Drilling and Related Operations

Drilling, Figure 1, is a machining operation used to create a round hole in a work part. This contrasts with boring, which can only be used to enlarge an existing hole. Most drilling operations are performed using a rotating cylindrical tool that has two cutting edges on its working end. The tool is called a ***drill*** or ***drill bit***, the most common form of which is the twist drill. The rotating drill feeds into the stationary work part to form a hole whose diameter is equal to the drill diameter.

Drilling is customarily performed on a ***drill press***, although other machine tools also perform this operation.

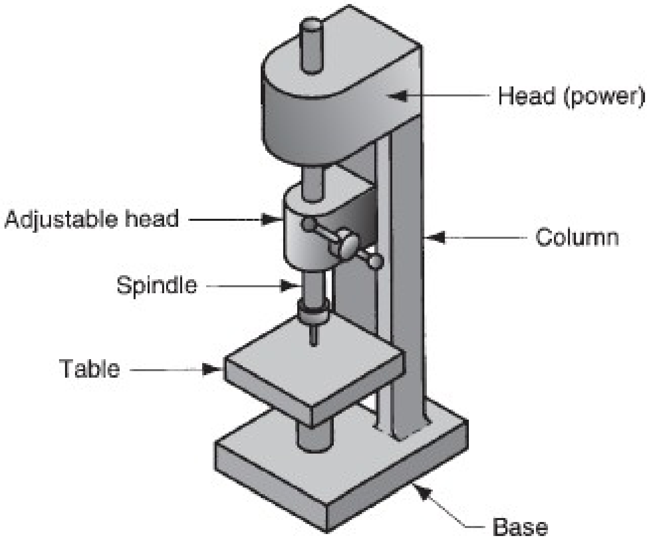

Figure 1: Drilling machine

Drills

Various cutting tools are available for hole making, but the ***twist drill*** is by far the most common. It comes in diameters ranging from about 0.15 mm to as large as 75 mm. Twist drills are widely used in industry to produce holes rapidly and economically. The standard twist drill geometry is illustrated in Figure 2. The body of the drill has two spiral ***flutes*** (the spiral gives the twist drill its name). The angle of the spiral flutes is called the ***helix angle***, a typical value of which is around 30°. While drilling, the flutes act as passageways for extraction of chips from the hole. Although it is desirable for the flute openings to be large to provide maximum clearance for the chips, the body of the drill must be supported over its length. This support is provided by the ***web***, which is the thickness of the drill between the flutes.

The point of the twist drill has a conical shape. A typical value for the ***point angle*** is 118°. The point can be designed in various ways, but the most common design is a ***chisel edge***, as in Figure 2. Connected to the chisel edge are two cutting edges (sometimes called lips) that lead into the flutes. The portion of each flute adjacent to the cutting edge acts as the rake face of the tool.

The cutting action of the twist drill is complex. The rotation and feeding of the drill bit result in relative motion between the cutting edges and the workpiece to form the chips. The cutting speed along each cutting edge varies as a function of the distance from the axis of rotation. Accordingly, the efficiency of the cutting action varies, being most efficient at the outer diameter of the drill and least efficient at the center. In fact, the relative velocity at the drill point is zero, so no cutting takes place.

Instead, the chisel edge of the drill point pushes aside the material at the center as it penetrates into the hole; a large thrust force is required to drive the twist drill forward into the hole. Also, at the beginning of the operation, the rotating chisel edge tends to wander on the surface of the work part, causing loss of positional accuracy.

Various alternative drill point designs have been developed to address this problem. Chip removal can be a problem in drilling. The cutting action takes place inside the hole, and the flutes must provide sufficient clearance throughout the length of

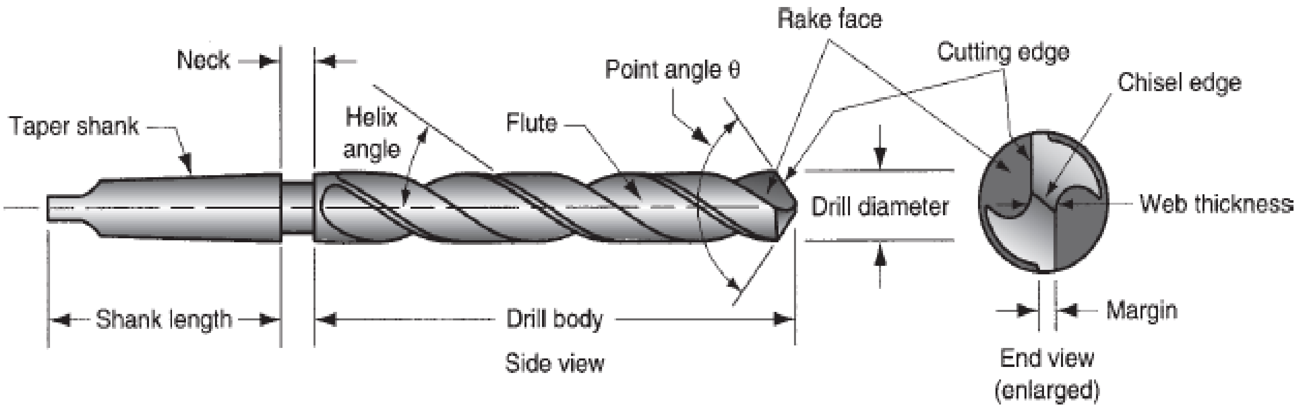

Figure 2: Standard geometry of a twist drill

The drill to allow the chips to be extracted from the hole. As the chip is formed it is forced through the flutes to the work surface. Friction makes matters worse in two ways. In addition to the usual friction in metal cutting between the chip and the rake face of the cutting edge, friction also results from rubbing between the outside diameter of the drill bit and the newly formed hole. This increases the temperature of the drill and work. Delivery of cutting fluid to the drill point to reduce the friction and heat is difficult because the chips are fl owing in the opposite direction. Because of chip removal and heat, a twist drill is normally limited to a hole depth of about four times its diameter. Twist drills are normally made of high-speed steel. The geometry of the drill is fabricated before heat treatment, and then the outer shell of the drill (cutting edges and friction surfaces) is hardened while retaining an inner core that is relatively tough. Grinding is used to sharpen the cutting edges and shape the drill point.

Although twist drills are the most common hole-making tools, other drill types are also available. ***Straight-flute drills*** operate like twist drills except that the flutes for chip removal are straight along the length of the tool rather than spiraled. The simpler design of the straight-flute drill permits carbide tips to be used as the cutting edges, either as brazed or indexable inserts. Figure 3 illustrates the straight-flute indexable-insert drill. The cemented carbide inserts allow higher cutting speeds and greater production rates than HSS twist drills. However, the inserts limit how small the drills can be made. Thus, the diameter range of commercially available indexable- insert drills runs from about 16 mm to about 127 mm.

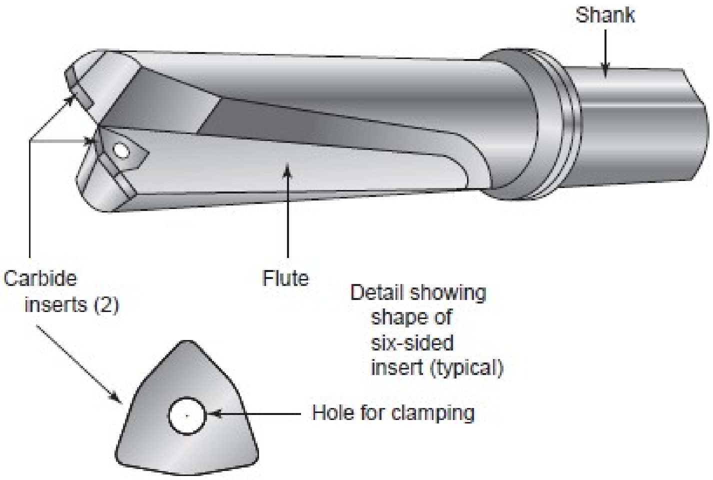

Figure 3: Straight-flute drill that uses indexable inserts

CUTTING CONDITIONS IN DRILLING

The cutting speed in a drilling operation is the surface speed at the outside diameter of the drill. It is specified in this way for convenience, even though nearly all of the cutting is actually performed at lower speeds closer to the axis of rotation. To set the desired cutting speed in drilling, it is necessary to determine the rotational speed of the drill. Letting N represent the spindle rev/min,

$$N = \frac{V}{\pi D} \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(1)}$$

Where v = cutting speed, mm/min; and D = the drill diameter, mm. In some drilling operations, the workpiece is rotated about a stationary tool, but the same formula applies.

Feed f in drilling is specified in mm/rev. Recommended feeds are roughly proportional to drill diameter; higher feeds are used with larger diameter drills. Since there are (usually) two cutting edges at the drill point, the uncut chip thickness (chip load) taken by each cutting edge is half the feed. Feed can be converted to feed rate using the same equation as for turning:

$$f_r = Nf \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(2)}$$

Where fr = feed rate, mm/min.

Drilled holes are either through holes or blind holes, shown in Figure 4 with a twist drill at the beginning of the operation. In ***through holes***, the drill exits the opposite side of the work; in ***blind holes***, it does not. The machining time required to drill a through hole can be determined by the following formula:

$$T_m = \frac{t + A}{f_r} \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(3)}$$

Where Tm= machining (drilling) time, min; t = work thickness, mm; f_r = feed rate, mm/min; and A = an approach allowance that accounts for the drill point angle, representing the distance the drill must feed into the work before reaching full diameter, Figure 4(a). This allowance is given by

$$A = 0.5D\tan(90 - \frac{\theta}{2}) \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (4)$$

Where A = approach allowance, mm; and θ = drill point angle, °. In drilling a through hole, the feed motion usually proceeds slightly beyond the opposite side of the work, thus making the actual duration of the cut greater than Tm in Equation (3) by a small amount.

In a blind-hole, hole depth d is defined as the distance from the work surface to the depth of the full diameter, Figure 4(b). Thus, for a blind hole, machining time is given by

$$T_m = \frac{d + A}{f_r} \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (5)$$

Where A = the approach allowance by Equation (4).

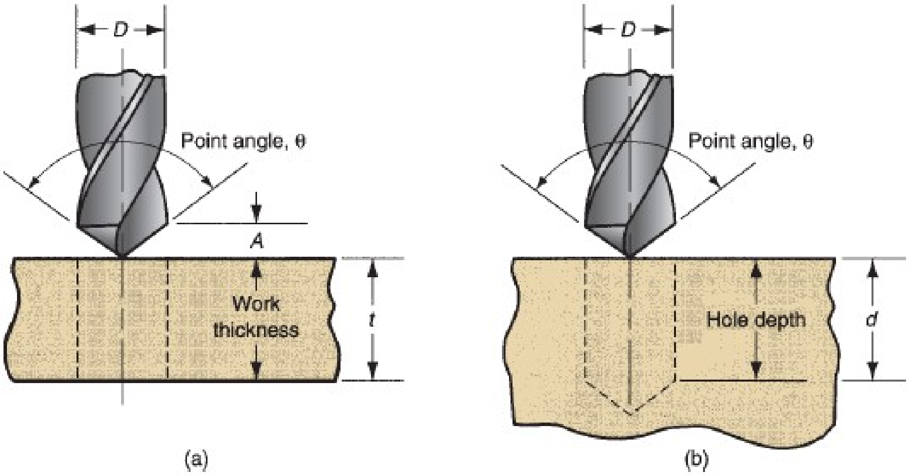

Figure 4: Two hole types: (a) through hole and (b) blind hole.

The rate of metal removal in drilling is determined as the product of the drill cross-sectional area and the feed rate:

$$R_{mR} = \frac{\pi D^2 f_r}{4} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (6)$$

Example: A drilling operation is performed to create a through hole on a steel plate that is 15 mm thick. Cutting speed = 0.5 m/s, and feed = 0.22 mm/rev. The 20-mm diameter twist drill has a point angle of 118º. Determine (a) the machining time and (b) metal removal rate once the drill reaches full diameter.

Solution: (a)

$N = v/\pi D = 0.5(103)/\ \pi(20) = 7.96$ rev/s

$f_r = N f = 7.96(0.22) = 1.75$ mm/s

$A = 0.5(20)\tan(90 - 118/2) = 6.01$ mm

$Tm = (t + A)/f_r = (15 + 6.01)/1.75 = 12.0$ s = **0.20 min**

(b)

$RMR = \pi\,(20)^2(1.75)/4 =$ **549.8 mm^3/s**

OPERATIONS RELATED TO DRILLING

Several operations related to drilling are illustrated in Figure 5 and described in this section. Most of the operations follow drilling; a hole must be made first by drilling, and then the hole is modified by one of the other operations. Centering and spot facing are exceptions to this rule. All of the operations use rotating tools.

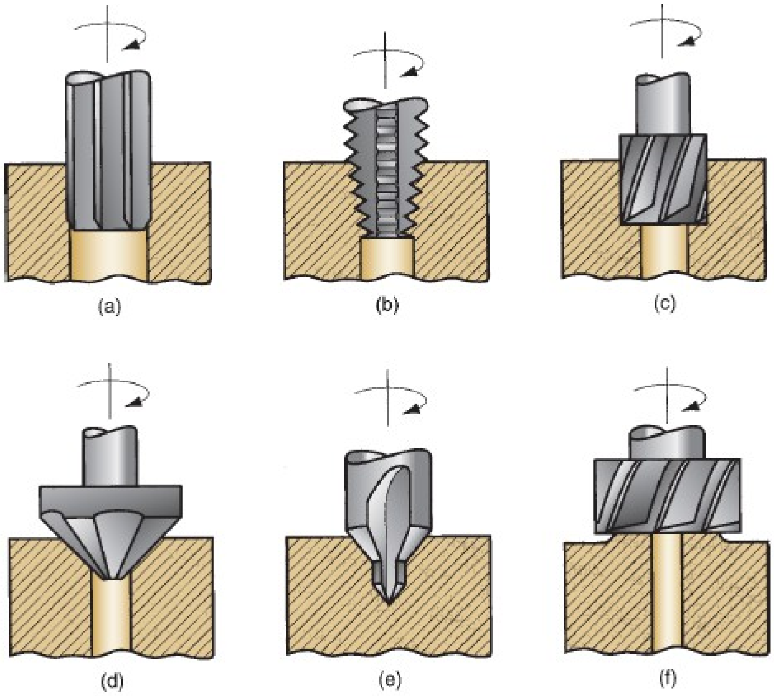

Figure 5: Machining operations related to drilling: (a) Reaming, (b) Tapping, (c) Counterboring, (d) Countersinking, (e) Centering, and (d) Spot facing

(a) ***Reaming***. Reaming is used to slightly enlarge a hole, to provide a better tolerance on its diameter, and to improve its surface finish. The tool is called a ***reamer***, and it usually has straight flutes.

(b) ***Tapping***. This operation is performed by a ***tap*** and is used to provide internal screw threads on an existing hole.

(c) ***Counterboring***. Counterboring provides a stepped hole, in which a larger diameter follows a smaller diameter partially into the hole. A counterbored hole is used to seat a bolt head into a hole so the head does not protrude above the surface.

(d) ***Countersinking***. This is similar to counterboring, except that the step in the hole is cone-shaped for flat head screws and bolts.

(e) ***Centering***. Also called center drilling, this operation drills a starting hole to accurately establish its location for subsequent drilling. The tool is called a ***center drill***.

(f) ***Spot facing***. Spot facing is similar to milling. It is used to provide a flat machined surface on the work part in a localized area.