly on the machine. The tool holder is made from a less expensive steel than the cutting tool. It is heat treated to increase its strength and to resist the penetration of the screws which clamp it in place. Although there is a great variety of tool holders, only representative examples of commonly used designs are illustrated here. One common but very effective tool holder for holding high-speed-steel tool bits, is shown in A, Fig. 7-5. Cemented carbide tips are sometimes brazed on steel shanks as in B, Fig. 7-5.

Most cemented carbide cutting tools are in the form of indexable inserts, which are also called disposable inserts and throw-away inserts. They are held in pockets machined in the tool holders which nest the insert. Three typical indexable insert tool holders are shown in Fig. 7-6. Each insert has several cutting edges. After a cutting edge has become dull, the insert is indexed to bring another cutting edge into the position on the tool holder where it can cut. When all the cutting edges are used up, the insert is replaced. There are positive rake and negative rake tool holders and inserts. A positive rake tool holder is shown at A, Fig. 7-7. The positive rake insert must be provided with a relief angle

Fig. 7-5. A. (*Left*) Toolholder for high-speed steel tool bits. B. (*Right*) A cemented-carbide cutting tool with a brazed-on insert.

Courtesy of Carboloy Systems Dept.-General Electric Co.

Fig. 7-6. Three typical indexable insert toolholders.

on the flank. For this reason only the edges on the top face of the insert can be used as cutting edges. The relief angle on the negative rake insert, shown at B, Fig. 7-7, is provided by the inclination at which the insert is held on the tool holder. Held in this position, the flanks of the insert can be perpendicular with both faces and all of the edges on both faces can be used as cutting edges. Therefore, negative rake inserts have twice as many cutting edges as comparable positive rake inserts. Another advantage of negative rake inserts is that they are stronger than positive rake inserts and more able to withstand shock loads, such as encountered when taking interrupted cuts. The advantage of positive rake inserts is that when cutting, the cutting force is significantly less. When the cutting force must be kept as low as possible, as when cutting thin material sections, a positive rake insert should be used. Positive-negative rake inserts, view C, Fig. 7-7, are held on negative rake tool holder but have an effective positive rake angle, provided by a groove on the face of the insert. As shown in Fig. 7-8, the groove also acts as a chip breaker. Ductile materials produce an unbroken continuous chip. This chip will move off from the tool very rapidly when cutting at a high speed with cemented carbides, thereby creating a hazard to the machine operator. To eliminate this hazard, the chips must be broken into small segments by providing a chip breaker on the tool.

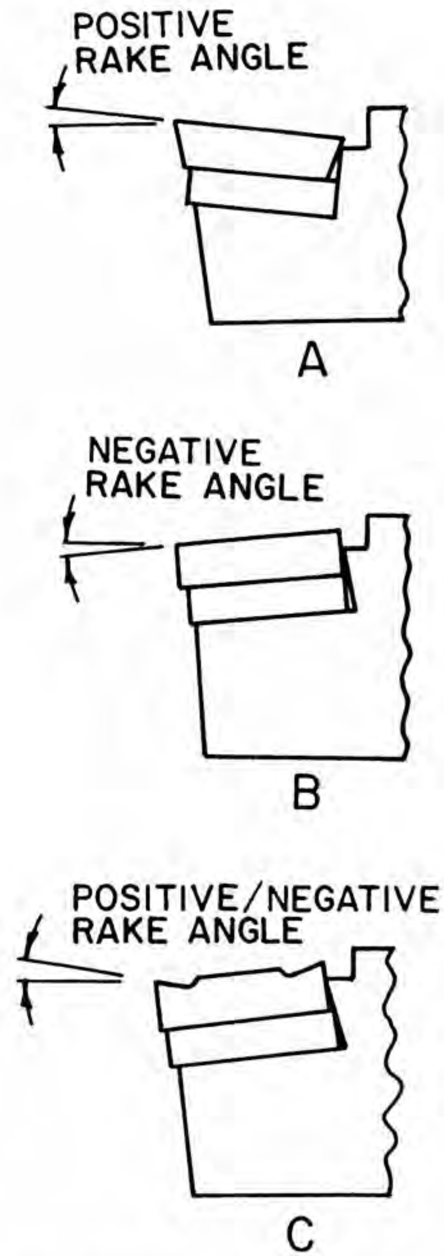
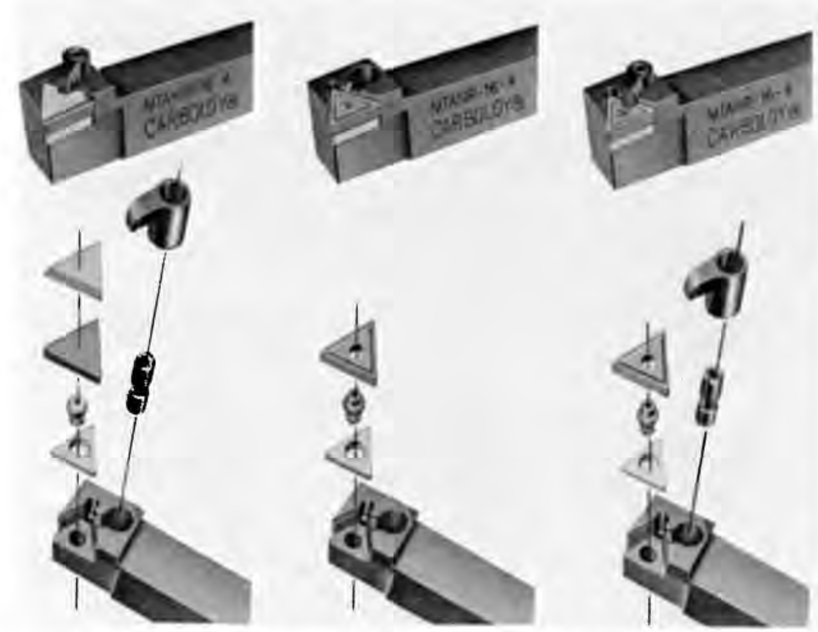


Fig. 7-7. Orientation of inserts in toolholder pockets. A. Positive rake insert; B. Negative rake insert; C. Positive/Negative rake insert.

Courtesy of the Valenite Division of the Valeron Corp.

Fig. 7-8. Positive/Negative rake insert shown cutting ductile material (steel). Groove on face provides positive rake angle and acts as chipbreaker.

The methods of clamping the inserts on the tool holder are shown in Fig. 7-9. The insert is always placed on a cemented carbide seat which rests on the bottom of the tool holder and provides a firm flat bearing surface for the insert. A pin, pin-type cam, or a screw holds the seat in place on the holder. In Fig. 7-9A the insert is clamped by a top clamp. When cutting ductile materials, a plate-type chip breaker is clamped on top of the insert. The insert at B has a chip breaker groove on the face and, therefore, does not require a separate plate-type chip breaker. It is clamped to the tool holder by a pin-type cam which fits into the hole provided in the insert. By turning the pin-type cam with an Allen wrench, the insert is clamped firmly to the seat and against the sides of the pocket. Other pin-type clamping arrangements are also used. The tool holder in Fig. 7-9 can be adapted to clamp the insert by both methods. At A it is used as a top clamp-type tool holder. At B it is a pin-type tool holder, and at C both methods are used simultaneously.



Courtesy of Carboloy Systems Dept.-General Electric Co.

Fig. 7-9. Methods of clamping indexable inserts on toolholder. A. Top clamp; B. Pin-type clamp; C. Using top clamp and pin-type clamp simultaneously.

Indexable inserts used to cut screw threads and shallow grooves are usually mounted on the side of the tool holder, as shown in Fig. 7-10. A tool used for cut-off and for deep grooving operations is shown in Fig. 7-11. Any of four different heads holding an equal number of different cemented carbide cut-off and deep grooving inserts can be mounted on a single shank. A group of typical boring bars used to hold cemented carbide indexable inserts are shown in Fig. 7-12. Two typical boring bars for holding high-speed steel tool bits are shown in Fig. 7-13.

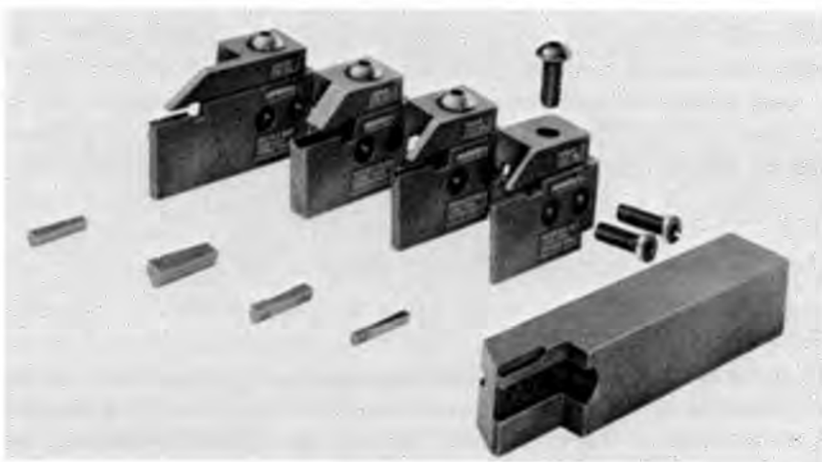
Tool Shapes

Many different tool shapes have been successfully used for turning operations but only a few of the basic shapes can be shown here. Regardless of the shape of the single-point tool, it must have the optimum tool angles that will allow it to penetrate the metal and cut efficiently. This section will deal only with the *shape* of the tool, since the tool angles have been described in a previous section.

High-speed steel single point cutting tools are usually ground to shape from solid blanks called tool bits. The basic high-speed steel cutting tool

Courtesy of Carboloy Systems Dept.-General Electric Co.

Fig. 7-10. Toolholder and inserts for thread cutting and for cutting shallow grooves. Insert is clamped on side of toolholder.



Courtesy of Carboloy Systems Dept.-General Electric Co.

Fig. 7-11. Deep grooving and cut-off tool, showing the cemented-carbide inserts and the four styles of heads that may be assembled on a single shank.

Courtesy of Carboloy Systems Dept.-General Electric Co.

Fig. 7-12. Typical cemented-carbide, indexable insert boring bars.

shapes and how they are applied are shown in Fig. 7-14 and are described below:

A. *Turning Tool.* This is a basic high-speed steel single point cutting tool shape; it is a right-hand turning tool used for both rough and finish turning operations. The side cutting edge angle is zero degrees providing a zero degree lead angle when the tool is positioned as shown. It is often used to turn a cylindrical surface and to form a square shoulder at the end of the cut.

B. *The Lead Angle Turning Tool.* The lead angle (side cutting edge angle) turning tool is one of the most frequently used single point cutting tools for roughing and for finishing operations. It may be positioned at an angle, as shown at C, to form a square shoulder at the end of a turning cut.

Courtesy of Armstrong Brothers Tool Company

Fig. 7-13. Boring bars for high-speed steel tool bits.

C. *Turning Tool with Chip Control.* This tool is used to turn ductile materials, which form continuous chips that sometimes have a tendency to snarl. The chip control groove will curl the chip and cause it to flow away from the workpiece. There are several variations of this tool, one of which does not have the small flat on the face of the cutting edge. However, the flat is frequently used as it strengthens the cutting edge and tends to increase the tool life. It should be approximately .016 to .047 inch (0.40 to 1.2 mm) wide. The width of the flat and size of the groove depend on the material being cut and the feed rate used.

D. *Round Nose Turning Tool.* The round nose turning tool is used extensively when turning large diameter workpieces; however, it can be used equally effectively on smaller diameter workpieces. The curved side cutting edge is, in effect, a large lead angle allowing a heavy feed rate to be used, while the large nose radius produces a very good surface finish on the workpiece. It is used for both heavy rough turning cuts and for light finish turning cuts.

E. *Turn and Facing Tool.* This is merely a variation of the application of the turning tool at A, although for this purpose the tool is often provided with a slightly larger end cutting edge angle. Positioned as shown, it is used to turn a cylindrical surface up to a shoulder. When against the shoulder it is fed outward away from the workpiece center to take a facing cut—called back facing—in order to cut the shoulder perfectly square. If necessary, a facing

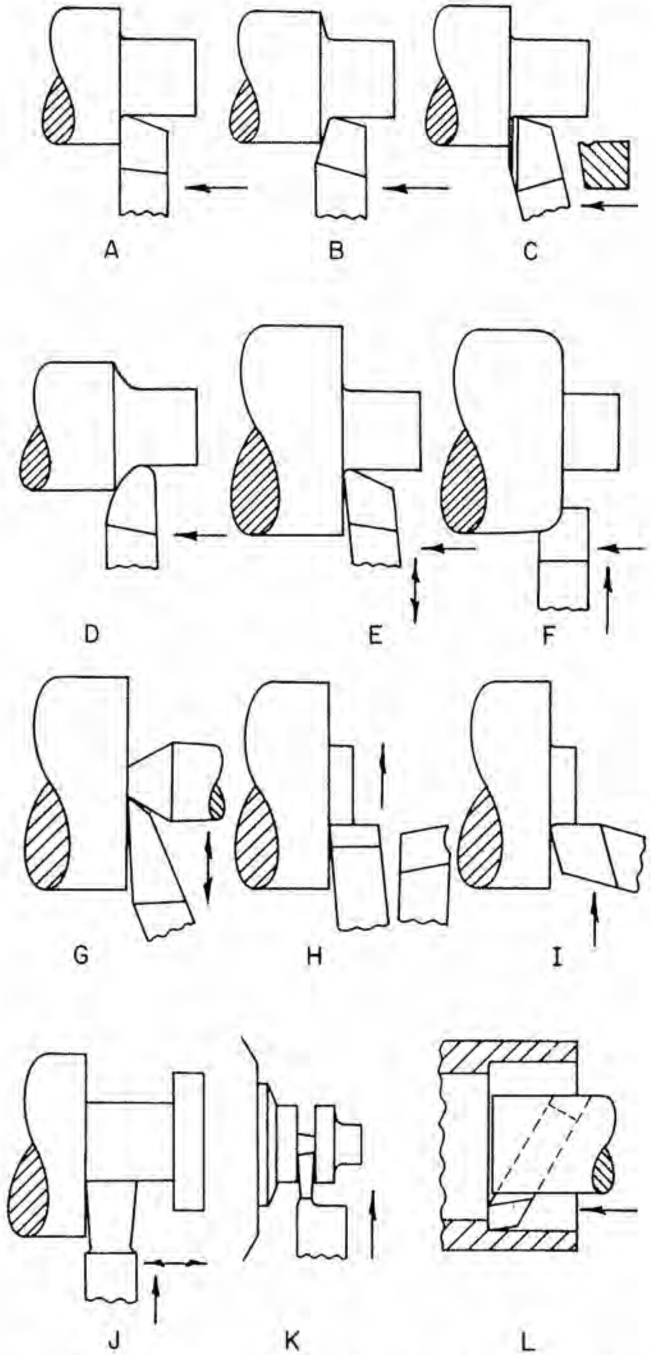


Fig. 7-14. Typical tool shapes ground on high-speed tool bits and how they are used.

cut can also be taken by feeding the tool inward toward the center of the workpiece.

F. *Form Tool.* Form tools are used to turn a contoured shape on the workpiece, such as the radius in the illustration.

G. *End Facing Tool.* This tool is used to face the end of the workpiece while it is mounted on a tailstock center (see also Fig. 8-19).

H. *Facing Tool.* The facing tool shown is used to take heavy facing cuts requiring a large amount of stock removal. It has a "hook" ground on its face to provide a rake angle to the end cutting edge. Sometimes it is provided with a narrow flat on the cutting edge, similar to the flat on the tool shown at C. The side flank adjacent to the shoulder has a 5- to 8-degree relief angle.

I. *Facing Tool.* Actually this is a left-hand turning tool which when used as shown is sometimes called a facing tool. It is used to take heavy or light facing cuts. Having a nose radius it will usually produce a better surface finish than the tool at H.

J. *Grooving or Necking Tool.* The end cutting edge is the primary cutting edge of this tool; the sides have a slight back clearance angle to prevent their rubbing against the sides of the groove. Wide grooves are cut by taking several plunge cuts into the workpiece. The groove is then finished by taking light finishing cuts on the cylindrical surface using the carriage feed and on the sides using the cross slide feed. Narrow grooves are cut by a single plunge into the workpiece, with the width of the tool being made equal to the width of the groove in the workpiece.

K. *Cut-Off Tool.* Cut-off tools, also called parting tools, are used to cut off the ends of stock in a lathe. Usually the portion that is cut off is a partially or completely finished workpiece.

L. *Boring Tool.* Boring tools are used to enlarge holes. There are many different styles of boring tools of which the tool illustrated is one example. Three features of these tools must be carefully controlled: 1. the end relief angle must be large enough to provide an adequate clearance with respect to the wall of the hole; 2. the nose radius should be as small as possible; and, 3. the lead angle should be small, preferably zero degree for small diameter boring bars. The nose radius and the lead angle tend to deflect the boring bar away from the wall of the hole. They should, therefore, be kept as small as possible.

Figure 7-15 illustrates a group of typical cemented carbide indexable insert cutting tools and how they are applied. When selecting negative rake indexable insert cutting tools it is necessary to know where the relief angles are located since the tool can only cut with an edge that has a

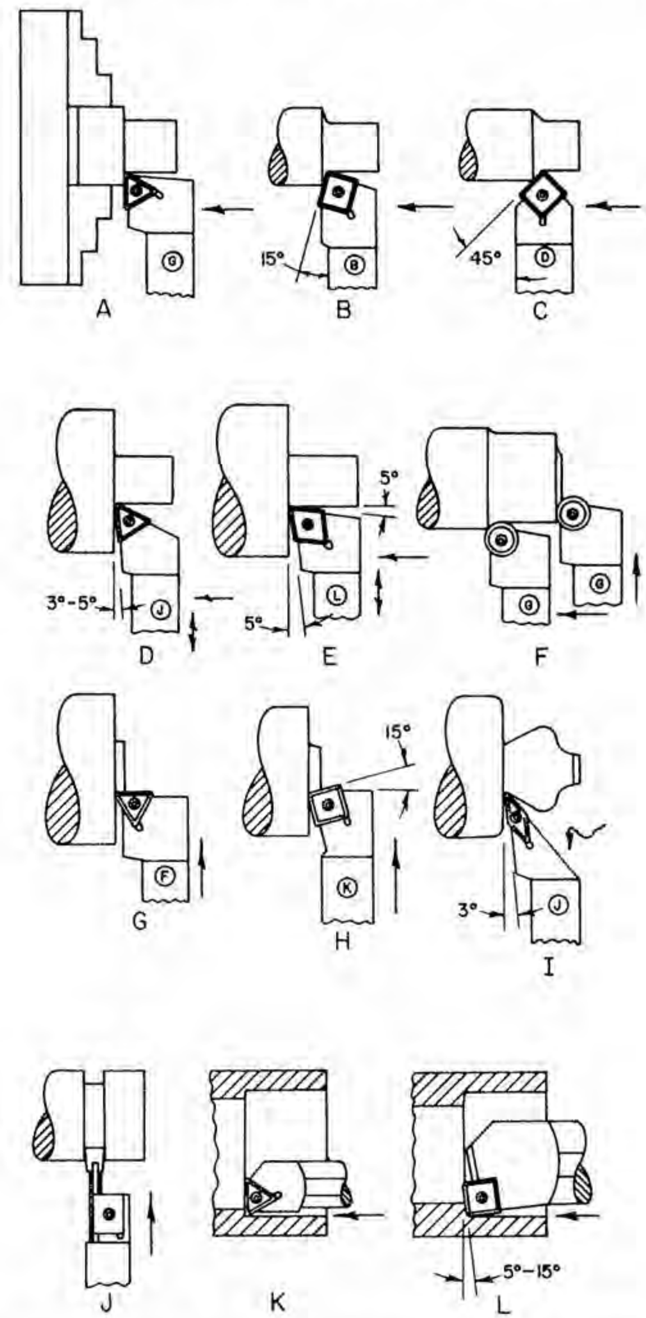


Fig. 7-15. Typical tool shapes and applications for cemented-carbide, indexable-insert cutting tools. Encircled letters are the standard toolholder designations.

relief angle. The relief angle on these inserts is provided by holding them at a compound angle on the tool holder. There will always be one or two edges of the insert that have a negative relief angle. Usually these edges are positioned on the body of the tool holder, but not always. The arrows on the illustrations show the direction in which negative rake tools will cut. Since the relief angles are ground or formed on positive rake inserts, they are not a cause of concern if the tool holder is correctly made. Standard indexable insert tool holders are designated by a letter symbol that is specified by an American National Standard (ANSI B94.26-1969). The ANSI designation of the tool holders shown in Fig. 7-15 are given by the letter enclosed within the circle. Each tool in Fig. 7-15 will be described below on the basis of the operations they are performing:

A. *Turning.* The G style tool holder illustrated has an offset to allow the triangular insert to cut close to chuck jaws or to a large shoulder. An A style tool holder is similar but does not have the offset. These tool holders have a zero degree lead angle and can be used to form a square shoulder following a turning operation.
B. *Turning.* The B style tool holder provides a 15-degree lead angle and allows a square insert to be used, which has more cutting edges available than the triangular insert.
C. *Turning.* The D style tool holder provides a 45-degree lead angle which allows a heavier feed rate to be used.
D. *Turn and Back Face.* This operation is used to produce shoulders that are precise with respect to squareness and location. The J style tool holder positions the insert to have a negative lead angle; i.e., it will cut with the nose leading. The procedure used is to turn the cylindrical surface to the shoulder, and then to backface the shoulder. If necessary, this tool can also be used to take a light facing cut by feeding inward toward the center of the workpiece.
E. *Turn and Backface.* In all respects the operation of this tool is identical to the tool shown at D. The insert is diamond shaped having an acute angle of 80 degrees. When held on the L style tool holder it has only two cutting edges per face available. However, this holder holds the insert very firmly and allows a heavier turning cut to be taken than is usually possible with the tool at D. The size of the backfacing cut is dependent primarily on the negative lead angle, although on tough materials the L style holder will hold the insert more firmly when backfacing.
F. *Turn and Facing.* Round inserts provide more usable cutting edge, or indexes, per face before they are used up than the other insert shapes. Furthermore, the large radius produces a good surface finish. When used on the G style tool holder, the round inserts can be used to take medium and light turning and facing cuts.

G. *Facing.* The F style tool holder and the triangular insert are used to take light and heavy facing cuts.
H. *Facing.* The K style tool holder utilizes a square insert to take light and heavy facing cuts.
I. *Contour Turning.* The diamond insert shown is specifically designed for contour turning, although other inserts are also used for this purpose depending on the shape of the contour. There are two types of diamond inserts designed for this purpose: one has a 35-degree acute angle while the other has a 55-degree acute angle. A J style tool holder is shown; other style tool holders are available which hold the insert in a different position.
J. *Cut-Off and Grooving.* Cut-off and grooving tool holders do not have an ANSI designation. Many different designs of carbide cut-off and grooving tools are available (see Figs. 7-10 and 7-11).
K. *Boring.* A large selection of indexable insert boring (Fig. 7-12) bars are available. When the lead angle is zero degrees a triangular insert is used, as shown.
L. *Boring.* Boring bars having a lead angle can utilize square inserts. The lead angle should be kept small on smaller diameter bars.

Some very useful single point tool shapes are shown in Fig. 7-16. The tool at A, is a combination "rough and finish" turning tool. A small flat is ground on the end-cutting edge, which is used to take a very light finishing or scraping cut, while the side-cutting edge is used to take a heavier cut. The width of the flat should be from $\frac{1}{16}$ to $\frac{3}{16}$ inch. The side rake angle of this tool must be zero degrees. It may be given a positive back rake angle up to 10 degrees. A similar tool, having a 45-degree, side-cutting-edge angle is shown at B. Either tool can be made from high-speed steel or from cemented carbide. The flat of each of these tools can be used to take a light finishing cut when desired, although, primarily, they are designed to take combined roughing and finishing cuts.

A broad nose finishing tool is shown at C, Fig. 7-16. When properly ground, honed, and applied it will produce an excellent surface finish. The high-speed steel, broad-nose tool has a deep hook ground on the face, which, in effect, gives it a very large rake angle. A small radius is ground or honed at each end of the cutting edge. The cutting edge should be honed to a sharp edge and a fine finish. Cemented carbide broad-nose tools should have a back rake angle from -6 to $+15$ degrees, depending on the hardness of the material being cut. The tool must be positioned with the end-cutting edge parallel to the work, as shown in the illustration, in order to contact the cylindrical work surface along the tool's entire length. It is used with a very light depth of cut which varies from approximately .001 to .005 inch. The recommended feed rate is .015 to .750 inch per revolution. Sometimes, even heavier feeds are used. The lighter feeds generally result in a better surface finish, while the heavier feeds result in higher production rates. The cutting speed used with broad-

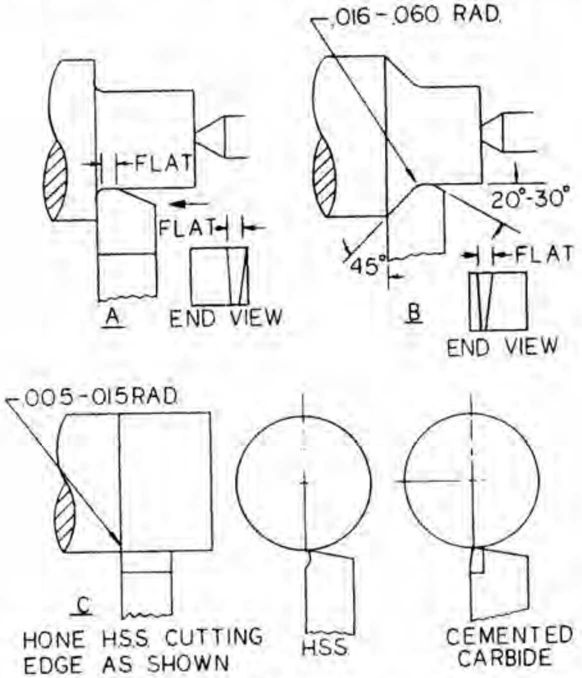


Fig. 7-16. A. Combination rough- and finish-turning tool. B. Combination rough- and finish-turning tool with lead angle. C. Broadnose finishing tool.

nose tools should generally be lower than recommended for turning tools. The high feed rates possible with this tool cause it to traverse the length of the turned surface much more rapidly than the turning tools; therefore, there is less wear on the cutting edge which results in greater accuracy when turning long, cylindrical surfaces. The short cutting time, and the greater accuracy obtained, make the broad-nose tool very useful when turning long, cylindrical or tapered surfaces.

Grinding Single-Point Cutting Tools

Single-point cutting tools are generally ground by hand on pedestal or bench type grinders. Some of these are specifically designed to grind single-point cutting tools and have a table on which the tools can be placed while they are held against the grinding wheel by hand. The grinding wheel used on these machines is designed for grinding on the side of the wheel in order to produce a true plane surface on the workpiece. The table can be set at an angle, with respect to the side of the wheel, thereby enabling the desired angle to be ground on the tool. Provisions are usually made to have a flow of coolant available in order to prevent the tool from overheating while grinding. Most single-point, cemented-carbide cutting tools are ground on this type of grinder using a diamond-

impregnated grinding wheel. High-speed-steel cutting tools can also be ground on these machines using an aluminum oxide grinding wheel.

Most high-speed-steel cutting tools, however, are ground on a simple pedestal or bench grinder. The tool bit is held by hand against the peripheral face of the wheel, at the angle required. This method is called off-hand grinding and it requires a considerable amount of skill. This skill can be acquired by analyzing what is to be accomplished and by practice.

A grinding wheel suitable for grinding high-speed steel should be mounted on the pedestal grinder. For this type of grinding a typical wheel specification is A-36-O-5-V for rough grinding, and A-60-M-5-V for finish grinding. (See Chapter 14, Volume 2, for the meaning of grinding-wheel specifications.) The grinding wheel should be dressed so that it is straight and that it runs true. Occasionally, it must be redressed in order to keep it sharp and true. A dull grinding wheel will cut more slowly and can cause the tool to become overheated. It is desirable to use a grinder equipped to supply a flow of coolant over the face of the wheel. If a coolant is not available on the grinder, the tool should be dipped in water, occasionally, to cool it. It should never be allowed to overheat to the extent of becoming discolored as this can cause metallurgical damage which will reduce the life of the tool. The most effective way of preventing the tool from becoming overheated is to control the pressure used to hold the tool against the face of the grinding wheel. A very heavy pressure can cause the tool to overheat even if a coolant is used on the machine and the grinding wheel is sharp.

A general-purpose, high-speed steel turning tool is shown in Fig. 7-17. Starting with a high-speed steel tool bit, the procedure for grinding this tool is shown in Fig. 7-18. When grinding, the tool bit should be held with the cutting edge facing upward so that it can be seen. Held in this manner, the grinding wheel will "run on" to the cutting edge instead of running off from it. This prevents the possibility of chipping and reduces the tendency to form a fine almost microscopic feather on the cutting edge. All of the angles on the tool should first be rough ground almost up to the cutting edges; then the angles are finish ground and the cutting edges sharpened. The nose radius is ground last. It is ground by holding the tool at the correct angle with a light pressure and then swinging the tool bit slowly back and forth between the side and end flanks.

Chip Formation

The three basic types of chips formed in metal cutting are the continuous chip, the discontinuous chip, and the segmental chip, Fig. 7-19. When the cutting tool first makes contact with the metal, it compresses the metal ahead of the cutting edge. As the tool advances, the metal ahead of the cutting edge is stressed to the point where it will shear internally, causing the grains of the metal to deform and to flow plastically along a plane called the shear plane, or roughly perpendicular to the

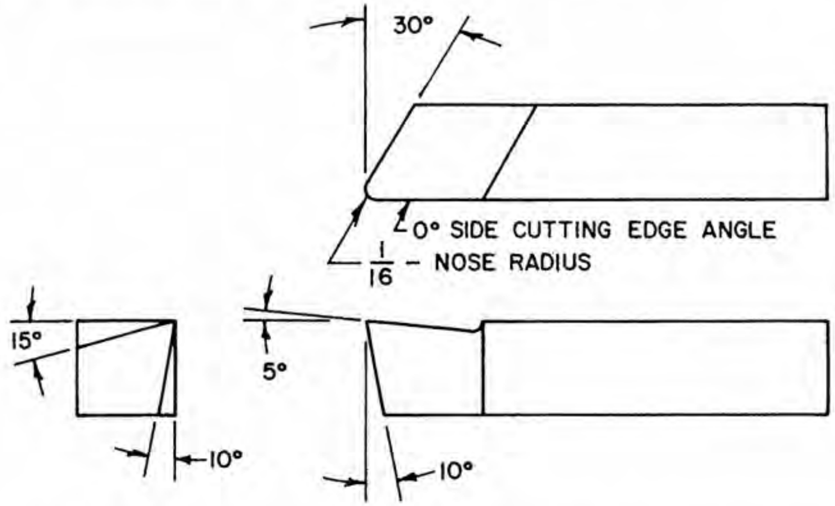


Fig. 7-17. General-purpose, high-speed steel turning tool. The sequence of grinding this tool is shown in Fig. 7-18.

face of the tool.* When the type of metal that is being cut is ductile, such as steel, the chip will come off in a continuous ribbon, as at A, Fig. 7-19. The internal shearing action of this type of chip, called a continuous chip, causes plastic flow but it does not cause the metal to separate, or fracture; the chip escapes by flowing over the face of the tool. The movement of the chip over the tool face, under high pressure, does further work to that portion of the chip in contact with the face. This surface, located on the underside of the chip, is work hardened and will have a shiny appearance.

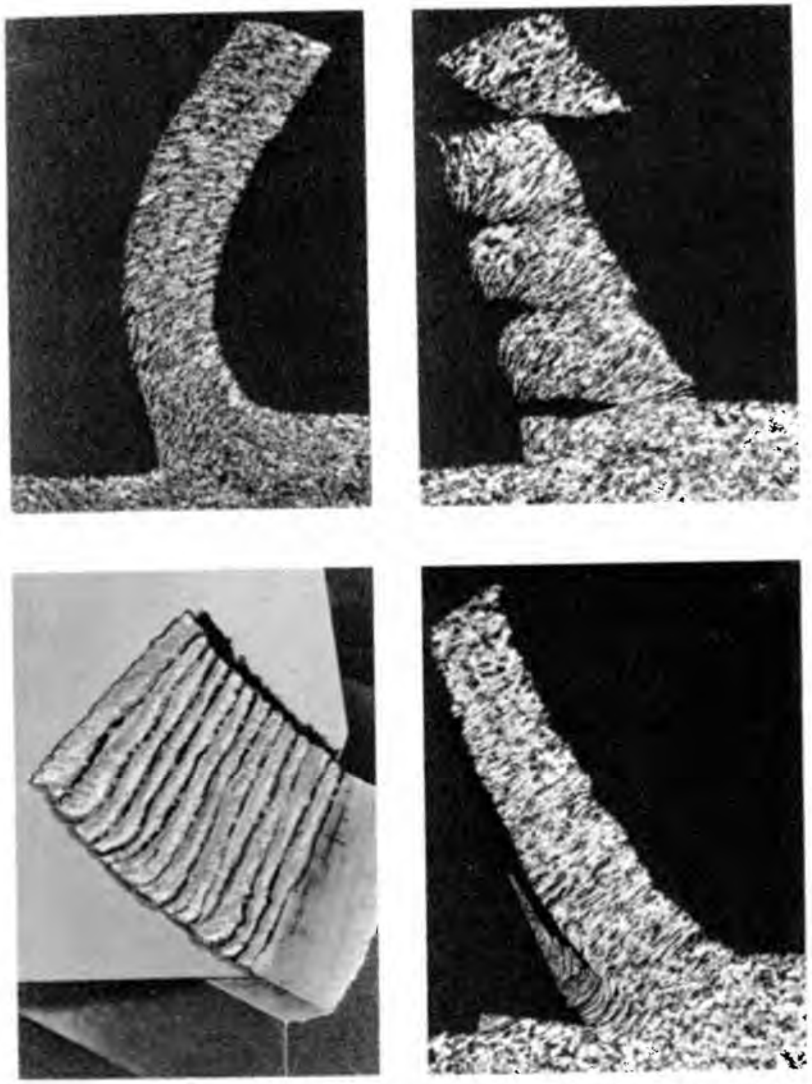
If the metal being cut is brittle, such as gray cast-iron, the stresses on the shear plane will eventually, cause the metal to fracture because it is unable to flow plastically as can a more ductile material. This causes the chip to break up into particles as shown at B, Fig. 7-19. Such a chip is called a discontinuous chip. The formation of a discontinuous chip is illustrated in Fig. 7-20.

A segmental chip is frequently encountered when machining ductile materials. It is shown at C, Fig. 7-19. When a heavy feed is used, the stresses on the shear plane cause the metal in this plane to eventually fracture internally. This may be further abetted either by low frequency and high amplitude vibrations of the cutting tool or the work, or both. The internal fractures will spread to the outside surfaces; however, they are also quickly welded together again by the action of the pressures existing in this region. The result is a continuous chip that has a file-

* The shear plane is not a true plane inasmuch as it usually is slightly curved. Furthermore, it is not a surface, but a region within the metal along which internal shear occurs.

* Fig. 7-18. Sequence for grinding general-purpose turning tool. (Upper left) Top view of simultaneously grinding the end cutting edge and end relief angle. (Upper right) Side view of same operation. (Center left) Top view of simultaneously grinding side-relief angle and side cutting edge angle (0°). (Center right) Side view of same operation. (Lower left) Simultaneously grinding the side-rake and back-rake angles. (Lower right) Grinding the nose radius.

* The views in Fig. 7-18 were obtained through the kind cooperation of Mr. Harold D. Manning, Chairman, Industrial Arts Dept., and Mr. Paul Vogel, student, at the James E. Sperry Senior High School, Henrietta, N.Y.

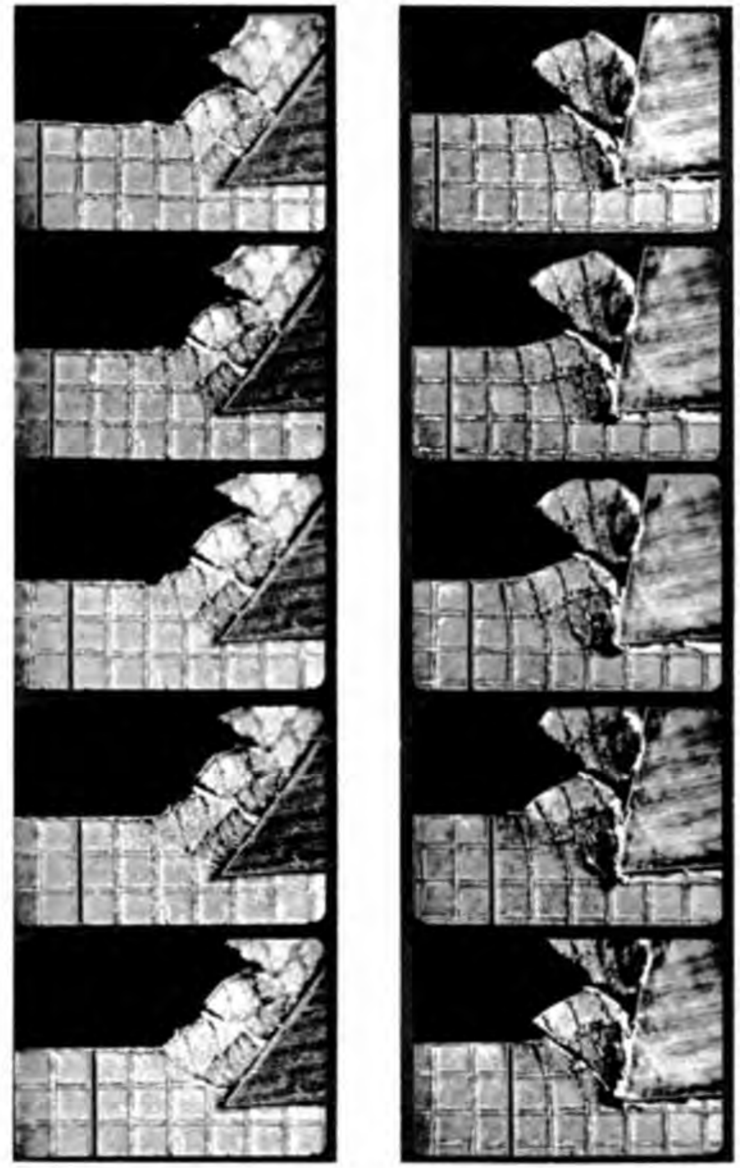


Courtesy of Cincinnati Milacron

Fig. 7-19. A (Upper left) Continuous chip. B (Upper right) Discontinuous chip. C (Lower left) Segmental chip. D (Lower right) Continuous chip with built-up edge.

tooth appearance on one side. The side of the chip that passes over the tool face is more completely welded together, and is burnished by the face so that it has a smooth appearance. The degree and depth of the segmentation of a chip can vary greatly because of the feed, the cutting speed, and the nature of the metal being cut.

Another phenomenon associated with the formation of the chip is the



Courtesy of Cincinnati Milacron

Fig. 7-20. The formation of a discontinuous chip. (Left) Cutting tool with large rake angle. (Right) Cutting tool with small rake angle.

built up edge shown at D, Fig. 7-19. This generally occurs when cutting ductile materials at relatively low (approximately, 150 to 200 fpm, or less) cutting speeds. The built-up edge is a highly work-hardened layer of the work material that has been welded onto the cutting edge. It starts out as a small layer that gradually builds up in size by having more work material continuously added to it by pressure welding. Eventually, the built-up edge enlarges to the point where it becomes unstable and it breaks off. When this happens, it may pass off with the chip or it may wedge between the work and the cutting edge to score the work surface. It can also pluck out small particles of the cutting tool material as it breaks off, resulting in a deterioration, or wear, of the cutting edge. When the built-up edge has broken off, a new built-up edge starts to form and this cycle is repeated. In some cases, the size of the built-up edge becomes quite large and protrudes beyond the cutting edge to score the surface of the work while it is still attached to the tool.

The effect of the built-up edge is to cause the machined surface of the work to be rough. *The built-up edge is the principal cause of surface roughness on the machined surface.* Efforts to achieve a good surface finish should be directed toward eliminating the formation of the built-up edge. It will *not* form when the cutting speed exceeds a certain critical rate which can be easily reached with cemented-carbide cutting tools. When these tools are operated above this speed, an excellent surface finish is obtained without difficulty. As high-speed steel must usually operate below this cutting speed, in order to obtain a good surface finish, a cutting fluid is used with high-speed steel tools, to reduce or to eliminate the built-up edge. Using a large rake angle, a light feed rate, and having a smooth surface finish on the cutting edge will also help. Since the formation of the built-up edge requires a metallurgical interaction between the work material and the cutting tool material, using a cutting tool material that does not react with the work material will also reduce or eliminate the built-up edge. Industrial diamonds fall into this category; they will produce excellent surface finishes; however, their use is restricted to very light finishing cuts.

Chip Control

Chip control is not a problem when cutting metals that form a discontinuous chip. It is only troublesome when machining metals that form a continuous or a segmental chip with high-speed-steel cutting tools. Chip control, on the other hand, is essential when forming continuous or segmental chips with cemented-carbide cutting tools. The high cutting speeds used cause such a rapid formation of the chip that both the machine operator and the equipment are endangered.

The action of a plate type chip breaker is shown in Fig. 7-21. The plate deflects the chip, causing it to curl, and to strike against the flank of the tool holder or the chip breaker. When the chip strikes the tool holder it breaks off into short lengths which are curled and have an appearance like a figure 9. The size of the broken off chips depends on the feed rate and the design of the chip breaker. Chip breakers may be ground into high speed steel and into cemented-carbide tools to cause the chip

Courtesy of Kennametal Inc.

Fig. 7-21.

to curl and break off as described. The three basic types of chip breakers are shown in Fig. 7-22. Many cemented carbide inserts for turning ductile materials that form a continuous chip have pressed-in chip breakers or a chip control geometry that is formed in the insert when it is made. A chip breaker plate is not used with these inserts.

Tool Wear

All metal cutting tools begin to wear at the instant they start to cut. Tool wear is unavoidable. However, the rate of tool wear must be kept within acceptable limits to obtain a reasonable tool life. Fundamentally, tool wear is caused by some very complex mechanical, chemical, and metallurgical interactions between the cutting tool material and the work material. The visible manifestation of these interactions provides useful information in improving the performance of the cutting tool. They will now be described.

Flank Wear. This type of tool wear pattern is normal and will always occur. It cannot be avoided, but the rate at which it progresses must be controlled to obtain a reasonable tool life. Flank wear is illustrated in Fig. 7-24A. The worn surface on the flank just below the cutting edge is called the flank wear land. It is a very uniform band that can readily be seen with a low powered magnifying glass or magnifying mirror. The width of the wear land can be measured with a low powered toolmakers microscope, and the progress of the wear as the tool cuts can be shown on a graph, as in Fig. 7-23. When a new tool starts to cut it will wear very rapidly during the first minute or two of its operation; this is called the

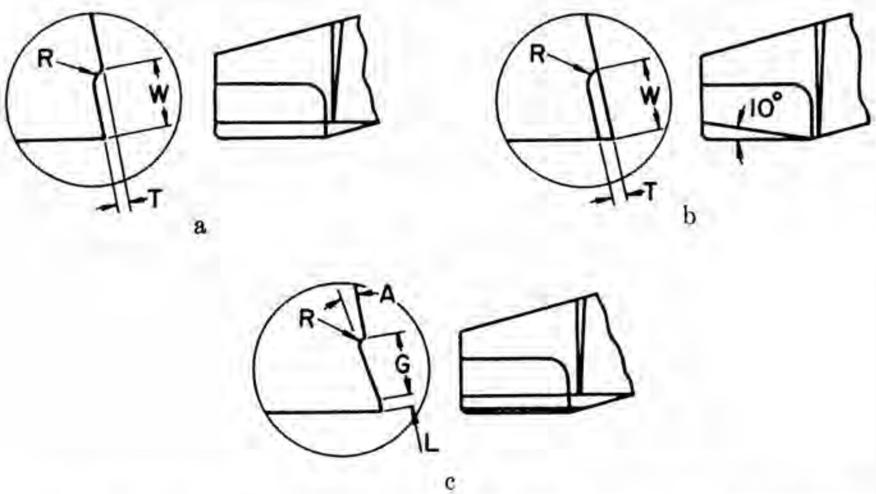


Fig. 7-22. Basic types of chip breakers: a. Parallel, b. Angular, c. Groove type.

initial or break-in flank wear. While the rate of wear during this period is rapid, under normal operating conditions, the amount of wear is not usually great. After the break-in wear has occurred, there is a long period of time during which the rate of wear is very uniform; i.e., the width of the wear land increases at a uniform rate. The rate at which the wear land increases is primarily affected by the cutting speed, then on the feed rate, and least of all on the depth of cut. As the tool cuts, the wear land rubs against the work material and the resulting friction creates heat. When the wear land is small, the heat generated by the rubbing action is much less than the heat generated in forming the chip; however, when the wear land reaches a certain width (approximately .035 to .050 in., or 0.89 to 1.27 mm), the heat generated by the flank wear land increases rather rapidly causing the flank wear rate to also increase rapidly. Then the combined effect of the heat generated by the flank wear land and by the formation of the chip causes the temperature of the cutting edge to rise to the point where the tool material becomes soft and unable to cut. At this point a catastrophic failure of the cutting edge occurs. Before catastrophic failure occurs, the rubbing action of the wide wear land will burnish the workpiece, giving it a shiny appearance.

The cutting tool should be replaced or resharpened before catastrophic failure takes place. Experience has shown that this should be done when the width of the wear land reaches .010 to .020 in. (0.25 to 0.50 mm) for finishing cuts, .030 in. (0.76 mm) for most medium and heavy roughing cuts, and .040 in. (1.02 mm) for extremely heavy roughing cuts, such as

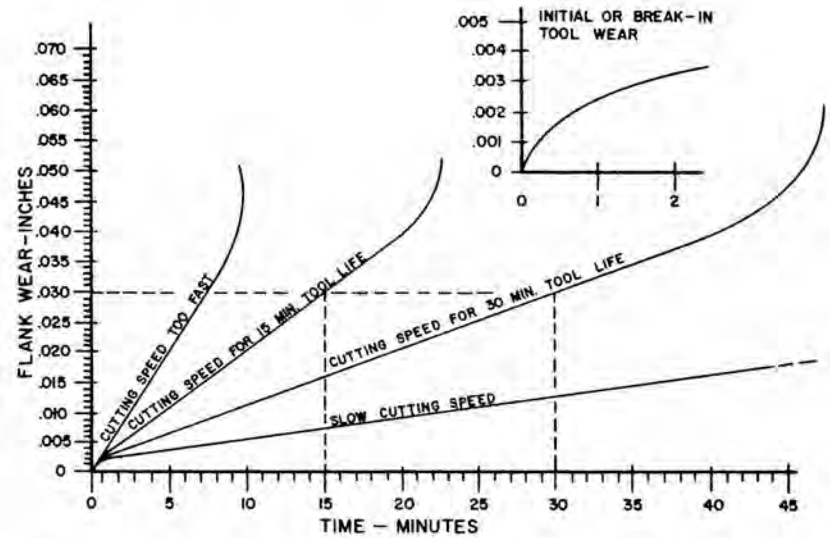


Fig. 7-23. Typical graph showing how the flank of a metal cutting tool wears. Higher cutting speeds cause flank wear rate to increase; slower cutting speeds reduce flank wear rate. Insert shows initial, or break-in, wear occurring during first minute or two of cutting time.

encountered when turning steel mill rolls. The length of time that the tool will cut before reaching the prescribed amount of flank wear defines its tool life. Therefore, decreasing the rate of flank wear will increase the tool life, as shown in Fig. 7-23. Flank wear rate and tool life depend on the combined effect of the work material, the work material hardness and heat treatment, the cutting tool material, the cutting speed, feed, and depth of cut. For a given combination of work material and tool material, the most effective way to increase the tool life is to decrease the cutting speed. Next in effectiveness is to reduce the feed rate and least effective is to reduce the depth of cut. A coolant-type cutting fluid will usually reduce the flank wear rate and increase the tool life. The flank wear rate is very sensitive to the grade of cemented carbide used in the cutting tool; tool life can be increased significantly by selecting the best grade of carbide for the particular application.

Built-Up Edge. The built-up edge (Fig. 7-24B) is the principal cause of roughness on the machined surface. It also damages the cutting edge and contributes to tool wear. How the built-up edge forms, grows, and breaks off has been described in this chapter under "Chip Formation" and will not be repeated here. Very often, when the built-up edge is broken off, it will pluck out a piece of the cutting edge to which it has been welded,

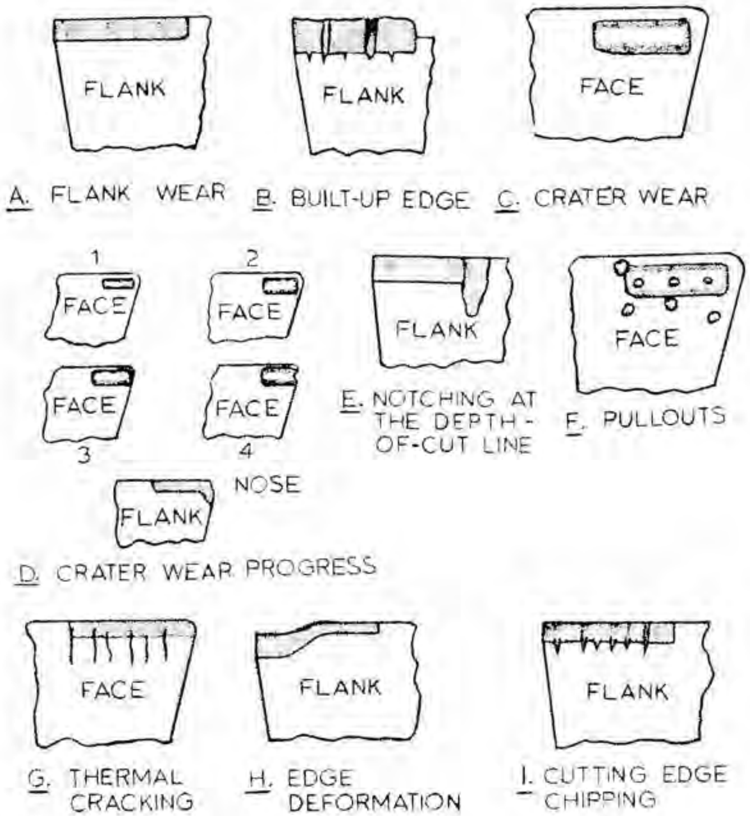


Fig. 7-24. Types of tool wear.

thereby forming a notch on the flank that may extend below the flank wear line giving it a jagged appearance. The voids thus created are quickly filled with work material which is welded to the tool material and work hardened by the pressure of the cut. Surprisingly, controlled tests have shown that on high speed steel tools, the built-up edge may actually prolong the life of the cutting tool by "repairing" the damaged areas with work material that is hardened and welded in place. These tests show that the tool life may be somewhat erratic; however, the tool life is not shortened and may be increased. Studies have shown that the tool *may* cut for periods of time without any increase in the width of the wear land. Apparently the built-up edge protects the flank of the tool from exposure to wear.

The built-up edge is not a problem when rough turning with high speed steel cutting tools, except when turning certain very soft materials which form an excessively large built-up edge. Since it does cause roughness on

the machined surface, on finishing cuts steps should be taken to reduce or eliminate the built-up edge. For high speed steel cutting tools, the most effective means of combating the built-up edge is to use a mineral oil base cutting fluid having good anti-weld properties. The rake angle on the tool should be made as large as possible and the surfaces on the face and flank should be honed smooth. Unless required to add strength, the cutting edge should not be honed round or to a bevel edge; it should remain sharp. When cutting with cemented carbide cutting tools, the most effective and easiest way to eliminate the built-up edge is to increase the cutting speed above the speed where the built-up edge will not form. For many materials this cutting speed is approximately 150 to 250 fpm (45 to 75 m/min.).

Crater Wear. The combined effect of heat, pressure, and abrasion caused by the movement of the chip, and some very complex chemical and metallurgical interactions between the work material and the tool material cause the surface on the face of the cutting tool near the cutting edge to erode. When this erosion becomes severe, a crater will form on the face near the cutting edge. The severity of the erosion is dependent on the work material, tool material, and the cutting speed. A typical crater is shown in Fig. 7-24C, and the progress of crater wear is shown at D. The formation of the crater starts a short distance behind the cutting edges. As the tool continues to cut, the crater will enlarge, in depth as well as spreading out. In time the crater approaches the cutting edges; usually it will approach the end cutting edge first. When this occurs the end cutting edge will be weakened and will break off in the region of the crater. Since this is close to the nose of the tool, the nose will fail shortly afterwards. With the nose having failed, the failure of the cutting tool is complete. This type of tool failure is called crater breakthrough. When machining titanium and its alloys the crater forms very close to the side cutting edge; thus, the crater will usually break through at the side cutting edge to cause tool failure.

When straight tungsten carbides cut steel, a crater will form very rapidly and crater breakthrough on the end cutting edge will occur in a short time. Steel should always be cut with a crater resistant carbide or with a coated carbide. Straight titanium carbide cutting tools have excellent crater resistance, but they tend to be brittle. Crater breakthrough can be retarded by using a tool having a small end cutting edge angle. If all else fails to prevent excessive cratering, the cutting speed should be reduced.

Notching at the Depth of Cut Line. This form of tool wear is characterized by a deep notch that forms at the depth of cut line, as shown in Fig. 7-24E. The notch is caused by cutting through hard and abrasive scale, or by a layer of highly work hardened material. Scale that is formed on the surface of castings, forgings, hot rolled stock, and heat treated parts is composed of a variety of oxides that are very abrasive and hard. Some materials, such as the austenitic stainless steels, work harden very easily when they are machined or cold worked, forming a layer of very hard material. High temperature alloys also cause excessive notching at the depth of cut line.

Notching at the depth of cut line is very difficult to prevent when it occurs. The most common corrective action is to use a tool having a large lead angle or, when this cannot be done, to reduce the feed rate. Honing a bevel on the cutting edge at the depth of cut line may be helpful, and using a harder grade of carbide may also help.

Pullouts. Pullouts (Fig. 7-24F) are very small craters that form on the face of the tool when machining materials that will readily weld to the tool material, such as high temperature alloys. Usually a larger crater will also form and the pullouts occur around the crater. Pullouts will also occur within the crater; however, they will usually be filled with work material that is welded in place. Pullouts are caused by particles of the chip that are welded firmly to the face of the tool. As the chip continues to sweep by, it will literally knock off the welded-on particle, which will pluck out a small particle of the tool material as it breaks off, leaving a small crater behind. While pullouts must be considered a form of tool wear, they are generally not harmful to tool life, except when they enlarge the crater or when they occur near the cutting edge. When they occur near the depth of cut line, pullouts can contribute to the formation of the notch in this region. Usually no steps are taken to prevent the formation of pullouts. The remedial measures are to decrease the cutting speed and to use a suitable cutting fluid.

Thermal Cracking. Thermal cracks (Fig. 7-24G) occur when the cutting zone of the tool is rapidly heated and cooled while the adjacent tool material is not unduly heated. This causes large differences in temperature, or temperature gradients, between adjacent layers of material on the tool, which in turn cause the cracks to form. Since the large temperature differences do not occur at the cutting edge but a short distance away, the cracks will first form a short distance away from the cutting edge. As the tool continues to cut, the cracks will propagate toward the cutting edge until it is finally reached. When the cracks reach the cutting edge, the tool will ultimately, but not necessarily immediately, fail. Thermal cracking generally occurs when machining at high speeds with cemented carbide tools. They may be caused by an interrupted cut or when milling, when the tool is alternately heated and cooled. A coolant that reaches the hot cutting tool only intermittently can cause thermal cracks to form. In this case a constant coolant supply to the tool must be obtained, or no coolant should be used. Thermal cracking may be prevented by a large reduction in the cutting speed. First, however, a tougher and more shock resistant grade of carbide should be tried.

Another major cause of thermal cracking is improper tool grinding. Any cutting tool can be severely damaged in this manner before it is used. Grinding (thermal) cracks are caused by pressing the tool hard against the grinding wheel, or by using too much infeed. A dull grinding wheel may also cause thermal cracks; always use a sharp grinding wheel when grinding a tool. Cemented carbides may be damaged by using an incorrect type of abrasive; a diamond abrasive is recommended for grinding carbide tools.

Edge Deformation. When taking a very heavy cut with a cemented carbide cutting tool, the carbide in the cutting zone may be heated to the point where it softens and deforms under the cutting load. As shown in Fig. 7-24H, this often occurs in the region of the nose. Deformation will result in the formation of a large wear land after a very short cutting time. Tantalum carbide is added to the tungsten carbide to resist deformation. Whenever deformation is apt to occur, a grade of carbide containing tantalum carbide should be used. Sometimes a coolant may be used to cool the tool, which helps to prevent deformation. If these preventative measures will not help, the feed and depth of cut must be reduced.

Cutting Edge Chipping. This type of tool failure is not always easy to recognize because the chipping may be minute, although more massive chipping may also occur. Often a ragged wear land will be an indication of chipping, as shown in Fig. 7-24I. Chipping is the mechanical failure of the cutting edge caused by overloading or by inherent lack of strength. (Chipping can also be caused by the built-up edge, but this type of chipping will not be considered here.) Anything that will strengthen the cutting edge will help to overcome chipping. Honing the cutting edge will strengthen it and often prevent chipping. This is easy to do on the job, or pre-honed inserts can be purchased when this type of tooling is used. Honing too much off the cutting edge should be avoided as this will reduce the tool life. One method is to hone a flat land on the cutting edge that is .002 to .005 in. (0.05 to 0.13 mm) wide and at an angle of 30 degrees with respect to the face. Other steps to strengthen the cutting edge may also be taken. A softer but tougher grade of carbide, or a high-speed steel that does not contain cobalt may be used as a cutting tool material. The relief angle on the tool should be as small as possible. Using a negative rake angle is often the most effective way to stop chipping. Chipping may not be the fault of the cutting tool. It may be caused by machine tool vibration, chatter, or a lack of rigidity of the setup.

Oxidation: Tool wear is caused by abrasion and by complex chemical-metallurgical reactions occurring at the tool-chip interface and on the flank of the tool. It is also caused by oxidation of the heated tool material which is exposed to the atmosphere. Oxidation of the tool material may be a significant cause of tool wear whenever the tool material is heated to a very high temperature, such as when operating at very high cutting speeds.

Metal Cutting Fluids

The functions of metal cutting fluids are: 1. to cool the cutting tool as a primary function, and also to cool the workpiece; 2. to reduce the heat generated by friction through lubrication; 3. to provide anti-weld properties in order to prevent welding of the chip to the tool; and 4. to wash away the chips. There are three basic types of cutting fluids: 1. water soluble oils; 2. straight cutting oils; and 3. chemical fluids.

Water soluble oils, also called emulsified oils, are mixed with water in a water-to-oil ratio ranging from 5 to 1 to as much as 50 to 1. The mixture has an appearance ranging from milky to cloudy, or almost clear. Water is the best coolant, but has many disadvantages as a cutting fluid, the primary disadvantage being that it causes rust. Soluble oils prevent rust, are excellent coolants, and have a measure of lubricity. Additives may be added to soluble oils to enhance their performance and to prevent bacterial growth. Soluble oils can be used for almost all light, medium, and heavy-duty machining operations. They can be used over the entire range of cutting speeds, including the higher speeds where cemented carbide tools are used. They are not recommended for certain severe cutting operations requiring greater lubricity and anti-weld properties. Soluble oils are also recommended for almost all precision grinding operations.

Straight cutting oils are made from mineral oil that is usually compounded with additives to increase their effectiveness. Uncompounded mineral oils are sometimes used on very light duty machining operations on aluminum, magnesium, brass, and sulfurized free cutting steels. However, in most instances these materials are machined with mineral oils having fatty oil additives. Fatty oil additives add lubricity to the cutting oil. They are largely composed of lard oil, caster oil, sperm oil, and fatty esters. Active additives will react mildly with metal to form an oxide film. Sulfur in the base mineral oil is usually not active, but when added to the oil it is an active additive. Chlorine is an active additive. Active sulfur will not become active until a higher temperature is reached, while chlorine becomes active at a lower temperature. Singularly or combined, active sulfur and chlorine provide anti-weld properties to the cutting oil to prevent the chip from welding and to prevent or reduce the formation of the built-up edge, thereby improving the finish on the machined surface. Straight oils should be used when the cutting speed is moderate or low and on work where soluble oils do not give a satisfactory performance. They are particularly recommended on jobs where the chip is crowded, such as thread cutting. Straight oils are used on some precision grinding operations; however, provisions must be made to prevent the vapors from exploding.

Chemical fluids, also called synthetic cutting fluids, are proprietary chemical compounds to which water is added. There are two kinds; those having a wetting agent and those that do not. Those that do not have a wetting agent are restricted to rough grinding operations, where they tend to keep the grinding wheel open and free cutting. Those having a wetting agent have some lubricity to allow the machine tool slides and other machine parts to function smoothly. Chemical fluids are excellent coolants and are recommended for machining operations that are in the moderate to higher cutting speed range.

In general, cutting fluids should be applied in a continuous stream that floods the cutting tool. An intermittant supply on a hot cutting tool will likely cause thermal cracks to form on the tool. It is not, however, poor practice to apply a cutting fluid manually with a brush or a squirt can whenever the cutting speed is moderate or low. There are many occasions in the toolroom or in a job shop where this practice is the most practical method of application. At these speeds the tool is not likely to be damaged and the application of the cutting fluid is beneficial.

The only real measure of a cutting fluid is how it performs on the job. When selecting a cutting fluid, the cutting speed should be considered. In general, when the cutting speed exceeds about 75 to 100 fpm (25 to 30 m/min) a water soluble oil or a chemical fluid should be used. Below this speed range, first consider a water soluble oil; then if the job is too difficult for a water soluble oil, select a straight cutting oil.

Estimating the Power Required for Turning

A knowledge of the power required to perform machining operations is useful when planning new machining operations, optimizing existing machining operations, and developing specifications for new machine tools to be acquired. The measure of power in customary inch units is the horsepower. In SI metric units the kilowatt is the measure of both mechanical and electrical power. The power required to cut a material by machining depends on the rate at which the material is being removed and by an experimentally determined power constant, K_p, which is also called the unit horsepower, unit power, or specific power consumption. The power constant is equal to the horsepower required to cut a material at a rate of one cubic inch per minute. In SI metric units, the power constant is equal to the power in kilowatts required to cut a material at a rate of one cubic shell reamer is separate from the body of the reamer. The advantage of this design is that reamers of different size can be held on the same arbor and when a body is worn out the arbor does not require replacement.

Counterbores, Countersinks, and Spotfacers

Counterbores, countersinks, and spotfacers are frequently used in drill-press work in order to modify an existing hole. The counterbore, A, in Fig. 4-16, is used to enlarge the end of a hole by cutting a cylindrical surface that is concentric with the original hole. The concentricity is obtained by the pilot located on the end of the counterbore and which acts to guide the tool as it penetrates into the work. A countersink, B, in Fig. 4-16, is used to make a cone-shaped enlargement in the end of the hole. A spotfacing tool is shown at C. The purpose of the spotfacing operation

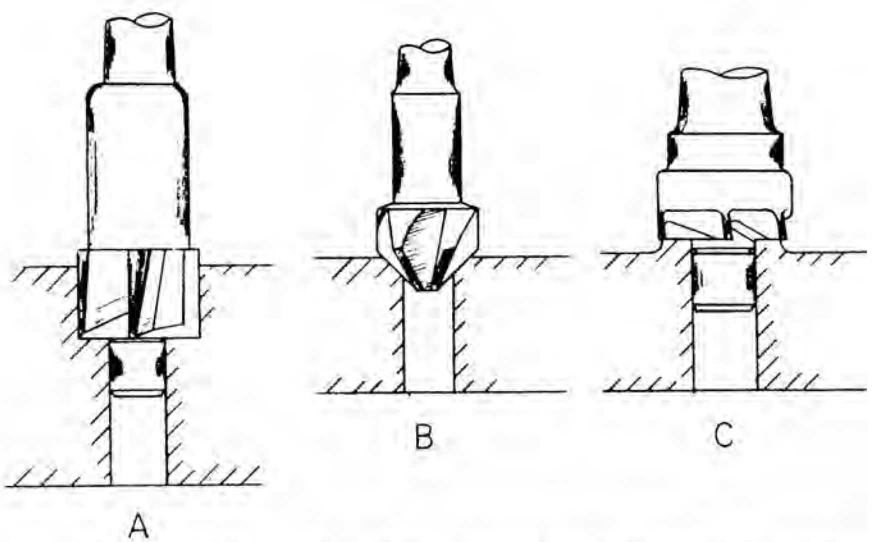
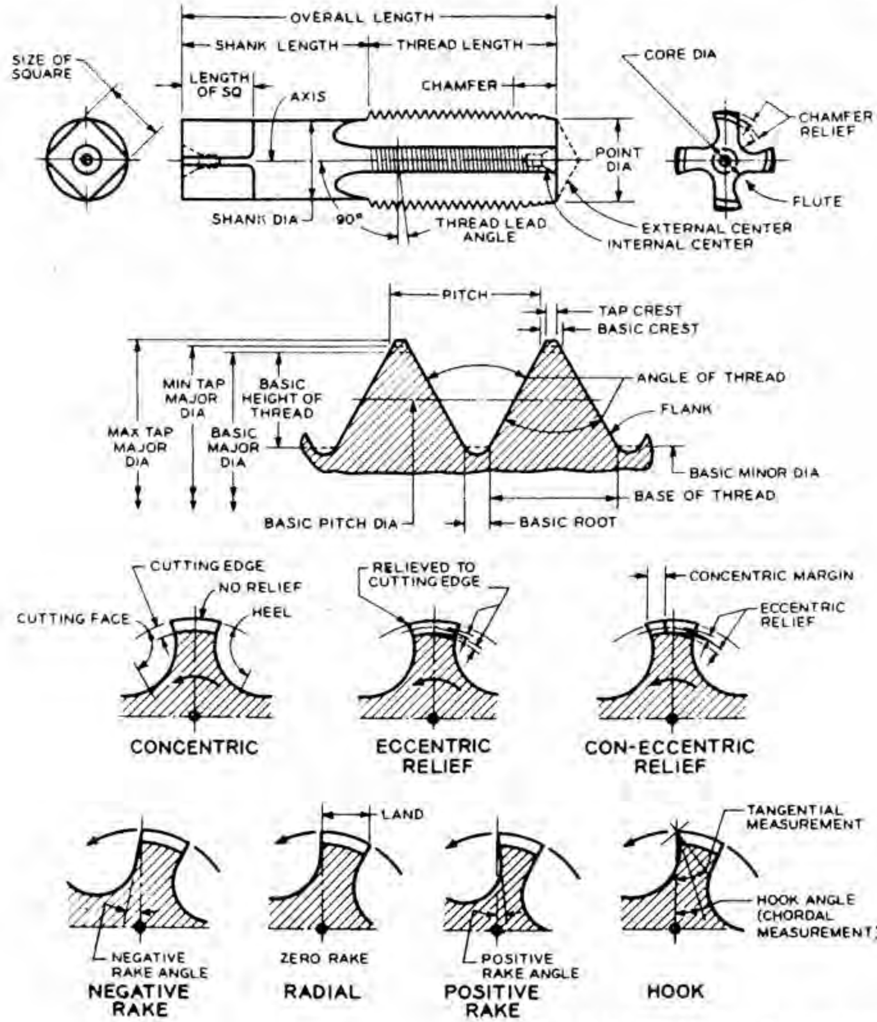


Fig. 4-16. A. Counterboring a hole. B. Countersinking a hole. C. Spot-facing a hole.

is to cut a smooth, flat surface which is perpendicular to the axis of the hole. Usually, this surface serves as a seat for the head of a cap screw or bolt; or as a seat for a nut.

Taps

The basic nomenclature of a tap is shown in Fig. 4-17. A tap is used to produce internal threads. It has thread-shaped teeth which cut a thread of similar form when it is screwed into a drilled hole. The first step in tapping an internal thread is to drill a hole with a tap drill. This is an ordinary drill that is appropriately sized so that it will leave just the right amount of metal in the hole for the tap to remove and finish



Courtesy of the Metal Cutting Tool Institute

Fig. 4-17. Basic nomenclature of a tap.

the thread forms to required size. The tap is then screwed into the workpiece by hand, with the aid of a tap wrench, or by power, as applied by a machine tool such as a lathe or a drilling machine. The feed of the tap must always be equal to the lead of the thread being cut. The lead of a thread is the distance that it advances in one revolution. A tap will generally feed itself, once it has started, as it will tend to follow that portion of the thread that already has been cut in starting.

The size of the tap drill to be used to tap the hole is dependent upon the diameter of the thread, the size of the thread tooth (or pitch of the thread), the material of which the thread is made, the length of engagement of the thread, and the available standard drill sizes. For ordinary manufacturing, the internal thread in a tapped hole need not be more than 75 or 80 per cent of the theoretical, standard thread depth. For some classes of work, not more than 50 per cent of the thread depth is required. The diameter of the tap drill should not be smaller than is necessary to give the required strength of thread, as every decrease in the diameter of the tap drill increases the power required for tapping, and, incidentally, the percentage of broken taps. Recommended tap drill sizes and hole sizes for tapping Unified Screw Threads can be found in various standard machine shop handbooks such as Machinery's Handbook.

Theoretical tap drill sizes for inch and metric threads can be calculated by the following formulas, in which the "Percent Full Thread" is expressed as a decimal; e.g., 75% is written .75 in the formula. The drill size nearest to the calculated hole size is used.

For American Unified Thread Form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.08253 \times \text{percent full thread}}{\text{Number of threads per inch}} \qquad (4\text{-}1)$$

For American National Thread form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.299 \times \text{percent full thread}}{\text{Number of threads per inch}} \qquad (4\text{-}2)$$

For ISO Metric Threads (all dimensions in millimeters):

$$\text{Hole size} = \text{Basic major dia.} - (1.08253 \times \text{pitch} \times \text{percent full thread}) \qquad (4\text{-}3)$$

Example 4-1:

Find the tap drill diameter for a ½-13 Unified thread and for an M12-1.75 thread. The M12-1.75 thread designation means that it is metric (M), it has a diameter equal to 12 mm, and the pitch of the thread is 1.75 mm. In each case the tapped hole is to have a 60 percent full thread.

For the ½-13 Unified thread:

$$\text{Hole size} = .500 - \frac{1.08253 \times .60}{13} = .450 \text{ in.}$$

For the M12-1.75 metric thread:

$$\text{Hole size} = 12 - (1.08253 \times 1.75 \times .60) = 10.86 \text{ mm}$$

The nearest drill sizes are 29/64 in. and 11.0 mm, respectively.

Hand taps have a cylindrical shank and a driving square on the end to which a tap wrench can be attached. These taps are used for most machine tapping applications, as well as for hand tapping. They can be obtained as ground taps, which are finished by precision grinding, or as cut taps, which are finished by a cutting operation.

Hand taps are classified into the following types: taper, plug, and bottoming. Each type is distinguished by the amount of chamfer on the end. The angle formed between the chamfer (see Fig. 4-17) and the axis of the tap is called the chamfer angle. It determines the number of threads having their height reduced on the end of the tap. The chamfered threads are relieved behind the face and do all of the cutting to form the thread on the part. The number of flutes and the length of the chamfer determine the chip load, or the amount of material removed by each tooth.

Taper taps have a chamfer of 4½° which reduces the height of approximately 8 to 10 threads. Plug taps have a chamfer angle of 10½° with 3 to 5 threads reduced in height. Bottoming taps have a chamfer angle of 25° to 29°, which reduces approximately 1½ threads. The most useful tap for general tapping applications is the plug tap. Plug taps can be used to tap through holes and also blind holes where a limited number of imperfect teeth are allowed. Bottoming taps are used to finish the bottom of blind holes where only a few imperfect teeth are permitted. Taper taps are used primarily for hand tapping thin workpieces, for tapping through holes in some abrasive materials, and for tapping the larger thread sizes and thread forms, where the chip thickness per tooth must be kept low. When a particularly hard, tough material is encountered, it is sometimes helpful to use a taper and a plug tap alternately. The taper tap starts easily but may stick after penetrating deeper; it is then removed and a plug tap is inserted to relieve the long cut made by the taper tap. For some very difficult jobs, a series of succeedingly larger taps are used to tap the hole, such taps being made specially for each job. On the large majority of jobs, however, the plug tap can be used alone.

On a limited number of softer, ductile materials, threaded holes can be produced on a machine with a forming tap. This is usually associated with high volume production operations. Forming taps have no flutes of the conventional type. Instead of being round, the body of the tap is radially relieved to form a lobed construction, the number of lobes depending on the tap diameter. The nose end of the tap is tapered to allow it to enter the hole. Forming taps produce threads by plastic flow of the metal near the hole walls instead of by cutting. A chip is not formed. The thread form produced is not perfect, but usually there is no reduction in the strength of the thread.

A cutting fluid should always be used with all materials when tapping except on cast iron and plastics. When hand tapping a hole, the tap should be advanced approximately one turn and then rotated in the reverse direction until the chip is broken. By repeating this sequence until the hole is tapped, a smooth thread will be produced with minimum effort. Machine-driven taps are generally screwed into the hole while rotating in the forward direction and are reversed to remove the tap when the job is finished. On very difficult jobs the tap is sometimes reversed when cutting into the hole in a manner similar to hand tapping.

Spade Drills

Spade drills, Fig. 4-18, are frequently used to drill large diameter holes. They are made in sizes ranging from one to five inches. A spade drill consists of a holder made from heat-treated steel and a high-speed steel, or cemented-carbide, tipped blade. The blade is seated in a slot in the end of the holder which is clamped together by a screw to secure the blade in place. Each holder can hold a range of different blade sizes. The shank of the holder may be straight or have a standard Morse taper. Some spade drill holders have coolant holes drilled lengthwise which supply a stream of coolant close to the cutting edges. Notches or chip breakers are ground on the cutting edges to reduce the width of the chip. These notches are unevenly spaced so that they do not occur at the same radial distances on the two cutting edges. Instead of a single chip, the notches cause several smaller chips to form on each cutting edge that are easier to break. A chip curling groove (chip curler) is ground on the face of each cutting edge which tends to curl and break the chips for easy disposal.

Several different types of blades are shown in Fig. 4-18 to show the versatility of this method of drilling. Besides drilling holes through solid metal, stepped holes can be drilled, and both counterboring and spot-

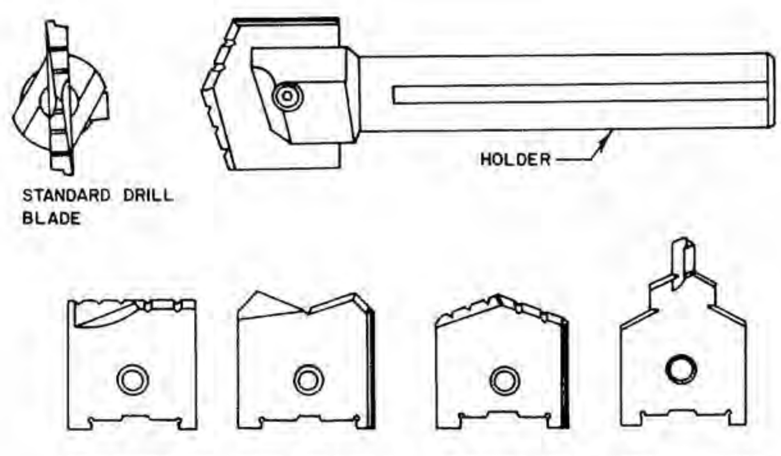


Fig. 4-18. Spade drill, including examples of standard and special blades.

facing can be done with spade drills. An important application of these drills is core drilling; i.e., enlarging an existing hole which may be a rough cored hole of a casting. The core drilling blade used for this operation has spurs ground on the end which bite into the metal and guide the drill, thereby drilling a hole that closely follows the path along which the drill is fed. Without the spurs, a core drill will be influenced by any eccentricity of the existing hole and may not drill a true hole. Core drilling operations may be performed on drilling machines having a power feed, on turret lathes, vertical turret lathes, on both horizontal and vertical boring mills, and on numerically controlled machines.

A power feed should be used when drilling with a spade drill. Only by applying extreme care and skill can spade drilling be accomplished with manual feed. The feed rate is a most important factor in successfully applying a spade drill and the constant feed rate supplied by a power feeding mechanism should be used whenever possible; for deep holes it should always be used. The feed rate should be adjusted so that the chips break into forms resembling a small "c" or number "9."

Spade drills are self starting if they are held in short stubby spade drill holders. Longer spade drills are started by first drilling a start hole or "cone" with a spade drill held in a stub holder. If the start drill is the same diameter as the second spade drill, the start drill should drill a hole that is to the full diameter of the spade drill. If the start drill is smaller than the second drill it should not drill to its full diameter, but should leave only a conical surface against which the second drill can start. Never use a center drill or drill a pilot hole with a twist drill to start a spade drill. When the spade drill contacts the sharp edge of the hole formed by these drills, it will tend to "hop around" which can cause the cutting edges of the spade drill to chip.

The size of the hole that can be drilled with a spade drill is limited only by the power and rigidity of the machine. When insufficient power is available to drill a large hole, the hole may be drilled in two passes using a smaller spade drill to drill the first hole. The power, torque, and thrust force required to feed the spade drill may be estimated by the procedures provided further on in this chapter.

Gun Drills

Gun drills are customarily used to drill deep holes where the length of the hole is five or six times its diameter. These are special drills and are usually used on special deep-hole drilling machines. They have a single flute and a single cutting edge which may be high-speed steel or a brazed-on cemented carbide tip. The cutting edge may be designed to cut to the center of the drill. Some gun drills have a cutting edge that does not cut to the center of the drill, thereby leaving a core of metal which can be easily removed if the hole is a through hole. This method of pro-

See that there is sufficient extra material for finishing on large castings to permit them to be "cleaned up," even though they warp. In such castings, make sure that the metal thickness will be sufficient after finishing, even though the castings do warp.

Make sure that sufficient sections are shown so that the pattern makers and molders will not be compelled to make assumptions about the form of any part of the casting. These details are particularly important when a number of sections of the casting are similar in form, while others differ slightly.

Checking Machined Parts: Study the sequences of operations in machining and see that all finish marks are indicated.

See that the finish marks are placed on the lines to which dimensions are given.

See that methods of machining are indicated where necessary.

Give all drill, reamer, tap, and rose bit sizes.

See that jig and gage numbers are indicated at the proper places.

See that all necessary bosses, lugs, and openings are provided for lifting, handling, clamping, and machining the piece.

See that adequate wrench room is provided for all nuts and bolt heads.

Avoid special tools, such as taps, drills, reamers, etc., unless such tools are specifically authorized.

Where parts are right- and left-hand, be sure that the hand is correctly designated. When possible, mark parts as symmetrical, so as to avoid having them right- and left-hand, but do not sacrifice correct design or satisfactory operation on this account.

When heat-treatment is required, the heat-treatment should be specified.

Check the title, size of machine, the scale, and the drawing number on both the drawing and the drawing record card.

ALLOWANCES AND TOLERANCES FOR FITS

Limits and Fits.—Fits between cylindrical parts, i.e., cylindrical fits, govern the proper assembly and performance of many mechanisms. Clearance fits permit relative freedom of motion between a shaft and a hole—axially, radially, or both. Interference fits secure a certain amount of tightness between parts, whether these are meant to remain permanently assembled or to be taken apart from time to time. Or again, two parts may be required to fit together snugly—without apparent tightness or looseness. The designer's problem is to specify these different types of fits in such a way that the shop can produce them. Establishing the specifications requires the adoption of two manufacturing limits for the hole and two for the shaft, and, hence, the adoption of a manufacturing tolerance on each part.

In selecting and specifying limits and fits for various applications, it is essential in the interests of interchangeable manufacturing that 1) standard definitions of terms relating to limits and fits be used; 2) preferred basic sizes be selected wherever possible to reduce material and tooling costs; 3) limits be based upon a series of preferred tolerances and allowances; and 4) a uniform system of applying tolerances (preferably unilateral) be used. These principles have been incorporated in both the American and British standards for limits and fits. Information about these standards is given beginning on page 627.

Basic Dimensions.—The basic size of a screw thread or machine part is the theoretical or nominal standard size from which variations are made. For example, a shaft may have a *basic* diameter of 2 inches, but a maximum variation of minus 0.010 inch may be permitted. The minimum hole should be of basic size wherever the use of standard tools represents the greatest economy. The maximum shaft should be of basic size wherever the use of standard purchased material, without further machining, represents the greatest economy, even though special tools are required to machine the mating part.

Tolerances.—Tolerance is the amount of variation permitted on dimensions or surfaces of machine parts. The tolerance is equal to the difference between the maximum and minimum limits of any specified dimension. For example, if the maximum limit for the diameter of a shaft is 2.000 inches and its minimum limit 1.990 inches, the tolerance for this diameter is 0.010 inch. The extent of these tolerances is established by determining the maximum and minimum clearances required on operating surfaces. As applied to the fitting of machine parts, the word tolerance means the amount that duplicate parts are allowed to vary in size in connection with manufacturing operations, owing to unavoidable imperfections of workmanship. Tolerance may also be defined as the amount that duplicate parts are permitted to vary in size to secure sufficient accuracy without unnecessary refinement. The terms "tolerance" and "allowance" are often used interchangeably, but, according to common usage, *allowance* is a difference in dimensions prescribed to secure various classes of fits between different parts.

Unilateral and Bilateral Tolerances.—The term "unilateral tolerance" means that the total tolerance, as related to a basic dimension, is in *one* direction only. For example, if the basic dimension were 1 inch and the tolerance were expressed as 1.000 − 0.002, or as 1.000 + 0.002, these would be unilateral tolerances because the total tolerance in each is in one direction. On the contrary, if the tolerance were divided, so as to be partly plus and partly minus, it would be classed as "bilateral."

Thus,

$$1.000\begin{array}{l}+0.001\\-0.001\end{array}$$

is an example of bilateral tolerance, because the total tolerance of 0.002 is given in two directions—plus and minus.

When unilateral tolerances are used, one of the three following methods should be used to express them:

1) Specify, limiting dimensions only as

Diameter of hole: 2.250, 2.252

Diameter of shaft: 2.249, 2.247

2) One limiting size may be specified with its tolerances as

Diameter of hole: 2.250 + 0.002, −0.000

Diameter of shaft: 2.249 + 0.000, −0.002

3) The nominal size may be specified for both parts, with a notation showing both allowance and tolerance, as

Diameter of hole: $2\frac{1}{4}$ + 0.002, −0.000

Diameter of shaft: $2\frac{1}{4}$ − 0.001, −0.003

Bilateral tolerances should be specified as such, usually with plus and minus tolerances of equal amount. An example of the expression of bilateral tolerances is

$$2 \pm 0.001 \quad \text{or} \quad 2\begin{matrix}+0.001\\-0.001\end{matrix}$$

Application of Tolerances.—According to common practice, tolerances are applied in such a way as to show the permissible amount of dimensional variation in the direction that is less dangerous. When a variation in either direction is equally dangerous, a bilateral tolerance should be given. When a variation in one direction is more dangerous than a variation in another, a unilateral tolerance should be given in the less dangerous direction.

For nonmating surfaces, or atmospheric fits, the tolerances may be bilateral, or unilateral, depending entirely upon the nature of the variations that develop in manufacture. On mating surfaces, with few exceptions, the tolerances should be unilateral.

Where tolerances are required on the distances between holes, usually they should be bilateral, as variation in either direction is normally equally dangerous. The variation in the distance between shafts carrying gears, however, should always be unilateral and plus; otherwise, the gears might run too tight. A slight increase in the backlash between gears is seldom of much importance.

One exception to the use of unilateral tolerances on mating surfaces occurs when tapers are involved; either bilateral or unilateral tolerances may then prove advisable, depending upon conditions. These tolerances should be determined in the same manner as the tolerances on the distances between holes. When a variation either in or out of the position of the mating taper surfaces is equally dangerous, the tolerances should be bilateral. When a variation in one direction is of less danger than a variation in the opposite direction, the tolerance should be unilateral and in the less dangerous direction.

Locating Tolerance Dimensions.—Only one dimension in the same straight line can be controlled within fixed limits. That dimension is the distance between the cutting surface of the tool and the locating or registering surface of the part being machined. Therefore, it is incorrect to locate any point or surface with tolerances from more than one point in the same straight line.

Every part of a mechanism must be located in each plane. Every operating part must be located with proper operating allowances. After such requirements of location are met, all other surfaces should have liberal clearances. Dimensions should be given between those points or surfaces that it is essential to hold in a specific relation to each other. This restriction applies particularly to those surfaces in each plane that control the location of other component parts. Many dimensions are relatively unimportant in this respect. It is good practice to establish a common locating point in each plane and give, as far as possible, such dimensions from these common locating points. The locating points on the drawing, the locatingor registering points used for machining the surfaces and the locating points for measuring should all be identical.

The initial dimensions placed on component drawings should be the exact dimensions that would be used if it were possible to work without tolerances. Tolerances should be given in that direction in which variations will cause the least harm or danger. When a variation in either direction is equally dangerous, the tolerances should be of equal amount in both directions, or bilateral. The initial clearance, or allowance, between operating parts should be as small as the operation of the mechanism will permit. The maximum clearance should be as great as the proper functioning of the mechanism will permit.

Direction of Tolerances on Gages.—The extreme sizes for all plain limit gages shall not exceed the extreme limits of the part to be gaged. All variations in the gages, whatever their cause or purpose, shall bring these gages within these extreme limits.

The data for gage tolerances on page 656 cover gages to inspect workpieces held to tolerances in the American National Standard ANSI B4.4M-1981.

Allowance for Forced Fits.—The allowance per inch of diameter usually ranges from 0.001 inch to 0.0025 inch, 0.0015 being a fair average. Ordinarily the allowance per inch decreases as the diameter increases; thus the total allowance for a diameter of 2 inches might be 0.004 inch, whereas for a diameter of 8 inches the total allowance might not be over 0.009 or 0.010 inch. The parts to be assembled by forced fits are usually made cylindrical, although sometimes they are slightly tapered. The advantages of the taper form are that the possibility of abrasion of the fitted surfaces is reduced; that less pressure is required in assembling; and that the parts are more readily separated when renewal is required. On the other hand, the taper fit is less reliable, because if it loosens, the entire fit is free with but little axial movement. Some lubricant, such as white lead and lard oil mixed to the consistency of paint, should be applied to the pin and bore before assembling, to reduce the tendency toward abrasion.

Pressure for Forced Fits.—The pressure required for assembling cylindrical parts depends not only upon the allowance for the fit, but also upon the area of the fitted surfaces, the pressure increasing in proportion to the distance that the inner member is forced in. The approximate ultimate pressure in tons can be determined by the use of the following formula in conjunction with the accompanying table of "Pressure Factors." Assuming that A = area of surface in contact in "fit"; a = total allowance in inches; P = ultimate pressure required, in tons; F = pressure factor based upon assumption that the diameter of the hub is twice the diameter of the bore, that the shaft is of machine steel, and that the hub is of cast iron:

$$P = \frac{A \times a \times F}{2}$$

Pressure Factors

Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor
1	500	$3\frac{1}{2}$	132	6	75	9	48.7	14	30.5
$1\frac{1}{4}$	395	$3\frac{3}{4}$	123	$6\frac{1}{4}$	72	$9\frac{1}{2}$	46.0	$14\frac{1}{2}$	29.4
$1\frac{1}{2}$	325	4	115	$6\frac{1}{2}$	69	10	43.5	15	28.3
$1\frac{3}{4}$	276	$4\frac{1}{4}$	108	$6\frac{3}{4}$	66	$10\frac{1}{2}$	41.3	$15\frac{1}{2}$	27.4
2	240	$4\frac{1}{2}$	101	7	64	11	39.3	16	26.5
$2\frac{1}{4}$	212	$4\frac{3}{4}$	96	$7\frac{1}{4}$	61	$11\frac{1}{2}$	37.5	$16\frac{1}{2}$	25.6
$2\frac{1}{2}$	189	5	91	$7\frac{1}{2}$	59	12	35.9	17	24.8
$2\frac{3}{4}$	171	$5\frac{1}{4}$	86	$7\frac{3}{4}$	57	$12\frac{1}{2}$	34.4	$17\frac{1}{2}$	24.1
3	156	$5\frac{1}{2}$	82	8	55	13	33.0	18	23.4
$3\frac{1}{4}$	143	$5\frac{3}{4}$	78	$8\frac{1}{2}$	52	$13\frac{1}{2}$	31.7	…	…

Allowance for Given Pressure.—By transposing the preceding formula, the approximate allowance for a required ultimate tonnage can be determined. Thus, $a = \frac{2P}{AF}$. The average ultimate pressure in tons commonly used ranges from 7 to 10 times the diameter in inches.

Expansion Fits.—In assembling certain classes of work requiring a very tight fit, the inner member is contracted by sub-zero cooling to permit insertion into the outer member and a tight fit is obtained as the temperature rises and the inner part expands. To obtain the sub-zero temperature, solid carbon dioxide or "dry ice" has been used but its temperature of about 109 degrees F. below zero will not contract some parts sufficiently to permit insertion in holes or recesses. Greater contraction may be obtained by using high purity liquid nitrogen which has a temperature of about 320 degrees F. below zero. During a temperature reduction from 75 degrees F. to −321 degrees F., the shrinkage per inch of diameter varies from about 0.002 to 0.003 inch for steel; 0.0042 inch for aluminum alloys; 0.0046 inch for magnesium alloys; 0.0033 inch for copper alloys; 0.0023 inch for monel metal; and 0.0017 inch for cast iron (not alloyed). The cooling equipment may vary from an insulated bucket to a special automatic unit, depending upon the kind and quantity of work. One type of unit is so arranged that parts are precooled by vapors from the liquid nitrogen before immersion. With another type, cooling is entirely by the vapor method.

Shrinkage Fits.—General practice seems to favor a smaller allowance for shrinkage fits than for forced fits, although in many shops the allowances are practically the same for each, and for some classes of work, shrinkage allowances exceed those for forced fits. The shrinkage allowance also varies to a great extent with the form and construction of the part that has to be shrunk into place. The thickness or amount of metal around the hole is the most important factor. The way in which the metal is distributed also has an influence on the results. Shrinkage allowances for locomotive driving wheel tires adopted by the American Railway Master Mechanics Association are as follows:

Center diameter, inches	38	44	50	56	62	66
Allowances, inches	0.040	0.047	0.053	0.060	0.066	0.070

Whether parts are to be assembled by forced or shrinkage fits depends upon conditions. For example, to press a tire over its wheel center, without heating, would ordinarily be a rather awkward and difficult job. On the other hand, pins, etc., are easily and quickly forced into place with a hydraulic press and there is the additional advantage of knowing the exact pressure required in assembling, whereas there is more or less uncertainty connected with a shrinkage fit, unless the stresses are calculated. Tests to determine the difference in the quality of shrinkage and forced fits showed that the resistance of a shrinkage fit to slippage for an axial pull was 3.66 times greater than that of a forced fit, and in rotation or torsion, 3.2 times greater. In each comparative test, the dimensions and allowances were the same.

Allowances for Shrinkage Fits.—The most important point to consider when calculating shrinkage fits is the stress in the hub at the bore, which depends chiefly upon the shrinkage allowance. If the allowance is excessive, the elastic limit of the material will be exceeded and permanent set will occur, or, in extreme conditions, the ultimate strength of the metal will be exceeded and the hub will burst. The intensity of the grip of the fit and the resistance to slippage depends mainly upon the thickness of the hub; the greater the thickness, the stronger the grip, and *vice versa.* Assuming the modulus of elasticity for steel to be 30,000,000, and for cast iron, 15,000,000, the shrinkage allowance per inch of nominal diameter can be determined by the following formula, in which A = allowance per inch of diameter; T = true tangential tensile stress at inner surface of outer member; C = factor taken from one of the accompanying tables, *Factors for Calculating Shrinkage Fit Allowances.*

For a cast-iron hub and steel shaft:

$$A = \frac{T(2+C)}{30{,}000{,}000} \qquad (1)$$

When both hub and shaft are of steel:

$$A = \frac{T(1+C)}{30{,}000{,}000} \qquad (2)$$

If the shaft is solid, the factor C is taken from Table 1; if it is hollow and the hub is of steel, factor C is taken from Table 2; if it is hollow and the hub is of cast iron, the factor is taken from Table 3.

Table 1. Factors for Calculating Shrinkage Fit Allowances

Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub	Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub
1.5	0.227	0.234	2.8	0.410	0.432
1.6	0.255	0.263	3.0	0.421	0.444
1.8	0.299	0.311	3.2	0.430	0.455
2.0	0.333	0.348	3.4	0.438	0.463
2.2	0.359	0.377	3.6	0.444	0.471
2.4	0.380	0.399	3.8	0.450	0.477
2.6	0.397	0.417	4.0	0.455	0.482

Values of factor C for solid steel shafts of nominal diameter D_1, and hubs of steel or cast iron of nominal external and internal diameters D_2 and D_1, respectively.

Example 1: A steel crank web 15 inches outside diameter is to be shrunk on a 10-inch solid steel shaft. Required the allowance per inch of shaft diameter to produce a maximum tensile stress in the crank of 25,000 pounds per square inch, assuming the stresses in the crank to be equivalent to those in a ring of the diameter given.

The ratio of the external to the internal diameters equals 15 ÷ 10 = 1.5; T = 25,000 pounds; from Table 1, C = 0.227. Substituting in Formula (2):

$$A = \frac{25{,}000 \times (1 + 0.227)}{30{,}000{,}000} = 0.001 \text{ inch}$$

Example 2: Find the allowance per inch of diameter for a 10-inch shaft having a 5-inch axial through hole, other conditions being the same as in Example 1.

The ratio of external to internal diameters of the hub equals 15 ÷ 10 = 1.5, as before, and the ratio of external to internal diameters of the shaft equals 10 ÷ 5 = 2. From Table 2, we find that factor C = 0.455; T = 25,000 pounds. Substituting these values in Formula (2):

$$A = \frac{25{,}000(1 + 0.455)}{30{,}000{,}000} = 0.0012 \text{ inch}$$

The allowance is increased, as compared with Example 1, because the hollow shaft is more compressible.

Table 2. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.468	2.4	2.0	0.798	3.4	2.0	0.926
	2.5	0.368		2.5	0.628		2.5	0.728
	3.0	0.322		3.0	0.549		3.0	0.637
	3.5	0.296		3.5	0.506		3.5	0.587
1.6	2.0	0.527	2.6	2.0	0.834	3.6	2.0	0.941
	2.5	0.414		2.5	0.656		2.5	0.740
	3.0	0.362		3.0	0.574		3.0	0.647
	3.5	0.333		3.5	0.528		3.5	0.596
1.8	2.0	0.621	2.8	2.0	0.864	3.8	2.0	0.953
	2.5	0.488		2.5	0.679		2.5	0.749
	3.0	0.427		3.0	0.594		3.0	0.656
	3.5	0.393		3.5	0.547		3.5	0.603
2.0	2.0	0.696	3.0	2.0	0.888	4.0	2.0	0.964
	2.5	0.547		2.5	0.698		2.5	0.758
	3.0	0.479		3.0	0.611		3.0	0.663
	3.5	0.441		3.5	0.562		3.5	0.610
2.2	2.0	0.753	3.2	2.0	0.909	...	...	...
	2.5	0.592		2.5	0.715		...	...
	3.0	0.518		3.0	0.625		...	...
	3.5	0.477		3.5	0.576		...	...

Values of factor C for hollow steel shafts and cast-iron hubs.
Notation as in Table 1.

Table 3. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.455	2.4	2.0	0.760	3.4	2.0	0.876
	2.5	0.357		2.5	0.597		2.5	0.689
	3.0	0.313		3.0	0.523		3.0	0.602
	3.5	0.288		3.5	0.481		3.5	0.555
1.6	2.0	0.509	2.6	2.0	0.793	3.6	2.0	0.888
	2.5	0.400		2.5	0.624		2.5	0.698
	3.0	0.350		3.0	0.546		3.0	0.611
	3.5	0.322		3.5	0.502		3.5	0.562
1.8	2.0	0.599	2.8	2.0	0.820	3.8	2.0	0.900
	2.5	0.471		2.5	0.645		2.5	0.707
	3.0	0.412		3.0	0.564		3.0	0.619
	3.5	0.379		3.5	0.519		3.5	0.570
2.0	2.0	0.667	3.0	2.0	0.842	4.0	2.0	0.909
	2.5	0.524		2.5	0.662		2.5	0.715
	3.0	0.459		3.0	0.580		3.0	0.625
	3.5	0.422		3.5	0.533		3.5	0.576
2.2	2.0	0.718	3.2	2.0	0.860	...	...	...
	2.5	0.565		2.5	0.676		...	...
	3.0	0.494		3.0	0.591		...	...
	3.5	0.455		3.5	0.544		...	...

Values of factor C for hollow steel shafts of external and internal diameters D_1 and D_0, respectively, and steel hubs of nominal external diameter D_2.

Example 3: If the crank web in Example 1 is of cast iron and 4000 pounds per square inch is the maximum tensile stress in the hub, what is the allowance per inch of diameter?

$$\frac{D_2}{D_1} = 1.5 \qquad T = 4000$$

In Table 1, we find that $C = 0.234$. Substituting in Formula (1), for cast-iron hubs, $A = 0.0003$ inch, which, owing to the lower tensile strength of cast iron, is endout one-third the shrinkage allowance in Example 1, although the stress is two-thirds of the elastic limit.

Temperatures for Shrinkage Fits.—The temperature to which the outer member in a shrinkage fit should be heated for clearance in assembling the parts depends on the total expansion required and on the coefficient α of linear expansion of the metal (i.e., the increase in length of any section of the metal in any direction for an increase in temperature of 1 degree F). The total expansion in diameter that is required consists of the total allowance for shrinkage and an added amount for clearance. The value of the coefficient α is, for nickel-steel, 0.000007; for steel in general, 0.0000065; for cast iron, 0.0000062. As an example, take an outer member of steel to be expanded 0.005 inch per inch of internal diameter, 0.001 being the shrinkage allowance and the remainder for clearance. Then

$$\alpha \times t^{\circ} = 0.005$$

$$t = \frac{0.005}{0.0000065} = 769 \text{ degrees F}$$

The value t is the number of degrees F that the temperature of the member must be raised above that of the room temperature.

ANSI Standard Limits and Fits (ANSI B4.1-1967 (R1994)).—This American National Standard for Preferred Limits and Fits for Cylindrical Parts presents definitions of terms applying to fits between plain (non threaded) cylindrical parts and makes recommendations on preferred sizes, allowances, tolerances, and fits for use wherever they are applicable. This standard is in accord with the recommendations of American-British-Canadian (ABC) conferences up to a diameter of 20 inches. Experimental work is being carried on with the objective of reaching agreement in the range above 20 inches. The recommendations in the standard are presented for guidance and for use where they might serve to improve and simplify products, practices, and facilities. They should have application for a wide range of products.

As revised in 1967, and reaffirmed in 1979, the definitions in ANSI B4.1 have been expanded and some of the limits in certain classes have been changed.

Factors Affecting Selection of Fits.—Many factors, such as length of engagement, bearing load, speed, lubrication, temperature, humidity, and materials must be taken into consideration in the selection of fits for a particular application, and modifications in the ANSI recommendations may be required to satisfy extreme conditions. Subsequent adjustments may also be found desirable as a result of experience in a particular application to suit critical functional requirements or to permit optimum manufacturing economy.

Definitions.—The following terms are defined in this standard:

Nominal Size: The nominal size is the designation used for the purpose of general identification.

Dimension: A dimension is a geometrical characteristic such as diameter, length, angle, or center distance.

Size: Size is a designation of magnitude. When a value is assigned to a dimension, it is referred to as the size of that dimension. (It is recognized that the words "dimension" and "size" are both used at times to convey the meaning of magnitude.)

Allowance: An allowance is a prescribed difference between the maximum material limits of mating parts. (See definition of *Fit*). It is a minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

Tolerance: A tolerance is the total permissible variation of a size. The tolerance is the difference between the limits of size.

Basic Size: The basic size is that size from which the limits of size are derived by the application of allowances and tolerances.

Design Size: The design size is the basic size with allowance applied, from which the limits of size are derived by the application of tolerances. Where there is no allowance, the design size is the same as the basic size.

Actual Size: An actual size is a measured size.

Limits of Size: The limits of size are the applicable maximum and minimum sizes.

Maximum Material Limit: A maximum material limit is that limit of size that provides the maximum amount of material for the part. Normally it is the maximum limit of size of an external dimension or the minimum limit of size of an internal dimension.*

Minimum Material Limit: A minimum material limit is that limit of size that provides the minimum amount of material for the part. Normally it is the minimum limit of size of an external dimension or the maximum limit of size of an internal dimension.*

Tolerance Limit: A tolerance limit is the variation, positive or negative, by which a size is permitted to depart from the design size.

Unilateral Tolerance: A unilateral tolerance is a tolerance in which variation is permitted in only one direction from the design size.

Bilateral Tolerance: A bilateral tolerance is a tolerance in which variation is permitted in both directions from the design size.

Unilateral Tolerance System: A design plan that uses only unilateral tolerances is known as a Unilateral Tolerance System.

Bilateral Tolerance System: A design plan that uses only bilateral tolerances is known as a Bilateral Tolerance System.

Fit: Fit is the general term used to signify the range of tightness that may result from the application of a specific combination of allowances and tolerances in the design of mating parts.

Actual Fit: The actual fit between two mating parts is the relation existing between them with respect to the amount of clearance or interference that is present when they are assembled. (Fits are of three general types: clearance, transition, and interference.)

Clearance Fit: A clearance fit is one having limits of size so specified that a clearance always results when mating parts are assembled.

Interference Fit: An interference fit is one having limits of size so specified that an interference always results when mating parts are assembled.

Transition Fit: A transition fit is one having limits of size so specified that either a clearance or an interference may result when mating parts are assembled.

Basic Hole System: A basic hole system is a system of fits in which the design size of the hole is the basic size and the allowance, if any, is applied to the shaft.

Basic Shaft System: A basic shaft system is a system of fits in which the design size of the shaft is the basic size and the allowance, if any, is applied to the hole.

* An example of exceptions: an exterior corner radius where the maximum radius is the minimum material limit and the minimum radius is the maximum material limit.

Preferred Basic Sizes.—In specifying fits, the basic size of mating parts may be chosen from the decimal series or the fractional series in the following table.

Table 1. Preferred Basic Sizes

Decimal			Fractional					
0.010	2.00	8.50	1/64	0.015625	2 1/4	2.2500	9 1/2	9.5000
0.012	2.20	9.00	1/32	0.03125	2 1/2	2.5000	10	10.0000
0.016	2.40	9.50	1/16	0.0625	2 3/4	2.7500	10 1/2	10.5000
0.020	2.60	10.00	3/32	0.09375	3	3.0000	11	11.0000
0.025	2.80	10.50	1/8	0.1250	3 1/4	3.2500	11 1/2	11.5000
0.032	3.00	11.00	5/32	0.15625	3 1/2	3.5000	12	12.0000
0.040	3.20	11.50	3/16	0.1875	3 3/4	3.7500	12 1/2	12.5000
0.05	3.40	12.00	1/4	0.2500	4	4.0000	13	13.0000
0.06	3.60	12.50	5/16	0.3125	4 1/4	4.2500	13 1/2	13.5000
0.08	3.80	13.00	3/8	0.3750	4 1/2	4.5000	14	14.0000
0.10	4.00	13.50	7/16	0.4375	4 3/4	4.7500	14 1/2	14.5000
0.12	4.20	14.00	1/2	0.5000	5	5.0000	15	15.0000
0.16	4.40	14.50	9/16	0.5625	5 1/4	5.2500	15 1/2	15.5000
0.20	4.60	15.00	5/8	0.6250	5 1/2	5.5000	16	16.0000
0.24	4.80	15.50	11/16	0.6875	5 3/4	5.7500	16 1/2	16.5000
0.30	5.00	16.00	3/4	0.7500	6	6.0000	17	17.0000
0.40	5.20	16.50	7/8	0.8750	6 1/2	6.5000	17 1/2	17.5000
0.50	5.40	17.00	1	1.0000	7	7.0000	18	18.0000
0.60	5.60	17.50	1 1/4	1.2500	7 1/2	7.5000	18 1/2	18.5000
0.80	5.80	18.00	1 1/2	1.5000	8	8.0000	19	19.0000
1.00	6.00	18.50	1 3/4	1.7500	8 1/2	8.5000	19 1/2	19.5000
1.20	6.50	19.00	2	2.0000	9	9.0000	20	20.0000
1.40	7.00	19.50	…	…	…	…	…	…
1.60	7.50	20.00	…	…	…	…	…	…
1.80	8.00	…	…	…	…	…	…	…

All dimensions are in inches.

Preferred Series of Tolerances and Allowances (In thousandths of an inch)

0.1	1	10	100	0.3	3	30	…
…	1.2	12	125	…	3.5	35	…
0.15	1.4	14	…	0.4	4	40	…
…	1.6	16	160	…	4.5	45	…
…	1.8	18	…	0.5	5	50	…
0.2	2	20	200	0.6	6	60	…
…	2.2	22	…	0.7	7	70	…
0.25	2.5	25	250	0.8	8	80	…
…	2.8	28	…	0.9	9	…	…

Standard Tolerances.—The series of standard tolerances shown in Table 1 are so arranged that for any one grade they represent approximately similar production difficulties throughout the range of sizes. This table provides a suitable range from which appropriate tolerances for holes and shafts can be selected and enables standard gages to be used. The tolerances shown in Table 1 have been used in the succeeding tables for different classes of fits.

Table 1. ANSI Standard Tolerances *ANSI B4.1-1967 (R1987)*

Nominal Size, Inches		Grade									
		4	5	6	7	8	9	10	11	12	13
Over	To	Tolerances in thousandths of an inch[a]									
0	0.12	0.12	0.15	0.25	0.4	0.6	1.0	1.6	2.5	4	6
0.12	0.24	0.15	0.20	0.3	0.5	0.7	1.2	1.8	3.0	5	7
0.24	0.40	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	6	9
0.40	0.71	0.2	0.3	0.4	0.7	1.0	1.6	2.8	4.0	7	10
0.71	1.19	0.25	0.4	0.5	0.8	1.2	2.0	3.5	5.0	8	12
1.19	1.97	0.3	0.4	0.6	1.0	1.6	2.5	4.0	6	10	16
1.97	3.15	0.3	0.5	0.7	1.2	1.8	3.0	4.5	7	12	18
3.15	4.73	0.4	0.6	0.9	1.4	2.2	3.5	5	9	14	22
4.73	7.09	0.5	0.7	1.0	1.6	2.5	4.0	6	10	16	25
7.09	9.85	0.6	0.8	1.2	1.8	2.8	4.5	7	12	18	28
9.85	12.41	0.6	0.9	1.2	2.0	3.0	5.0	8	12	20	30
12.41	15.75	0.7	1.0	1.4	2.2	3.5	6	9	14	22	35
15.75	19.69	0.8	1.0	1.6	2.5	4	6	10	16	25	40
19.69	30.09	0.9	1.2	2.0	3	5	8	12	20	30	50
30.09	41.49	1.0	1.6	2.5	4	6	10	16	25	40	60
41.49	56.19	1.2	2.0	3	5	8	12	20	30	50	80
56.19	76.39	1.6	2.5	4	6	10	16	25	40	60	100
76.39	100.9	2.0	3	5	8	12	20	30	50	80	125
100.9	131.9	2.5	4	6	10	16	25	40	60	100	160
131.9	171.9	3	5	8	12	20	30	50	80	125	200
171.9	200	4	6	10	16	25	40	60	100	160	250

[a] All tolerances above heavy line are in accordance with American-British-Canadian (ABC) agreements.

Table 2. Relation of Machining Processes to Tolerance Grades

MACHINING OPERATION	4	5	6	7	8	9	10	11	12	13
Lapping & Honing	■	■								
Cylindrical Grinding		■	■	■						
Surface Grinding		■	■	■	■					
Diamond Turning		■	■	■						
Diamond Boring		■	■	■						
Broaching		■	■	■	■					
Reaming			■	■	■	■	■			
Turning				■	■	■	■	■	■	■
Boring					■	■	■	■	■	■
Milling							■	■	■	■
Planing & Shaping							■	■	■	■
Drilling							■	■	■	■

This chart may be used as a general guide to determine the machining processes that will under normal conditions, produce work within the tolerance grades indicated.

(See also *Relation of Surface Roughness to Tolerances* starting on page 702.)

ANSI Standard Fits.—Tables 3 through 9 inclusive show a series of standard types and classes of fits on a unilateral hole basis, such that the fit produced by mating parts in any one class will produce approximately similar performance throughout the range of sizes. These tables prescribe the fit for any given size, or type of fit; they also prescribe the standard limits for the mating parts that will produce the fit. The fits listed in these tables contain all those that appear in the approved American-British-Canadian proposal.

Selection of Fits: In selecting limits of size for any application, the type of fit is determined first, based on the use or service required from the equipment being designed; then the limits of size of the mating parts are established, to insure that the desired fit will be produced.

Theoretically, an infinite number of fits could be chosen, but the number of standard fits shown in the accompanying tables should cover most applications.

Designation of Standard Fits: Standard fits are designated by means of the following symbols which, facilitate reference to classes of fit for educational purposes. The symbols are not intended to be shown on manufacturing drawings; instead, sizes should be specified on drawings.

The letter symbols used are as follows:

RC = Running or Sliding Clearance Fit

LC = Locational Clearance Fit

LT = Transition Clearance or Interference Fit

LN = Locational Interference Fit

FN = Force or Shrink Fit

These letter symbols are used in conjunction with numbers representing the class of fit; thus FN 4 represents a Class 4, force fit.

Each of these symbols (two letters and a number) represents a complete fit for which the minimum and maximum clearance or interference and the limits of size for the mating parts are given directly in the tables.

Description of Fits.—The classes of fits are arranged in three general groups: running and sliding fits, locational fits, and force fits.

Running and Sliding Fits (RC): Running and sliding fits, for which limits of clearance are given in Table 2, are intended to provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes. The clearances for the first two classes, used chiefly as slide fits, increase more slowly with the diameter than for the other classes, so that accurate location is maintained even at the expense of free relative motion.

These fits may be described as follows:

RC 1 *Close sliding fits* are intended for the accurate location of parts that must assemble without perceptible play.

RC 2 *Sliding fits* are intended for accurate location, but with greater maximum clearance than class RC 1. Parts made to this fit move and turn easily but are not intended to run freely, and in the larger sizes may seize with small temperature changes.

RC 3 *Precision running fits* are about the closest fits that can be expected to run freely, and are intended for precision work at slow speeds and light journal pressures, but are not suitable where appreciable temperature differences are likely to be encountered.

RC 4 *Close running fits* are intended chiefly for running fits on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

RC 5 and RC 6 *Medium running fits* are intended for higher running speeds, or heavy journal pressures, or both.

RC 7 *Free running fits* are intended for use where accuracy is not essential, or where large temperature variations are likely to be encountered, or under both these conditions.

RC 8 and RC 9 *Loose running fits* are intended for use where wide commercial tolerances may be necessary, together with an allowance, on the external member.

Locational Fits (LC, LT, and LN): Locational fits are fits intended to determine only the location of the mating parts; they may provide rigid or accurate location, as with interference fits, or provide some freedom of location, as with clearance fits. Accordingly, they are divided into three groups: clearance fits (LC), transition fits (LT), and interference fits (LN).

These are described as follows:

LC *Locational clearance fits* are intended for parts which are normally stationary, but that can be freely assembled or disassembled. They range from snug fits for parts requiring accuracy of location, through the medium clearance fits for parts such as spigots, to the looser fastener fits where freedom of assembly is of prime importance.

LT *Locational transition fits* are a compromise between clearance and interference fits, for applications where accuracy of location is important, but either a small amount of clearance or interference is permissible.

LN *Locational interference fits* are used where accuracy of location is of prime importance, and for parts requiring rigidity and alignment with no special requirements for bore pressure. Such fits are not intended for parts designed to transmit frictional loads from one part to another by virtue of the tightness of fit. These conditions are covered by force fits.

Force Fits: (FN): Force or shrink fits constitute a special type of interference fit, normally characterized by maintenance of constant bore pressures throughout the range of sizes. The interference therefore varies almost directly with diameter, and the difference between its minimum and maximum value is small, to maintain the resulting pressures within reasonable limits.

These fits are described as follows:

FN 1 *Light drive fits* are those requiring light assembly pressures, and produce more or less permanent assemblies. They are suitable for thin sections or long fits, or in cast-iron external members.

FN 2 *Medium drive fits* are suitable for ordinary steel parts, or for shrink fits on light sections. They are about the tightest fits that can be used with high-grade cast-iron external members.

FN 3 *Heavy drive fits* are suitable for heavier steel parts or for shrink fits in medium sections.

FN 4 and FN 5 *Force fits* are suitable for parts that can be highly stressed, or for shrink fits where the heavy pressing forces required are impractical.

Graphical Representation of Limits and Fits.—A visual comparison of the hole and shaft tolerances and the clearances or interferences provided by the various types and classes of fits can be obtained from the diagrams on page 633. These diagrams have been drawn to scale for a nominal diameter of 1 inch.

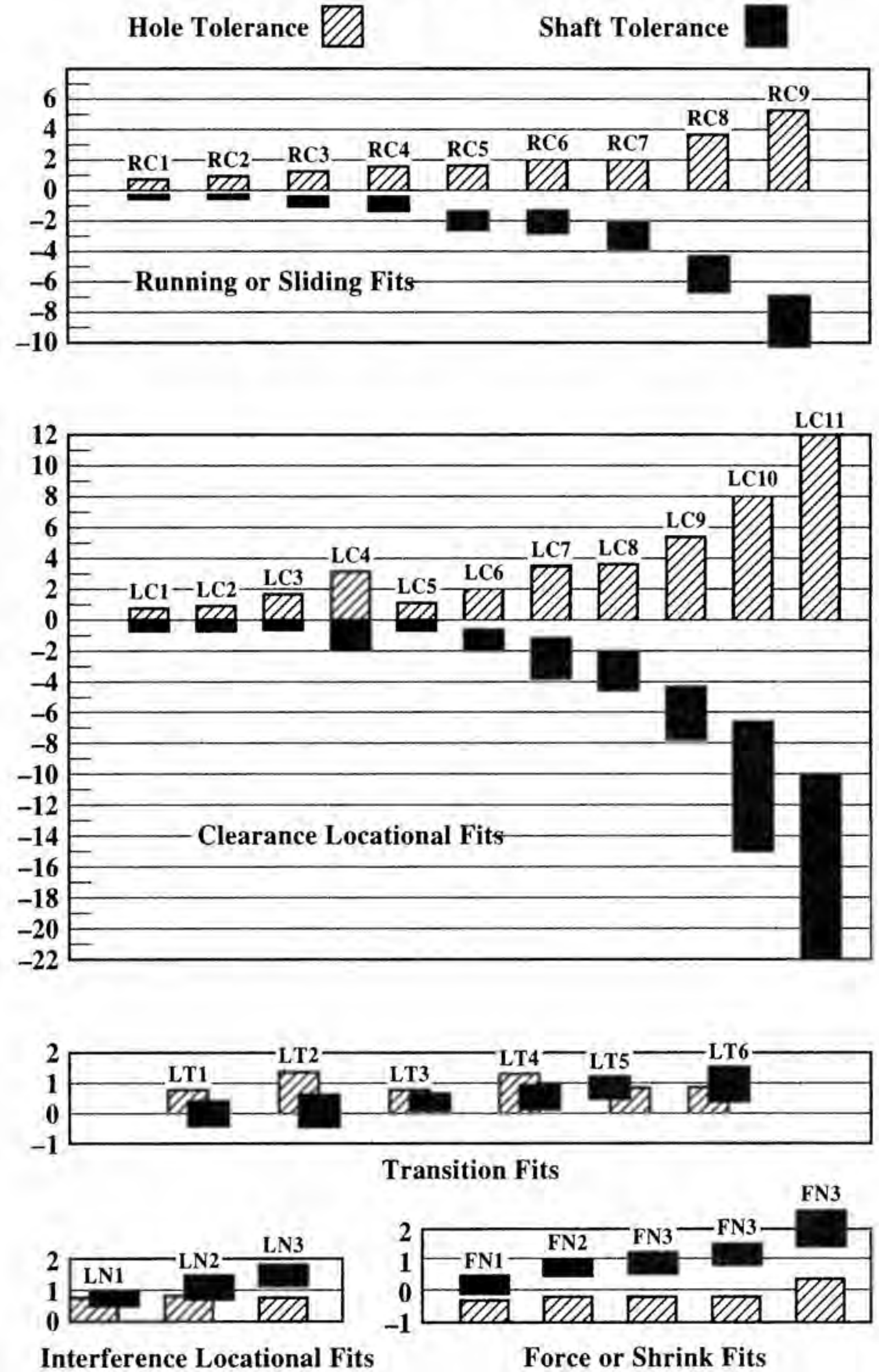
Use of Standard Fit Tables.—*Example 1:* A Class RC 1 fit is to be used in assembling a mating hole and shaft of 2-inch nominal diameter. This class of fit was selected because the application required accurate location of the parts with no perceptible play (see *Description of Fits*, RC 1 close sliding fits). From the data in Table 2, establish the limits of size and clearance of the hole and shaft.

Maximum hole = 2 + 0.0005 = 2.0005; minimum hole = 2 inches

Maximum shaft = 2 − 0.0004 = 1.9996; minimum shaft = 2 − 0.0007 = 1.9993 inches

Minimum clearance = 0.0004; maximum clearance = 0.0012 inch

Graphical Representation of ANSI Standard Limits and Fits



Diagrams show disposition of hole and shaft tolerances (in thousandths of an inch) with respect to basic size (0) for a diameter of 1 inch.

Table 3. American National Standard Running and Sliding Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class RC 1			Class RC 2			Class RC 3			Class RC 4		
	Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits	
		Hole H5	Shaft g4		Hole H6	Shaft g5		Hole H7	Shaft f6		Hole H8	Shaft f7
Over To	Values shown below are in thousandths of an inch											
0 – 0.12	0.1	+0.2	−0.1	0.1	+0.25	−0.1	0.3	+0.4	−0.3	0.3	+0.6	−0.3
	0.45	0	−0.25	0.55	0	−0.3	0.95	0	−0.55	1.3	0	−0.7
0.12 – 0.24	0.15	+0.2	−0.15	0.15	+0.3	−0.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4
	0.5	0	−0.3	0.65	0	−0.35	1.12	0	−0.7	1.6	0	−0.9
0.24 – 0.40	0.2	+0.25	−0.2	0.2	+0.4	−0.2	0.5	+0.6	−0.5	0.5	+0.9	−0.5
	0.6	0	−0.35	0.85	0	−0.45	1.5	0	−0.9	2.0	0	−1.1
0.40 – 0.71	0.25	+0.3	−0.25	0.25	+0.4	−0.25	0.6	+0.7	−0.6	0.6	+1.0	−0.6
	0.75	0	−0.45	0.95	0	−0.55	1.7	0	−1.0	2.3	0	−1.3
0.71 – 1.19	0.3	+0.4	−0.3	0.3	+0.5	−0.3	0.8	+0.8	−0.8	0.8	+1.2	−0.8
	0.95	0	−0.55	1.2	0	−0.7	2.1	0	−1.3	2.8	0	−1.6
1.19 – 1.97	0.4	+0.4	−0.4	0.4	+0.6	−0.4	1.0	+1.0	−1.0	1.0	+1.6	−1.0
	1.1	0	−0.7	1.4	0	−0.8	2.6	0	−1.6	3.6	0	−2.0
1.97 – 3.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4	1.2	+1.2	−1.2	1.2	+1.8	−1.2
	1.2	0	−0.7	1.6	0	−0.9	3.1	0	−1.9	4.2	0	−2.4
3.15 – 4.73	0.5	+0.6	−0.5	0.5	+0.9	−0.5	1.4	+1.4	−1.4	1.4	+2.2	−1.4
	1.5	0	−0.9	2.0	0	−1.1	3.7	0	−2.3	5.0	0	−2.8
4.73 – 7.09	0.6	+0.7	−0.6	0.6	+1.0	−0.6	1.6	+1.6	−1.6	1.6	+2.5	−1.6
	1.8	0	−1.1	2.3	0	−1.3	4.2	0	−2.6	5.7	0	−3.2
7.09 – 9.85	0.6	+0.8	−0.6	0.6	+1.2	−0.6	2.0	+1.8	−2.0	2.0	+2.8	−2.0
	2.0	0	−1.2	2.6	0	−1.4	5.0	0	−3.2	6.6	0	−3.8
9.85 – 12.41	0.8	+0.9	−0.8	0.8	+1.2	−0.8	2.5	+2.0	−2.5	2.5	+3.0	−2.5
	2.3	0	−1.4	2.9	0	−1.7	5.7	0	−3.7	7.5	0	−4.5
12.41 – 15.75	1.0	+1.0	−1.0	1.0	+1.4	−1.0	3.0	+2.2	−3.0	3.0	+3.5	−3.0
	2.7	0	−1.7	3.4	0	−2.0	6.6	0	−4.4	8.7	0	−5.2
15.75 – 19.69	1.2	+1.0	−1.2	1.2	+1.6	−1.2	4.0	+2.5	−4.0	4.0	+4.0	−4.0
	3.0	0	−2.0	3.8	0	−2.2	8.1	0	−5.6	10.5	0	−6.5

[a] Pairs of values shown represent minimum and maximum amounts of clearance resulting from application of standard tolerance limits.

Table 4. American National Standard Running and Sliding Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class RC 5			Class RC 6			Class RC 7			Class RC 8			Class RC 9		
	Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits		Clearance[a]	Standard Tolerance Limits	
		Hole H8	Shaft e7		Hole H9	Shaft e8		Hole H9	Shaft d8		Hole H10	Shaft c9		Hole H11	Shaft
Over To	Values shown below are in thousandths of an inch														
0 – 0.12	0.6	+0.6	− 0.6	0.6	+1.0	− 0.6	1.0	+1.0	− 1.0	2.5	+1.6	− 2.5	4.0	+2.5	− 4.0
	1.6	0	− 1.0	2.2	0	− 1.2	2.6	0	− 1.6	5.1	0	− 3.5	8.1	0	− 5.6
0.12 – 0.24	0.8	+0.7	− 0.8	0.8	+1.2	− 0.8	1.2	+1.2	− 1.2	2.8	+1.8	− 2.8	4.5	+3.0	− 4.5
	2.0	0	− 1.3	2.7	0	− 1.5	3.1	0	− 1.9	5.8	0	− 4.0	9.0	0	− 6.0
0.24 – 0.40	1.0	+0.9	− 1.0	1.0	+1.4	− 1.0	1.6	+1.4	− 1.6	3.0	+2.2	− 3.0	5.0	+3.5	− 5.0
	2.5	0	− 1.6	3.3	0	− 1.9	3.9	0	− 2.5	6.6	0	− 4.4	10.7	0	− 7.2
0.40 – 0.71	1.2	+1.0	− 1.2	1.2	+1.6	− 1.2	2.0	+1.6	− 2.0	3.5	+2.8	− 3.5	6.0	+4.0	− 6.0
	2.9	0	− 1.9	3.8	0	− 2.2	4.6	0	− 3.0	7.9	0	− 5.1	12.8	0	− 8.8
0.71 – 1.19	1.6	+1.2	− 1.6	1.6	+2.0	− 1.6	2.5	+2.0	− 2.5	4.5	+3.5	− 4.5	7.0	+5.0	− 7.0
	3.6	0	− 2.4	4.8	0	− 2.8	5.7	0	− 3.7	10.0	0	− 6.5	15.5	0	−10.5
1.19 – 1.97	2.0	+1.6	− 2.0	2.0	+2.5	− 2.0	3.0	+2.5	− 3.0	5.0	+4.0	− 5.0	8.0	+6.0	− 8.0
	4.6	0	− 3.0	6.1	0	− 3.6	7.1	0	− 4.6	11.5	0	− 7.5	18.0	0	−12.0
1.97 – 3.15	2.5	+1.8	− 2.5	2.5	+3.0	− 2.5	4.0	+3.0	− 4.0	6.0	+4.5	− 6.0	9.0	+7.0	− 9.0
	5.5	0	− 3.7	7.3	0	− 4.3	8.8	0	− 5.8	13.5	0	− 9.0	20.5	0	−13.5
3.15 – 4.73	3.0	+2.2	− 3.0	3.0	+3.5	− 3.0	5.0	+3.5	− 5.0	7.0	+5.0	− 7.0	10.0	+9.0	−10.0
	6.6	0	− 4.4	8.7	0	− 5.2	10.7	0	− 7.2	15.5	0	−10.5	24.0	0	−15.0
4.73 – 7.09	3.5	+2.5	− 3.5	3.5	+4.0	− 3.5	6.0	+4.0	− 6.0	8.0	+6.0	− 8.0	12.0	+10.0	−12.0
	7.6	0	− 5.1	10.0	0	− 6.0	12.5	0	− 8.5	18.0	0	−12.0	28.0	0	−18.0
7.09 – 9.85	4.0	+2.8	− 4.0	4.0	+4.5	− 4.0	7.0	+4.5	− 7.0	10.0	+7.0	−10.0	15.0	+12.0	−15.0
	8.6	0	− 5.8	11.3	0	− 6.8	14.3	0	− 9.8	21.5	0	−14.5	34.0	0	−22.0
9.85 – 12.41	5.0	+3.0	− 5.0	5.0	+5.0	− 5.0	8.0	+5.0	− 8.0	12.0	+8.0	−12.0	18.0	+12.0	−18.0
	10.0	0	− 7.0	13.0	0	− 8.0	16.0	0	−11.0	25.0	0	−17.0	38.0	0	−26.0
12.41 – 15.75	6.0	+3.5	− 6.0	6.0	+6.0	− 6.0	10.0	+6.0	−10.0	14.0	+9.0	−14.0	22.0	+14.0	−22.0
	11.7	0	− 8.2	15.5	0	− 9.5	19.5	0	−13.5	29.0	0	−20.0	45.0	0	−31.0
15.75 – 19.69	8.0	+4.0	− 8.0	8.0	+6.0	− 8.0	12.0	+6.0	−12.0	16.0	+10.0	−16.0	25.0	+16.0	−25.0
	14.5	0	−10.5	18.0	0	−12.0	22.0	0	−16.0	32.0	0	−22.0	51.0	0	−35.0

Tolerance limits given in body of table are added to or subtracted from basic size (as indicated by + or − sign) to obtain maximum and minimum sizes of mating parts.

All data above heavy lines are in accord with ABC agreements. Symbols H5, g4, etc. are hole and shaft designations in ABC system. Limits for sizes above 19.69 inches are also given in the ANSI Standard.

Table 5. American National Standard Clearance Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LC 1			Class LC 2			Class LC 3			Class LC 4			Class LC 5		
		Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits	
	Clearance[a]	Hole H6	Shaft h5	Clearance[a]	Hole H7	Shaft h6	Clearance[a]	Hole H8	Shaft h7	Clearance[a]	Hole H10	Shaft h9	Clearance[a]	Hole H7	Shaft g6
Over To	Values shown below are in thousandths of an inch														
0– 0.12	0 0.45	+0.25 0	0 −0.2	0 0.65	+0.4 0	0 −0.25	0 1	+0.6 0	0 −0.4	0 2.6	+1.6 0	0 −1.0	0.1 0.75	+0.4 0	−0.1 −0.35
0.12– 0.24	0 0.5	+0.3 0	0 −0.2	0 0.8	+0.5 0	0 −0.3	0 1.2	+0.7 0	0 −0.5	0 3.0	+1.8 0	0 −1.2	0.15 0.95	+0.5 0	−0.15 −0.45
0.24– 0.40	0 0.65	+0.4 0	0 −0.25	0 1.0	+0.6 0	0 −0.4	0 1.5	+0.9 0	0 −0.6	0 3.6	+2.2 0	0 −1.4	0.2 1.2	+0.6 0	−0.2 −0.6
0.40– 0.71	0 0.7	+0.4 0	0 −0.3	0 1.1	+0.7 0	0 −0.4	0 1.7	+1.0 0	0 −0.7	0 4.4	+2.8 0	0 −1.6	0.25 1.35	+0.7 0	−0.25 −0.65
0.71– 1.19	0 0.9	+0.5 0	0 −0.4	0 1.3	+0.8 0	0 −0.5	0 2	+1.2 0	0 −0.8	0 5.5	+3.5 0	0 −2.0	0.3 1.6	+0.8 0	−0.3 −0.8
1.19– 1.97	0 1.0	+0.6 0	0 −0.4	0 1.6	+1.0 0	0 −0.6	0 2.6	+1.6 0	0 −1	0 6.5	+4.0 0	0 −2.5	0.4 2.0	+1.0 0	−0.4 −1.0
1.97– 3.15	0 1.2	+0.7 0	0 −0.5	0 1.9	+1.2 0	0 −0.7	0 3	+1.8 0	0 −1.2	0 7.5	+4.5 0	0 −3	0.4 2.3	+1.2 0	−0.4 −1.1
3.15– 4.73	0 1.5	+0.9 0	0 −0.6	0 2.3	+1.4 0	0 −0.9	0 3.6	+2.2 0	0 −1.4	0 8.5	+5.0 0	0 −3.5	0.5 2.8	+1.4 0	−0.5 −1.4
4.73– 7.09	0 1.7	+1.0 0	0 −0.7	0 2.6	+1.6 0	0 −1.0	0 4.1	+2.5 0	0 −1.6	0 10.0	+6.0 0	0 −4	0.6 3.2	+1.6 0	−0.6 −1.6
7.09– 9.85	0 2.0	+1.2 0	0 −0.8	0 3.0	+1.8 0	0 −1.2	0 4.6	+2.8 0	0 −1.8	0 11.5	+7.0 0	0 −4.5	0.6 3.6	+1.8 0	−0.6 −1.8
9.85– 12.41	0 2.1	+1.2 0	0 −0.9	0 3.2	+2.0 0	0 −1.2	0 5	+3.0 0	0 −2.0	0 13.0	+8.0 0	0 −5	0.7 3.9	+2.0 0	−0.7 −1.9
12.41– 15.75	0 2.4	+1.4 0	0 −1.0	0 3.6	+2.2 0	0 −1.4	0 5.7	+3.5 0	0 −2.2	0 15.0	+9.0 0	0 −6	0.7 4.3	+2.2 0	−0.7 −2.1
15.75– 19.69	0 2.6	+1.6 0	0 −1.0	0 4.1	+2.5 0	0 −1.6	0 6.5	+4 0	0 −2.5	0 16.0	+10.0 0	0 −6	0.8 4.9	+2.5 0	−0.8 −2.4

[a] Pairs of values shown represent minimum and maximum amounts of interference resulting from application of standard tolerance limits.

Table 6. American National Standard Clearance Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LC 6			Class LC 7			Class LC 8			Class LC 9			Class LC 10			Class LC 11		
		Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits	
	Clearance[a]	Hole H9	Shaft f8	Clearance[a]	Hole H10	Shaft e9	Clearance[a]	Hole H10	Shaft d9	Clearance[a]	Hole H11	Shaft c10	Clearance[a]	Hole H12	Shaft	Clearance[a]	Hole H13	Shaft
Over To	Values shown below are in thousandths of an inch																	
0 – 0.12	0.3 1.9	+1.0 0	−0.3 −0.9	0.6 3.2	+1.6 0	− 0.6 − 1.6	1.0 2.0	+1.6 0	− 1.0 − 2.0	2.5 6.6	+2.5 0	− 2.5 − 4.1	4 12	+4 0	− 4 − 8	5 17	+6 0	− 5 − 11
0.12 – 0.24	0.4 2.3	+1.2 0	−0.4 −1.1	0.8 3.8	+1.8 0	− 0.8 − 2.0	1.2 4.2	+1.8 0	− 1.2 − 2.4	2.8 7.6	+3.0 0	− 2.8 − 4.6	4.5 14.5	+5 0	− 4.5 − 9.5	6 20	+7 0	− 6 −13
0.24 – 0.40	0.5 2.8	+1.4 0	−0.5 −1.4	1.0 4.6	+2.2 0	− 1.0 − 2.4	1.6 5.2	+2.2 0	− 1.6 − 3.0	3.0 8.7	+3.5 0	− 3.0 − 5.2	5 17	+6 0	− 5 −11	7 25	+9 0	− 7 −16
0.40 – 0.71	0.6 3.2	+1.6 0	−0.6 −1.6	1.2 5.6	+2.8 0	− 1.2 − 2.8	2.0 6.4	+2.8 0	− 2.0 − 3.6	3.5 10.3	+4.0 0	− 3.5 − 6.3	6 20	+7 0	− 6 −13	8 28	+10 0	− 8 −18
0.71 – 1.19	0.8 4.0	+2.0 0	−0.8 −2.0	1.6 7.1	+3.5 0	− 1.6 − 3.6	2.5 8.0	+3.5 0	− 2.5 − 4.5	4.5 13.0	+5.0 0	− 4.5 − 8.0	7 23	+8 0	− 7 −15	10 34	+12 0	−10 −22
1.19 – 1.97	1.0 5.1	+2.5 0	−1.0 −2.6	2.0 8.5	+4.0 0	− 2.0 − 4.5	3.6 9.5	+4.0 0	− 3.0 − 5.5	5.0 15.0	+6 0	− 5.0 − 9.0	8 28	+10 0	− 8 −18	12 44	+16 0	−12 −28
1.97 – 3.15	1.2 6.0	+3.0 0	−1.0 −3.0	2.5 10.0	+4.5 0	− 2.5 − 5.5	4.0 11.5	+4.5 0	− 4.0 − 7.0	6.0 17.5	+7 0	− 6.0 −10.5	10 34	+12 0	−10 −22	14 50	+18 0	−14 −32
3.15 – 4.73	1.4 7.1	+3.5 0	−1.4 −3.6	3.0 11.5	+5.0 0	− 3.0 − 6.5	5.0 13.5	+5.0 0	− 5.0 − 8.5	7 21	+9 0	− 7 −12	11 39	+14 0	−11 −25	16 60	+22 0	−16 −38
4.73 – 7.09	1.6 8.1	+4.0 0	−1.6 −4.1	3.5 13.5	+6.0 0	− 3.5 − 7.5	6 16	+6 0	− 6 −10	8 24	+10 0	− 8 −14	12 44	+16 0	−12 −28	18 68	+25 0	−18 −43
7.09 – 9.85	2.0 9.3	+4.5 0	−2.0 −4.8	4.0 15.5	+7.0 0	− 4.0 − 8.5	7 18.5	+7 0	− 7 −11.5	10 29	+12 0	−10 −17	16 52	+18 0	−16 −34	22 78	+28 0	−22 −50
9.85 – 12.41	2.2 10.2	+5.0 0	−2.2 −5.2	4.5 17.5	+8.0 0	− 4.5 − 9.5	7 20	+8 0	− 7 −12	12 32	+12 0	−12 −20	20 60	+20 0	−20 −40	28 88	+30 0	−28 −58
12.41 – 15.75	2.5 12.0	+6.0 0	−2.5 −6.0	5.0 20.0	+9.0 0	− 5 −11	8 23	+9 0	− 8 −14	14 37	+14 0	−14 −23	22 66	+22 0	−22 −44	30 100	+35 0	−30 −65
15.75 – 19.69	2.8 12.8	+6.0 0	−2.8 −6.8	5.0 21.0	+10.0 0	− 5 −11	9 25	+10 0	− 9 −15	16 42	+16 0	−16 −26	25 75	+25 0	−25 −50	35 115	+40 0	−35 −75

Tolerance limits given in body of table are added or subtracted to basic size (as indicated by + or − sign) to obtain maximum and minimum sizes of mating parts.

All data above heavy lines are in accordance with American-British-Canadian (ABC) agreements. Symbols H6, H7, s6, etc. are hole and shaft designations in ABC system. Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI Standard.

Table 7. ANSI Standard Transition Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LT 1			Class LT 2			Class LT 3			Class LT 4			Class LT 5			Class LT 6		
		Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits	
Over To	Fit[a]	Hole H7	Shaft js6	Fit[a]	Hole H8	Shaft js7	Fit[a]	Hole H7	Shaft k6	Fit[a]	Hole H8	Shaft k7	Fit[a]	Hole H7	Shaft n6	Fit[a]	Hole H7	Shaft n7
	Values shown below are in thousandths of an inch																	
0 – 0.12	−0.12 +0.52	+0.4 0	+0.12 −0.12	−0.2 +0.8	+0.6 0	+0.2 −0.2							−0.5 +0.15	+0.4 0	+0.5 +0.25	−0.65 +0.15	+0.4 0	+0.65 +0.25
0.12 – 0.24	−0.15 +0.65	+0.5 0	+0.15 −0.15	−0.25 +0.95	+0.7 0	+0.25 −0.25							−0.6 +0.2	+0.5 0	+0.6 +0.3	−0.8 +0.2	+0.5 0	+0.8 +0.3
0.24 – 0.40	−0.2 +0.8	+0.6 0	+0.2 −0.2	−0.3 +1.2	+0.9 0	+0.3 −0.3	−0.5 +0.5	+0.6 0	+0.5 +0.1	−0.7 +0.8	+0.9 0	+0.7 +0.1	−0.8 +0.2	+0.6 0	+0.8 +0.4	−1.0 +0.2	+0.6 0	+1.0 +0.4
0.40 – 0.71	−0.2 +0.9	+0.7 0	+0.2 −0.2	−0.35 +1.35	+1.0 0	+0.35 −0.35	−0.5 +0.6	+0.7 0	+0.5 +0.1	−0.8 +0.9	+1.0 0	+0.8 +0.1	−0.9 +0.2	+0.7 0	+0.9 +0.5	−1.2 +0.2	+0.7 0	+1.2 +0.5
0.71 – 1.19	−0.25 +1.05	+0.8 0	+0.25 −0.25	−0.4 +1.6	+1.2 0	+0.4 −0.4	−0.6 +0.7	+0.8 0	+0.6 +0.1	−0.9 +1.1	+1.2 0	+0.9 +0.1	−1.1 +0.2	+0.8 0	+1.1 +0.6	−1.4 +0.2	+0.8 0	+1.4 +0.6
1.19 – 1.97	−0.3 +1.3	+1.0 0	+0.3 −0.3	−0.5 +2.1	+1.6 0	+0.5 −0.5	−0.7 +0.9	+1.0 0	+0.7 +0.1	−1.1 +1.5	+1.6 0	+1.1 +0.1	−1.3 +0.3	+1.0 0	+1.3 +0.7	−1.7 +0.3	+1.0 0	+1.7 +0.7
1.97 – 3.15	−0.3 +1.5	+1.2 0	+0.3 −0.3	−0.6 +2.4	+1.8 0	+0.6 −0.6	−0.8 +1.1	+1.2 0	+0.8 +0.1	−1.3 +1.7	+1.8 0	+1.3 +0.1	−1.5 +0.4	+1.2 0	+1.5 +0.8	−2.0 +0.4	+1.2 0	+2.0 +0.8
3.15 – 4.73	−0.4 +1.8	+1.4 0	+0.4 −0.4	−0.7 +2.9	+2.2 0	+0.7 −0.7	−1.0 +1.3	+1.4 0	+1.0 +0.1	−1.5 +2.1	+2.2 0	+1.5 +0.1	−1.9 +0.4	+1.4 0	+1.9 +1.0	−2.4 +0.4	+1.4 0	+2.4 +1.0
4.73 – 7.09	−0.5 +2.1	+1.6 0	+0.5 −0.5	−0.8 +3.3	+2.5 0	+0.8 −0.8	−1.1 +1.5	+1.6 0	+1.1 +0.1	−1.7 +2.4	+2.5 0	+1.7 +0.1	−2.2 +0.4	+1.6 0	+2.2 +1.2	−2.8 +0.4	+1.6 0	+2.8 +1.2
7.09 – 9.85	−0.6 +2.4	+1.8 0	+0.6 −0.6	−0.9 +3.7	+2.8 0	+0.9 −0.9	−1.4 +1.6	+1.8 0	+1.4 +0.2	−2.0 +2.6	+2.8 0	+2.0 +0.2	−2.6 +0.4	+1.8 0	+2.6 +1.4	−3.2 +0.4	+1.8 0	+3.2 +1.4
9.85 – 12.41	−0.6 +2.6	+2.0 0	+0.6 −6.6	−1.0 +4.0	+3.0 0	+1.0 −1.0	−1.4 +1.8	+2.0 0	+1.4 +0.2	−2.2 +2.8	+3.0 0	+2.2 +0.2	−2.6 +0.6	+2.0 0	+2.6 +1.4	−3.4 +0.6	+2.0 0	+3.4 +1.4
12.41 – 15.75	−0.7 +2.9	+2.2 0	+0.7 −0.7	−1.0 +4.5	+3.5 0	+1.0 −1.0	−1.6 +2.0	+2.2 0	+1.6 +0.2	−2.4 +3.3	+3.5 0	+2.4 +0.2	−3.0 +0.6	+2.2 0	+3.0 +1.6	−3.8 +0.6	+2.2 0	+3.8 +1.6
15.75 – 19.69	−0.8 +3.3	+2.5 0	+0.8 −0.8	−1.2 +5.2	+4.0 0	+1.2 −1.2	−1.8 +2.3	+2.5 0	+1.8 +0.2	−2.7 +3.8	+4.0 0	+2.7 +0.2	−3.4 +0.7	+2.5 0	+3.4 +1.8	−4.3 +0.7	+2.5 0	+4.3 +1.8

[a] Pairs of values shown represent maximum amount of interference (−) and maximum amount of clearance (+) resulting from application of standard tolerance limits.

All data above heavy lines are in accord with ABC agreements. Symbols H7, js6, etc., are hole and shaft designations in the ABC system.

Table 8. ANSI Standard Interference Location Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LN 1			Class LN 2			Class LN 3		
		Standard Limits			Standard Limits			Standard Limits	
Over To	Limits of Interference	Hole H6	Shaft n5	Limits of Interference	Hole H7	Shaft p6	Limits of Interference	Hole H7	Shaft r6
	Values shown below are given in thousandths of an inch								
0– 0.12	0 0.45	+0.25 0	+0.45 +0.25	0 0.65	+0.4 0	+0.65 +0.4	0.1 0.75	+0.4 0	+0.75 +0.5
0.12– 0.24	0 0.5	+0.3 0	+0.5 +0.3	0 0.8	+0.5 0	+0.8 +0.5	0.1 0.9	+0.5 0	+0.9 +0.6
0.24– 0.40	0 0.65	+0.4 0	+0.65 +0.4	0 1.0	+0.6 0	+1.0 +0.6	0.2 1.2	+0.6 0	+1.2 +0.8
0.40– 0.71	0 0.8	+0.4 0	+0.8 +0.4	0 1.1	+0.7 0	+1.1 +0.7	0.3 1.4	+0.7 0	+1.4 +1.0
0.71– 1.19	0 1.0	+0.5 0	+1.0 +0.5	0 1.3	+0.8 0	+1.3 +0.8	0.4 1.7	+0.8 0	+1.7 +1.2
1.19– 1.97	0 1.1	+0.6 0	+1.1 +0.6	0 1.6	+1.0 0	+1.6 +1.0	0.4 2.0	+1.0 0	+2.0 +1.4
1.97– 3.15	0.1 1.3	+0.7 0	+1.3 +0.8	0.2 2.1	+1.2 0	+2.1 +1.4	0.4 2.3	+1.2 0	+2.3 +1.6
3.15– 4.73	0.1 1.6	+0.9 0	+1.6 +1.0	0.2 2.5	+1.4 0	+2.5 +1.6	0.6 2.9	+1.4 0	+2.9 +2.0
4.73– 7.09	0.2 1.9	+1.0 0	+1.9 +1.2	0.2 2.8	+1.6 0	+2.8 +1.8	0.9 3.5	+1.6 0	+3.5 +2.5
7.09– 9.85	0.2 2.2	+1.2 0	+2.2 +1.4	0.2 3.2	+1.8 0	+3.2 +2.0	1.2 4.2	+1.8 0	+4.2 +3.0
9.85– 12.41	0.2 2.3	+1.2 0	+2.3 +1.4	0.2 3.4	+2.0 0	+3.4 +2.2	1.5 4.7	+2.0 0	+4.7 +3.5
12.41– 15.75	0.2 2.6	+1.4 0	+2.6 +1.6	0.3 3.9	+2.2 0	+3.9 +2.5	2.3 5.9	+2.2 0	+5.9 +4.5
15.75– 19.69	0.2 2.8	+1.6 0	+2.8 +1.8	0.3 4.4	+2.5 0	+4.4 +2.8	2.5 6.6	+2.5 0	+6.6 +5.0

All data in this table are in accordance with American-British-Canadian (ABC) agreements.

Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI Standard.

Symbols H7, p6, etc., are hole and shaft designations in the ABC system.

Tolerance limits given in body of table are added or subtracted to basic size (as indicated by + or − sign) to obtain maximum and minimum sizes of mating parts.

Table 9. ANSI Standard Force and Shrink Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches Over – To	Class FN 1 Interference[a]	Class FN 1 Standard Tolerance Limits Hole H6	Class FN 1 Standard Tolerance Limits Shaft	Class FN 2 Interference[a]	Class FN 2 Standard Tolerance Limits Hole H7	Class FN 2 Standard Tolerance Limits Shaft s6	Class FN 3 Interference[a]	Class FN 3 Standard Tolerance Limits Hole H7	Class FN 3 Standard Tolerance Limits Shaft t6	Class FN 4 Interference[a]	Class FN 4 Standard Tolerance Limits Hole H7	Class FN 4 Standard Tolerance Limits Shaft u6	Class FN 5 Interference[a]	Class FN 5 Standard Tolerance Limits Hole H8	Class FN 5 Standard Tolerance Limits Shaft x7
	Values shown below are in thousandths of an inch														
0– 0.12	0.05	+0.25	+0.5	0.2	+0.4	+0.85				0.3	+0.4	+0.95	0.3	+0.6	+1.3
	0.5	0	+0.3	0.85	0	+0.6				0.95	0	+0.7	1.3	0	+0.9
0.12– 0.24	0.1	+0.3	+0.6	0.2	+0.5	+1.0				0.4	+0.5	+1.2	0.5	+0.7	+1.7
	0.6	0	+0.4	1.0	0	+0.7				1.2	0	+0.9	1.7	0	+1.2
0.24– 0.40	0.1	+0.4	+0.75	0.4	+0.6	+1.4				0.6	+0.6	+1.6	0.5	+0.9	+2.0
	0.75	0	+0.5	1.4	0	+1.0				1.6	0	+1.2	2.0	0	+1.4
0.40– 0.56	0.1	+0.4	+0.8	0.5	+0.7	+1.6				0.7	+0.7	+1.8	0.6	+1.0	+2.3
	0.8	0	+0.5	1.6	0	+1.2				1.8	0	+1.4	2.3	0	+1.6
0.56– 0.71	0.2	+0.4	+0.9	0.5	+0.7	+1.6				0.7	+0.7	+1.8	0.8	+1.0	+2.5
	0.9	0	+0.6	1.6	0	+1.2				1.8	0	+1.4	2.5	0	+1.8
0.71– 0.95	0.2	+0.5	+1.1	0.6	+0.8	+1.9				0.8	+0.8	+2.1	1.0	+1.2	+3.0
	1.1	0	+0.7	1.9	0	+1.4				2.1	0	+1.6	3.0	0	+2.2
0.95– 1.19	0.3	+0.5	+1.2	0.6	+0.8	+1.9	0.8	+0.8	+2.1	+1.0	+0.8	+2.3	1.3	+1.2	+3.3
	1.2	0	+0.8	1.9	0	+1.4	2.1	0	+1.6	2.3	0	+1.8	3.3	0	+2.5
1.19– 1.58	0.3	+0.6	+1.3	0.8	+1.0	+2.4	1.0	+1.0	+2.6	1.5	+1.0	+3.1	1.4	+1.6	+4.0
	1.3	0	+0.9	2.4	0	+1.8	2.6	0	+2.0	3.1	0	+2.5	4.0	0	+3.0
1.58– 1.97	0.4	+0.6	+1.4	0.8	+1.0	+2.4	1.2	+1.0	+2.8	1.8	+1.0	+3.4	2.4	+1.6	+5.0
	1.4	0	+1.0	2.4	0	+1.8	2.8	0	+2.2	3.4	0	+2.8	5.0	0	+4.0
1.97– 2.56	0.6	+0.7	+1.8	0.8	+1.2	+2.7	1.3	+1.2	+3.2	2.3	+1.2	+4.2	3.2	+1.8	+6.2
	1.8	0	+1.3	2.7	0	+2.0	3.2	0	+2.5	4.2	0	+3.5	6.2	0	+5.0
2.56– 3.15	0.7	+0.7	+1.9	1.0	+1.2	+2.9	1.8	+1.2	+3.7	2.8	+1.2	+4.7	4.2	+1.8	+7.2
	1.9	0	+1.4	2.9	0	+2.2	3.7	0	+3.0	4.7	0	+4.0	7.2	0	+6.0
3.15– 3.94	0.9	+0.9	+2.4	1.4	+1.4	+3.7	2.1	+1.4	+4.4	3.6	+1.4	+5.9	4.8	+2.2	+8.4
	2.4	0	+1.8	3.7	0	+2.8	4.4	0	+3.5	5.9	0	+5.0	8.4	0	+7.0
3.94– 4.73	1.1	+0.9	+2.6	1.6	+1.4	+3.9	2.6	+1.4	+4.9	4.6	+1.4	+6.9	5.8	+2.2	+9.4
	2.6	0	+2.0	3.9	0	+3.0	4.9	0	+4.0	6.9	0	+6.0	9.4	0	+8.0

Table 9. *(Continued)* **ANSI Standard Force and Shrink Fits** *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches Over – To	Class FN 1 Interference[a]	Class FN 1 Standard Tolerance Limits Hole H6	Class FN 1 Standard Tolerance Limits Shaft	Class FN 2 Interference[a]	Class FN 2 Standard Tolerance Limits Hole H7	Class FN 2 Standard Tolerance Limits Shaft s6	Class FN 3 Interference[a]	Class FN 3 Standard Tolerance Limits Hole H7	Class FN 3 Standard Tolerance Limits Shaft t6	Class FN 4 Interference[a]	Class FN 4 Standard Tolerance Limits Hole H7	Class FN 4 Standard Tolerance Limits Shaft u6	Class FN 5 Interference[a]	Class FN 5 Standard Tolerance Limits Hole H8	Class FN 5 Standard Tolerance Limits Shaft x7
	Values shown below are in thousandths of an inch														
4.73– 5.52	1.2	+1.0	+2.9	1.9	+1.6	+4.5	3.4	+1.6	+6.0	5.4	+1.6	+8.0	7.5	+2.5	+11.6
	2.9	0	+2.2	4.5	0	+3.5	6.0	0	+5.0	8.0	0	+7.0	11.6	0	+10.0
5.52– 6.30	1.5	+1.0	+3.2	2.4	+1.6	+5.0	3.4	+1.6	+6.0	5.4	+1.6	+8.0	9.5	+2.5	+13.6
	3.2	0	+2.5	5.0	0	+4.0	6.0	0	+5.0	8.0	0	+7.0	13.6	0	+12.0
6.30– 7.09	1.8	+1.0	+3.5	2.9	+1.6	+5.5	4.4	+1.6	+7.0	6.4	+1.6	+9.0	9.5	+2.5	+13.6
	3.5	0	+2.8	5.5	0	+4.5	7.0	0	+6.0	9.0	0	+8.0	13.6	0	+12.0
7.09– 7.88	1.8	+1.2	+3.8	3.2	+1.8	+6.2	5.2	+1.8	+8.2	7.2	+1.8	+10.2	11.2	+2.8	+15.8
	3.8	0	+3.0	6.2	0	+5.0	8.2	0	+7.0	10.2	0	+9.0	15.8	0	+14.0
7.88– 8.86	2.3	+1.2	+4.3	3.2	+1.8	+6.2	5.2	+1.8	+8.2	8.2	+1.8	+11.2	13.2	+2.8	+17.8
	4.3	0	+3.5	6.2	0	+5.0	8.2	0	+7.0	11.2	0	+10.0	17.8	0	+16.0
8.86– 9.85	2.3	+1.2	+4.3	4.2	+1.8	+7.2	6.2	+1.8	+9.2	10.2	+1.8	+13.2	13.2	+2.8	+17.8
	4.3	0	+3.5	7.2	0	+6.0	9.2	0	+8.0	13.2	0	+12.0	17.8	0	+16.0
9.85– 11.03	2.8	+1.2	+4.9	4.0	+2.0	+7.2	7.0	+2.0	+10.2	10.0	+2.0	+13.2	15.0	+3.0	+20.0
	4.9	0	+4.0	7.2	0	+6.0	10.2	0	+9.0	13.2	0	+12.0	20.0	0	+18.0
11.03– 12.41	2.8	+1.2	+4.9	5.0	+2.0	+8.2	7.0	+2.0	+10.2	12.0	+2.0	+15.2	17.0	+3.0	+22.0
	4.9	0	+4.0	8.2	0	+7.0	10.2	0	+9.0	15.2	0	+14.0	22.0	0	+20.0
12.41– 13.98	3.1	+1.4	+5.5	5.8	+2.2	+9.4	7.8	+2.2	+11.4	13.8	+2.2	+17.4	18.5	+3.5	+24.2
	5.5	0	+4.5	9.4	0	+8.0	11.4	0	+10.0	17.4	0	+16.0	24.2	0	+22.0
13.98– 15.75	3.6	+1.4	+6.1	5.8	+2.2	+9.4	9.8	+2.2	+13.4	15.8	+2.2	+19.4	21.5	+3.5	+27.2
	6.1	0	+5.0	9.4	0	+8.0	13.4	0	+12.0	19.4	0	+18.0	27.2	0	+25.0
15.75– 17.72	4.4	+1.6	+7.0	6.5	+2.5	+10.6	+9.5	+2.5	+13.6	17.5	+2.5	+21.6	24.0	+4.0	+30.5
	7.0	0	+6.0	10.6	0	+9.0	13.6	0	+12.0	21.6	0	+20.0	30.5	0	+28.0
17.72– 19.69	4.4	+1.6	+7.0	7.5	+2.5	+11.6	11.5	+2.5	+15.6	19.5	+2.5	+23.6	26.0	+4.0	+32.5
	7.0	0	+6.0	11.6	0	+10.0	15.6	0	+14.0	23.6	0	+22.0	32.5	0	+30.0

[a] Pairs of values shown represent minimum and maximum amounts of interference resulting from application of standard tolerance limits.

All data above heavy lines are in accordance with American-British-Canadian (ABC) agreements. Symbols H6, H7, s6, etc., are hole and shaft designations in the ABC system. Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI standard.