Chapter 8
Drills & Drilling Operations

Upcoming Chapters

Metal Removal
Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining
Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes
Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining
Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Finishing Processes
Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Excutive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: http://www.ltu.edu
Prentice Hall: http://www.prenhall.com

8.1 Introduction

Drilling is the process most commonly associated with producing machined holes. Although many other processes contribute to the production of holes, including boring, reaming, broaching, and internal grinding, drilling accounts for the majority of holes produced in the machine shop. This is because drilling is a simple, quick, and economical method of hole production. The other methods are used principally for more accurate, smoother, larger holes. They are often used after a drill has already made the pilot hole.

Drilling is one of the most complex machining processes. The chief characteristic that distinguishes it from other machining operations is the combined cutting and extrusion of metal at the chisel edge in the center of the drill. The high thrust force caused by the feeding motion first extrudes metal under the chisel edge. Then it tends to shear under the action of a negative rake angle tool. Drilling of a single hole is shown in Figure 8.1 and high production drilling of a plate component is shown in Figure 8.2.

FIGURE 8.1: Drilling accounts for the majority of holes produced in industry today. (Courtesy Valenite Inc.)

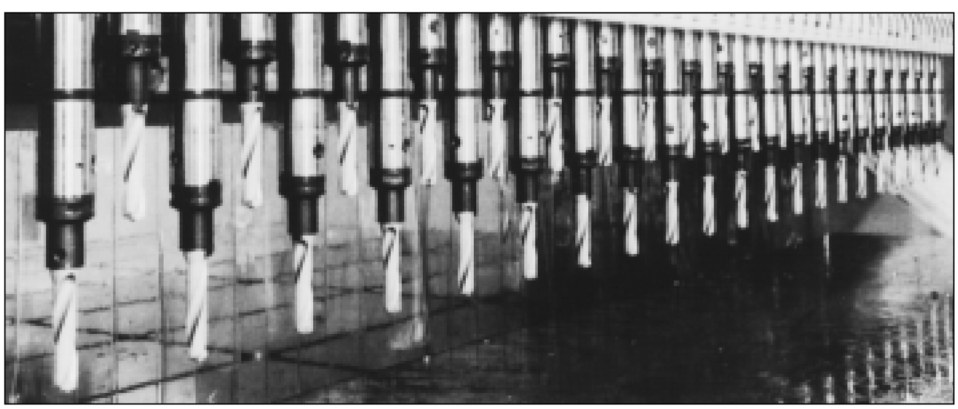


FIGURE 8.2: Holes can be drilled individually as shown in Figure 8.1, or many holes can be drilled at the same time as shown here. (Courtesy Sandvik Coromant Co.)

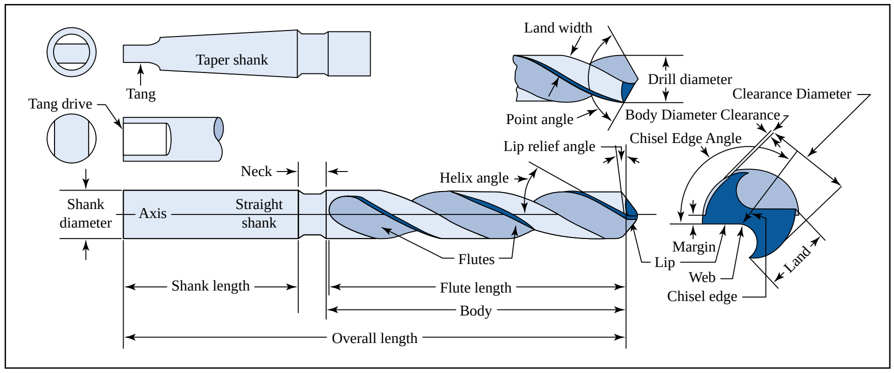


FIGURE 8.4: Nomenclature of a twist drill shown with taper and tang drives.

The cutting action along the lips of the drill is not unlike that in other machining processes. Due to variable rake angle and inclination, however, there are differences in the cutting action at various radii on the cutting edges. This is complicated by the constraint of the whole chip on the chip flow at any single point along the lip. Still, the metal removing action is true cutting, and the problems of variable geometry and constraint are present, but because it is such a small portion of the total drilling operation, it is not a distinguishing characteristic of the process. Many of the drills discussed in this chapter are shown in Figures 8.3.

The machine settings used in drilling reveal some important features of this hole producing operation. Depth of cut, a fundamental dimension in other cutting processes, corresponds most closely to the drill radius. The undeformed chip width is equivalent to the length of the drill lip, which depends on the point angle as well as the drill size. For a given set-up, the undeformed chip width is constant in drilling. The feed dimension specified for drilling is the feed per revolution of the spindle. A more fundamental quantity is the feed per lip. For the common two-flute drill, it is half the feed per revolution. The undeformed chip thickness differs from the feed per lip depending on the point angle.

The spindle speed is constant for any one operation, while the cutting speed varies all along the cutting edge. Cutting speed is normally computed for the outside diameter. At the center of the chisel edge the cutting speed is zero; at any point on the lip it is proportional to the radius of that point. This variation in cutting speed along the cutting edges is an important characteristic of drilling.

Once the drill engages the workpiece, the contact is continuous until the drill breaks through the bottom of the part or is withdrawn from the hole. In this respect, drilling resembles turning and is unlike milling. Continuous cutting means that steady forces and temperatures may be expected shortly after contact between the drill and the workpiece.

FIGURE 8.3: Many of the drills used in industry are shown here and described in this chapter. (Courtesy Cleveland Twist Drill Greenfield Industries)

8.2 Drill Nomenclature

The most important type of drill is the twist drill. The important nomenclature listed below and illustrated in Figure 8.4 applies specifically to these tools.

Drill: A drill is an end-cutting tool for producing holes. It has one or more cutting edges, and flutes to allow fluids to enter and chips to be ejected. The drill is composed of a shank, body, and point.

Shank: The shank is the part of the drill that is held and driven. It may be straight or tapered. Smaller diameter drills normally have straight shanks. Larger drills have shanks ground with a taper and a tang to insure accurate alignment and positive drive.

Tang: The tang is a flattened portion at the end of the shank that fits into a driving slot of the drill holder on the spindle of the machine.

Body: The body of the drill extends from the shank to the point, and contains the flutes. During sharpening, it is the body of the drill that is partially ground away.

Point: The point is the cutting end of the drill.

Flutes: Flutes are grooves that are cut or formed in the body of the drill to allow fluids to reach the point and chips to reach the workpiece surface. Although straight flutes are used in some cases, they are normally helical.

Land: The land is the remainder of the outside of the drill body after the flutes are cut. The land is cut back somewhat from the outside drill diameter in order to provide clearance.

Margin: The margin is a short portion of the land not cut away for clearance. It preserves the full drill diameter.

Web: The web is the central portion of the drill body that connects the lands.

Chisel Edge: The edge ground on the tool point along the web is called the chisel edge. It connects the cutting lips.

Lips: The lips are the primary cutting edges of the drill. They extend from the chisel point to the periphery of the drill.

Axis: The axis of the drill is the centerline of the tool. It runs through the web and is perpendicular to the diameter.

Neck: Some drills are made with a relieved portion between the body and the shank. This is called the drill neck.

In addition to the above terms that define the various parts of the drill, there are a number of terms that apply to the dimensions of the drill, including the important drill angles. Among these terms are the following:

Length: Along with its outside diameter, the axial length of a drill is listed when the drill size is given. In addition, shank length, flute length, and neck length are often used.(see Fig. 8.4)

Body Diameter Clearance: The height of the step from the margin to the land is called the body diameter clearance.

Web Thickness: The web thickness is the smallest dimension across the web. It is measured at the point unless otherwise noted. Web thickness will often increase in going up the body away from the point, and it may have to be ground down during sharpening to reduce the size of the chisel edge. This process is called 'web thinning'. Web thinning is shown in Figure 8.13.

Helix Angle: The angle that the leading edge of the land makes with the drill axis is called the helix angle. Drills with various helix angles are available for different operational requirements.

Point Angle: The included angle between the drill lips is called the point angle. It is varied for different workpiece materials.

Lip Relief Angle: Corresponding to the usual relief angles found on other tools is the lip relief angle. It is measured at the periphery.

Chisel Edge Angle: The chisel edge angle is the angle between the lip and the chisel edge, as seen from the end of the drill.

It is apparent from these partial lists of terms that many different drill geometries are possible.

8.3 Classes of Drills

There are different classes of drills for different types of operations. Workpiece materials may also influence the class of drill used, but it usually determines the point geometry rather than the general type of drill best suited for the job. It has already been noted that the twist drill is the most important class. Within the general class of twist drills there are a number of drill types made for different kinds of operations. Many of the special drills discussed below are shown in Figure 8.5.

High Helix Drills: This drill has a high helix angle, which improves cutting efficiency but weakens the drill body. It is used for cutting softer metals and other low strength materials.

Low Helix Drills: A lower than normal helix angle is sometimes useful to prevent the tool from 'running ahead' or 'grabbing' when drilling brass and similar materials.

Heavy-duty Drills: Drills subject to severe stresses can be made stronger by such methods as increasing the web thickness.

Left Hand Drills: Standard twist drills can be made as left hand tools. These are used in multiple drill heads where the head design is simplified by allowing the spindle to rotate in different directions.

Straight Flute Drills: Straight flute drills are an extreme case of low helix drills. They are used for drilling brass and sheet metal.

Crankshaft Drills: Drills that are especially designed for crankshaft work have been found to be useful for machining deep holes in tough materials. They have a heavy web and helix angle that is somewhat higher than normal. The heavy web prompted the use of a specially notched chisel edge that has proven useful on other jobs as well. The crankshaft drill is an example of a special drill that has found wider application than originally anticipated and has become standard.

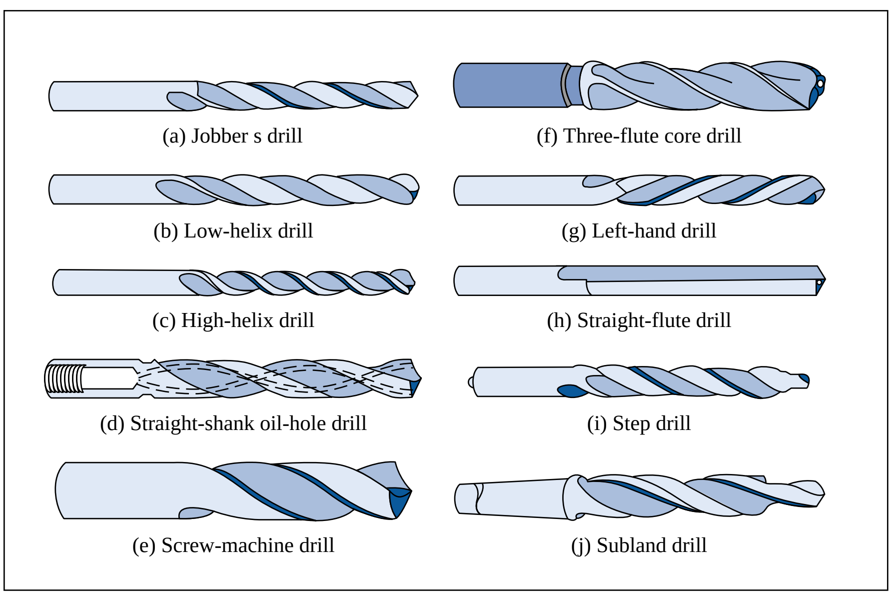


FIGURE 8.5: Special drills are used for some drilling operations.

Extension Drills: The extension drill has a long, tempered shank to allow drilling in surfaces that are normally inaccessible.

Extra-length Drills: For deep holes, the standard long drill may not suffice, and a longer bodied drill is required.

Step Drill: Two or more diameters may be ground on a twist drill to produce a hole with stepped diameters.

Subland Drill: The subland or multi-cut drill does the same job as the step drill. It has separate lands running the full body length for each diameter, whereas the step drill uses one land. A subland drill looks like two drills twisted together.

Solid Carbide Drills: For drilling small holes in light alloys and nonmetallic materials, solid carbide rods may be ground to standard drill geometry. Light cuts without shock must be taken because carbide is quite brittle.

Carbide Tipped Drills: Carbide tips may be used on twist drills to make the edges more wear resistant at higher speeds. Smaller helix angles and thicker webs are often used to improve the rigidity of these drills, which helps to preserve the carbide. Carbide tipped drills are widely used for hard, abrasive non-metallic materials such as masonry.

Oil Hole Drills: Small holes through the lands, or small tubes in slots milled in the lands, can be used to force oil under pressure to the tool point. These drills are especially useful for drilling deep holes in tough materials.

Flat Drills: Flat bars may be ground with a conventional drill point at the end. This gives very large chip spaces, but no helix. Their major application is for drilling railroad track.

Three and Four Fluted Drills: There are drills with three or four flutes which resemble standard twist drills except that they have no chisel edge. They are used for enlarging holes that have been previously drilled or punched. These drills are used because they give better productivity, accuracy, and surface finish than a standard drill would provide on the same job.

Drill and Countersink: A combination drill and countersink is a useful tool for machining 'center holes' on bars to be turned or ground between centers. The end of this tool resembles a standard drill. The countersink starts a short distance back on the body.

A double-ended combination drill and countersink, also called a center drill, is shown in Figure 8.6.

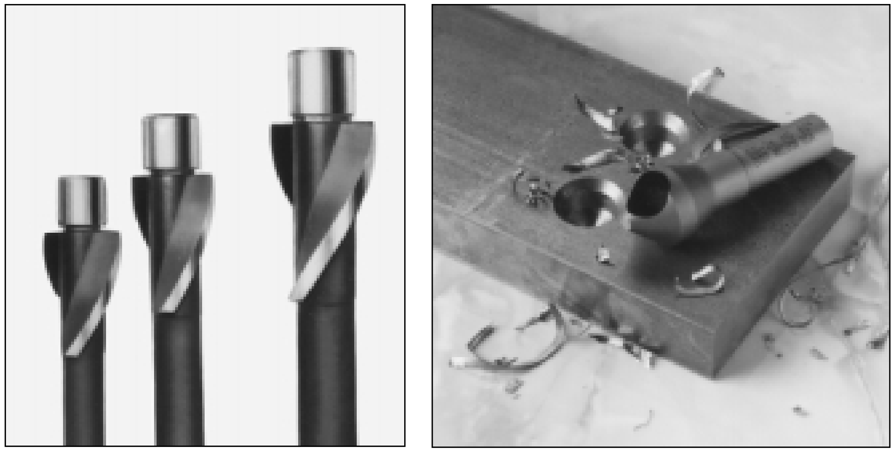


FIGURE 8.8: Counterboring tools (a) and countersinking operation (b) are shown here. (Courtesy The Weldon Tool Co.)

8.4 Related Drilling Operations

Several operations are related to drilling. In the following list, most of the operations follow drilling except for centering and spotfacing which precede drilling. A hole must be made first by drilling and then the hole is modified by one of the other operations. Some of these operations are described here and illustrated in Figure 8.7

Reaming: A reamer is used to enlarge a previously drilled hole, to provide a higher tolerance and to improve the surface finish of the hole.

Tapping: A tap is used to provide internal threads on a previously drilled hole.

Reaming and tapping are more involved and complicated than counterboring, countersinking, centering, and spot facing, and are therefore discussed in Chapter 11.

Counterboring: Counterboring produces a larger step in a hole to allow a bolt head to be seated below the part surface.

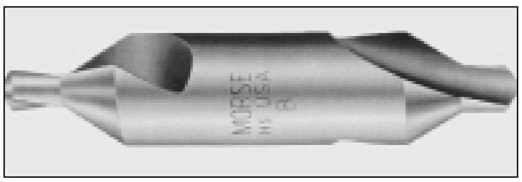


FIGURE 8.6: A double-ended combination drill and countersink, also called a center drill. (Courtesy Morse Cutting Tools)

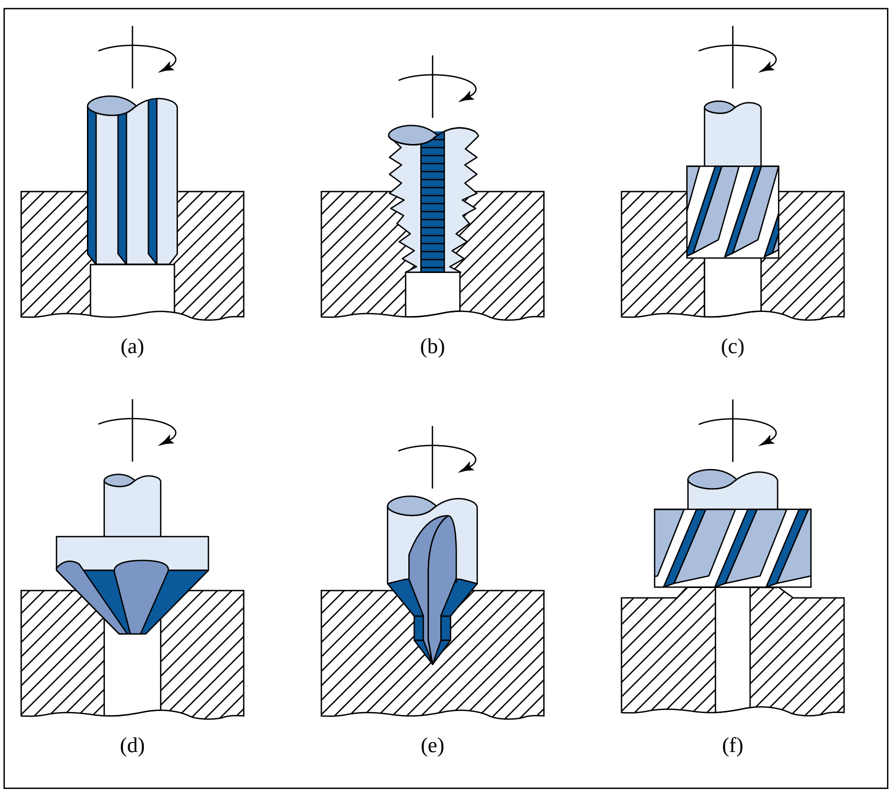


FIGURE 8.7: Related drilling operations: (a) reaming, (b) tapping, (c) counterboring, (d) countersinking, (e) centering, (f) spotfacing.

Countersinking: Countersinking is similar to counterboring except that the step is angular to allow flat-head screws to be seated below the surface.

Counterboring tools are shown in Figure 8.8a, and a counter- sinking tool with two machined holes is shown in Figure 8.8b.

Centering: Center drilling is used for accurately locating a hole to be drilled afterwards.

Spotfacing: Spotfacing is used to provide a flat-machined surface on a part.

8.5 Operating Conditions

The varying conditions, under which drills are used, make it difficult to give set rules for speeds and feeds. Drill manufacturers and a variety of reference texts provide recommendations for proper speeds and feeds for drilling a variety of materials. General drilling speeds and feeds will be discussed here and some examples will be given.

Drilling Speed: Cutting speed may be referred to as the rate that a point on a circumference of a drill will travel in 1 minute. It is expressed in surface feet per minute (SFPM). Cutting speed is one of the most important factors that determine the life of a drill. If the cutting speed is too slow, the drill might chip or break. A cutting speed that is too fast rapidly dulls the cutting lips. Cutting speeds depend on the following seven variables:

- The type of material being drilled. The harder the material, the slower the cutting speed.
- The cutting tool material and diame-

ter. The harder the cutting tool material, the faster it can machine the material. The larger the drill, the slower the drill must revolve.

- The types and use of cutting fluids allow an increase in cutting speed.
- The rigidity of the drill press.
- The rigidity of the drill (the shorter the drill, the better).
- The rigidity of the work setup.
- The quality of the hole to be drilled.

Each variable should be considered prior to drilling a hole. Each variable is important, but the work material and its cutting speed are the most important factors. To calculate the revolutions per minute (RPM) rate of a drill, the diameter of the drill and the cutting speed of the material must be considered.

The formula normally used to calculate cutting speed is as follows:

SFPM = (Drill Circumference) x (RPM)

Where:
SFPM = surface feet per minute, or the distance traveled by a point on the drill periphery in feet each minute.
Drill Circumference = the distance around the drill periphery in feet.
RPM = revolutions per minute

In the case of a drill, the circumference is:
Drill Circumference =
Pi/12 x (d) = .262 x d

Where:
Drill Circumference = the distance around the drill periphery in feet.
Pi = is a constant of 3.1416
d = the drill diameter in inches.

By substituting for the drill circumference, the cutting speed can now be written as:

SFPM = .262 x d x RPM

This formula can be used to determine the cutting speed at the periphery of any rotating drill.

For example: Given a .25 inch drill, what is the cutting speed (SFPM) drilling cast iron at 5000 RPM?
SFPM = .262 x d x RPM
SFPM = .262 x .25 x 5000
Answer = 327.5 or 327 SFPM

RPM can be calculated as follows:

Given a .75 inch drill, what is the RPM drilling low carbon steel at 400 SFPM?

$$RPM = \frac{SFPM}{.262 \times d} = \frac{400}{.262 \times .75} = \frac{400}{.1965}$$

Answer = 2035.62 or 2036 RPM

Drilling Feed: Once the cutting speed has been selected for a particular workpiece material and condition, the appropriate feed rate must be established. Drilling feed rates are selected to maximize productivity while maintaining chip control. Feed in drilling operations is expressed in inches per revolution, or IPR, which is the distance the drill moves in inches for each revolution of the drill. The feed may also be expressed as the distance traveled by the drill in a single minute, or IPM (inches per minute), which is the product of the RPM and IPR of the drill. It can be calculated as follows:

IPM = IPR x RPM

Where:
IPM = inches per minute
IPR = inches per revolution
RPM = revolutions per minute.

For example: To maintain a .015 IPR feed rate on the .75 inch drill discussed above, what would the IPM feed rate be?

IPM = PR x RPM
IPM = .015 x 2036
Answer = 30.54 or 31 IPM

The selection of drilling speed (SFPM) and drilling feed (IPR) for various materials to be machined often starts with recommendations in the form of application tables from manufacturers or by consulting reference books.

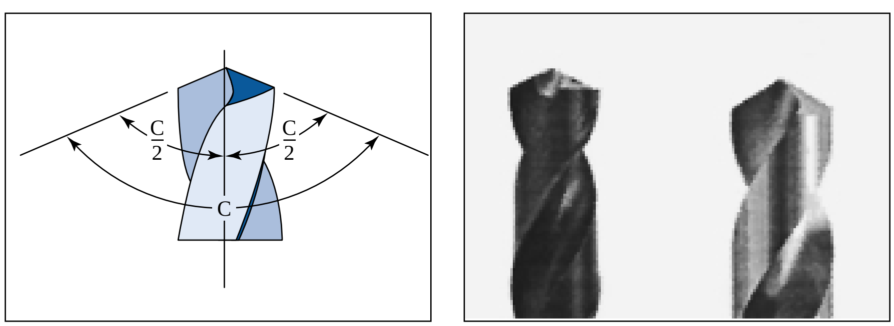
8.5.1 Twist Drill Wear

Drills wear starts as soon as cutting begins and instead of progressing at a constant rate, the wear accelerates continuously. Wear starts at the sharp corners of the cutting edges and, at the same time, works its way along the cutting edges to the chisel edge and up the drill margins. As wear progresses, clearance is reduced. The resulting rubbing causes more heat, which in turn causes faster wear.

Wear lands behind the cutting edges are not the best indicators of wear, since they depend on the lip relief angle. The wear on the drill margins actually determines the degree of wear and is not nearly as obvious as wear lands. When the corners of the drill are rounded off, the drill has been damaged more than is readily apparent. Quite possibly the drill appeared to be working properly even while it was wearing. The margins could be worn in a taper as far back as an inch from the point. To restore the tool to new condition, the worn area must be removed. Because of the accelerating nature of wear, the number of holes per inch of drill can sometimes be doubled by reducing, by 25 percent, the number of holes drilled per grind.

8.5.2 Drill Point Grinding

It has been estimated that about 90 percent of drilling troubles are due to improper grinding of the drill point. Therefore, it is important that care be taken when resharpening drills. A good drill point will have: both lips at the same angle to the axis of the drill; both lips the same length; correct clearance angle; and correct thickness of web.



(a) (b)

FIGURE 8.9: The included lip angle varies between 90 and 135 degrees (a): two drill points are shown in (b). (Courtesy Cleveland Twist Drill Greenfield Industries)

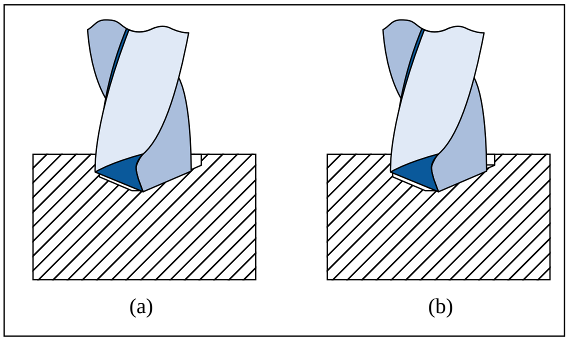


FIGURE 8.10: Drill with equal lip angle but unequal lip length (a), and drill with unequal lip angle and unequal lip length (b).

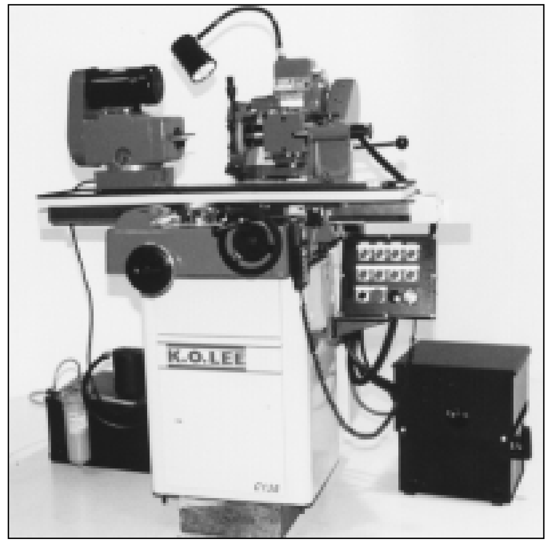


FIGURE 8.12: Tool and cutter grinders, are used to properly sharpen drills and other cutting tools. (Courtesy K. O. Lee Co.)

Lip Angle and Lip Length: When grinding the two cutting edges they should be equal in length and have the same angle with the axis of the drill as shown in Figure 8.9a. Figure 8.9b shows two ground drill points.

For drilling hard or alloy steels, angle C (Fig. 8.9a) should be 135 degrees. For soft materials and for general purposes, angle C should be 118 degrees. For aluminum, angle C should be 90 degrees.

If lips are not ground at the same angle with the axis, the drill will be subjected to an abnormal strain, because only one lip comes in contact with the work. This will result in unnecessary breakage and also cause the drill to dull quickly. A drill so sharpened will drill an oversized hole. When the point is ground with equal angles, but has lips of different lengths, a condition as shown in Figure 8.10a is produced.

A drill having cutting lips of different angles, and of unequal lengths, will be laboring under the severe conditions shown in Figure 8.10b.

Lip Clearance Angle: The clearance angle, or 'backing-off' of the point, is the next important thing to consider. When drilling steel this angle A (Fig. 8.11a) should be from 6 to 9 degrees. For soft cast iron and other soft materials, angle A may be increased to 12 degrees (or even 15 degrees in some cases)

This clearance angle should increase gradually as the center of the drill is approached. The amount of clearance at the center of the drill determines the chisel point angle B (Fig. 8.11b).

The correct combination of clearance and chisel point angles should be as follows: When angle A is made to be 12 degrees for soft materials, angle B should be made approximately 135 degrees; when angle A is 6 to 9 degrees for harder materials, angle B should be 115 to 125 degrees.

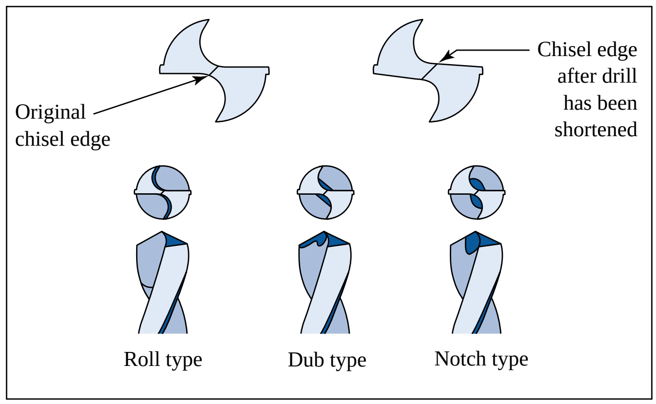


FIGURE 8.13: Web thinning restores proper web thickness after sharpening twist drills; three methods are shown.

While insufficient clearance at the center is the cause of drills splitting up the web, too much clearance at this point will cause the cutting edges to chip.

In order to maintain the necessary accuracy of point angles, lip lengths, lip clearance angle, and chisel edge angle, the use of machine point grinding is recommended. There are many commercial drill point grinders available today, which will make the accurate repointing of drills much easier. Tool and cutter grinders such as the one shown in Figure 8.12 are often used.

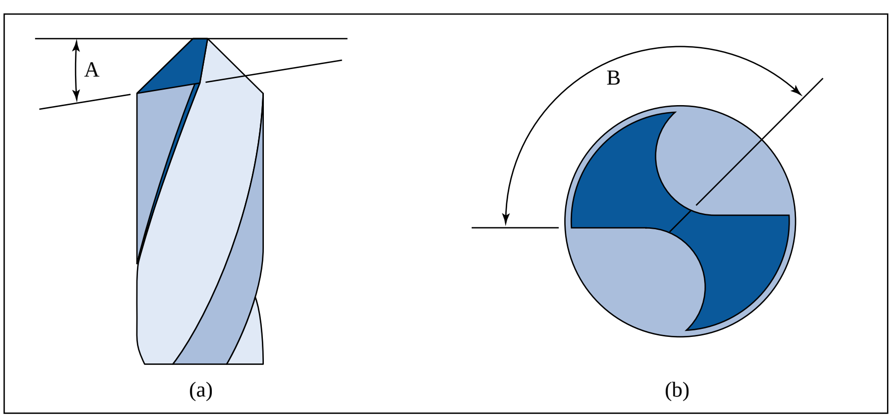


FIGURE 8.11: Drill lip clearance angle (a) and drill chisel point angle (b).

Twist Drill Web Thinning: The tapered web drill is the most common type manufactured. The web thickness increases as this type of drill is resharpened. This requires an operation called web thinning to restore the tool's original web thickness. Without the web thinning process, more thrust would be required to drill, resulting in additional generated heat and reduced tool life. Figure 8.13 illustrates a standard drill before and after the web thinning process. Thinning is accomplished with a radiused wheel and should be done so the thinned section tapers gradually from the point. This prevents a blunt wedge from being formed that would be detrimental to chip flow. Thinning can be done by hand, but since point centrality is important, thinning by machine is recommended.

8.6 Spade Drills

The tool generally consists of a cutting blade secured in a fluted holder (See Figure 8.14). Spade drills can machine much larger holes (up to 15 in. in diameter) than twist drills. Spade drills usually are not available in diameters smaller than 0.75 inch. The drilling depth capacity of spade drills, with length-to-diameter ratios over 100 to 1 possible, far exceeds that of twist drills. At the same time, because of their much greater feed capability, the penetration rates for spade drills exceed those of twist drills by 60 to 100 percent. However, hole finish generally suffers because of this. Compared to twist drills, spade drills are much more resistant to chatter under heavy feeds once they are fully engaged with the workpiece. Hole straightness is generally improved (with comparable size capability) by using a spade drill. However, these advantages can only be gained by using drilling machines of suitable capability and power.

The spade drill is also a very economical drill due to its diameter flexibility. A single holder will accommodate many blade diameters as shown in Figure 8.14. Therefore, when a diameter change is required, only the blade needs to be purchased which is far less expensive than buying an entire drill.

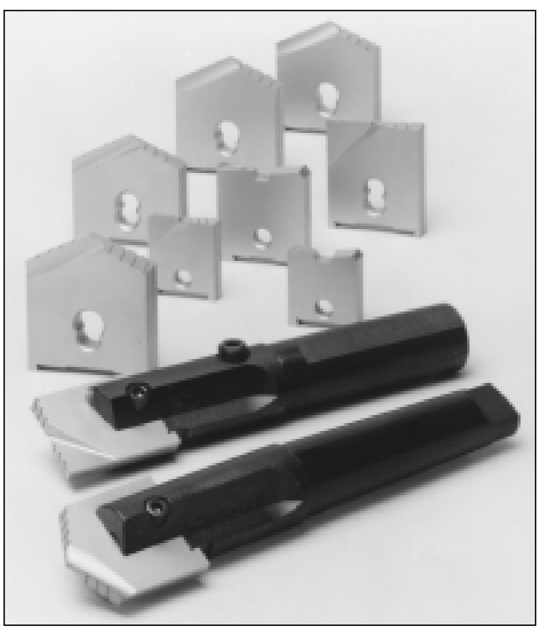


FIGURE 8.14: Spade drills with various cutting blades. (Courtesy Kennametal Inc.)

8.6.1 Spade Drill Blades

The design of spade drill blades varies with the manufacturer and the intended application. The most common design is shown in Figure 8.15. The locator length is ground to a precision dimension that, in conjunction with the ground thickness of the blade, precisely locates the blade in its holder. When the seating pads properly contact the holder, the holes in the blade and holder are aligned and the assembly can be secured with a screw.

The blade itself as shown in Figure 8.15, possesses all the cutting geometry necessary. The point angle is normally 130 degrees but may vary for special applications. In twist drill designs, the helix angle generally determines the cutting rake angle but since spade drills have no helix, the rake surface must be ground into the blade at the cutting edge angle that produces the proper web thickness. The cutting edge clearance angle is a constant type of relief, generally 6 to 8 degrees. After this clearance is ground, the chip breakers are ground, about 0.025 inch deep, in the cutting edge.

These chip breakers are necessary on spade drill blades and not optional as with twist drills. These notches make the chips narrow enough to flush around the holder. Depending on the feed rate, the grooves can also cause a rib to form in the chip. The rib stiffens the chip and causes it to fracture or break more easily which results in shorter, more easily removed chips. Margins on the blade act as bearing surfaces once the tool is in a bushing or in the hole being drilled. The width of the margins will vary from 1/16 to 3/16 inches, depending on the tool size. A slight back taper of 0.004 to 0.006 inch is normally provided and outside diameter clearance angles are generally 10 degrees.

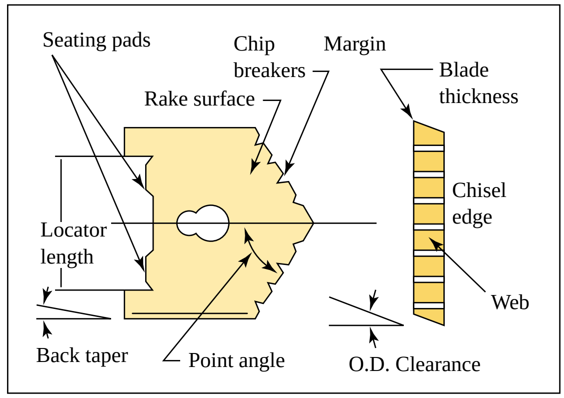


FIGURE 8.15: Spade drill cutting blades shows geometry specifications.

8.6.2 Spade Drill Blade Holders

The blade holder makes up the major part of the spade drill. The blade holder is made of heat-treated alloy steel and is designed to hold a variety of blades in a certain size range as shown in Figure 8.14. Two straight chip channels or flutes are provided for chip ejection.

The holder shank designs are available in straight, Morse taper, and various other designs to fit the machine spindles. The holders are generally supplied with internal coolant passages to ensure that coolant reaches the cutting edges and to aid chip ejection.

When hole position is extremely critical and requires the use of a starting bushing, holders with guide strips are available. These strips are ground to fit closely with the starting bushing to support the tool until it is fully engaged in the workpiece. The strips may also be ground to just below the drill diameter to support the tool in the hole when the set-up lacks rigidity.

8.6.3 Spade Drill Feeds and Speeds

The cutting speed for spade drills is generally 20 percent less than for twist drills. However, the spade drill feed capacity can be twice that of twist drills. The manufacturers of spade drills and other reference book publishers provide excellent recommendations for machining rates in a large variety of metals. These published rates should generally be observed. Spade drills work best under moderate speed and heavy feed. Feeding too lightly will result in either long, stringy chips or chips reduced almost to a powder. The drill cutting edges will chip and burn because of the absence of the thick, heat absorbing, C-shaped chips. Chips can possibly jam

FIGURE 8.16: Indexing drills were introduced using square inserts; three sizes are shown here. (Courtesy Kennametal Inc.)

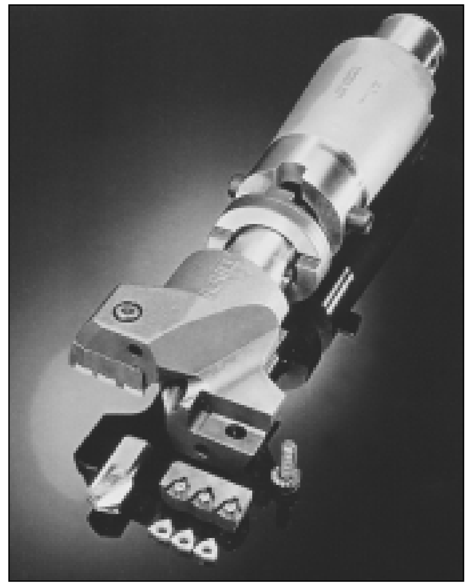


FIGURE 8.18: Indexable drill using six Trigon inserts for drilling large holes. (Courtesy Kennametal Inc.)

and pack, which can break the tool or the workpiece. If the machine cannot supply the required thrust to maintain the proper feed without severe deflection, a change in tool or machine may be necessary.

8.7 Indexable Carbide Drills

Indexable drilling has become so efficient and cost effective that in many cases it is less expensive to drill the hole rather than to cast or forge it. Basically, the indexable drill is a two fluted, center cutting tool with indexable carbide inserts. Indexable drills were introduced using square inserts (see Fig. 8.16). Shown in Figure 8.17a are indexable drills using the more popular Trigon Insert (see Fig. 8.17b). In most cases two inserts are used, but as size increases, more inserts are added with as many as eight inserts in very large tools. Figure 8.18 shows six inserts being used.

Indexable drills have the problem of zero cutting speed at the center even though speeds can exceed 1000 SFPM at the outermost inserts. Because speed generally replaces feed to some degree, thrust forces are usually 25 to 30 percent of those required by conventional tools of the same size. Indexable drills have a shank, body, and multi-edged point. The shank designs generally available are straight, tapered and number 50 V-flange.

The bodies have two flutes that are normally straight but may be helical. Because no margins are present to provide bearing support, the tools must rely on their inherent stiffness and on the balance in the cutting forces to maintain accurate hole size and straightness. Therefore, these tools are usually limited to length-to-diameter ratios of approximately 4 to 1.

The drill point is made of pocketed carbide inserts. These inserts are usually specially designed. The cutting rake can be negative, neutral, or positive, depending on holder and insert design. Coated and uncoated carbide grades are available for drilling a wide variety of work materials. Drills are sometimes combined with indexable or replaceable inserts to perform more than one operation, such as drilling, counterboring, and countersinking.

As shown in Figure 8.19a and Figure 8.19b, body mounted insert tooling can perform multiple operations. More examples will be shown and discussed in Chapter 10: Boring Operations and Machines.

The overall geometry of the cutting edges is important to the performance of indexable drills. As mentioned earlier, there are no supporting margins to keep these tools on line, so the forces required to move the cutting edges through the work material must be balanced to minimize tool deflection, particularly on starting, and to maintain hole size.

While they are principally designed for drilling, some indexable drills, as shown in Figure 8.20, can perform facing, and boring in lathe

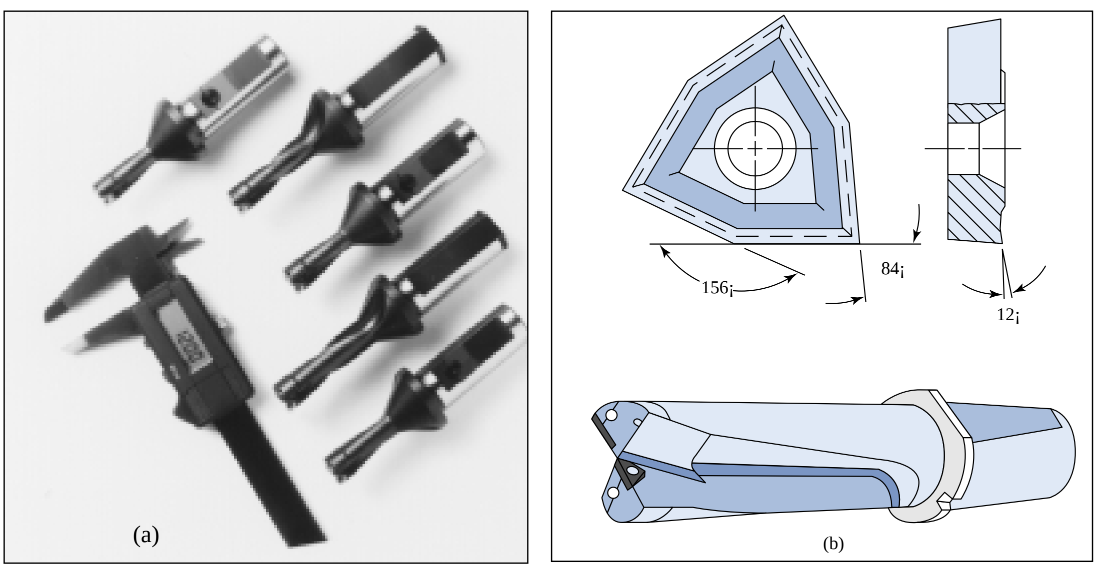
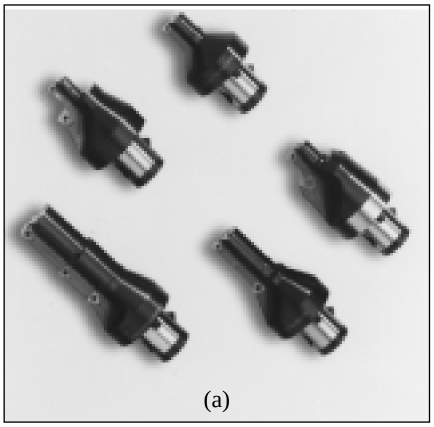


FIGURE 8.17: (a) Indexable drills using Trigon inserts. (b) A Trigon insert and holder. (Courtesy Komet of America, Inc.)



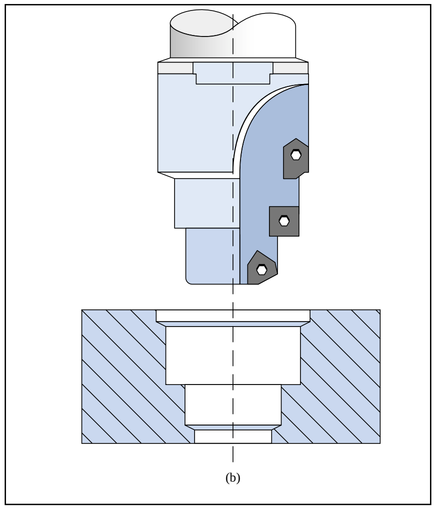


FIGURE 8.19: Body-mounted insert tooling can perform multiple operations. (Courtesy Komet of America, Inc.)

applications. How well these tools perform in these applications depends on their size, rigidity, and design.

8.7.1 Indexable Carbide Drill Operation

When used under the proper conditions, the performance of indexable drills is impressive. However, the manufacturer's recommendations must be carefully followed for successful applications.

Set-up accuracy and rigidity is most important to tool life and performance. Chatter will destroy drilling inserts just as it destroys turning or milling inserts. If the inserts fail when the tool is rotating in the hole at high speed, the holder and workpiece will be damaged. Even if lack of rigidity has only a minor effect on tool life, hole size and finish will be poor. The machine must be powerful, rigid and capable of high speed. Radial drill presses do not generally meet the rigidity requirements. Heavier lathes, horizontal boring mills, and N/C machining centers are usually suitable.

When installing the tool in the machine, the same good practice followed for other drill types should be observed for indexable drills. The shanks must be clean and free from burrs to ensure good holding and to minimize runout. Runout in indexable drilling is dramatically amplified because of the high operating speeds and high penetration rates.

When indexing the inserts is necessary, make sure that the pockets are clean and undamaged. A small speck of dirt or chip, or a burr will cause stress in the carbide insert and result in a microscopic crack, which in turn, will lead to early insert failure.

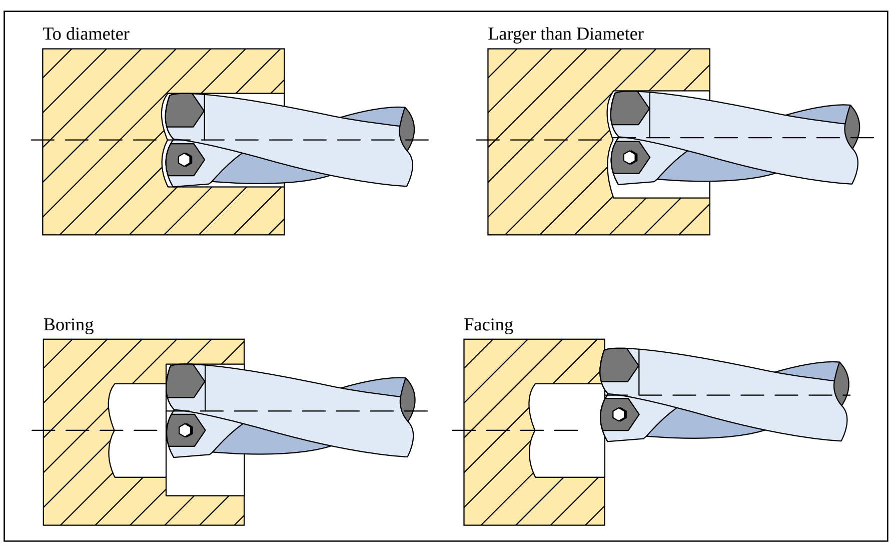


FIGURE 8.20: In addition to drilling, indexable drills can perform boring and facing operations.

8.7.2 Indexable Drill Feeds and Speeds

Indexable drills are very sensitive to machining rates and work materials. The feed and speed ranges for various materials, as recommended by some manufacturers of these tools, can be very broad and vague, but can be used as starting points in determining exact feed and speed rates. Choosing the correct feed and speed rates, as well as selecting the proper insert style and grade, requires some experimentation. Chip formation is a critical factor and must be correct.

In general, soft low carbon steel calls for high speed (650 SFPM or more), and low feed (0.004/0.006 IPR). Medium and high carbon steels, as well as cast iron, usually react best to lower speed and higher feed. The exact speed and feed settings must be consistent with machine and set-up conditions, hole size and finish requirements, and chip formation for the particular job.

8.8 Trepanning

In trepanning the cutting tool produces a hole by removing a disk shaped piece also called slug or core, usually from flat plates. A hole is produced without reducing all the material removed to chips, as is the case in drilling. The trepanning process can be used to make disks up to 6 in. in diameter from flat sheet or plate. A trepanning tool also called a "Rotabroach" with a core or slug is shown in Figure 8.21a and an end view of a Rotabroach is shown in Figure 8.21 b.

Trepanning can be done on lathes, drill presses, and milling machines, as well as other machines using single point or multi point tools. Figure 8.22 shows a Rotabroach cutter machining holes through both sides of a rectangular tube on a vertical milling machine.

Rotabroach drills provide greater tool life because they have more teeth than conventional drilling tools. Since more teeth are engaged in the workpiece, the material cut per hole is distributed over a greater number of cutting edges. Each cutting edge cuts less material for a given hole. This extends tool life significantly.

Conventional drills must contend with a dead center area that is prone to chip, thus reducing tool life. In the chisel-edge region of a conventional drill the cutting speed approaches zero. This

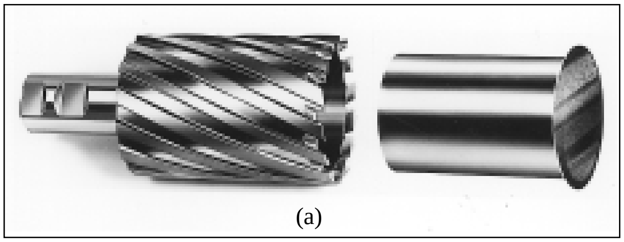
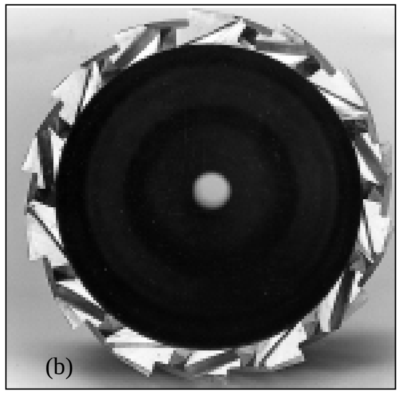


FIGURE 8.21: Trepanning tool also called Rotabroach with core or slug. (Courtesy Hougen Manufacturing, Inc.)



is quite different from the speed at the drill O.D. Likewise, thrust forces are high due to the point geometry. Rotabroach drills cut in the region from the slug O.D. to the drill O.D. Since only a small kerf is machined, cutting speeds are not so different across the face of a tooth. This feature extends tool life and provides uniform machinability. Figure 8.23 shows drilling holes with conventional drills and hole broaching drills.

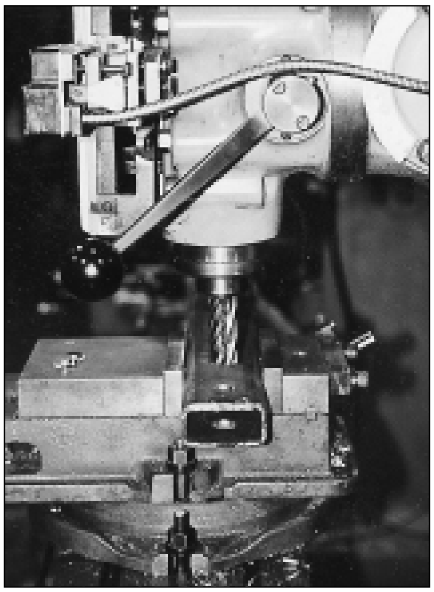


FIGURE 8.22: Rotabroach machining set-up on a milling machine. (Courtesy Hougen Manufacturing, Inc.)

8.8.1 Trepanning Operations

Trepanning is a roughing operation. Finishing work requires a secondary operation using reamers or boring bars to get a specified size and finish. Of the many types of hole-making operations, it competes with indexable carbide cutters and spade drilling.

Several types of tools are used to trepan. The most basic is a single or double point cutter (Fig. 8.24). It orbits the spindle centerline cutting the periphery of the hole. Usually, a pilot drill centers the tool and drives the orbiting cutter like a compass inscribing a circle on paper. Single/double point trepanning tools are often adjustable within their working diameter. They are efficient and versatile, but do begin to have rigidity problems when cutting large holes - 6 1/2 inches in diameter is about the maximum.

A hole saw is another tool that trepans holes. It is metalcutting's version of the familiar doorknob hole cutter used in wood. Hole saws have more teeth and therefore cut faster than single, or double-point tools. Both hole saws and single-point tools curl up a chip in the space, or gullet, between the teeth, and carry it with them in the cut.

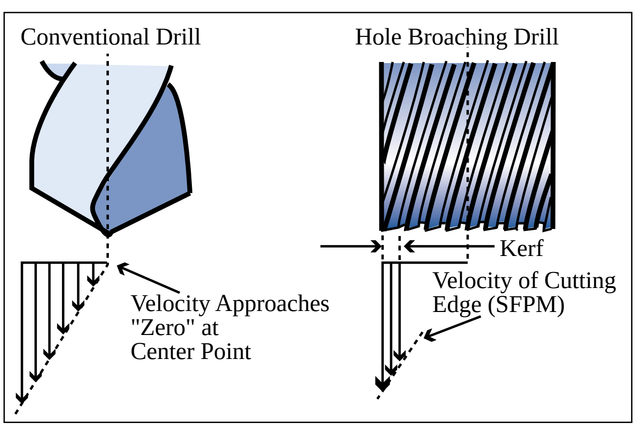


FIGURE 8.23: Drilling holes with conventional drill and hole broaching drill. Surface speed increases with distance from center.

Hole broaching tools are hybrid trepanners. (Fig. 8.21a and 8.21b) They combine spiral flutes like a drill with a broach-like progressive tool geometry that splits the chip so it exits the cut along the flutes. With this design, the larger number of cutting edges and chip evacuation, combine to reduce the chip load per tooth so this drill can cut at higher feed rates than trepanning tools and hole saws. Like the hole saw, a hole broaching tool has a fixed diameter. One size fits one hole.

8.8.2 Cutting Tool Material Selection

M2 High Speed Steel (HSS) is the standard Rotabroach cutting tool material. M2 has the broadest application range and is the most economical tool material. It can be used on ferrous and non-ferrous materials and is generally recommended for cutting materials up to 275 BHN. M2 can be applied to harder materials, but tool life is dramatically decreased.

TiN coated M2 HSS Rotabroach drills are for higher speeds, more endurance, harder materials or freer cutting action to reduce power consumption. The TiN coating reduces friction and operates at cooler temperatures while presenting a harder cutting edge surface. Increased cutting speeds of 15 to 25 % are recommended to obtain the benefits of this surface treatment. The reduction in friction and resistance to edge build-up are key benefits. The ability to run at higher speeds at less power is helpful for applications where the machine tool is slightly underpowered and TiN coated tools are recommended for these applications. TiN coated tools are recommended for applications on materials to 325 BHN.

Carbide cutting tool materials are

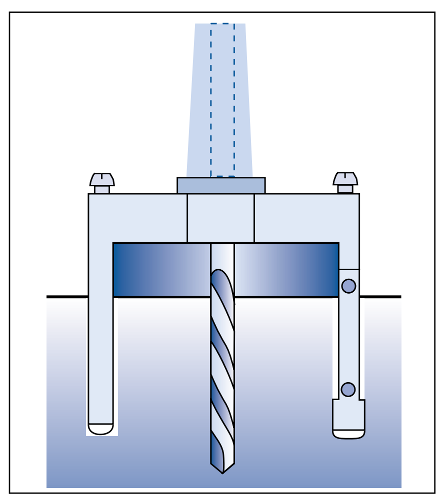


FIGURE 8.24: Traditional trepanning tool orbits around a center drill.

also available as a special option on Rotabroach drills. Carbide offers certain advantages over high-speed steel. Applications are limited and need to be discussed with a manufacturer's representative.

8.8.3 Rigidity and Hole Size Tolerance

Rotabroach drills were originally designed as roughing tools to compete with twist drills and provide similar hole tolerances. Many users have successfully applied Rotabroach drills in semi-finishing applications, reducing the number of passes from two or more to just one. A rigid machine tool and set-up are required to produce holes to these specifications. Tolerances will vary with the application and are impossible to pin point.

Spindle rigidity or "tightness" and workpiece rigidity are more crucial than with a twist drill. Even if a twist drill runs out slightly at first, the conical point tends to center itself before the O.D. of the tool engages the workpiece. The higher thrust of a twist drill also tends to " preload" the spindle and fixture. The trepanning cutter relies more on the rigidity of the system (workpiece, holder, and spindle). If excessive spindle runout or, worse yet, spindle play exists, the cutter may chatter on entry. At best this will cause a bell-mouthed hole with poor finish, but it can easily lead to drill breakage.

Hole tolerances are dependent on much more than the accuracy of any tool and its grind. The machine tool, workpiece, fixture, selection of speeds and feeds, projection and type of application also play an important part in determining overall results.

8.8.4 Chip Control

In material such as aluminum, tool steels and cast iron, proper selection of feeds and speeds usually causes the chips to break up and allows them to be flushed out of the cut by the cutting fluid. In many other materials, such as mild and alloy steels, the chips tend to be long and frequently wrap themselves around the drill to form a "bird's nest". In most manual operations this is an annoyance that is outweighed by the other benefits of the method. In automated operations, however, the build-up of chips around the drill cannot be tolerated. Besides the obvious problems that this can cause, the nest of chips impedes the flow of additional chips trying to escape from the flutes. This in turn can cause the flutes to pack and may result in drill breakage.

There are several methods that can be used to break up the chips if this cannot be accomplished by adjusting the feeds and speeds. One method is to use an interrupted feed cycle. It is recommended that the drill not be retracted as with a "peck" cycle, because chips may become lodged under the cutting edges. Instead employ an extremely short dwell approximately every two revolutions. This will produce a chip that is usually short enough not to wrap around the tool. A programmed dwell may not be necessary since some hesitation is probably inherent between successive feed commands in an NC system

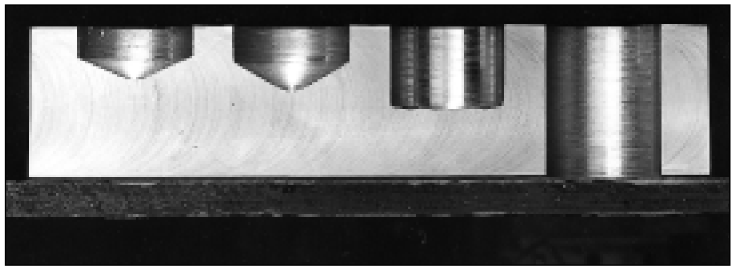


FIGURE 8.25: Trepanning produces holes faster than more conventional tooling. (Courtesy Hougen Manufacturing, Inc.)

8.8.5 Advantages of Trepanning Tools

The twist drill has a center point, which is not really a point at all - it's the intersecting line where two cutting edge angles meet at the web of the drill. This point is the so-called "dead zone" of a twist drill.

It's called a dead zone because the surface speed of the cutting edges (a factor of revolutions per minute and diameter of the drill) approaches zero as the corresponding diameter nears zero. Slower surface speed reduces cutting efficiency and requires increased feed pressure for the cutting edges to bite into the material. In effect, the center of the drill does not cut - it pushes its way through the material. The amount of thrust required to overcome the resistance of the workpiece often causes the stock to deform or dimple around the hole, and creates a second problem - burrs or flaking around the hole's breakthrough side. As material at the bottom of the hole becomes thinner and thinner, if the feed is not eased off, the drill will push through, typically leaving two jagged remnants of stock attached.

Trepanning tools produce holes faster than more conventional tooling as shown in Figure 8.25. From left to right are shown a 1 1/2 inch hole drilled into a 2 inch thick 1018 steel plate with: a spade drill, with a twist drill, with an indexable carbide drill, and with a Rotabroach. With approximately 50% to 80% faster drilling time, the cost per hole can be substantially lower.

An indirect yet significant source of savings attributable to trepanning tooling is the solid slug it provides. Separating chips from coolant and oils is increasingly called for by scrap haulers, In one application, while significant gains in productivity were made with hole-broaching tools, the savings in going from chips to a solid slug was enough to justify the change in process.

In Figure 8.26 the workpiece is a tube holder for an industrial heat exchanger. When this workpiece is finished, better than 60 per cent of the plate has been reduced to scrap.

Sixty percent of this heat exchanger plate was converted into chips by the sheer number of holes drilled. Besides increasing production, trepanning tooling's solid core by-product increased scrap value from $0.17 per pound of chips, to $0.37 per pound for the core metal.

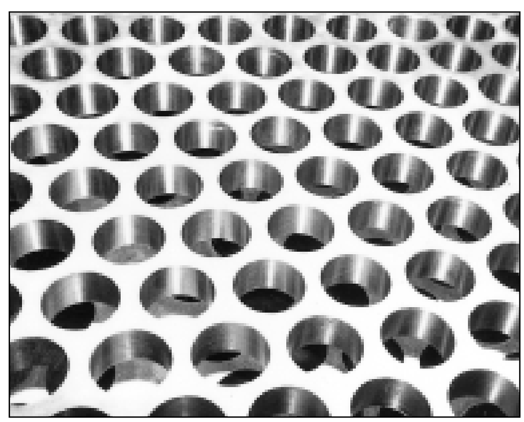


FIGURE 8.26: Sixty percent of this heat exchanger plate was converted to chips. (Courtesy Hougen Manufacturing, Inc.)