Basic Holemaking

Training Objectives

After watching the video and reviewing this printed material, the viewer will gain knowledge and understanding of the basics of the drills and machines used in a variety of holemaking operations.

- a description of the basic twist drill is featured
- the importance of coolants are detailed
- drill maintenance and sharpening is shown
- manual and numerically controlled holemaking equipment is described
- secondary drilling and finishing operations are shown

Holemaking

Drilling, the most common of holemaking processes, consumes half of the cutting tools used in all chip making processes. In most cases, the drill creates a cylindrical hole by rotational action, cutting into a fixed workpiece. When holemaking is done on a lathe, the drill is fixed while the work rotates. Because most of the cutting and chip generation takes place in the confined space of the hole, lubrication is critical. Coolants lubricate the cut, cool the drill point, and help flush out chips.

The main problem in holemaking is deflection of the drill. A rule of thumb states that the rigidity of a the drill is proportional to the fourth power of its diameter. Thus, a 1" (25mm) diameter drill is only 1/16 as rigid as a 2" (50mm) diameter drill.

Because of the many types of drilling tools and methods, the tool choice is determined by a number of factors, including:

- the ultimate hole size
- the hole's depth
- tolerances required
- the type of material being drilled
- the amount of holes needed

Holemaking processes are divided into either “short-hole” drilling, which is most common, and “deep hole” drilling.

A short hole is defined as a hole with a small ratio of depth to diameter. Typically that would include holes up to 1.2" (30mm) having a depth of no more than 5 or 6 times that diameter. For holes greater than 1.2" (30mm) in diameter, short holes are those on more than 2.5 times hole diameter. Short holes are usually drilled in one motion. Drilling deeper with conventional drills requires repeated withdrawal of the drill to clear the chips from the drill’s flutes. This repeated withdrawal of the drill is called “pecking.”

Deep hole drilling is more difficult, mainly in keeping the hole straight, and requires special drills, guides, equipment, and methods. One of these methods is called “trepanning.” This involves the making of a circular cut using a hollow-core cutting tool. The result is a cut in the form of an outer ring while a central core of material drops through and leaves the full hole. In this way, larger diameter holes may be cut using less power and chip production.
Holes come in many forms. A “through” hole goes completely through a workpiece. A “blind” hole is drilled only to a certain depth. “Interrupted” holes intersect at some point with other holes within a workpiece. Holes may be drilled for coolant passages or to provide access for part inspection. Holes may also be threaded for fasteners.

The Drill

By definition, a drill is a round, end-cutting tool with one or more cutting lips and one or more straight or helical flutes. The twist drill is the most common holemaking tool, and is made up of a shank, flutes, two cutting lips and a point. Such drills are made of high speed steel or carbide for drilling hard and abrasive materials. Twist drills are used in production for holes up to 5/8" (20mm) in diameter. Some are “indexable-insert” drills- steel drills with one or more carbide cutting edges seated in them.

The twist drill is held and driven by the shank. These shanks can be straight for use in collets or drill chucks, or tapered for mounting directly into machine spindles.

The helix angle of the drill’s flutes will vary according to the material being drilled. An angle of 25° to 33° is optimal for steel and cast iron. Angles ranging from 35° to 40° are used for softer metals like aluminum. Brass and plastics are best drilled with low helix drills having an angle of 15° to 20°.

The drill point is formed by the two cutting lips and does the work of metal cutting. The standard drill point angle is 118° with a clearance angle of between 10° and 20°. Variations in tip geometry have improved drill action by allowing cooler running and improved chip removal. Various high penetration drills have been developed with special point designs. These drills may also be coated high speed steel, carbide tipped, or all carbide.

Like other tools, drills will become dull over time. Indications of dulling include an increase of spindle pressure required to produce chips and a squealing in the cut. When required, drill sharpening should always be done on machines or with fixtures designed for that specific purpose.

There are many special purpose drills such as the center drill which is very short with a 60° countersink. The center drill is used to start holes. Spade drills have interchangeable flat blade inserts which will drill large diameter holes from 1" (25mm) to 6" (150mm). Core drills are multi-fluted drills used to enlarge and finish previously drilled or cast holes. A step drill presents increasingly larger diameters as it progresses through the material. Drilling end mills can drill to relatively shallow depths and then enlarge or reshape the hole according to movement of the workpiece on the machine table.

For increased accuracy and to help prevent drill breakage, drilling is often done using drill jigs and bushings. The jig positions the drill while the hardened bushing guides it. This keeps the drill straight and true as it enters the work.
Holemaking Machines And Parameters

The drill press is a familiar shop machine, but most production drilling is done on more productive and sophisticated equipment. This equipment includes lathes, knee mills, machining centers, and high-production transfer machines. For drilling large workpieces, a radial drill press can be used. The drill head can be moved in and out from the machine column and rotated around the column to almost any point. Gang drilling is done with machines having a number of drill press heads mounted on a common base and table. Self-feeding drilling units operate automatically for repetitive work. They can be set up for drilling as well as secondary hole finishing operations. Multi-spindle drilling machines are used for mass production. The rotary motion of the multi-spindle machine is generated from a central source and transmitted to multiple spindles. Drill and tap centers perform automated tapping as well as drilling operations.

The two parameters for accurate and efficient drilling include:

- cutting speed, which is measured at the periphery of the drill body and expressed in surface feet per minute
- feed or penetration rate, which is the distance of travel into the stock per unit of time and expressed as inches or millimeters per minute

As a rule of thumb, the harder the material, the slower the cutting speed and feed rates.

Initial hole drilling is usually followed by some kind of finishing operation. Reaming is one such method. A reamer removes just enough metal to provide a smooth finish and accurate size to the hole. Reamers may be right- or left-handed, straight or tapered. Boring is an internal turning operation that enlarges and finishes an existing hole. Boring uses a bar with a single-point cutting tool, or a tool with two or three edges. Boring can improve a hole's geometry and location, since the hole shape is primarily determined by machine tool motions. Roller burnishing is a polishing operation which smoothes irregularities on the hole wall and often produces a mirror-like finish. Honing is an abrasive method which gives very accurate final sizing and finish to the hole.

Other post-drilling operations include counterboring which enlarges hole diameter to a specific depth, countersinking which produces an angle entry to the hole to allow certain types of hardware to be flush to the material's surface, and spotfacing. Spotfacers, like counterbores, are pilot guided tools that provide a flat clean surface on angled or irregular surfaces.
Basic Holemaking

Review Questions

1. The main problem encountered in holemaking is:
   a. dulling of the drill's cutting edges
   b. the generation of long chips
   c. excessive heating of the workpiece
   d. deflection of the drill

2. Periodically withdrawing the drill to clear chips is called:
   a. stepping
   b. pecking
   c. burnishing
   d. high speed drilling

3. Trepanning requires the use of:
   a. a boring bar
   b. carbide tips
   c. a hollow-core cutting tool
   d. slower drilling speed

4. The standard drill point angle is:
   a. 60°
   b. 64°
   c. 118°
   d. 128°

5. The helix angle recommended for drilling aluminum is:
   a. 30° to 33°
   b. 35° to 40°
   c. 25° to 30°
   d. 60° to 66°

6. A drilling machine typically used on large workpieces is a:
   a. gang drilling machine
   b. multi-spindle drill machine
   c. knee mill
   d. radial drill

7. A post-holemaking operation that provides a very accurate final sizing and finish to a hole is called:
   a. roller burnishing
   b. reaming
   c. honing
   d. boring
Basic Holemaking

Answer Key

1. d
2. b
3. c
4. c
5. b
6. d
7. c