Adjustable reamers have cutting edges in the form of blades that are locked in a body. The blades can be adjusted over a greater range than expansion reamers. This permits adjustment for size and to compensate for regrinding. When the blades become too small from regrinding, they can be replaced. Both tool steel and carbide blades are used.

Taper reamers are used for finishing holes to an exact taper. They may have up to eight straight or spiral flutes. Standard tapers, such as Morse, Jarno, or Brown & Sharpe, come in sets of two. The roughing reamer has nicks along the cutting edge to break up the heavy chips that result as a cylindrical hole is cut to a taper. The finishing reamer has smooth cutting edges.

REAMING PRACTICE
If the material to be removed is free-cutting, reamers of fairly light construction will give satisfactory results. However, if the material is hard, then tough, solid-type reamers are recommended, even for fairly large holes.

To meet quality requirements, including both finish and accuracy (tolerances on diameter, roundness, straightness, and absence of bell-mouth at ends of holes), reamers must have adequate support for the cutting edges, and reamer deflection must be minimal. Reaming speed is usually two-thirds the speed for drilling the same materials. However, for close tolerances and fine finish, speeds should be slower.

Feeds are usually much higher than those for drilling and depend upon material. A feed of between 0.0015 and 0.004 in. per flute is recommended as a starting point. Use the highest feed that will still produce the required finish and accuracy. Recommended cutting fluids are the same as those for drilling. Reamers, like drills, should not be allowed to become dull. The chamfer must be reground long before it exhibits excessive wear. Sharpening is usually restricted to the starting taper or chamfer. Each flute must be ground exactly even, or the tool will cut oversize.

Reamers tend to chatter when not held securely, when the work or workholder is loose, or when the reamer is not properly ground. Irregularly spaced teeth may help reduce chatter. Other cures for chatter in reaming are to reduce the speed, vary the feed rate, chamfer the hole opening, use a piloted reamer, reduce the relief angle on the chamfer, or change cutting fluid. Any misalignment between the workpiece and the reamer will cause chatter and improper reaming.

Key Words

- center core drill
- chisel end
- chuck
- counterboring
- countersinking
- deep-hole drilling
- drill press
- drilling
- flute
- gang-drilling machine
- gundrill
- helix angle
- indexable insert drill
- jig
- lip
- multiple-spindle drilling machine
- radial drilling machine
- reaming, hand
- reaming, machine
- shell reamer
- spade drill
- spot facing
- subland drill
- tang
- thrust force
- trepanning
- turret drilling machine
- twist drill
- web

Review Questions

1. What functions are performed by the flutes on a standard twist drill?
2. What determines the rake angle of a drill? See Figure 23-2.
3. Basically, what determines what helix angle a drill should have?
4. When a large-diameter hole is to be drilled, why is a smaller-diameter hole often drilled first?
5. Equation 23-4 for the MRR for drilling can be thought of as ___ times ___ where \( f, N_r \) is the feed rate of the drill bit.
6. Are the recommended surface speeds for spade drills given in Table 23-3 typically higher or lower than those recommended for twist drills? How about the feeds? Why?
7. What can happen when an improperly ground drill is used to drill a hole?
8. Why are most drilled holes oversize with respect to the nominally specified diameter?
9. What are the two primary functions of a combination center drill?
10. What is the function of the margins on a twist drill?
11. What factors tend to cause a drill to "drift" off the centerline of a hole?
12. The drills shown in Figure 23-13 have coolant passages in the flutes. What is the purpose of these holes?
13. In drilling, the deeper the hole, the greater the torque. Why?
14. Why do cutting fluids for drilling usually have more lubricating qualities than those for most other machining operations?
15. How does a gang-drilling machine differ from a multiple-spindle drilling machine?
16. How does a multiple-spindle drilling machine differ from a NC drilling machine with a tool changer that would hold all the drills found in the multiple-spindle machine?
17. How does the thrust force vary with feed? Why?
18. Holding the workpiece by hand when drilling is not a good idea. Why?
19. What is the rationale behind the operation sequence shown in Figure 23-10?
20. In terms of thrust, what is unusual about the slot-point drill compared to other drills?

21. What is the purpose of spot facing?
22. How does the purpose of counterboring differ from that of spot facing?
23. What are the primary purposes of reaming?
24. What are the advantages of shell reamers?
25. A drill that operated satisfactorily for drilling cast iron gave very short life when used for drilling a plastic. What might be the reason for this?
26. What precautionary procedures should be used when drilling a deep, vertical hole in mild steel when using an ordinary twist drill?
27. What is the advantage of a spade drill? Is it really a drill?
28. What is a "pecking" action in drilling?
29. Why does drill feed increase with drill size?
30. Suppose you specified a drilling feed rate that was too large. What kinds of problems do you think this might cause? See Figure 23-6 and Table 23-4 for help.

**Problems**

1. Suppose you wanted to drill a 1.5-in.-diameter hole through a piece of 1020 cold-rolled steel that is 2 in. thick, using an indexable insert drill. What values of feed and cutting speed will you specify, along with an appropriate allowance. Is this the correct tool? What other drill types could be used?
2. How much time will be required to drill the hole in Problem 1 using the insert drill?
3. What is the metal removal rate when a 1.5-in.-diameter hole, 2 in. deep, is drilled in 1020 steel at a cutting speed of 200 fps with a feed of 0.010 ipr? What is the cutting time?
4. If the specific horsepower for the steel in Problem 3 is 0.9, what horsepower would be required, assuming 80% efficiency in the machine tool?
5. If the specific power of an AISI 1020 steel of 0.9, and 80% of the output of the 1.0-kW motor of a drilling machine is available at the tool, what is the maximum feed that can be used in drilling a 1-in.-diameter hole with a carbide drill? (Use the cutting speed suggested in Problem 3.)
6. Show how the approximate equation 23-5 for MRR in drilling was obtained. What assumption was needed?
7. A workpiece must have 10 holes finished in it. Manual layout time is 0.2 hr/piece. To drill and ream all the holes requires 1 hour on the machine for each piece, not counting layout or setup. The labor rate is $10/hr and the machine rate is $20/hr. If a jig is used, the labor cost to lay out each piece can be saved. Both methods give the same-quality product, but this jig saves 40 min in processing time on the machine. How large a lot justifies the use of a jig that costs $150 to make (labor and materials)?
8. A part has two holes located for drilling by manual layout. If a drill jig is used, 0.5 min in processing time is saved for each piece. The labor rate is $9/hr. The overhead rate on the labor saved is 100%. Setup time is no more than with out the jig. The combined rate for interest, insurance, taxes, and maintenance is 35%. The cost of the jig is $500.
   a. How many pieces must be made in one lot to make the jig worthwhile?
   b. How many pieces must be made on the jig in one lot each month to earn the cost of the jig in two years?
9. Manufacturer's charts will help determine the best feed and speed to run the drills. For example, a 1.5-in hole is to be drilled in 4140 steel annealed to Brinell 275. For the spade drill, speed is 80 sfm; feed, 0.009 ipr; and spindle rotation, 204 rpm. For the indexable insert drill, speed is 358 sfm; feed, 0.007 ipr; and spindle rotation, 891 rpm. Typically, an indexable insert drill can produce a hole four times faster than a spade drill but may cost (with inserts) 50% to 75% more than the equivalent spade blade and holder. For making only a couple of holes, the extra cost is not usually justified. Determine the number of holes needed to justify the extra cost of the indexable insert drill. Some additional cost data are given below.

   Ignore tool life and assume that the blades and the indexable drills make about the same number of holes. (Why is this a reasonable assumption?) The holes are 3 in. deep, with no allowance needed. Cost of drills:

<table>
<thead>
<tr>
<th>Spade drill</th>
<th>Indexable-insert Drill</th>
</tr>
</thead>
<tbody>
<tr>
<td>$139.00 holder</td>
<td>$273.00 drill</td>
</tr>
<tr>
<td>+21.50 per blade</td>
<td>+12.80 per two inserts</td>
</tr>
<tr>
<td>$160.90</td>
<td>$285.80</td>
</tr>
</tbody>
</table>

   Assume for this example that a machine rate of $45/hr includes the cost of labor and machine burden.

10. Assume that you are drilling eight holes, equally spaced in a bolt-hole circle. That is, there would be holes at 12, 3, 6, and 9 o'clock and four more holes equally spaced between them. The diameter of the bolt hole circle is 6 in. The designer says that the holes must be 45° ± 1° from each other around the circle.
   a. Compute the tolerance between hole centers.
   b. Do you think a typical multiple-spindle drill setup could be used to make this bolt circle—using eight drills all at once? Why or why not?
   c. Do you think that the use of a jig may help improve the situation?
   d. Do you think a CNC drilling process could do the holes best?
11. A part with seven holes can be machined on a numerically controlled turret drill press in 3 min (estimated time based on