

DRILLING, REAMING AND RELATED PROCESSES

DRILLING

The production of holes by drilling is one of the oldest and the most widely used of all machining processes. Holes are also produced by many other processes, including forging, casting, extruding, molding, and punching/piercing, as discussed in Volume II of this Handbook series; electrical discharge, electrochemical, laser, and electron beam machining, and other nontraditional processes, as discussed in Chapter 14, "Nontraditional Machining"; and flame cutting. Coverage in this section is confined to the production of holes by drilling.

THE DRILLING PROCESS

Drilling is basically the production or enlarging of holes by the relative motion of a cutting tool and the workpiece, which produces chips. The cutting tool, the workpiece, or both may rotate, with the tool generally being fed. Several different methods of drilling exist, including conventional, deep-hole, and small-hole drilling. The choice of a method depends upon the size, depth, tolerance, and finish needed; production requirements; and the machines available to perform the operations.

While drilling is fast and economical, its cutting action is difficult and inefficient. Cutting speed varies from a maximum at the periphery of the tool to zero at the center of the tool, thus varying the load on the cutting edges. Both chip ejection and flow of the cutting fluid are restricted in drilling. In addition, the production of small, deep holes can create problems with respect to necessary rigidity of the tools.

Machines Used for Drilling

Holes are often produced using a variety of drilling machines, as discussed later in this chapter. Considerable drilling, however, is also done on lathes and boring machines, discussed in Chapter 8, "Turning and Boring"; on automatics; on boring, drilling, and milling machines; on special and

transfer machines; and on machining centers, as discussed in Chapter 15, "Multifunction Machines."

When machining centers were first developed, many experts thought that drilling machines would become obsolete. Since then, however, it has been found that many hole-producing operations can still be done better and more economically on drilling machines, releasing machining centers for applications more suited to their capabilities (see Chapter 15).

Tools Used for Drilling

Cutting tools for drilling—drills—are rotary end-cutting tools having one or more cutting lips, and one or more helical or straight flutes for the passage of chips and the admission of a cutting fluid. They are available in a wide variety of types and geometries, as discussed later in this chapter. The most common are helically fluted twist drills with various shanks and tip geometries. Other types of drills include half-round, coolant-hole, spade, and indexable-insert drills. Gundrills, pressure-coolant drills, and trepanning tools are used extensively for precision and deep-hole operations. So-called pivot drills are used for small holes.

Cutting Action

Drilling is a complex three-dimensional cutting operation with conditions varying along the entire cutting edge. With twist drills, the rake angle normal to the cutting lip decreases from the periphery toward the drill center and cutting action improves along the cutting edge from the axis to the periphery of the drill. While the outer edge of the drill produces chips by shearing, workpiece material under the chisel edge of the tool is subject to more severe deformation, thus requiring greater thrust forces.

The limited chip space provided by a drill makes it desirable to have the chips produced in small

CHAPTER CONTENTS:

DRILLING	
Process	9-1
Accuracy	9-2
Machines	9-4
Twist Drills	9-14
Drill	
Geometries	9-40
Indexable-	
Insert Drills	9-48
Spade Drills	9-51
Coolant Drills	9-53
Toolholders	9-67
Workholders	9-69
Power	9-78
Speeds &	
Feeds	9-84
Cutting Fluids	9-89
Drill Recon-	
ditioning	9-93
Applications	9-102
REAMING	
Reamers	9-107
Reamers/	
Holders/	
Drivers	9-123
Workholding	9-125
Speeds &	
Feeds	9-127
COUNTER-	
BORES	9-134
SPOT-	
FACERS	9-139
COUNTER-	
SINKS	9-139

Contributors of sections of this chapter are: Robert H. Allen, Chief Engineer, Charles G. Allen Co.; Paul D. Bober, Sales Service Representative, Chicago-Latrobe Div., TRW, Inc.; Sebastian Ebenhoch, Supervisor—New Product Development, Waukesha Cutting Tools, Inc.; Geoffrey Y. Gill, Vice President, Muskegon Tool Industries; DeWitt F. Goodemote, CMfgE, P.E., Senior Manufacturing Engineer—Group Leader, Aerospace Div., Moog Inc.; Edward G. Hoffman, Technical Writer/Consultant, E. G. Hoffman Co.; Michael J. McGinty, Manager, Product Design & Application—Cutting Tools, Cleveland Twist Drill Co., Div. of Acme-Cleveland Corp.; Donald C. Noble, Senior Project Engineer, The Weldon Tool Co.; Roger J. Reddin, Burgmaster Div., Houdaille Industries, Inc.; A. J. Reitenbach, Supervisor, Metallurgical Quality Control, Cleveland Twist Drill Co.; J. K. Scholz, Vice President, Cleereman Machine Tool Div., Appleton Machine Co.; Ernest E. Torkilsen, Sales and Marketing Manager, Industrial Products Div., Dumore Corp.; Charles J. Trost, President, C.J.T. Industries, Inc.; Cyrus T. Wax, Manager, Engineering, Industrial Tools Div., The Bendix Corp.; Howard Whalley, Sales Manager, The George Whalley Co.; Jon A. Wolfe, Manager, Drilling Systems, Coromant Div., Sandvik Inc.; Lou Yane, Master Mechanic, Zagar, Inc.

THE DRILLING PROCESS

pieces. Coiling of the chips, especially in deep holes, causes packing of the drill flutes, interferes with chip ejection, and reduces the flow of cutting fluid to the drill tip. This causes excessive heat generation and premature dulling of the tool.

Effect of workpiece material. Ductility of the material to be drilled is a major factor in chip formation. When less ductile materials are being drilled, the chips tend to break into pieces; this is desirable and generally permits the use of lower cost, standard drills. When more ductile materials are being drilled, the chips tend to bend and coil and special tool designs and geometries must be employed to minimize this action.

Effect of feed rate. Chip thickness, which varies with the feed rate, also has a major influence on chip formation. Increasing the feed, which increases the chip thickness, minimizes the possibility of the flute clogging from coiled chips. Maximum feed, however, is limited by the structural strength of the drill and the capability of the machine. A woodpeckering or step-drilling technique—periodic withdrawal of the drill from the hole—may be required to remove chips when deep holes are being drilled, but this practice is not recommended for work-hardening materials.

SELECTION AND PERFORMANCE OF DRILLS

Selection of the type of drill to be used for a specific application depends upon many factors. The performance of a drill can be judged on the basis of efficiency (rate of penetration), accuracy maintained, surface finish produced, drill life, and most importantly, the cost per hole produced. These criteria vary with the type of drill used, the workpiece, the machine employed, the rigidity of the setup, the operating parameters, and the cutting fluid used.

Important variables that influence the selection and performance of drills are the material from which they are made and their physical properties, sizes, and geometries. Workpiece criteria include the material from which the workpiece is made, its physical properties and surface condition, the type of holes (diameter, depth, through or blind, etc.) to be drilled, production requirements (lot size, accuracy, and finish), and the rigidity of the setup.

Factors that should be considered with respect to the machine used include its design, condition, and accuracy. The operating parameters of cutting speed and feed rate play a

critical part in drill performance. Also important is the cutting fluid employed, including the type, rate of flow, temperature, method of application, and means for chip removal.

Accuracies and Finishes Attainable in Drilling

The accuracy of any hole produced by drilling depends upon many factors, including the type of drilling process, the workpiece, the tool and machine used, operating parameters, and rigidity of the setup. Accuracy is also largely dependent upon the sharpness and geometry of the drill point.

The accuracy of the drill itself, with respect to diameter and runout, is not too critical on machines on which drill bushings are used, but it is extremely important when the drills are to be used on numerically controlled and other precision machines. Standard, off-the-shelf drills may need regrinding for precision operations. Inaccurate grinding can produce unbalanced forces that may deflect the drill and result in hole errors.

Average accuracies. Because of the many variables, hole accuracies are almost impossible to specify precisely. The average accuracy of holes produced with twist drills are presented in Table 9-1. Closer tolerances than those listed can be maintained, but this generally means higher cost per hole produced.

One method of obtaining closer tolerances is to use drills with special tip geometries such as a four-facet grind. Another method is to use a starting hole (not center drilled) made with a stub-length drill of the same diameter as the follower drill and having a self-centering point. The starter hole should be deep enough so that the OD of the follower drill enters the hole before it begins to cut. Double-margin drills can also produce closer tolerances.

Common errors in hole geometry. Certain errors or imperfections can occur to various degrees in any drilling process because of common conditions (see Fig. 9-1). These include:

1. Errors in shape. This condition exists when the hole diameter is not uniform throughout the depth of the hole. Examples include bell-mouthed, ball-shaped, and concave holes, as well as crooked holes having inclined axes. Without special precautions and preparations, drilling can produce objectionable results such as these. The magnitude of the error often depends primarily on the diameter and length-to-diameter (L/d) ratio of the drill.
2. Burrs. A burr can form on the circumference of the

Reviewers of sections of this chapter are: Robert H. Allen, Chief Engineer, Chas. G. Allen Co.; Ronald C. Allen, Marketing Technician, Kennametal Inc.; James J. Arn, Domestic Sales Manager, Burgmaster Div., Houdaille Industries; Peter R. Arzt, Branch Manager, Manufacturing Research, McDonnell Douglas Corp.; Robert W. Berry, Chief Engineer, VR/Wesson Div., Fansteel, Inc.; Paul D. Bober, Sales Service Representative, Chicago-Latrobe Div., TRW, Inc.; D. J. Bohac, Sales Manager, Madison Industries, A Div. of Sandvik Inc.; William Breuning, President, R. B. Tool Co., Inc.; Clyde Brisk, Sales Engineer, Waukesha Cutting Tools, Inc.; Charles P. Brumder, Vice President-Marketing, Waukesha Cutting Tools, Inc.; Dr. Marvin Burnham, Associate Scientist, Research & Development, Rockwell International; George A. Clement, President, Welch Drill Bushing Co.; George Cleverly, Chief Engineer, DeHoff Div., W. H. Nichols Co.; John A. Cmar, Marketing Engineer, Erickson Tool Co., Subs. of Kennametal Inc.; Harry N. Cobb, General Manager, General Chipbreaker Drill Co.; J. A. Cupler, II, President, National Jet Sales Corp.; Ralph J. Dalusio, Service Manager - Research & Development, Eldorado Tool & Manufacturing Corp.; Douglas Doeden, President, Doler, Inc.; Colin Donaldson, Research & Development Manager, Mohawk Tools, Inc.; Richard Donarski, Product Sales Manager, Winslow Products, Giddings & Lewis—Bickford Machine Co.; Sebastian Ebenhock, Supervisor—New Product Development, Waukesha Cutting Tools, Inc.; Jack Fickers, Assistant Department Head, Shop Department, Los Alamos Scientific Laboratory; Dan Fogarty, Aro Corp.; Geoffrey Y. Gill, Vice President, Muskegon Tool Industries; DeWitt F. Goodemote, CMfgE, P.E., Senior Manufacturing Engineer-Group Leader, Aerospace Division-Manufacturing Engineering Dept., Moog Inc.; Herbert V. Gregg, Director of Engineering, Star Cutter Co.; Robert W. Hahn, Manager—Drilling Systems, The Valeron Corp.; Sazzadul Haque, Research Engineer, Carboloy Systems Dept., General Electric Co.; Edward G. Hoffman, Technical Writer/Consultant, E. G. Hoffman Co.; Werner K. Holl, Product Engineering Supervisor, Drill & End Mill Div., TRW Inc.; Thomas Hollis, Jr., President, The O.K. Tool Co., Inc.; Don Holt, President, Inmotion Quickchange Div., Diamond Tool & Die, Inc.; James N. Hood, Sales Manager, General Chipbreaker Drill Co.; Phillip Ireland, Application Engineer, Cutting Tool Div., The Ingersoll Milling Machine Co.; Charles A. Jorgensen, Chief Tool Engineer, Everede Tool Co.; Professor Ted Judson, General Motors Institute; Dr. Charles Kahng, Professor-Dept. of Mechanical Engineering, University of Colorado; Ray Kennedy, Tool Engineer, Cross-Fraser Div., The Cross Co.; Brian Kent, Vice President and General Manager, Radial Lip Machine, Inc.;

SELECTION AND PERFORMANCE OF DRILLS

TABLE 9-1
Average Accuracy of Holes Produced with Twist Drills

Drilling Condition	Diameter, in. (mm)					
	1/8-1/4 (3-6)		1/4-3/4 (6-19)		3/4-1 1/2 (19-38)	
	Oversize, in. (mm)	Location, in. (mm)	Oversize, in. (mm)	Location, in. (mm)	Oversize, in. (mm)	Location, in. (mm)
No center-drilled hole or bushing	0.003 (0.08)	±0.007 (0.18)	0.006 (0.15)	±0.008 (0.20)	0.008 (0.20)	±0.009 (0.23)
Center-drilled hole, no bushing	0.003 (0.08)	±0.004 (0.10)	0.003 (0.08)	±0.004 (0.10)	0.004 (0.10)	±0.005 (0.13)
With drill bushing	0.002 (0.05)	±0.002 (0.05)	0.003 (0.08)	±0.002 (0.05)	0.004 (0.10)	±0.003 (0.08)

(Metcut Research Associates, Inc.)

drilled hole at both the entrance to and the exit from the workpiece. Whether or not a burr forms and the height of the burr depends upon the workpiece and tool materials, their properties, the condition of the tools, and the cutting speed and feed rate used.

- Errors in hole locations. These occur when the centers of the drilled holes are not where they are supposed to be. The exactness of hole locations depend primarily upon the condition of the machines, tools, and cutting points. Approximate errors in location for various sizes of twist drills are given in Table 9-1.
- Errors in roundness. These errors occur when the roundness of a hole is distorted in comparison to its ideal form. Triangular, irregular, or oval-shaped forms are commonly observed when drilling. Accuracy of the spindle bearings on the machine, precise drill grinding, and rigidity of the setup (bushings, fixtures, etc.) are all critical factors if roundness errors are to be minimized. The roundness error may also be reduced by increasing the feed rate and using tools having a more rigid body.
- Errors in size. In most instances, the diameter of the hole produced is larger than the diameter of the drill. The approximate amounts of diameter oversize for twist drills of various sizes are presented in Table 9-1.

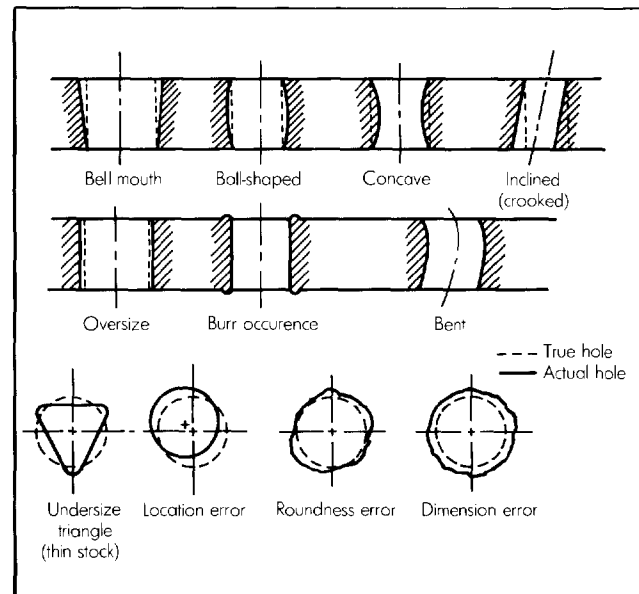


Fig. 9-1 Common errors in hole geometry experienced in drilling.

Reviewers, cont.: Edwin A. Klockowski, Senior Designer, Industrial Tools Div., The Bendix Corp.; Robert Koblesky, Technical Director, Cutting Tool Div., The Ingersoll Milling Machine Div.; Porter L. Laughlin, Supervisory Engineer, Industrial Tools Div., The Bendix Corp.; Robert S. Levin, Chairman of the Board, Louis Levin & Son, Inc.; Joseph J. Ley, Manager Technical Services, TRW Carbide Tools Div., TRW Inc.; Donald M. MacKay, Director of Research & Development, Union/Butterfield Div., Litton Industrial Products; Joe Martin, Assistant Manager, Sales Service Engineering, Morse Cutting Tools Div., Gulf & Western Manufacturing Co.; Mel Matusz, Project Engineer, Cross-Fraser Div., The Cross Co.; Donald J. McCarthy, Senior Product Engineer, The Jacobs Manufacturing Co.; Michael J. McGinty, Manager, Product Design and Application-Cutting Tools, Cleveland Twist Drill Co., Div. of Acme-Cleveland Corp.; Jan F. Miller, President, Miller Precision Jet Co., Inc.; J. H. Nymberg, Jr., Vice President and General Manager, United States Drill Head Co.; Donald C. Noble, Senior Project Engineer, Engineering Department, Weldon Tool Co.; Gene Orth, Director of Engineering, Waukesha Cutting Tools, Inc.; Carl J. Oxford, Jr., Vice President-Technology, National Twist Drill and Tool Div., Lear Siegler, Inc.; Ed Phillips, Jr., Plant Manager, National Jet Companies; Lee S. Pinkard, Jr., Product Marketing Manager, Power Tool Div., Ingersoll-Rand Co.; Donald Pratt, Plant Manager, Muskegon Tool Industries; Guy Priest, Vice President, Mohawk Tools, Inc.; Heinz Reich, President, Micro Technology Dept., Minitool, Inc.; James Rozell, Sales Manager, American Heller Corp.; Peter S. Rudd, Vice President-Sales, Eldorado Tool & Manufacturing Corp.; J. K. Scholz, Vice President, Cleereman Machine Tool Div., Appleton Machine Co.; James W. Seppala, Supervisor, Process Engineering, Detroit Diesel Allison Div., General Motors Corp.; Patrick J. Sharkus, Sales Engineer, Davis Tool Company, A Division of Giddings & Lewis, Inc.; Keith H. Smith, Marketing Manager, Greenleaf Corp.; Richard Sollich, Vice President, American Heller Corp.; John A. Stambach, Sales Engineering, National Automatic Tool Co.; Thomas C. Stokes, Chief Engineer, Star Cutter Co.; James Strolberg, Manager, Product Applications, Cutting Tool Div., The Ingersoll Milling Machine Co.; Robert E. Thaxton, Supervisor, Tool Design, Detroit Diesel Allison Div., General Motors Corp.; Charles J. Trost, President, C.J.T. Industries, Inc.; John W. Turton, Manager, Research & Development, Industrial Tools Division, The Bendix Corporation; John Von Hollen, President, Von Hollen Tool Co.; Cyrus T. Wax, Manager, Engineering, Industrial Tools Div., The Bendix Corp.; Charles Wells, Manufacturing Engineering Supervisor, Drill & End Mill Div., TRW Inc.; Matthew J. Wilding, Product Designer, Cleveland Twist Drill Co.; Jon A. Wolfe, Manager, Drilling Systems, Coromant Div., Sandvik Inc.; Louis Yane, Master Mechanic, Zagar, Inc.; Frank G. Zagar, President, Zagar, Inc.

CHAPTER 9

DRILLING MACHINES

Surface finishes produced. The surface finishes produced by drilling generally range from about 100-250 μ in. (2.54-6.35 μ m) or more. The many variables, similar to those previously discussed for accuracies, make it impractical to predict the actual finish attainable in different applications. The surface finish of many drilled holes is often improved by subsequent reaming, boring, grinding, honing, or roller burnishing.

Effect of Drilling on Workpieces

Drilling has little effect on the physical properties of the workpiece. There is, however, a thin layer of highly stressed material around the drilled hole, which is often removed by subsequent machining. If this material is not removed, the surfaces may be more susceptible to corrosion. Clamping of workpieces for drilling may also result in distortion.

DRILLING MACHINES

Machines designed specifically for drilling are available in many different types, sizes, and capacities. The types include light-duty (sensitive), heavy-duty upright, radial, gang, multi-spindle, turret, deep-hole, small-hole, and special purpose machines. In addition to drilling, many of these machines can also perform related operations, such as reaming, facing, chamfering, counterboring, countersinking, undercutting/recessing, roller burnishing, and tapping. In some cases, depending upon the design and rigidity, the machines are used for boring and milling operations.

Machine Requirements

Selection of a type of drilling machine for a specific application depends on many variables. Workpiece size, geometry, and material; production requirements (quantity, accuracy, and finish); and economic factors are major considerations.

The spindle, which holds the cutting tool, is probably the most important single component of a drilling machine. High-quality, rigid, accurate spindles are essential, especially for precision drilling. The spindle axis must be parallel to the machine ways (the direction of feed travel); any misalignment multiplies errors with increasing stroke length. Accuracy, along with a minimum of vibration, is also essential for the entire machine.

Power requirements for drilling machines depend primarily upon the diameters and depths of the holes to be produced, as well as the material to be drilled. Electric motors on these machines range from fractional horsepower to 75 hp (56 kW) or more. Details of the power required for drilling are discussed in the section on operating parameters later in this chapter.

A positive-feed mechanism (using a leadscrew, ballscrew, cam, rack, hydraulic cylinder, or other device) is recommended to provide consistently steady advancement of the cutting tool. This is especially critical when small diameter or deep holes are being drilled. Any surge or jump may break the tool because of too heavy a chip load. Hand feeding is used for most light-duty sensitive machines, but power feeding is employed for most other drilling machines.

Machines with high-production capabilities are desirable for many applications. Versatility, however, can be equally important when the requirements are for a wide variety of workpieces and hole sizes.

The tables on drilling machines should be large and sturdy to avoid deflection under cutting pressures. A backup bar at the

rear of the table is sometimes desirable to help control the workpieces. Wear plates, generally a machined cast-iron plate the size of the table and bolted to it, are sometimes provided on machines subject to rough usage. These removable plates can be easily restored to their original accuracy when necessary.

Cutting fluid systems may be required, depending upon the applications for which the machine is intended.

Control Systems

Drilling machines are available for manual, semiautomatic, and automatic operation. NC and CNC are used on many drilling machines, and these types of control are particularly suitable for producing patterns of holes in various workpieces. NC or CNC is usually standard on circuit board, tube sheet, and other special-purpose drilling machines. Programmable controllers are also used on some drilling machines and drillheads.

NC/CNC requirements for drilling machines vary, depending upon the application. Simple two-axis (X and Y) positioning systems are used only for table movement on some machines, with the drilling depth (Z axis) controlled manually, electrically, electromechanically, or mechanically. Many drilling machines are equipped with more complex systems that can control table positioning, drilling depths, spindle speeds, feed rates, and other functions. Some systems provide for the compensation of varying tool lengths. A comprehensive discussion of NC/CNC systems is presented in Chapter 5, "Machine Controls."

Cincinnati Milacron offers a continuous monitoring control system for its NC machining centers and special drilling machines. This system automatically lowers the feed rate whenever torque on the drill reaches a preset limit or motor power exceeds the rated amount. Advantages include faster hole production with the same tool life, reduced tool breakage, improved chip-breaking action, and protection of the motor from overloads.

This closed-loop control system continually derives torque at the drill from measured horsepower and spindle speed. The calculated cutting torque is compared to the breaking torque punched into the NC tape. Whenever the cutting torque exceeds one-half the preset limit, the feed rate is automatically reduced. The feed rate is also lowered whenever the horsepower required exceeds the rated motor capacity.

Macotech Corp. offers a retrofittable adaptive control for NC drilling machines and machining centers. This is a microprocessor-based system with the capability for combining soft-wired modules for drilling, milling, boring, and tapping in the same unit. The drilling unit uses torque as a feedback.

Light-Duty, Sensitive Drilling Machines

Machines of this type are the most common of all drilling machines. These general-purpose machines are most often used for drilling one hole at a time in small workpieces, with hole diameters to about 1" (25 mm) diam. They are often referred to as drill presses and are employed extensively for many toolroom, machine shop, and maintenance applications, as well as for some production operations.

The typical light-duty drilling machine (see Fig. 9-2) has a vertical, base-mounted, round column; a table that can be adjusted vertically and tilted; and a powered spindle that holds the cutting tool. The round column allows the table to be swung aside when not required. These machines are available in both floor and bench models.

Most light-duty drilling machines are manually fed with a rack-and-pinion mechanism and a hand-operated lever, but

DRILLING MACHINES

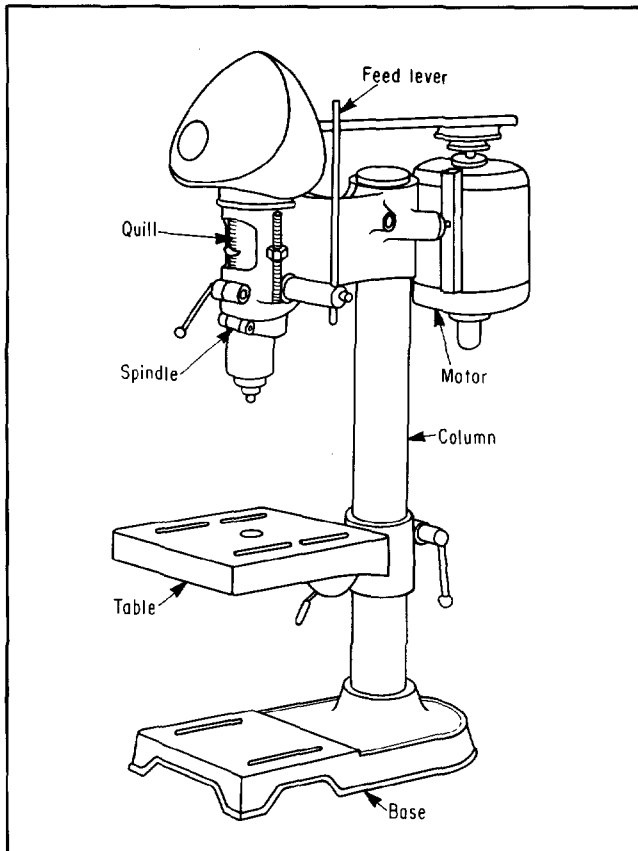


Fig. 9-2 Light-duty, sensitive-type drilling machine with hand feed.

automatic feeds are available on some models. Hand feeding is the reason these machines are often referred to as sensitive types; it permits the operators to change feed rates as they sense soft or hard spots in the workpieces from the feel of the cutting action.

The spindles on these machines are generally belt driven, with either stepped or variable-pitch pulleys for speed changes, but geared-head machines are also available. Multispindle attachments, usually with up to about 12 spindles, can be used on these machines for increased production requirements. Spindle noses and adjustable adapters for multispindle drilling heads are specified in ANSI Standard B5.11-1954 (reaffirmed 1973), published by ASME.

Upright (Vertical) Drilling Machines

Upright or vertical drilling machines, such as the one illustrated in Fig. 9-3, are similar to the light-duty machines just discussed, but differ in that they are more massive for heavy-duty applications. They permit the production of larger diameter and deeper holes with improved accuracy and quality. Practically all upright drilling machines are equipped with power feed. They are most suitable for workpieces that can be quickly positioned under the tool, require short cycle times, or need only a few holes per part.

Some upright drilling machines have a round column, while others (see Fig. 9-3) have a box column for increased rigidity. Machines with round columns generally have circular tables that can be rotated about the columns and their own centers, as well as raised and lowered. Most box-column machines are

equipped with square tables, some of which have knee-type supporting members and one or two screws for positioning the tables vertically. Power elevating systems are available on some machines. Most tables are furnished with T-slots for clamping drilling fixtures or workpieces.

Cross compound tables are also available with scales and/or dials to facilitate drilling holes in various locations. More sophisticated versions, sometimes called layout drilling machines (see Fig. 9-4), are equipped with precision slides for the saddle and table, and digital readout systems. These machines can also be used for precision boring and milling.

Most box-column upright machines are designed so that their upper halves can be removed for shipping and alignment purposes. These units above the base assemblies are called top columns and carry the spindle drive and some of the feed gearing. A counterbalanced sliding-head assembly, which carries the spindle and feed mechanism, is clamped to dovetail ways on the top column.

Types of spindle drives. Various spindle-drive systems, all suitable for drilling, are used on upright machines. These systems include the following:

1. Four-speed motor drive. The motor is mounted above and in line with the spindle or is belted to the spindle. Speeds are quickly and easily changed by the operator who can stand at floor level. This is a good system for a tapping or reversing drive, using the lower motor speeds, and is suitable for toolroom or medium-duty manufacturing. Horsepower available, however, is limited for heavy-duty drilling.

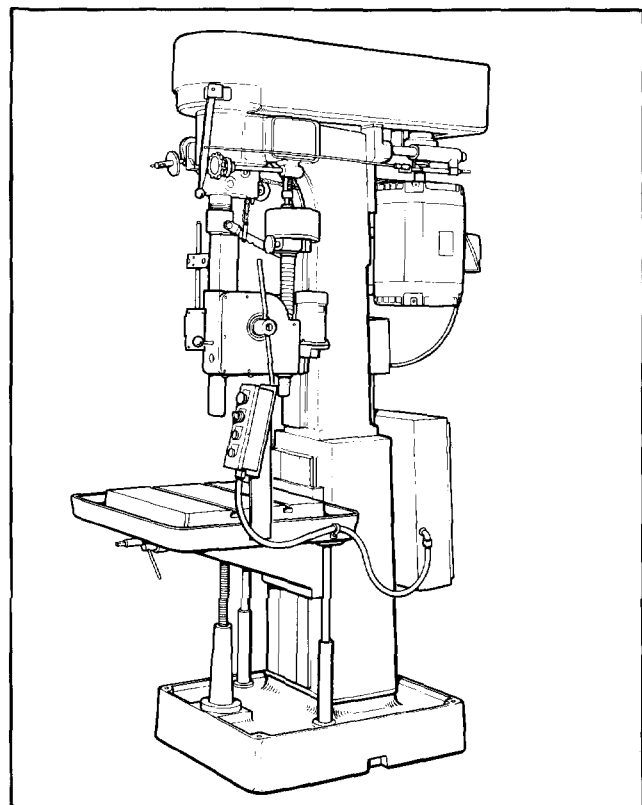


Fig. 9-3 Upright (vertical) drilling and tapping machine with power feed and power traverse. (Chas. G. Allen Co.)

DRILLING MACHINES

2. Multiple V-belt drive. This is a single-speed drive, generally used on single-purpose machines. It is a good reversing drive and can transmit high horsepower.
3. Step-pulley V-belt drive. Speed changes with this drive are slow because the operator has to climb a ladder to make a change. It is, however, a good general-purpose and reversing drive, and can transmit high horsepower.
4. Ten-step, quick-change V-belt drive. This drive system provides ten speeds, arranged in a geometric progression, that can be quickly and easily changed from floor level. It is a good general-purpose drive for drilling and tapping and can transmit high horsepower.
5. Variable-speed drive. With this drive system, speeds are quickly and easily changed at floor level. This system provides a good general-purpose drive for drilling. It is not, however, a good reversing drive because of the inertia of the belts and pulleys and because only moderate horsepowers can be transmitted.
6. Gearbox drive. With this general-purpose drive, four or more speeds can be belted or geared to the spindle and speed changes can be made at floor level. Low-speed torque is excellent, high horsepowers can be transmitted, and it can be used for moderate-duty reversing.
7. Back-gear drive. This system, usually with a reduction from 4:1 to 6:1, doubles the number of speeds available from the various spindle drives. It also extends their range to lower speeds for large slow-running tools that require increased torque.

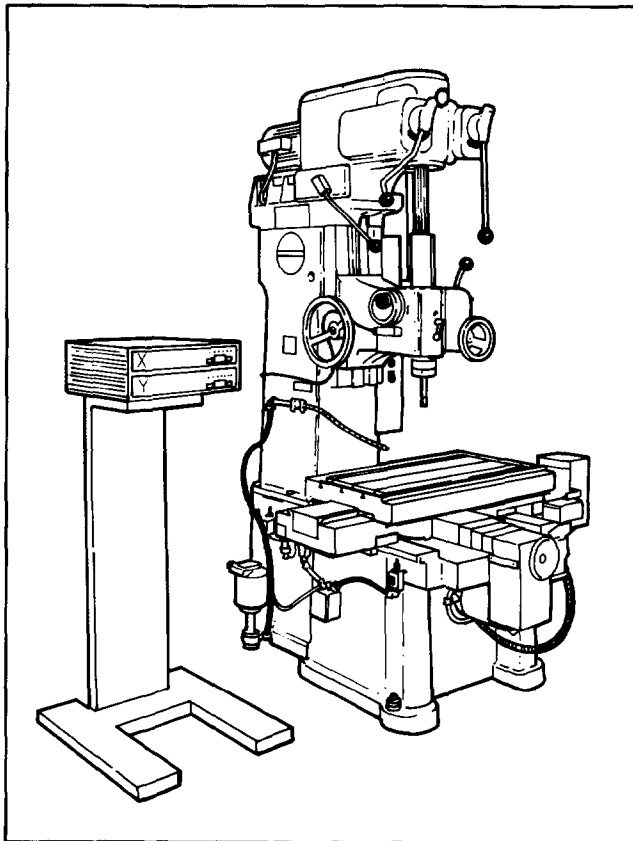


Fig. 9-4 Bed-type layout drilling machine with digital readout system. (Cleereman Machine Tool Div., Appleton Machine Co.)

Power-feed systems. Power-feed sliding heads for upright drilling machines are practically all the mechanical type, which provides a constant feed rate. This type of head can be quickly changed for various applications with no modification in the structure of the head or the gearing. The heads should provide at least three different feed rates, and more feeds are often desirable.

With a semiautomatic feed mode, the spindle is usually fed manually to within close proximity of the workpiece. Then, the power feed is engaged manually or by means of a trip mechanism. The tool continues feeding into the workpiece until a depth trip or stop is contacted, at which point the power feed is disengaged and a counterweight returns the spindle to its starting position.

For automatic operation, power traverse can be provided by mounting an electric motor on the sliding head and gearing it to the power-feed mechanism. When the cycle-start pushbutton is pressed, this motor rapidly traverses the spindle until an adjustable limit switch is tripped. At this point, the traverse motor stops and the power-feed mechanism carries the tool to the required depth. At this point, the power feed is disengaged and the spindle is rapidly returned to its starting position. The rate of power rapid traverse is about 200 ipm (5080 mm/min).

Remote control of power feed and traverse is possible for increased operator safety, with initiation on the closing of a jig, seating of a pin, or from a cycle-start button. When the power traverse and an adjustable timer with suitable controls are used, the woodpeckering technique (periodic withdrawal of the tool) can be employed. The distance drilled (step) before withdrawal is a function of the timer setting.

Also available is an automatic adjustable dwell device that keeps the tool from advancing at the end of the preset feed stroke for a length of time set on a dwell timer. This eliminates feed marks when counterboring or spotfacing is being done, and provides precision depth control within $\pm 0.0005"$ (0.013 mm).

Other attachments. A lathe-type feed or lead gearbox can be mounted on the feed shaft entering the power-feed head. This attachment provides a selection of up to 48 feed rates. Used in conjunction with tapping controls, it establishes the lead of the tap and is helpful on materials in which taps are hard to get started, as well as to the operators who are inexperienced. Tapping controls can be used with either hand or power-feed sliding heads.

Cam-feed sliding heads are used for high-production applications in which each cycle must be exactly the same. All characteristics of the cycle are incorporated in the cam profile, which guides the spindle. A different cam and possibly a gearing change are required for each application, resulting in added costs which prohibit the use of these heads for general-purpose applications.

When multispindle drillheads or bushing plates are used on upright machines, their weight may exceed the lifting capacity of the spindle counterweight system. Air-actuated counterbalance units are available to provide additional lift. With these units, the lifting capacity of the machine can be varied by adjusting the air supply pressure.

Gang Drilling Machines

A gang drilling machine consists of two or more independent, light-duty or upright drilling machines mounted on a common base or table. One machine with six upright spindles is illustrated in Fig. 9-5. The table on this unit is power elevated by two screws.

DRILLING MACHINES

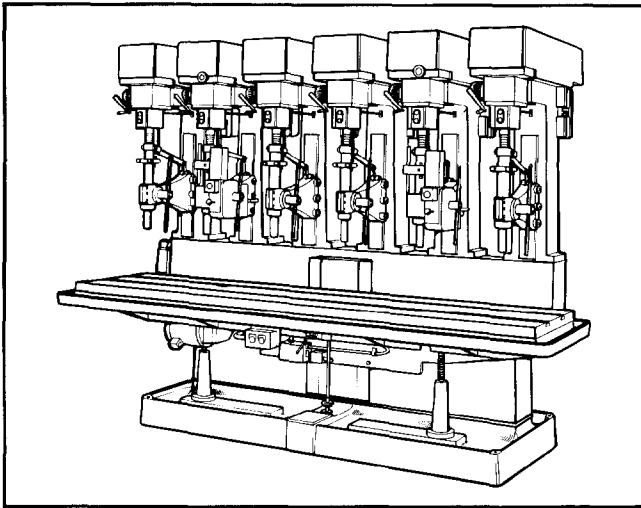


Fig. 9-5 Gang-type, six-spindle drilling machine with both hand and power-feed top columns equipped with power elevating table. (Chas. G. Allen Co.)

Gang drilling machines permit higher production rates on workpieces which have multiple holes or require multiple operations by saving time that would be required to change tools on single-spindle machines. Each spindle on a gang drilling machine can hold the same tools for simultaneous operations on a number of workpieces. More often, however, a different size or type of cutting tool is mounted in each spindle for sequential operations. In these cases, the workpieces are manually moved from one spindle to the next. Depending upon

the application, one operator can handle operations using several spindles or individual operators can be employed for each spindle.

The individual top columns on a gang drilling setup do not have to be the same. Some might be mounted on risers to provide more room under their spindles; some may be hand fed; and others may be power fed. The right-hand top column in the gang is often a hand-feed type and provided with tapping controls. Individual top columns may have fixed center-to-center locations or the design may allow for repositioning the top columns on a slide. Multispindle drillheads can also be mounted on any of the spindles.

Radial Drilling Machines

Excellent versatility is an important advantage of radial drilling machines. These machines are used extensively for drilling holes in large and irregularly shaped workpieces that cannot be easily positioned or repositioned. A number of smaller workpieces can also be clamped to the base or floor plate. Flexibility of these machines permits drilling holes in workpiece surfaces that cannot be reached with other drilling machines. Time is saved by repositioning the drill instead of the workpiece for each operation.

A radial drilling machine (see Fig. 9-6) consists of a horizontal arm, from 2-12 ft (0.6-3.7 m) or more in length, supported by a round, vertical column. The arm can be raised, lowered, and rotated around the column axis. The drillhead mounted on the arm contains the speed-changing and power-feed mechanisms, as well as the spindle, and can be repositioned along the arm and clamped in any desired location. Some machine designs permit tilting the head to produce angular holes.

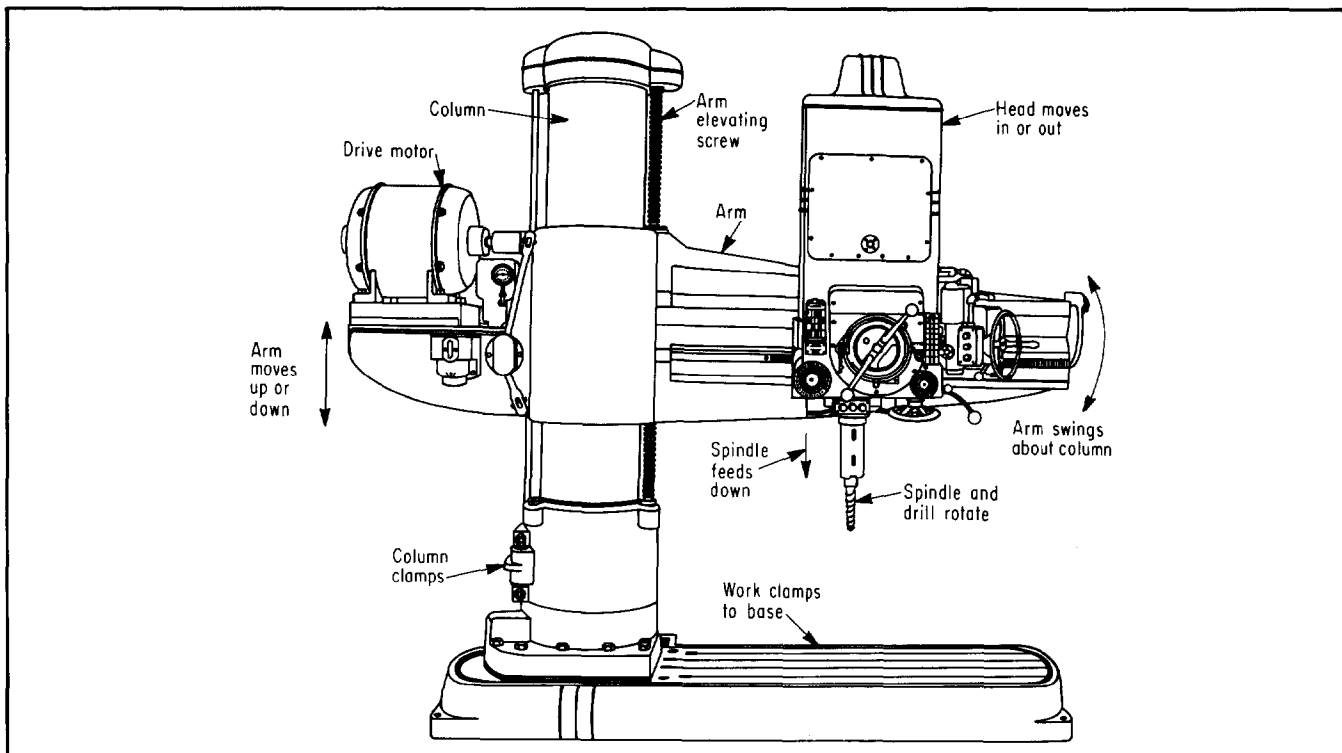


Fig. 9-6 Radial drilling machine. Arm which carries adjustable-position drillhead can be raised, lowered, and pivoted around column.

DRILLING MACHINES

Multispindle Drilling Machines and Heads

In addition to multispindle drillheads for use on single-spindle machines, multispindle drilling machines are available for high-production requirements. Major time savings can be realized with both heads and machines having multiple spindles by performing a number of machining operations simultaneously and minimizing the need for toolchanging. Multispindle machines are used primarily for three general types of production operations:

1. Multiple operations (drilling, reaming, chamfering, spot-facing, etc.) in a single hole. Machines used for these applications are often equipped with hand-positioned tables, shuttle tables, or rotary indexing tables.
2. One operation in multiple holes which are the same size or different sizes and on the same or different planes. Machines used for these operations may require a rotary indexing table if hole center distances are close. Multiple-plane operations are often performed with multiple-position workholding fixtures.
3. Multiple operations in multiple holes which generally require that the machine be equipped with a rotary indexing table or other type of table, especially when tapping is one of the operations to be performed.

Multispindle machines. While some of these machines are sold as standard models, many are designed for a specific workpiece or family of parts. They are built with from 2-100 or more spindles on one head, all powered from a single drive which is usually an electric motor. Hydraulic motors are used on some machines. Power is distributed to the individual spindles through gears or a gearless crank drive, discussed in the next section on drillheads.

Multispindle machines are made in various types, including an open-side type with way or quill design and a four-post type with vertical, horizontal, and angular construction. An open-side machine with way design is illustrated in Fig. 9-7. Machines of way or four-post design, with drive motors to 100 hp (74.6 kW), are suitable for heavy-duty drilling. The way and four-post machines may have the drillhead movable and the workholding fixture stationary, or they may have the fixture movable and the drillhead stationary. Multispindle drilling generally requires slightly greater hole tolerances than can be obtained with single-spindle machines.

Multispindle drillheads. These units are designed for various applications and have geared or gearless (crank-type) drives. All tools on a head are fed into the workpiece together, but tool lengths are sometimes staggered so that cutting loads are applied progressively.

Three major types of geared drillheads exist. These are as follows:

1. Adjustable-arm, universal-joint drillheads which are generally used for frequent changeover or limited production requirements. They provide maximum flexibility because the spindles can be adjusted to any desired location within the head housing. Spindles can be removed or added, up to the capacity of the head. The adjustable-arm spindles are rotated by universally jointed connecting rods that transmit power from a fixed-center, gear or crank-driven head. All spindles rotate at the same speed, which is normally limited to a maximum of 2000 rpm primarily because of joint wear problems and the method of lubrication. Some standard machines come equipped

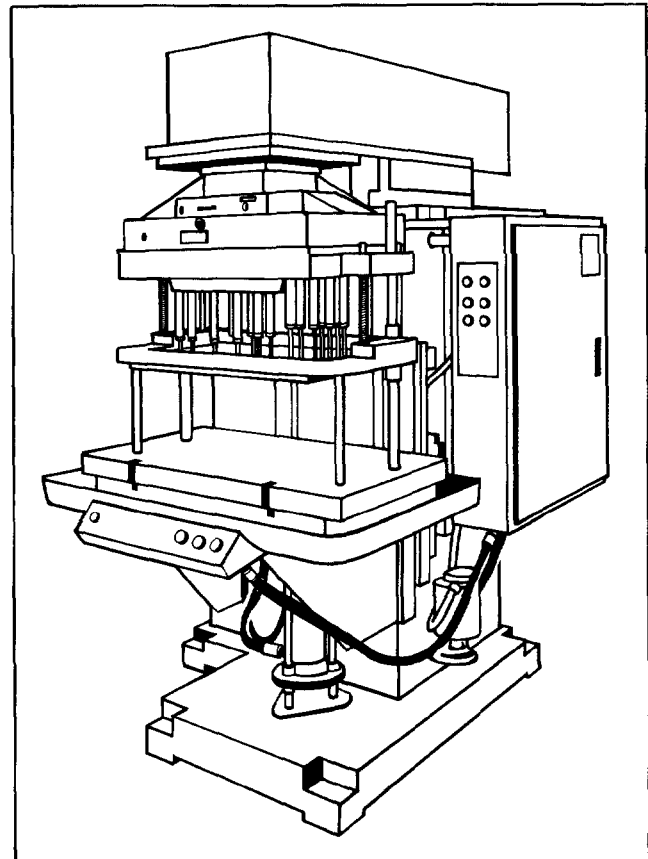


Fig. 9-7 Multispindle drilling machine of the open-side type with way design. (Zagar, Inc.)

- with universal-joint heads having a high, neutral, and low-speed shifter range for each spindle. Special heads can be designed to incorporate this speed selector feature.
2. Slip-plate drillheads which are often used for repetitive hole patterns and longer production runs. The same type of universal rods as those used on adjustable-arm heads drive slip spindles mounted in a slip plate that can be changed for different hole patterns. The capability of producing various hole patterns on fixed centers can make slip-plate heads more desirable than fixed-center heads (discussed next), but greater center distances are sometimes required between spindles and locational accuracy of the spindles is not as close. The spindles normally rotate at the same speed unless the head is of special design. These heads are generally used for drilling holes $1/16$ to $2 1/4$ " (1.6-57 mm) or more in diameter in cast irons and steels.
3. Fixed-center drillheads which are used for dedicated, high-production machines. The number of spindles can vary from 2-2000 or more, depending upon the number of holes required in the workpieces or the ease with which the workpieces can be handled. Gearless heads are required for large numbers of spindles, and all spindles operate at the same speed for light-duty operations. These heads are stronger and more rigid than either the adjustable-arm or slip-plate head because the spindles are permanently fixed for required hole patterns.

DRILLING MACHINES

Gearless, fixed-center, crank-driven drillheads are used when the need for close-center holes makes the use of gears and universal joints impractical or impossible. Some heads of this type are capable of drilling holes with center distances as close as 0.19" (4.8 mm) and diameters generally ranging from 0.020-0.750" (0.51-19.05 mm). In one interesting application, a gearless drillhead having 1800 spindles is mounted on the ram of a punch press to drill acoustical tiles. A 50 hp (37.3 kW) electric drive motor is mounted on top of the press.

One design of a gearless drillhead is illustrated in Fig. 9-8. The driver, *A*, in the drilling machine spindle rotates the drive crank, *B*, in the drillhead. This crank moves the oscillator, *C*, in a short oscillating motion. The oscillating motion rotates the individual drill spindles, *D*, in the same direction and at the same speed as the drive crank, *B*.

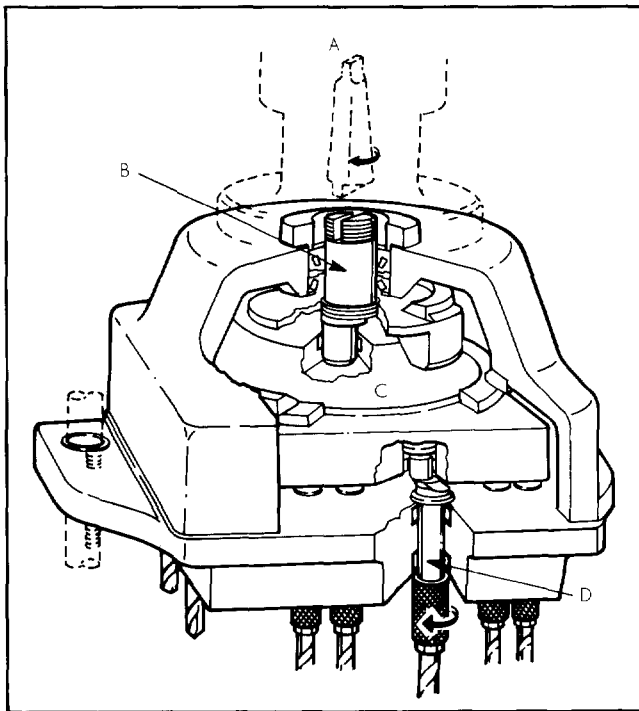


Fig. 9-8 Gearless drillhead in which an oscillator, *C*, rotates drill spindles, *D*. (Zagar, Inc.)

Turret Drilling Machines

Upright drilling machines, both bench and floor types as well as hand and power-feed types, are available with indexing drums or turrets. The turret typically has six or eight faces, although machines are available with turrets having four and ten faces. Each face has a spindle for holding a drill or other cutting tool. The turrets can be indexed manually or automatically to bring the spindles into operating position with respect to the workpiece. A solid-bed, sliding-head drilling machine with a turret having eight tooling faces and two/three-axis NC is shown in Fig. 9-9. Some NC turret machines are classified as machining centers, discussed in Chapter 15, "Multifunction Machines."

Turret drilling machines permit performing a number of operations in a hole or group of holes without the need for changing tools. Various tools on the turret are sequenced into cutting position with every feed stroke of the machine. Turret

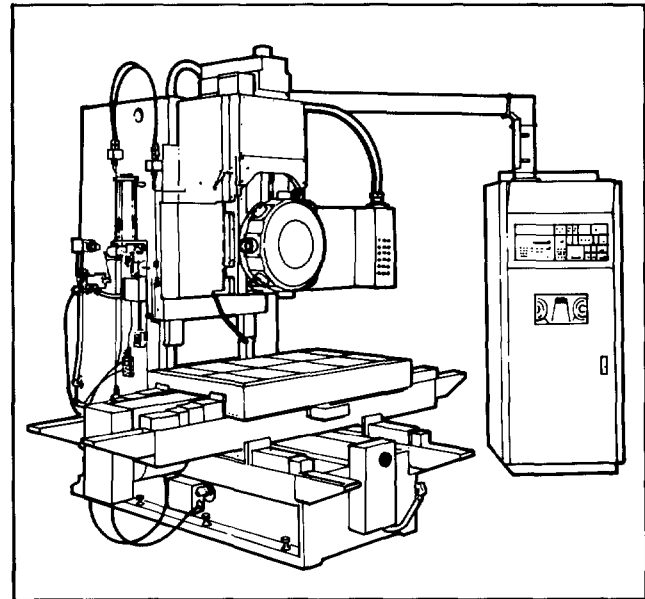


Fig. 9-9 Solid-bed, sliding-head drilling machine with eight-station tooling turret and two/three-axis NC (Burgmaster Div., Houdaille Industries)

drilling machines are suitable for heavier workpieces that can remain stationary, as well as for operations in which quick-change tooling is not practical. The capability of these machines can be expanded by adding indexing, rotating, or shuttling fixtures.

These machines are also made with dual (pancaked) turret heads for small workpieces having many multiple-operation holes. This arrangement reduces the need for toolchanging and two symmetrical workpieces can be machined simultaneously; thus, cycle time is cut in half. Drilling capacity per head, however, is reduced to half that when a single head is used because the spindle drive is not increased.

A manually fed, bench-type drilling machine of tubular column construction, with a six-station turret, is illustrated in Fig. 9-10. Spindles in the turret head of this machine are driven by helical gears through a clutch and variable pulley drive transmission from a two-speed a-c motor supplying four speeds in three ranges. Turret head indexing is accomplished by a Geneva mechanism mounted on the back of the head, which is actuated at the top of its stroke by a tripper pin engaging a drive dog in the camshaft. A counterbalanced, manual-feed lever is provided on the side of the turret head.

Automatic models are essentially the same except for the addition of power-feed mechanisms and control stations. On heavy-duty machines with higher horsepower motors, cast iron columns are used instead of tubes and the carriages slide on fully engaged ways. Manual or power-operated movement of the carriage is available. NC machines have various size, automatically controlled, positioning tables and can have two or three-axis control. Traveling-column versions, with the column moving along ways in the X axis, are built for handling long workpieces.

Deep-Hole Drilling Machines

Deep-hole drilling with gundrilling, gunboring, trepanning, and other self-guided tools using high-pressure cutting fluids is sometimes done by converting conventional machines such as

CHAPTER 9

DRILLING MACHINES

suitably designed lathes and drilling, boring, and milling machines. Most applications, however, are performed on machines specifically designed for deep-hole drilling. These machines are generally horizontal, but some, used for shorter operations on smaller workpieces, have vertical or angular spindles.

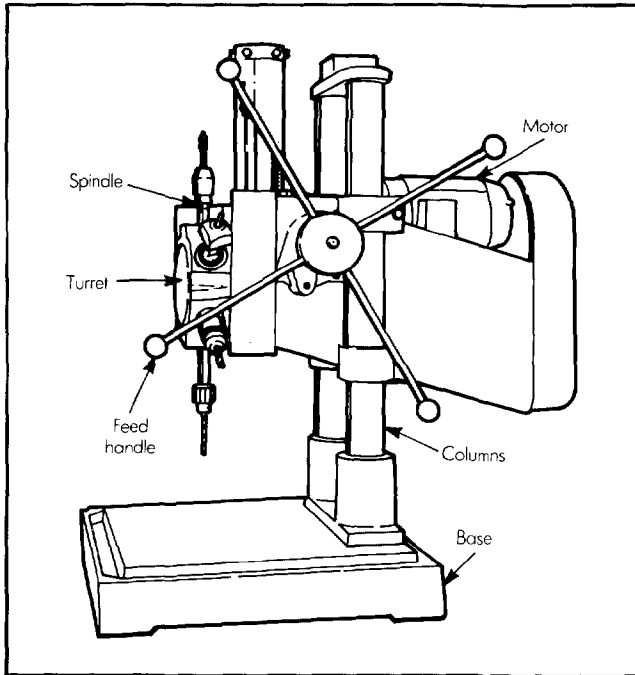


Fig. 9-10 Manually fed, bench-type turret drilling machine of tubular column construction. (Burgmaster Div., Houdaille Industries)

Most deep-hole drilling machines have a rotating tool that is fed into a workpiece mounted on a table at one end of the machine (see top view in Fig. 9-11). Some machines, used to drill long slender parts, rotate the workpiece while a nonrotating drill is fed into it (bottom view, Fig. 9-11). For some precision applications, both the workpiece and tool rotate, but in opposite directions.

Deep-hole drilling machines have several basic requirements which are not necessary on machines used for the more conventional hole-making processes. For optimum tool performance, the machine must:

1. *Be sufficiently rigid* so that moving parts which affect alignment do not deviate from their true path as the machine operates at its designated capabilities and capacity.
2. *Have ample power*, especially in view of the relatively high penetration rates associated with these tools.
3. *Have a system of controls that make precision work feasible*. The most desirable controls include a spindle-load meter, a feed-rate meter, a fluid-pressure gage or pressure switch, and a fluid-flow-control meter.
4. *Have a precision spindle*. The spindle must be designed to operate at the highest speed anticipated in use with an absolute minimum of end play. These machines are available with spindle speeds up to 20,000 rpm. Since very fine feed graduations are characteristic of pressure-coolant tools, spindle end play can result in the tool cutting too deep in one revolution, and then not cutting for the next several revolutions.
5. *Have a feed mechanism that is constant*, finely enough controlled to accommodate the expected range of conditions, and readily adjustable.
6. *Have a start bushing setup that allows near-perfect alignment*.

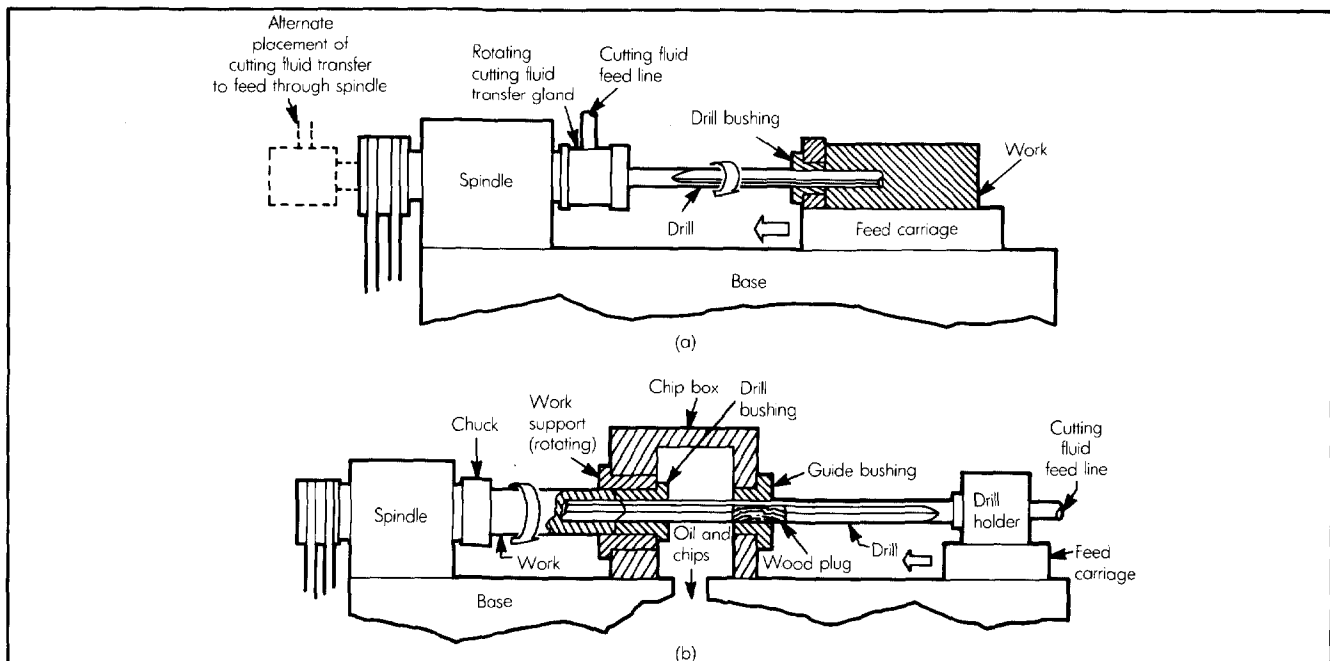


Fig. 9-11 Two methods of gundrilling: (a) tool rotating and workpiece stationary; and (b) workpiece rotating, tool nonrotating. (Eldorado Tool & Mfg. Corp.)

DRILLING MACHINES

7. *Be capable of close-tolerance alignment that can be easily set up and then maintained.* Shops doing precise gun-drilling may require a maximum of 0.0002" (0.005 mm) eccentricity per foot, measured from the spindle.
8. *Have a cutting fluid and fine-filtration system.*

Machine motions. The simplest type of machine, the fixed-table, advancing-spindle machine (see Fig. 9-12), is generally used for general-purpose applications. The fixed table requires a fixture to hold and position the workpiece.

Advancing-spindle machines are also available with two-axis tables, which travel vertically and horizontally, perpendicular to the tool, and with three-axis tables, which travel vertically and horizontally, both parallel and perpendicular to the tool. The way-type, advancing-spindle machine has the spindle mounted on a column for vertical positioning. The table positions horizontally, parallel, and perpendicular to the tool. Machines are also available with a fixed spindle and a table that advances toward the tool on ways. This type of machine is economical when loading time is short relative to cutting time. The basic machines are also available with automatic-control equipment for high production and more than one spindle.

Trepanning machines. Since trepanning starts at hole diameters where the high-pressure coolant drills leave off, trepanning usually requires a larger machine. Trepanning may be done on any one of several types of machines modified in varying degrees to suit the process. For relatively shallow holes, engine or turret lathes, horizontal or vertical boring mills, or heavy-duty drill presses are used. For deeper holes, engine lathes and some deep-hole drilling machines are suitable. Whatever type of machine is used, it must be extremely rigid and should have infinitely variable and independent feed control.

Coolant systems. Gundrilling, trepanning, and other pressure-coolant tools for deep-hole drilling require a coolant system with the machine which includes a pump, a filtering unit, and sometimes a water chilling or refrigeration system.

Two general types of pumps are used with high pressure coolant tool systems: those with a fixed volume and those with a

variable volume. The fixed type costs less initially, but is economical only for long-run high-production situations in which fluid-volume requirements are relatively unvaried. Using a fixed-volume pump, for example, on a job requiring a volume below capacity would require that the excess volume be dumped through a bypass valve. This in turn would require a very high horsepower motor that would overheat the fluid and would certainly require a refrigeration system.

Because the coolant collects and circulates considerable quantities of both coarse and fine chips, it must be very carefully filtered in the interest of quality finish, increased tool life, reduced machine downtime, constant hole size, and minimum wear of the pump and the coolant-transfer unit. Depending largely upon long-run production needs, coolant-filtration systems vary considerably in fluid capacity and filtering devices. Manufacturers offer a wide range of coolant systems with varying filtering capacities. Detailed information on cutting fluids and an applications chart may be found in Chapter 4, "Cutting Fluids and Industrial Lubricants."

Small-Hole Drilling Machines

Major problems in the mechanical drilling of small holes include compensating for the decreased rigidity of the delicate tools required, controlling chip removal, minimizing runout, and maintaining a uniform feed rate. This type of drilling can often be done on conventional machine tools such as lathes, drilling and boring machines, machining and turning centers, and transfer machines. Sensitive drill presses are used extensively for drilling small holes. They are almost always hand fed, with either a sliding quill to advance the rotating drill or an elevating table to force the work into the drill.

Automatic screw machines, especially Swiss types, discussed in Chapter 15, "Multifunction Machines," are used for small-hole drilling, often with both the workpiece and drill rotating. These machines, as well as small turret lathes, permit using several tools to produce the holes. For example, a center drill can be used to start the hole, then a pivot drill to enlarge and deepen the hole, and finally a straight-shank drill to enlarge the hole further and finish it to final depth.

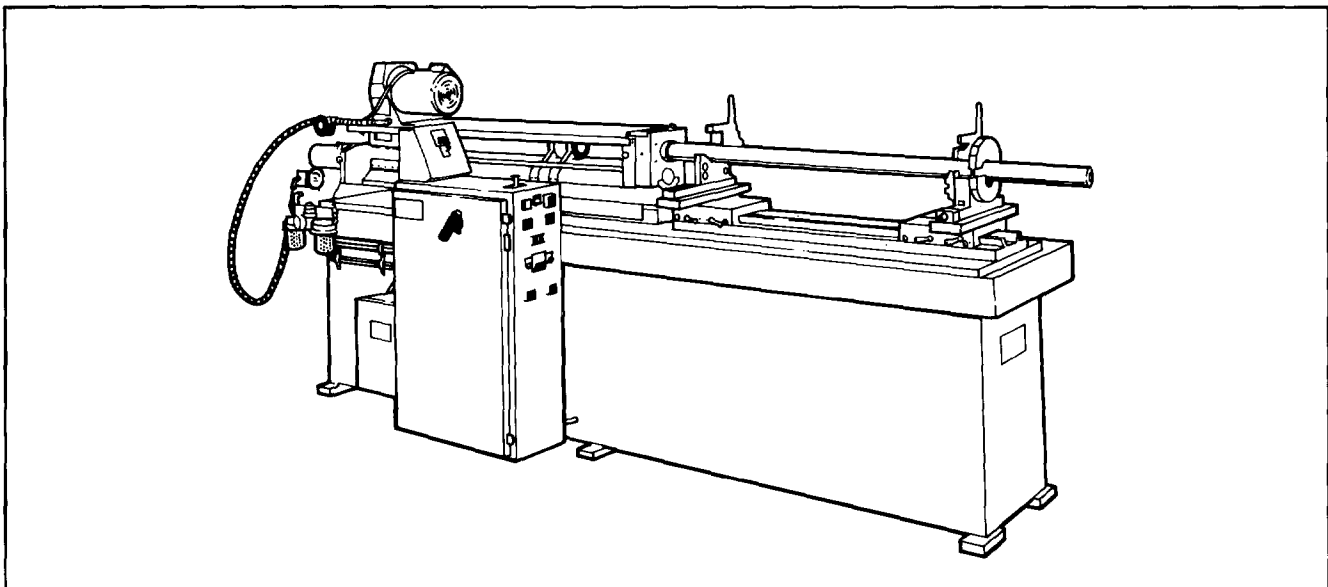


Fig. 9-12 Fixed-table, advancing-spindle machine for deep-hole drilling.

CHAPTER 9

DRILLING MACHINES

Small holes with close tolerances, however, may require special machines with increased rigidity, more accurate spindles, and sensitive feeding arrangements. Microdrilling machines are available that rotate both the workpiece and the tool and can produce holes of less than 0.001" (0.03 mm) diam. An independently driven, rotating spindle for the drill is mounted in a tailstock on these machines. This rotating, collet-holding spindle assembly is supported on a pair of dovetail slides equipped with micrometer adjustments for precise alignment of the tool centerline with the rotational axis of the workpiece. Sensitive feeding is accomplished by rotating a handwheel on the feedscrew while maintaining pressure on the slide handle.

The ultrasensitive, precision drilling machine illustrated in Fig. 9-13 can be used with various accessories, including a digital readout system and a binocular-type stereoscopic microscope as shown. A pivot-mounted drill-feed mechanism minimizes friction and lost motion and amplifies the operator's sensitivity of touch by 20 times. The spindle is automatically retracted by a calibrated spring when the operator releases the feed lever.

Tool flexing and breakage due to eccentric rotation are minimized on these machines by mounting the drills in mandrels which rotate on two in-line diamond V-bearings. A small collet-type pulley is fitted over the mandrel and connected by an endless belt to the pulley of an isolated vibration-dampened motor. Pull of the belt holds the mandrel firmly against the V-bearings for continuous contact, as shown in the view at the right of Fig. 9-13.

Cam-controlled and CNC models of precision, small-hole drilling machines are also available.

Special-Purpose Drilling Machines

Many drilling machines are built in a wide variety of designs and configurations for special-purpose applications. Special-purpose dedicated machines are often used when large quantities of parts require multiple operations. These include

shuttle transfer, dial index, ring index, trunnion index, and in-line transfer machines, discussed in Chapter 15, "Multi-function Machines."

Special-purpose machines for drilling of pilot holes in tubes and pipes and for forming of 90° branch collars or bosses, all in one cycle, are built by T-Drill, Inc. This method eliminates the need for tube or pipe cutting and end preparation for welding T-fittings. As soon as the cutting tool penetrates the wall of the workpiece, during the downfeed stroke, two formation pins are extended. Then, with the feed reversed, the collar is formed as the rotating pins are withdrawn from the hole. Extension and retraction of the formation pins are accomplished by cam action, with lugs on the upper ends of the pins traveling in spiral grooves in an adjustment cone.

Portable Drilling Units

Considerable drilling is done with portable tools that are powered pneumatically, electrically, or hydraulically. Such tools are used extensively, particularly in the aircraft/aerospace and automotive industries, when workpieces cannot be easily taken to drilling machines. Drilling of fastener holes in airframes with portable tools is probably the most common manufacturing operation required in the aircraft industry, with millions of holes drilled each month.

Some portable tools are available with magnetic bases or other devices for mounting or clamping to the work. In addition to being used for drilling, portable tools are also employed for screwdrivers, nutrunners, impact wrenches, hammers, routers, and grinders, as well as for tapping, nibbling, shearing, sanding, scraping, filing, sawing, and broaching operations.

Tool selection. Selecting a type of portable power tool for drilling depends upon many factors, the most important of which include the following:

1. Power source existing in the plant. When sufficient compressed air capacity is available, pneumatically

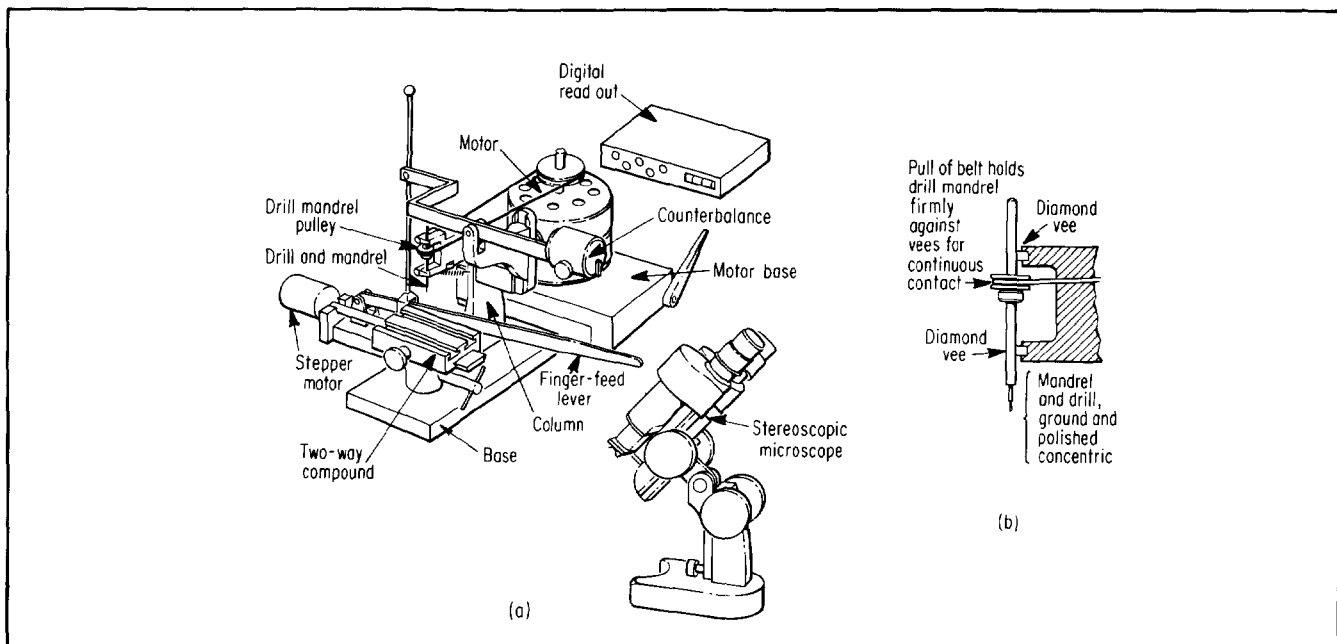


Fig. 9-13 Ultrasensitive drilling machine for producing small holes can be used with digital readout and microscope. Tool breakage is minimized by mounting shown in view at right.

DRILLING MACHINES

powered tools are often preferred. Air capacity refers to adequate pressure and flow while the tools are running.

2. Type of operation to be performed. Power requirements and whether the operation is continuous or intermittent are important considerations. In some applications in which safety is critical, such as drilling in hazardous locations, air tools are preferable to electric tools to eliminate any danger of explosion or fire. Speed/torque relationships are critical—the tool must be powerful enough for the drill employed and the material to be drilled.
3. Size, power/weight ratio, and the speed/feed ranges are critical factors.
4. Ease and cost of servicing and maintenance must also be considered.

Air drills. Portable drills powered by air are the most widely used. Advantages include lighter weight, smaller size, less vibration, cooler operation, variable speeds, no shock hazard, good torque characteristics, instant reversibility, and in general, lower maintenance costs. Possible limitations include higher initial cost, speed decreasing under load, air exhaust problems, and in some cases, higher operating costs. It is recommended that these tools be operated with clean, dry, and lubricated air, generally at a pressure of about 90 psi (621 kPa), with an air consumption varying from 10-70 cfm (0.3-2 m³/min).

Electric drills. These portable tools are available for operation on normal (60 Hz) or high-frequency (120, 360, or 400 Hz) current. Advantages of tools that operate on normal frequency current include lower initial cost, readily available power supply, and high efficiency. Disadvantages include a higher weight/power ratio, speed decreasing under load, limited torque characteristics, and possible shock hazard. Many electric drills are now available with electronically controlled, infinitely variable speeds, and tool weights have been reduced. Advantages of tools that operate on high-frequency current include lighter weight, faster speeds, and the ability to maintain speed under load. Disadvantages include the need for a special power source (frequency changer or motor-generator set) and special wiring.

Hydraulic drills. Portable tools with hydraulic motors provide variable speed and feed control. They generally operate at pressures to 1500 psi (10 342 kPa) and require special hydraulic power units.

Hand-feed portable drills. These tools are generally used only for producing small diameter holes in thin materials when accuracy is not critical. They can be equipped with self-locking nosepieces and used in conjunction with sliding racks to allow the operator to keep the drill at any desired angle to the workpiece.

Self-feeding units. Self-feeding or automatic drill units (see Fig. 9-14) can be mounted on drill stands or fixtures at any angle and are light enough to be carried from one job to another. Several units can be mounted at different angles to drill multiple holes on two or more planes in sequence or simultaneously.

Automatic drilling units have several advantages over traditional drill presses. Unlike the drill press operated by hand, self-feeding units reduce labor costs because they do not require a skilled operator. Once the stroke, speed, thrust, and feed have been set, the entire drilling or tapping operation is automatic and repetitive. This means that one operator can manage several machines or stations. Multispindle drillheads can be attached to automatic drilling equipment to produce two or more holes simultaneously with precise hole-to-hole accuracy and even greater cost savings.

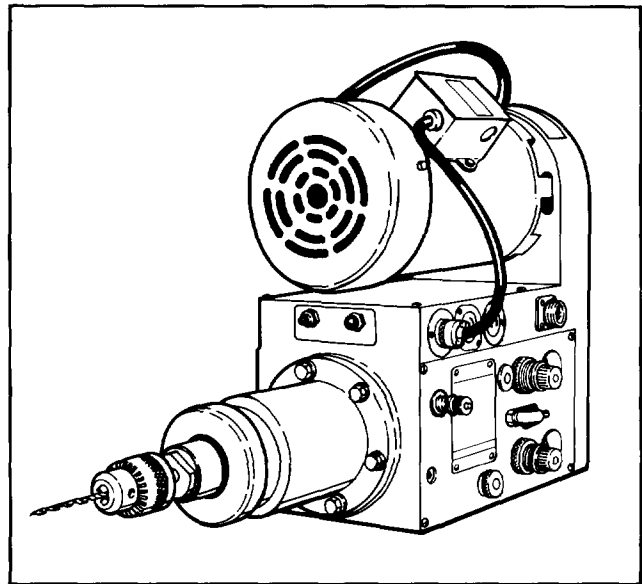


Fig. 9-14 Automatic self-feeding drill unit. (Dumore Corp.)

Other advantages include higher production, increased drilling consistency, fewer scrapped parts, and less tool wear and breakage. Automatic drills can be used for drilling, tapping, reaming, boring, spotfacing, chamfering, counter-boring, and countersinking. Self-feeding drilling and tapping units are available in several types, including electromechanical with air feed, air powered, mechanical, and hydraulic.

Electromechanical units with air feed. Electrically driven, air-controlled units provide the most drilling flexibility by combining the advantages of electric power with the benefits of economical air feed. Electric drive provides high drilling speeds, and good accuracy and productivity. Air used for thrust and feed results in low installation costs, minimal maintenance, fewer components to wear out, no contamination of parts, and less drill breakage.

Unlike air-powered units, the air-feed, electric-powered types are not affected by line pressure drops and they do not require a large compressor. Lightweight and portable, they can easily be moved from one setup to another. Most units accept multi-spindle drillheads for even greater efficiency and cost savings. While they have a more limited stroke and offer limited spindle options, they are generally the most cost effective approach to self-feeding drilling and tapping.

Air powered units. These units are generally smaller and lighter than other types, and they are lower in initial cost. Closer center distances can be obtained with air units. There are, however, several disadvantages. They are more costly to operate and maintain, and they have a high noise level. Since a 7 hp (5.2 kW) electric motor is needed to power the compressor for each horsepower (0.746 kW) drilling ability, air-powered units are inefficient. A larger compressor may be needed because heavy demand on the air supply may cause a drop in line pressure. Air motors require more frequent lubrication and maintenance, and they tend to wear faster than electric units.

Portable pneumatic drills with a positive screw-feed spindle and wormgear mechanism are sometimes used for producing large holes in thick or work-hardening materials. With such units, the feed rate cannot be changed unless the tools are disassembled and the drive gears replaced.

CHAPTER 9

TWIST DRILLS

Mechanical units. Drilling units with mechanical feed have greater thrust and stroke capability, and they can drill larger holes—3/4" (19 mm) and more in diameter. Their disadvantages are that they are more complex and, hence, more costly. There are more components, some units require two electric motors, and they are less portable.

Hydraulic units. Hydraulic drilling units are the most expensive, but they are capable of even greater thrust than mechanical systems. They are also more compact and have a higher cycle rate. Their disadvantages include increased noise, susceptibility to leaks and contamination, higher maintenance costs, higher installation costs, and less portability. An electric motor is required to drive the hydraulic pump, and a hydraulic motor to drive the spindle.

TWIST DRILLS

Drills are defined as rotary end-cutting tools having one or more cutting lips and one or more helical or straight flutes for the passage of chips and the admission of a cutting fluid. These cutting tools are made in a wide variety of types with many different forms, dimensions, and tolerances.

Twist drills are not considered to be precision cutting tools; rather, they are tools designed to produce holes rapidly and economically. When precision is required, subsequent operations such as boring or reaming are generally required. Drilling, using twist drills having tapered webs, is also generally limited to hole depths of about three to five times the hole diameter unless the woodpeckering technique of periodic tool withdrawal is employed or coolant-fed twist drills (discussed later in this chapter) are used.

Classification of Twist Drills

Twist drills can be classified by the material from which they are made, kinds of shank, number of flutes, hand of cut, length, diameter, and point geometry.

Based on the kind of shank, twist drills can be classified as:

1. **Straight-shank drills.** Those having cylindrical shanks which may be the same or different diameter than the body of the drill. The shanks may be provided with or without driving flats, tangs, grooves, or threads.
2. **Taper-shank drills.** Those having conical shanks suitable for direct fitting into tapered holes in machine spindles, driving sleeves, or sockets. Taper-shank drills generally have a tang to assist in driving and to permit removing the drill from the spindle or holder.

Based on the number of flutes, twist drills can be classified as:

1. **Single-flute drills.** These tools, having only one flute, are used for originating holes and for drilling plastics.
2. **Two-flute drills.** These are the conventional type drills also used for originating holes.
3. **Three or four-flute drills (core drills).** These are commonly used for enlarging and finishing drilled, cast, or punched holes. They do not produce original holes.

Based on hand of cut, twist drills can be classified as:

1. **Right-hand cut.** As viewed when looking toward the point of these drills, with their shanks extending away, they must be rotated in a counterclockwise direction in order to cut. Most drills are made for right-hand rotation.
2. **Left-hand cut.** When viewed from the cutting point, clockwise rotation is necessary for cutting.

Nomenclature

The following terminology is extracted from ANSI B94.11-M-1979, "Twist Drills—Straight Shank and Taper Shank, Combined Drills and Countersinks," with the permission of the publisher, ASME (see Fig. 9-15). Many of these terms also apply to other types of drills.

axis The imaginary straight line which forms the longitudinal centerline of the drill.

back taper A slight decrease in diameter from point to back in the body of the drill.

body The portion of the drill extending from the shank or neck to the outer corners of the cutting lips.

body-diameter clearance That portion of the land that has been cut away so it will not rub against the walls of the hole.

chisel edge The edge at the end of the web that connects the cutting lips.

chisel edge angle The angle included between the chisel edge and the cutting lip as viewed from the end of the drill.

clearance diameter The diameter over the cutaway portion of the drill lands.

drill diameter The diameter over the margins of the drill measured at the point.

flutes Helical or straight grooves cut or formed in the body of the drill to provide cutting lips, to permit removal of chips, and to allow cutting fluid to reach the cutting lips.

flute length The length from the outer corners of the cutting lip to the extreme back end of the flutes. It includes the sweep of the tool used to generate the flutes and therefore does not indicate the useable length of the flutes.

helix angle The angle made by the leading edge of the land with a plane containing the axis of the drill.

land The peripheral portion of the drill body between adjacent flutes.

land width The distance between the leading edge and the heel of the land measured at a right angle to the leading edge.

lips The cutting edges of a two-flute drill extending from the chisel edge to the periphery. On core drills, the lips are the cutting edges extending from the bottom of the chamfer to the periphery.

lip relief The axial relief on the drill point.

lip relief angle The axial relief angle at the outer corner of the lip. It is measured by projection into a plane tangent to the periphery at the outer corner of the lip. Lip relief angle is usually measured across the margin of the twist drill.

margin The cylindrical portion of the land which is not cut away to provide clearance.

neck The section of reduced diameter between the body and the shank of a drill.

overall length The length from the extreme end of the shank to the outer corners of the cutting lips. It does not include the conical shank end often used on straight-shank drills, nor does it include the conical cutting point used on both straight and taper-shank drills. For core drills with an external center on the cutting end, overall length is the same as for two-flute drills. For core drills with internal centers on the cutting end, the overall length is to the extreme ends of the tool.

point The cutting end of the drill, made up of the ends of the lands, the web, and the lips. In form, it resembles a cone, but it departs from a true cone to furnish clearance behind the cutting lips.

point angle The angle included between the lips projected upon a plane parallel to the drill axis and parallel to the cutting lips.

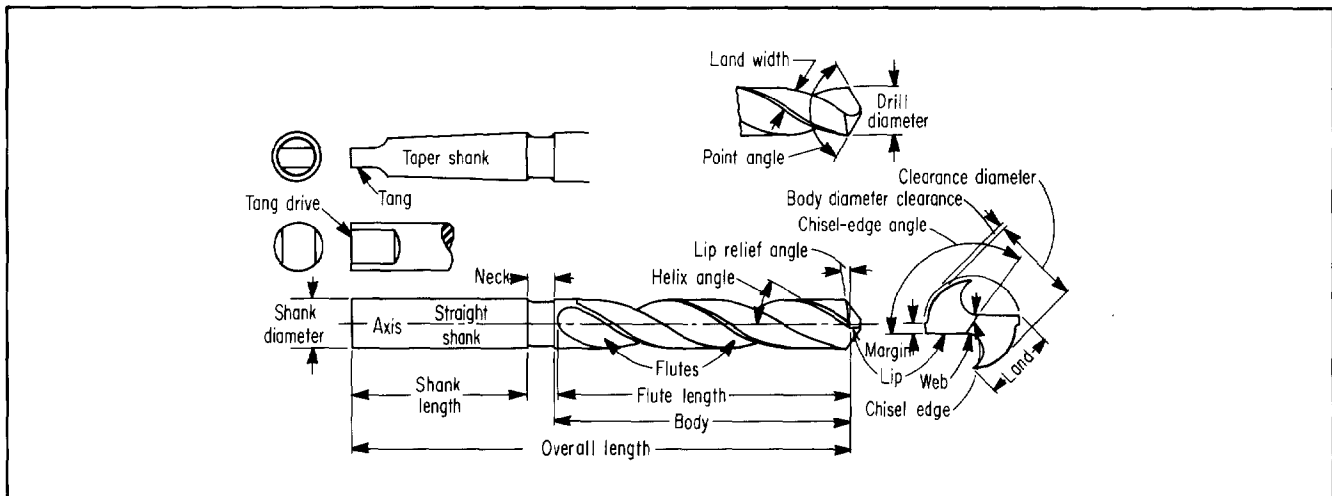


Fig. 9-15 Standard terms used to describe elements of twist drills.

shank The part of the drill by which it is held and driven.

tang The flattened end of a taper shank, intended to fit into a driving slot in the socket.

tang drive Two opposite and parallel driving flats on the extreme end of a straight shank.

web The central portion of the body that joins the lands. The extreme end of the web forms the chisel edge on a two-flute drill.

web thickness The thickness of the web at the point unless another specific location is indicated.

web thinning The operation of reducing the web thickness at the point to reduce drilling thrust.

Materials and Surface Treatments for Twist Drills

Desirable characteristics for materials used for manufacturing twist drills include good wear resistance, toughness, and hot hardness, as discussed in Chapter 3, "Cutting Tool Materials." Two basic groups of cutting tool materials that meet these criteria are high-speed steels and sintered tungsten carbides. High-speed steel is the most common material used for drills, but carbide drills, both tipped and solid, are employed for many applications.

Carbon and low-alloy tool steels. These materials, used for practically all cutting tools prior to 1900, are no longer employed for production metalcutting, primarily because of their low hot hardness. Applications for drills made from these materials are limited to nonproduction applications and the drilling of wood and some soft, nonferrous metals.

High-speed steels. A comprehensive discussion of the compositions and heat treatment of the various high-speed steels is presented in Chapter 3. Twist drills for general industrial applications are most commonly made from one of four molybdenum-based high-speed steels: M1, M2, M7, and M10. Drills made from these materials are effective in drilling a majority of materials.

For more difficult operations in which increased hot hardnesses are required because of the higher heat generated, HSS's containing cobalt, such as types M33, M35, M36, M42, and T15, are often employed. Cobalt-containing HSS drills allow faster operation compared with regular HSS drills. Typical applications for tools made from these materials

include nickel-based alloys, titanium alloys, and steels having a hardness over R_{C40} .

High-speed steels containing cobalt are, more expensive. As a result, they are generally only employed when productivity can be improved, or when necessary. HSS's now being made by powder metallurgy (PM) processes have superior grindability, as well as other advantages (see Chapter 3). Powder metallurgy HSS's, however, are not normally used for standard twist drills, but they are employed for some special twist drills and spade drills (discussed later in this section).

Heat treatment. The heat treatment (hardening, quenching, and tempering) of HSS's (see Chapter 3) is critical to the quality and performance of twist drills. A particular HSS of the same analysis may be heat treated differently by different manufacturers to develop desired properties for the specific application of a certain style of drill.

Twist drills made from general-purpose HSS's such as M1 and M7 are usually heat treated to a hardness range of R_{C64-66} . Drills made of HSS's containing cobalt generally have a hardness range of R_{C65-67} , with a higher hardness sometimes for unusual applications.

Surface treatments. Various surface treatments (see Chapter 3) are applied to HSS drills to either increase the hardness of the outer layer of material or reduce the friction between the drill and the workpiece or chips in the flute. These treatments are normally applied after the drills have been finish ground. The category of treatments that produces a thin hard layer in the outer layers includes nitriding, cyaniding, and, less frequently, carbonitriding and carburizing. These processes are carried out in either a liquid or gaseous media at elevated temperatures to accelerate the absorption of the element(s) into the outer layers. These layers, while being hard, are also brittle, so they are kept thin to reduce the probability of chipping. Nitride layers range from 0.0005-0.002" (0.013-0.05 mm) thick, whereas carburized layers are several times that thickness. Carburizing and carbonitriding usually require temperatures above the tempering range, while gas nitriding or cyaniding can be done at about 1000° F (538° C).

In addition to the hard surfaces produced in the outer layers of the drill material by the processes described, there is some small usage of the processes for producing surfaces on the exterior of drills for the same purpose. These surfaces are

CHAPTER 9

TWIST DRILLS

applied by electroplating, chemical plating, vapor deposition, and flame or arc deposition. Included in such platings are hard chromium, nickel, tungsten, tungsten carbide, titanium nitride, and various other wear-resistant compounds. Considerable care is required during the processing to assure that the deposition is uniform, continuous, and tightly adhering and that the tempering temperature of the HSS is not exceeded.

Surface treatments for reducing friction or improving lubrication include the limited penetration of oxygen or sulfur into tool surfaces in a controlled-atmosphere furnace at an elevated temperature, and vapor or liquid processing.

The most widely used surface treatment on drills consists of the development of a thin surface oxide which has been found to act as a solid lubricant and to prevent welding of chips to the drill. While various oxide films can be produced in air at temperatures above 400° (204° C), oxide films most advantageous for improved performance are formed in a dry or superheated steam atmosphere at temperatures of 800-1050° F (427-566° C). Oxides created in salt baths are not as effective as those produced by steam. Tools having the dark gray to blue-black surface oxide layers produced in this way are often used for drilling ferrous materials.

Although surface treatments have been discussed as separate entities, they are often used in combination. The most widely used combination treatment for drills is a surface oxide over a nitride layer. A general discussion of surface treatments and coatings is presented in Volume III of this Handbook series, "Materials and Finishing."

Cast cobalt-based alloys. These materials (see Chapter 3) are used only occasionally for special drills to produce holes in case-hardened steels, manganese steels, chilled irons, and tool steels having a hardness over R_{C50} .

Carbide drills. Cemented carbides have a much higher hardness and greater resistance to abrasive wear than hardened HSS's. As a result, drills (both solid and tipped) are commonly used for producing holes in cast irons, aluminum and other soft nonferrous metals, highly abrasive materials such as reinforced plastics, and steels harder than R_{C48} . Drilling of steels in the soft annealed condition or in a heat-treated condition with moderate hardness levels is not successful with carbide tools.

Advantages of using carbide drills, when applicable, include longer tool life (up to 10 times that of HSS drills), higher speed capabilities (about 2 1/2 times HSS drills), and faster penetration rates (to 100% faster). Limiting characteristics are brittleness, which requires rigid and accurate setups and machines. Pointing of the drills is critical, especially for drilling very hard materials.

Although there are many types of carbide materials, the straight tungsten carbide types, C-1 and C-2 classifications, have been found to be most applicable for drills. Coated carbides, as well as classifications C-5 through C-8, have not proved advantageous. Solid carbide drills, normally made from C-2 carbide, are available from 0.0135 to 1/2" (0.343 to 12.7 mm) diam.

Carbide-tipped drills, made from C-1 or C-2 carbide, are available from 3/32 to 1 1/2" (2.4-38 mm) diam. The C-1 carbide is generally employed for tipped drills for which greater resistance to chipping, particularly in handling, is required. In making tipped twist drills, formed tips conforming to ANSI Standard B94.20-1977 are brazed to bodies of hardened HSS or alloy steel of high strength and hardness.

Carbide-tipped drills are used more extensively than solid carbide drills due to economic considerations and the fact that

the tipped tools can be used with less-rigid setups since the tool bodies absorb some deflection without fracturing. While tipped drills are less costly than solid carbide drills in larger diameters, solid drills cost less than tipped tools in the small-diameter range. Solid carbide drills provide more rigidity and stiffness than tipped drills for a given size. This is important in smaller-diameter sizes or other sizes which must be operated without a bushing.

Standard solid carbide drills are normally much shorter than a comparable-size tipped drill because the flexing capability of solid carbide is lower than that of the steel body of a tipped drill. Useable regrind lengths of tipped drills are approximately equal to the drill diameters.

Types of Twist Drills

Twist drills are manufactured in a wide variety of types, some of which are illustrated in Fig. 9-16, and in many different sizes. To produce a hole of any given diameter, twist drills are commercially available with variations in length, flute and shank configuration, point geometry, and web thickness. In some cases, a dozen or more drills may be available to produce the same size hole.

Drills are made in many different diameter sizes—fractional, number (wire gage), letter, and metric—ranging from 0.0059" (0.150 mm) to 3 1/2" (89 mm). However, data compiled by National Twist Drill, based on sales of more than 50-million standard twist drills, showed that a median 90% of all sales (5% were for larger sizes, and 5% for smaller) fall between 0.050 and 0.400" (1.27 and 10.16 mm) diam. The most common drill size,

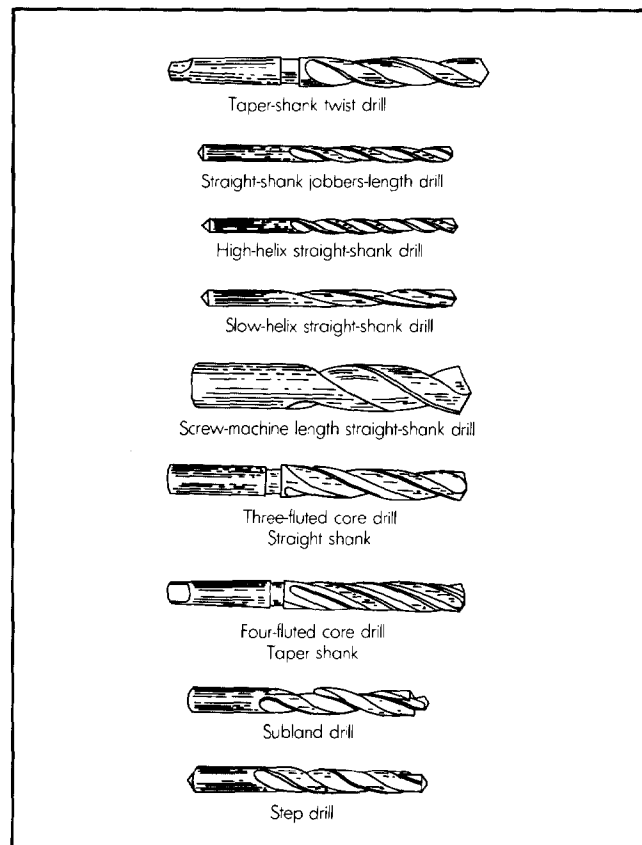


Fig. 9-16 Some conventional and special-purpose twist drills.

especially for rivet holes in the aircraft/aerospace industry, is a No. 30 (0.1285", 3.264 mm), with a 1/8" (3.2 mm) diam a close second. Only about 1% of all twist drills sold exceed 3/4" (19 mm) diam.

For simplification purposes, many styles of drills can be classified separated into general-purpose and heavy-duty categories. General-purpose drills are the most widely used. Slight alterations of the original point angles sometimes improve performance for given speeds and feeds. Heavy-duty drills are designed to provide greater torsional strength and rigidity than general-purpose drills. They can be used to drill steel forgings, hard castings, and high-hardness ferrous alloys.

Dimensional tolerances for drill diameters at their points, shank diameters (for straight-shank drills), back tapers, flute lengths, and overall lengths for HSS general-purpose twist drills with two, three, and four flutes are presented in Table 9-2. Tolerances on elements—the included angles of their points, lip heights, centrality of their webs, and flute spacings—for HSS, general-purpose, two-flute, twist drills are given in Table 9-3. The tolerances in both of these tables are extracted from ANSI B94.11M-1979 with the permission of the publisher, ASME.

Taper-shank twist drills. These tools are used for general-purpose drilling in all kinds of machines for which an ANSI taper shank is required. Diameters, flute lengths, and overall lengths of some standard taper-shank twist drills are presented in Table 9-4. Some sizes of these drills are available with shorter and longer shanks than standard, which decreases and increases their overall lengths. Amount of the shank taper, which varies with the drill diameter, and detail dimensions are given in Table 9-5. The HSS drill body and the alloy steel shank (when used) on these drills are usually joined by electrical butt or friction welding.

Straight-shank twist drills. These tools are made in three series, based on their range of lengths: jobbers length (a medium-length range), taper length (a long-length range), and screw-machine length (a short-length range). Diameters, flute lengths, and overall lengths of some standard straight-shank twist drills in all three length series are presented in Table 9-6. Heavy-duty drills are also available in screw-machine lengths with 135° split points.

The screw-machine length series of straight-shank twist drills are used in screw machines of all types when conditions require the use of short drills for maximum rigidity. Other applications for these short drills include portable and sheet-metal drilling, body work, and the drilling of tougher, harder steels such as stainless and high-manganese steels.

High-helix (fast-spiral) twist drills. These tools have higher helix angles (35-40°) than conventional drills (25-33°). These two-flute twist drills are designed for drilling deep holes in materials having low tensile strengths, such as aluminum, magnesium, copper, diecast metals, and some plastics, and are also used successfully for producing holes in soft steels and free-machining brasses and bronzes. The wide, ground, and/or polished flutes assist in clearing chips from the holes. The drills are available in fractional, number, letter, and metric sizes.

Low-helix (slow-spiral) twist drills. These two-flute tools have lower helix angles (15-20°) than conventional tools. They are made in two types: (1) for drilling plastics and (2) for drilling brass. Low-helix drills are also successful in applications for shallow drilling of some aluminum and magnesium alloys. This type of twist drill is also available in solid or tipped carbide, heavy-duty design for producing holes in cast iron.

Core drills. These three or four-flute tools are intended for

enlarging cored, punched, or drilled holes or increasing the ID's of short lengths of metal tubing or pipe. With proper flute construction, core drills are capable of enlarging holes which are only two-thirds of their final desired size. They will not cut holes in solid material because the cutting edges do not extend to the center of the drill. Cutting is accomplished by a chamfered edge at the end of each flute.

Core drills produce more finished holes with better quality than those cut by ordinary two-flute twist drills when all cutting edges of the core drill are sharp and have the same length and point angle. They are very rigid, and because of the greater number of flutes (three or four), can be used at higher feed rates than normal. Diameters, flute lengths, and overall lengths of some three and four-flute core drills are presented in Table 9-7.

The cutting action of core drills is much the same as that of rose reamers, which are practically obsolete. As a result, it is often practical to use core drills as roughing reamers or to correct the location of previously produced holes. In correcting hole locations, rigid machines and fixtures with drill bushings are required to guide the tools. When used for this purpose, it is recommended that the drills be ground to a flat (180°) point, similar to that of an end mill. The core drills should be kept very sharp and have little or no chamfer on the corners of the point. This reduces the tendency to follow incorrectly located holes.

Core drills normally have a fairly heavy web (up to 50% of the diameter) since they are not designed for center cutting. They are available with three or four flutes, straight or taper shanks, and in fractional sizes only. Straight-shank core drills are made with diameters from 1/4 to 1 1/4" (6.35 to 31.75 mm), flute lengths from 3 3/4 to 7 7/8" (95 to 200 mm), and overall lengths from 6 1/8 to 12 1/2" (156 to 317 mm). Taper-shank core drills are made with diameters from 1/4 to 2 1/2" (6.35 to 63.5 mm), flute lengths from 2 7/8 to 11 1/4" (73 to 286 mm), and overall lengths from 6 1/8 to 18 3/4" (156 to 476 mm).

Larger diameter core drills usually embody a fluted cutter interchangeably mounted on the end of a fluted body. Such drills often incorporate pilots to assure accuracy of hole alignment. Four-flute core drills are generally preferred unless the greater chip room of a three-flute tool is needed.

Multidiameter drills. Many manufactured parts require concentric holes having different diameters over portions of their lengths. In such cases, significant reductions in machining costs are often possible by using multidiameter drills instead of a succession of various diameter drills. Multidiameter tools are also used to perform two or more operations in a single feed stroke—for example, drilling combined with countersinking, counterboring, reaming, and tapping.

There are two different types of construction used in multidiameter tools—step and subland (see Fig. 9-17). These types are often confused both as to nomenclature and application. The type of construction used relates to the effect of the regrinding process and the desire to maximize useable tool life rather than to the material being machined. Step construction is the simpler type and generally less expensive; subland construction is more difficult and costly to manufacture.

The ratio of step length to resharpening allowance is a major factor in determining whether step or subland construction should be used. If the step length (length of small diameter) is fairly long, it is practically as easy to resharpen a step tool as a subland tool and there is no advantage to the use of subland construction. However, tools with relatively short steps and with step diameters considerably smaller than the large

CHAPTER 9

TWIST DRILLS

diameters can be expensive to regrind. This, coupled with the excessive loss of length in resharpener, can make the more costly subland tools preferable.

Step drills. Step drills have two or more diameters ground into the lands of the drill. Normally, the drill will be of the two-flute style, although it is possible to have a three or four-flute step drill. These various diameters or steps are separated from each other by angular cutting edges which conform to the shape desired in the part. Optimum tool performance of step or combination drills is obtained when the tool is manufactured to provide for proper balance of web thickness and chip room, even with the various diameters present. Step drills can frequently be made by grinding down and stepping conventional drills. Some thinning of the web is usually required.

As a general rule, multiple-diameter drills can be manufactured in the step drill style as long as the ratio between diameters does not exceed 4:1. A higher ratio between diameters presents problems in achieving good mechanical strength, and the tool often fails due to breakage. Such applications should be approached with caution.

Resharpener the cutting angle or shoulder of step drills results in a slight undercutting or nicking of the smaller diameter at that point. Eventually the reground point of the small diameter will begin to approach this nicked area, and it becomes the limit as to the number of regrinds possible. Obviously, when the step length is small, the regrinds available are limited.

Step drills are often used to drill holes to close tolerances, sometimes eliminating the need for subsequent reaming operations. When the smaller diameter pilot portion of a step drill enters the workpiece, it may produce a hole that is larger than its diameter, as illustrated at the left in Fig. 9-18. When the larger-diameter body portion of the step drill begins to cut, it centers the pilot, shown at the right, thus producing a hole to close tolerances. For such applications, the pilot and body portions of the step drill must be concentric and the pilot should have a diameter three-fourths that of the body and a length about two-thirds of the body diameter. For some applications, the body portion of the step drill is carbide tipped.

Subland drills. Subland-type construction of a multidiameter

TABLE 9-2
Dimensional Tolerances for HSS General-Purpose Twist Drills with Two, Three, and Four Flutes

DRILL DIAMETER AT POINT					
Diameter of Drill		Tolerance			
		Inches		Millimeters	
Inches	Millimeters	Plus (+)	Minus (-)	Plus (+)	Minus (-)
From No. 97 thru No. 81	From 0.15 thru 0.33	0.0002	0.0002	0.005	0.005
Over No. 81 thru 1/8	Over 0.33 thru 3.18	0.0000	0.0005	0.000	0.013
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0000	0.0007	0.000	0.018
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0000	0.0010	0.000	0.025
Over 1/2 thru 1	Over 12.70 thru 25.40	0.0000	0.0012	0.000	0.030
Over 1 thru 2	Over 25.40 thru 50.80	0.0000	0.0015	0.000	0.038
Over 2 thru 3 1/2	Over 50.80 thru 88.90	0.0000	0.0020	0.000	0.051

SHANK DIAMETER (STRAIGHT-SHANK DRILLS)					
Diameter of Drill		Tolerance			
		Inches		Millimeters	
Inches	Millimeters	Plus (+)	Minus (-)	Plus (+)	Minus (-)
From No. 97 thru No. 81	From 0.15 thru 0.33	0.0002	0.0002	0.005	0.005
Over No. 81 thru 1/8	Over 0.33 thru 3.18	0.0000	0.0025	0.000	0.064
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0005	0.0030	0.013	0.076
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0005	0.0045	0.013	0.114
Over 1/2 thru 2	Over 12.70 thru 50.80	0.0005	0.0030	0.013	0.076

BACK TAPER		
Diameter of Drill		Allowance
Inches	Millimeters	Per Unit of Length
From No. 97 thru No. 81	From 0.15 thru 0.33	None
Over No. 81 thru 1/8	Over 0.33 thru 3.18	0.0000 to 0.0008
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0002 to 0.0008
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0002 to 0.0009
Over 1/2 thru 1	Over 12.70 thru 25.40	0.0003 to 0.0011
Over 1 thru 3 1/2	Over 25.40 thru 88.90	0.0004 to 0.0015

TABLE 9-2—Continued

FLUTE LENGTH					
Diameter of Drill		Tolerance			
		Inches		Millimeters	
Inches	Millimeters	Plus (+)	Minus (-)	Plus (+)	Minus (-)
From No. 97 thru No. 81	From 0.15 thru 0.33	1/64	1/64	.4	.4
Over No. 81 thru 1/8	Over 0.33 thru 3.18	1/8	1/16	3.2	1.6
Over 1/8 thru 1/2	Over 3.18 thru 12.70	1/8	1/8	3.2	3.2
Over 1/2 thru 1	Over 12.70 thru 25.40	1/4	1/8	6.4	3.2
Over 1 thru 2	Over 25.40 thru 50.80	1/4	1/4	6.4	6.4
Over 2 thru 3 1/2	Over 50.80 thru 88.90	3/8	3/8	9.5	9.5

OVERALL LENGTH					
Diameter of Drill		Tolerance			
		Inches		Millimeters	
Inches	Millimeters	Plus (+)	Minus (-)	Plus (+)	Minus (-)
From No. 97 thru No. 81	From 0.15 thru 0.33	1/32	1/32	0.8	0.8
Over No. 81 thru 1/8	Over 0.33 thru 3.18	1/8	1/16	3.2	1.6
Over 1/8 thru 1/2	Over 3.18 thru 12.70	1/8	1/8	3.2	3.2
Over 1/2 thru 1	Over 12.70 thru 25.40	1/4	1/8	6.4	3.2
Over 1 thru 2	Over 25.40 thru 50.80	1/4	1/4	6.4	6.4
Over 2 thru 3 1/2	Over 50.80 thru 88.90	3/8	3/8	9.5	9.5

(ANSI Standard B94.11M-1979, published by ASME)

drill is specifically intended to overcome resharpener problems. The subland drill performs the same type of drilling operations as step drills, but the construction of the drill has been changed to prevent nicking of the small diameter. On the step drill, the various diameters are on the same set of lands. With subland drills, an independent set of lands is provided down the entire flute length for each diameter present in the form. Thus, in regrinding the shoulder angle to the largest diameter, it is not necessary to come into contact with lands associated with the other diameter(s).

Although the subland drill is initially more expensive to produce, an advantage of this construction is that the number of potential regrinds is increased when the step length is relatively short. A drill of subland construction, however, tends to be weaker than the same drill of step construction. For this reason, subland construction is generally not used when the ratio of the largest to smallest diameter exceeds 2:1. Given an acceptable relationship between the largest and smallest diameters, the decision as to whether to use a subland drill is related to the length of the smaller diameters. If the length of the smaller diameters of a step drill will not give acceptable regrind life, then a subland drill should be used.

Subland construction is of limited value for carbide-tipped drills because tool life is normally restricted to the useable carbide length (about equal to one diameter) of the smaller diameter of the drill. Step drills, however, are often made with carbide tips on one or more of the diameters, depending upon the application.

Combined drills and countersinks. These step tools probably represent the most common type of multidiameter drills. They are used to form the seats or bearings in the ends of workpieces which will be machined between centers. The regular drill and countersink consists of a single drill diameter, followed by a 60° included angle which forms the bearing surface. In those

instances in which accuracy is important or in which the centers will be used for several machining operations, it is important to retain a good, smooth bearing surface. This is often done by using a bell-type center, which provides an additional 120° angle to protect the 60° bearing surface. Plain and bell-type, combined drills and countersinks are illustrated in Fig. 9-19.

One drill manufacturer offers a two-piece step drill consisting of a countersink sleeve made of high-speed steel that can be combined with a standard general-purpose drill (see Fig. 9-20). A setting device with a clamping stop-nut permits varying the step lengths. This design can reduce resharpener costs since the drill usually becomes dull before the countersink section, and can be resharpened separately. The two-piece step drills are particularly suitable for drilling materials that form short chips, such as cast irons.

Other multidiameter tools. Multidiameter tools are also available to combine drilling with counterboring, reaming, and tapping. One type of combined drill and tap, made by Regal-Beloit Corp., consists of a twist drill with a 25° helix angle followed by a helical-fluted, self-leading tap. Through holes up to two times the diameter can be drilled and tapped in a single pass. Length of the drill section is sufficient to assure that it passes through the workpiece before tapping begins, and to provide stock for resharpener. Machines used for these tools must be capable of reversing to withdraw the tap. Standard tools have straight shanks with a square tap drive for use with straight toolholders without float.

Other twist drill types. There are many other types of twist drills available, most developed for special-purpose applications. Only the more common types are discussed in this section.

Center drills. These straight-shank twist drills are often used to center the ends of shaft-type workpieces. Their short flute and overall lengths as well as oversize shank diameters provide

CHAPTER 9

TWIST DRILLS

TABLE 9-3
Tolerances on Elements for HSS General-Purpose Two-Flute Twist Drills*

INCLUDED ANGLE OF POINT			
Diameter of Drill		Included Angle	Tolerance
Inches	Millimeters		
From 1/16 thru 1/2	From 1.59 thru 12.70	118°	±5°
Over 1/2 thru 1 1/2	Over 12.70 thru 38.10	118°	±3°
Over 1 1/2 thru 3 1/2	Over 38.10 thru 88.90	118°	±2°

LIP HEIGHT			
Diameter of Drill		Tolerance	
		Total Indicator Variation (TIV)	
Inches	Millimeters	Inches	Millimeters
From 1/16 thru 1/8	From 1.59 thru 3.18	0.0020	0.051
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0030	0.076
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0040	0.102
Over 1/2 thru 1	Over 12.70 thru 25.40	0.0050	0.127
Over 1 thru 3 1/2	Over 25.40 thru 88.90	0.0060	0.152

Method of Measurement: Rotate the drill in a V-block against a back end stop. Measure the cutting lip height variation on a comparator, or with an indicator at a location approximately 75% of the distance from the center to the periphery of the drill.

CENTRALITY OF WEB			
Diameter of Drill		Tolerance	
		Total Indicator Variation (TIV)	
Inches	Millimeters	Inches	Millimeters
From 1/16 thru 1/8	From 1.59 thru 3.18	0.0030	0.076
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0040	0.102
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0050	0.127
Over 1/2 thru 1	Over 12.70 thru 25.40	0.0070	0.178
Over 1 thru 3 1/2	Over 25.40 thru 88.90	0.0100	0.254

Method of Measurement: Rotate the drill in a close-fitting bushing. Record the difference in indicator readings of the web at the point as the drill is indexed 180°.

FLUTE SPACING					
Diameter of Drill		Tolerance			
		Total Indicator Variation (TIV)		Actual Deviation	
Inches	Millimeters	Inches	Millimeters	Inches	Millimeters
From 1/16 thru 1/8	From 1.59 thru 3.18	0.0030	0.076	0.0015	0.038
Over 1/8 thru 1/4	Over 3.18 thru 6.35	0.0060	0.152	0.0030	0.076
Over 1/4 thru 1/2	Over 6.35 thru 12.70	0.0100	0.254	0.0050	0.127
Over 1/2 thru 1	Over 12.70 thru 25.40	0.0140	0.356	0.0070	0.178
Over 1 thru 3 1/2	Over 25.40 thru 88.90	0.0260	0.660	0.0130	0.330

Method of Measurement: Place the drill in a V-block against a back end stop, and rotate it against a radial finger stop. Take an indicator reading at the leading edge of the margin on the opposite flute. Repeat for the other flute and note the difference between the two readings. The deviation in flute spacing is equal to one-half the difference between the two readings.

(ANSI Standard B94.11M-1979, published by ASME)

* In any one lot, 95% of the drills fall within the given tolerances.

TABLE 9-4
General Dimensions of Taper-Shank Twist Drills, Regular Shank, Fractional and Metric Sizes

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
1/8	3.00	0.1181	3.000	1	1 7/8	48	5 1/8	130
		0.1250	3.175	1	1 7/8	48	5 1/8	130
	3.20	0.1260	3.200	1	2 1/8	54	5 3/8	137
	3.50	0.1378	3.500	1	2 1/8	54	5 3/8	137
9/64		0.1406	3.571	1	2 1/8	54	5 3/8	137
5/32	3.80	0.1496	3.800	1	2 1/8	54	5 3/8	137
		0.1562	3.967	1	2 1/8	54	5 3/8	137
	4.00	0.1575	4.000	1	2 1/2	64	5 3/4	146
	4.20	0.1654	4.200	1	2 1/2	64	5 3/4	146
11/64		0.1719	4.366	1	2 1/2	64	5 3/4	146
3/16	4.50	0.1772	4.500	1	2 1/2	64	5 3/4	146
		0.1875	4.762	1	2 1/2	64	5 3/4	146
	4.80	0.1890	4.800	1	2 3/4	70	6	152
	5.00	0.1969	5.000	1	2 3/4	70	6	152
13/64		0.2031	5.159	1	2 3/4	70	6	152
7/32	5.20	0.2047	5.200	1	2 3/4	70	6	152
	5.50	0.2165	5.500	1	2 3/4	70	6	152
		0.2183	5.558	1	2 3/4	70	6	152
	5.80	0.2223	5.800	1	2 7/8	73	6 1/8	156
15/64		0.2344	5.954	1	2 7/8	73	6 1/8	156
1/4	6.00	0.2362	6.000	1	2 7/8	73	6 1/8	156
	6.20	0.2441	6.200	1	2 7/8	73	6 1/8	156
		0.2500	6.350	1	2 7/8	73	6 1/8	156
	6.50	0.2559	6.500	1	3	76	6 1/4	159
17/64		0.2656	6.746	1	3	76	6 1/4	159
9/32	6.80	0.2677	6.800	1	3	76	6 1/4	159
	7.00	0.2756	7.000	1	3	76	6 1/4	159
		0.2812	7.142	1	3	76	6 1/4	159
	7.20	0.2835	7.200	1	3 1/8	79	6 3/8	162
19/64	7.50	0.2953	7.500	1	3 1/8	79	6 3/8	162
		0.2969	7.541	1	3 1/8	79	6 3/8	162
	7.80	0.3071	7.800	1	3 1/8	79	6 3/8	162
		0.3125	7.938	1	3 1/8	79	6 3/8	162
5/16	8.00	0.3150	8.000	1	3 1/4	83	6 1/2	165
	8.20	0.3228	8.200	1	3 1/4	83	6 1/2	165
		0.3281	8.334	1	3 1/4	83	6 1/2	165
	8.50	0.3346	8.500	1	3 1/4	83	6 1/2	165
11/32		0.3438	8.733	1	3 1/4	83	6 1/2	165
23/64	8.80	0.3465	8.800	1	3 1/2	89	6 3/4	171
	9.00	0.3543	9.000	1	3 1/2	89	6 3/4	171
		0.3594	9.129	1	3 1/2	89	6 3/4	171
	9.20	0.3622	9.200	1	3 1/2	89	6 3/4	171
3/8	9.50	0.3740	9.500	1	3 1/2	89	6 3/4	171
		0.3750	9.525	1	3 1/2	89	6 3/4	171
	9.80	0.3858	9.800	1	3 5/8	92	7	178

CHAPTER 9

TWIST DRILLS

TABLE 9-4—Continued

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
25/64		0.3906	9.921	1	3 5/8	92	7	178
	10.00	0.3937	10.000	1	3 5/8	92	7	178
	10.20	0.4016	10.200	1	3 5/8	92	7	178
13/32		0.4062	10.320	1	3 5/8	92	7	178
	10.50	0.4134	10.500	1	3 7/8	98	7 1/4	184
		0.4219	10.716	1	3 7/8	98	7 1/4	184
27/64		0.4252	10.800	1	3 7/8	98	7 1/4	184
	10.80	0.4331	11.000	1	3 7/8	98	7 1/4	184
	11.00	0.4375	11.112	1	3 7/8	98	7 1/4	184
7/16		0.4409	11.200	1	4 1/8	105	7 1/2	190
	11.20	0.4528	11.500	1	4 1/8	105	7 1/2	190
	11.50	0.4531	11.509	1	4 1/8	105	7 1/2	190
29/64		0.4646	11.800	1	4 1/8	105	7 1/2	190
	11.80	0.4688	11.900	1	4 1/8	105	7 1/2	190
		0.4724	12.000	2	4 3/8	111	8 1/4	210
15/32		0.4803	12.200	2	4 3/8	111	8 1/4	210
	12.00	0.4844	12.304	2	4 3/8	111	8 1/4	210
	12.20	0.4921	12.500	2	4 3/8	111	8 1/4	210
31/64		0.5000	12.700	2	4 3/8	111	8 1/4	210
	12.50	0.5034	12.800	2	4 5/8	117	8 1/2	216
		0.5118	13.000	2	4 5/8	117	8 1/2	216
1/2		0.5156	13.096	2	4 5/8	117	8 1/2	216
	13.00	0.5197	13.200	2	4 5/8	117	8 1/2	216
		0.5312	13.492	2	4 5/8	117	8 1/2	216
33/64		0.5315	13.500	2	4 5/8	117	8 1/2	216
	13.20	0.5433	13.800	2	4 7/8	124	8 3/4	222
		0.5469	13.891	2	4 7/8	124	8 3/4	222
17/32		0.5572	14.000	2	4 7/8	124	8 3/4	222
	13.50	0.5610	14.250	2	4 7/8	124	8 3/4	222
		0.5625	14.288	2	4 7/8	124	8 3/4	222
35/64		0.5709	14.500	2	4 7/8	124	8 3/4	222
	14.00	0.5781	14.684	2	4 7/8	124	8 3/4	222
		0.5807	14.750	2	4 7/8	124	8 3/4	222
9/16		0.5906	15.000	2	4 7/8	124	8 3/4	222
	14.50	0.5938	15.083	2	4 7/8	124	8 3/4	222
		0.6004	15.250	2	4 7/8	124	8 3/4	222
37/64		0.6094	15.479	2	4 7/8	124	8 3/4	222
	15.00	0.6102	15.500	2	4 7/8	124	8 3/4	222
		0.6201	15.750	2	4 7/8	124	8 3/4	222
19/32		0.6250	15.815	2	4 7/8	124	8 3/4	222
	15.25	0.6299	16.000	2	5 1/8	130	9	229
		0.6398	16.250	2	5 1/8	130	9	229
5/8		0.6406	16.271	2	5 1/8	130	9	229
	16.00	0.6496	16.500	2	5 1/8	130	9	229
		0.6562	16.667	2	5 1/8	130	9	229
41/64								
	16.25							
	16.50							
21/32								

TABLE 9-4—Continued

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
43/64	16.75	0.6594	16.750	2	5 3/8	137	9 1/4	235
	17.00	0.6693	17.000	2	5 3/8	137	9 1/4	235
		0.6719	17.066	2	5 3/8	137	9 1/4	235
	17.25	0.6791	17.250	2	5 3/8	137	9 1/4	235
11/16		0.6875	17.462	2	5 3/8	137	9 1/4	235
45/64	17.50	0.6880	17.500	2	5 5/8	143	9 1/2	241
		0.7031	17.859	2	5 5/8	143	9 1/2	241
	18.00	0.7087	18.000	2	5 5/8	143	9 1/2	241
23/32		0.7188	18.258	2	5 5/8	143	9 1/2	241
	18.50	0.7283	18.500	2	5 7/8	149	9 3/4	248
47/64		0.7344	18.654	2	5 7/8	149	9 3/4	248
	19.00	0.7480	19.000	2	5 7/8	149	9 3/4	248
3/4		0.7500	19.050	2	5 7/8	149	9 3/4	248
49/64		0.7656	19.446	2	6	152	9 7/8	251
	19.50	0.7677	19.500	2	6	152	9 7/8	251
25/32		0.7812	19.843	2	6	152	9 7/8	251
	20.00	0.7821	20.000	3	6 1/8	156	10 3/4	273
51/64		0.7969	20.241	3	6 1/8	156	10 3/4	273
	20.50	0.8071	20.500	3	6 1/8	156	10 3/4	273
13/16		0.8125	20.638	3	6 1/8	156	10 3/4	273
	21.00	0.8268	21.000	3	6 1/8	156	10 3/4	273
53/64		0.8281	21.034	3	6 1/8	156	10 3/4	273
27/32		0.8438	21.433	3	6 1/8	156	10 3/4	273
	21.50	0.8465	21.500	3	6 1/8	156	10 3/4	273
55/64		0.8594	21.829	3	6 1/8	156	10 3/4	273
	22.00	0.8661	22.000	3	6 1/8	156	10 3/4	273
7/8		0.8750	22.225	3	6 1/8	156	10 3/4	273
	22.50	0.8858	22.500	3	6 1/8	156	10 3/4	273
57/64		0.8906	22.621	3	6 1/8	156	10 3/4	273
	23.00	0.9055	23.000	3	6 1/8	156	10 3/4	273
29/32		0.9062	23.017	3	6 1/8	156	10 3/4	273
59/64		0.9219	23.416	3	6 1/8	156	10 3/4	273
	23.50	0.9252	23.500	3	6 1/8	156	10 3/4	273
15/16		0.9375	23.862	3	6 1/8	156	10 3/4	273
	24.00	0.9449	24.000	3	6 3/8	162	11	279
61/64		0.9531	24.209	3	6 3/8	162	11	279
	24.50	0.9646	24.500	3	6 3/8	162	11	279
31/32		0.9688	24.608	3	6 3/8	162	11	279
	25.00	0.9843	25.000	3	6 3/8	162	11	279
63/64		0.9844	25.004	3	6 3/8	162	11	279
1		1.0000	25.400	3	6 3/8	162	11	279
	25.50	1.0039	25.500	3	6 1/2	165	11 1/8	283
1 1/64		1.0156	25.796	3	6 1/2	165	11 1/8	283
	26.00	1.0236	26.000	3	6 1/2	165	11 1/8	283
1 1/32		1.0312	26.192	3	6 1/2	165	11 1/8	283

CHAPTER 9

TWIST DRILLS

TABLE 9-4—Continued

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
	26.50	1.0433	26.500	3	6 5/8	168	11 1/4	286
1 3/64		1.0469	26.591	3	6 5/8	168	11 1/4	286
1 1/16		1.0625	26.988	3	6 5/8	168	11 1/4	286
	27.00	1.0630	27.000	3	6 5/8	168	11 1/4	286
1 5/64		1.0781	27.384	4	6 7/8	175	12 1/2	318
	27.50	1.0827	27.500	4	6 7/8	175	12 1/2	318
1 3/32		1.0938	27.783	4	6 7/8	175	12 1/2	318
	28.00	1.1024	28.000	4	7 1/8	181	12 3/4	324
1 7/64		1.1094	28.179	4	7 1/8	181	12 3/4	324
	28.50	1.1220	28.500	4	7 1/8	181	12 3/4	324
1 1/8		1.1250	28.575	4	7 1/8	181	12 3/4	324
1 9/64		1.1406	28.971	4	7 1/4	184	12 7/8	327
	29.00	1.1417	29.000	4	7 1/4	184	12 7/8	327
1 5/32		1.1562	29.367	4	7 1/4	184	12 7/8	327
	29.50	1.1614	29.500	4	7 3/8	187	13	330
1 1/64		1.1719	29.766	4	7 3/8	187	13	330
	30.00	1.1811	30.000	4	7 3/8	187	13	330
1 3/16		1.1875	30.162	4	7 3/8	187	13	330
	30.50	1.2008	30.500	4	7 1/2	190	13 1/8	333
1 13/64		1.2031	30.559	4	7 1/2	190	13 1/8	333
1 7/32		1.2188	30.958	4	7 1/2	190	13 1/8	333
	31.00	1.2205	31.000	4	7 7/8	200	13 1/2	343
1 15/64		1.2344	31.354	4	7 7/8	200	13 1/2	343
	31.50	1.2402	31.500	4	7 7/8	200	13 1/2	343
1 1/4		1.2500	31.750	4	7 7/8	200	13 1/2	343
	32.00	1.2598	32.000	4	8 1/2	216	14 1/8	359
1 17/64		1.2656	32.146	4	8 1/2	216	14 1/8	359
	32.50	1.2795	32.500	4	8 1/2	216	14 1/8	359
1 9/32		1.2812	32.542	4	8 1/2	216	14 1/8	359
1 19/64		1.2969	32.941	4	8 5/8	219	14 1/4	362
	33.00	1.2992	33.000	4	8 5/8	219	14 1/4	362
1 5/16		1.3125	33.338	4	8 5/8	219	14 1/4	362
	33.50	1.3189	33.500	4	8 3/4	222	14 3/8	365
1 21/64		1.3281	33.734	4	8 3/4	222	14 3/8	365
	34.00	1.3386	34.000	4	8 3/4	222	14 3/8	365
1 11/32		1.3438	34.133	4	8 3/4	222	14 3/8	365
	34.50	1.3583	34.500	4	8 7/8	225	14 1/2	368
1 23/64		1.3594	34.529	4	8 7/8	225	14 1/2	368
1 3/8		1.3750	34.925	4	8 7/8	225	14 1/2	368
	35.00	1.3780	35.000	4	9	229	14 5/8	371
1 25/64		1.3906	35.321	4	9	229	14 5/8	371
	35.50	1.3976	35.500	4	9	229	14 5/8	371
1 13/32		1.4062	35.717	4	9	229	14 5/8	371
	36.00	1.4173	36.000	4	9 1/8	232	14 3/4	375
1 27/64		1.4219	36.116	4	9 1/8	232	14 3/4	375

TABLE 9-4—Continued

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
	36.50	1.4370	36.500	4	9 1/8	232	14 3/4	375
1 7/16		1.4375	36.512	4	9 1/8	232	14 3/4	375
1 29/64		1.4531	36.909	4	9 1/4	235	14 7/8	378
	37.00	1.4567	37.000	4	9 1/4	235	14 7/8	378
1 15/32		1.4688	37.308	4	9 1/4	235	14 7/8	378
	37.50	1.4764	37.500	4	9 3/8	238	15	381
1 31/64		1.4844	37.704	4	9 3/8	238	15	381
	38.00	1.4961	38.000	4	9 3/8	238	15	381
1 1/2		1.5000	38.100	4	9 3/8	238	15	381
1 33/64		1.5156	38.496	---	---	---	---	---
1 17/32		1.5312	38.892	5	9 3/8	238	16 3/8	416
	39.00	1.5354	39.000	5	9 5/8	244	16 5/8	422
1 35/64		1.5469	39.291	---	---	---	---	---
1 9/16		1.5625	39.688	5	9 5/8	244	16 5/8	422
	40.00	1.5748	40.000	5	9 7/8	251	16 7/8	429
1 37/64		1.5781	40.084	---	---	---	---	---
1 19/32		1.5938	40.483	5	9 7/8	251	16 7/8	429
1 39/64		1.6094	40.879	---	---	---	---	---
	41.00	1.6142	41.000	5	10	254	17	432
1 5/8		1.6250	41.275	5	10	254	17	432
1 41/64		1.6406	41.671	---	---	---	---	---
	42.00	1.6535	42.000	5	10 1/8	257	17 1/8	435
1 21/32		1.6562	42.067	5	10 1/8	257	17 1/8	435
1 43/64		1.6719	42.466	---	---	---	---	---
1 11/16		1.6875	42.862	5	10 1/8	257	17 1/8	435
	43.00	1.6929	43.000	5	10 1/8	257	17 1/8	435
1 45/64		1.7031	43.259	---	---	---	---	---
1 23/32		1.7188	43.658	5	10 1/8	257	17 1/8	435
	44.00	1.7323	44.000	5	10 1/8	257	17 1/8	435
1 47/64		1.7344	44.054	---	---	---	---	---
1 3/4		1.7500	44.450	5	10 1/8	257	17 1/8	435
	45.00	1.7717	45.000	5	10 1/8	257	17 1/8	435
1 25/32		1.7812	45.242	5	10 1/8	257	17 1/8	435
	46.00	1.8110	46.000	5	10 1/8	257	17 1/8	435
1 13/16		1.8125	46.038	5	10 1/8	257	17 1/8	435
1 27/32		1.8438	46.833	5	10 1/8	257	17 1/8	435
	47.00	1.8504	47.000	5	10 3/8	264	17 3/8	441
1 7/8		1.8750	47.625	5	10 3/8	264	17 3/8	441
	48.00	1.8898	48.000	5	10 3/8	264	17 3/8	441
1 29/32		1.9062	48.417	5	10 3/8	264	17 3/8	441
	49.00	1.9291	49.000	5	10 3/8	264	17 3/8	441
1 15/16		1.9375	49.212	5	10 3/8	264	17 3/8	441
	50.00	1.9625	50.000	5	10 3/8	264	17 3/8	441
1 32/32		1.9688	50.008	5	10 3/8	264	17 3/8	441
2		2.0000	50.800	5	10 3/8	264	17 3/8	441

CHAPTER 9

TWIST DRILLS

TABLE 9-4—Continued

Fraction	Drill Diameter			Morse Taper No.	Flute Length		Overall Length	
	mm	in.	mm		in.	mm	in.	mm
2 1/32	51.00	2.0079	51.000	5	10 3/8	264	17 3/8	441
		2.0312	51.592	5	10 3/8	264	17 3/8	441
2 1/16	52.00	2.0472	52.000	5	10 1/4	260	17 3/8	441
		2.0625	52.388	5	10 1/4	260	17 3/8	441
2 3/32	53.00	2.0866	53.000	5	10 1/4	260	17 3/8	441
		2.0938	53.183	5	10 1/4	260	17 3/8	441
2 1/8	54.00	2.1250	53.975	5	10 1/4	260	17 3/8	441
		2.1260	54.000	5	10 1/4	260	17 3/8	441
2 5/32	55.00	2.1562	54.767	5	10 1/4	260	17 3/8	441
		2.1654	55.000	5	10 1/4	260	17 3/8	441
2 3/16	56.00	2.1875	55.512	5	10 1/4	260	17 3/8	441
		2.2000	56.000	5	10 1/8	257	17 3/8	441
2 7/32	57.00	2.2188	56.358	5	10 1/8	257	17 3/8	441
		2.2441	57.000	5	10 1/8	257	17 3/8	441
2 1/4	58.00	2.2500	57.150	5	10 1/8	257	17 3/8	441
		2.2835	58.000	5	10 1/8	257	17 3/8	441
2 5/16	59.00	2.3125	58.738	5	10 1/8	257	17 3/8	441
		2.3228	59.000	5	10 1/8	257	17 3/8	441
2 3/8	60.00	2.3622	60.000	5	10 1/8	257	17 3/8	441
		2.3750	60.325	5	10 1/8	257	17 3/8	441
2 7/16	61.00	2.4016	61.000	5	11 1/4	286	18 3/4	476
		2.4375	61.912	5	11 1/4	286	18 3/4	476
2 1/2	62.00	2.4409	62.000	5	11 1/4	286	18 3/4	476
		2.4803	63.000	5	11 1/4	286	18 3/4	476
2 9/16	63.00	2.5000	63.500	5	11 1/4	286	18 3/4	476
		2.5197	64.000	5	11 7/8	302	19 1/2	495
2 5/8	64.00	2.5591	65.000	5	11 7/8	302	19 1/2	495
		2.5625	65.088	5	11 7/8	302	19 1/2	495
2 11/16	65.00	2.5984	66.000	5	11 7/8	302	19 1/2	495
		2.6250	66.675	5	11 7/8	302	19 1/2	495
2 3/4	66.00	2.6378	67.000	5	12 3/4	324	20 3/8	518
		2.6772	68.000	5	12 3/4	324	20 3/8	518
2 13/16	67.00	2.6875	68.262	5	12 3/4	324	20 3/8	518
		2.7165	69.000	5	12 3/4	324	20 3/8	518
2 7/8	68.00	2.7500	69.850	5	12 3/4	324	20 3/8	518
		2.7559	70.000	5	13 3/8	340	21 1/8	537
2 15/16	69.00	2.7953	71.000	5	13 3/8	340	21 1/8	537
		2.8125	71.438	5	13 3/8	340	21 1/8	537
3	70.00	2.8346	72.000	5	13 3/8	340	21 1/8	537
		2.8740	73.000	5	13 3/8	340	21 1/8	537
3 1/8	71.00	2.8750	73.025	5	13 3/8	340	21 1/8	537
		2.9134	74.000	5	14	356	21 3/4	552
3 1/4	72.00	2.9375	74.612	5	14	356	21 3/4	552
		2.9528	75.000	5	14	356	21 3/4	552
3 1/2	73.00	2.9921	76.000	5	14	356	21 3/4	552
		3.0000	76.200	5	14	356	21 3/4	552
3 3/4	74.00	3.0315	77.000	6	14 5/8	371	24 1/2	622
		3.0709	78.000	6	14 5/8	371	24 1/2	622
3 7/8	75.00	3.1250	79.375	6	14 5/8	371	24 1/2	622
		3.2500	82.550	6	15 1/2	394	25 1/2	648

(ANSI Standard B94.11M-1979, published by ASME)

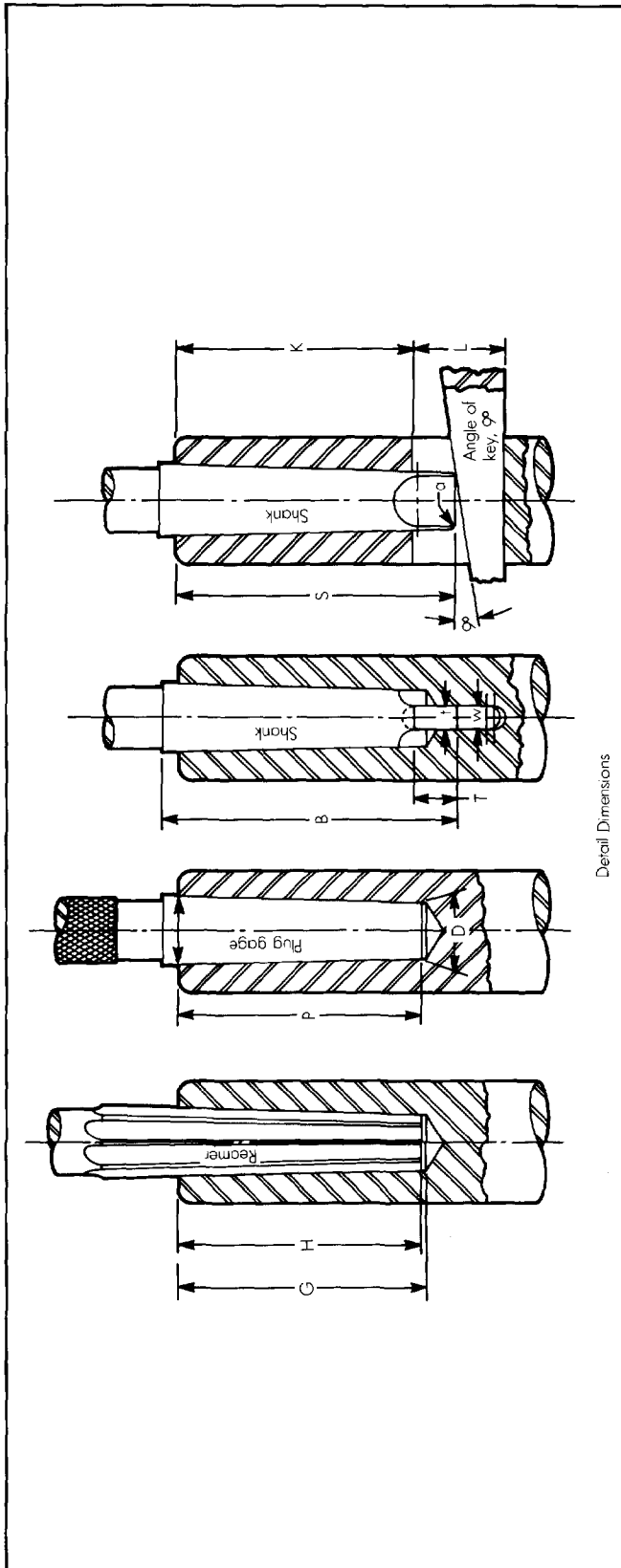


TABLE 9-5
Detail Dimensions of Morse Taper Shanks

Shank										Tang				Tang Slot				End of Socket to Tang Slot	Taper per Inch	Taper per Foot
Diam. of Plug at Small End	Diam. at End of Socket	Whole Length	Depth	Depth of Drilled Hole	Depth of Reamed Hole	Standard Plug Depth	Thickness	Length	Radius	Radius	a	W	L							
D	A	B	S	G	H	P	t	T	R	R	a	W	L	K						
0*	0.25200	0.35610	2 11/32	2 7/32	2 1/16	2 1/32	2	5/32	1/4	5/32	3/64	11/64	9/16	1 15/16	0.052050	0.62460				
1	0.36900	0.47500	2 9/16	2 7/16	2 3/16	2 5/32	2 1/8	13/64	3/8	3/16	3/64	7/32	3/4	2 1/16	0.049882	0.59858				
2	0.57200	0.30000	3 1/8	2 15/16	2 21/32	2 39/64	2 9/16	1/4	7/16	1/4	1/16	17/64	7/8	2 1/2	0.049951	0.59941				
3	0.77800	0.93800	3 7/8	3 11/16	3 5/16	3 1/4	3 3/16	5/16	9/16	9/32	5/64	21/64	1 3/16	3 1/16	0.050196	0.60235				
4	1.02000	1.23100	4 7/8	4 5/8	4 3/16	4 1/8	4 1/16	15/32	5/8	5/16	3/32	31/64	1 1/4	3 7/8	0.051938	0.62326				
4 1/2	1.26600	1.50000	5 3/8	5 1/8	4 5/8	4 9/16	4 1/2	9/16	11/16	3/8	1/8	37/64	1 3/8	4 5/16	0.052000	0.62400				
5	1.47500	1.74800	6 1/8	5 7/8	5 5/16	5 1/4	5 3/16	5/8	3/4	3/8	1/8	21/32	1 1/2	4 15/16	0.052626	0.63151				
5 1/2	2.11600	2.49400	8 9/16	8 1/4	7 13/32	7 21/64	7 1/4	3/4	1 1/8	1/2	5/32	25/32	1 3/4	7	0.052138	0.62565				
6	2.75000	3.27000	11 5/8	11 1/4	10 5/32	10 5/64	10	1 1/8	1 3/8	3/4	3/16	1 5/32	2 5/8	9 1/2	0.052000	0.62400				

(Cleveland Twist Drill Co.)

*Size 0 taper shank not listed in American National Standards.

CHAPTER 9

TWIST DRILLS

TABLE 9-6
Dimensions of Straight-Shank Twist Drills

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
97		0.0059	3/64"	3/4"				
	0.15	0.0059	1.25mm	19mm				
96		0.0063	3/64"	3/4"				
	0.16	0.0063	1.25mm	19mm				
95		0.0067	1/16"	3/4"				
	0.17	0.0067	1.25mm	19mm				
94		0.0071	1/16"	3/4"				
	0.18	0.0071	1.50mm	19mm				
93		0.0075	1/16"	3/4"				
	0.19	0.0075	1.50mm	19mm				
92		0.0079	1/16"	3/4"				
	0.20	0.0079	1.50mm	19mm				
91		0.0083	1/16"	3/4"				
90		0.0087	1/16"	3/4"				
	0.22	0.0087	1.75mm	19mm				
89		0.0091	1/16"	3/4"				
88		0.0095	5/64"	3/4"				
	0.25	0.0098	2.00mm	19mm				
87		0.0100	5/64"	3/4"				
86		0.0105	5/64"	3/4"				
85		0.0110	5/64"	3/4"				
	0.28	0.0110	2.25mm	19mm				
84		0.0115	3/32"	3/4"				
	0.30	0.0118	2.50mm	19mm				
83		0.0120	3/32"	3/4"				
82		0.0125	3/32"	3/4"				
	0.32	0.0126	2.50mm	19mm				
81		0.0130	3/32"	3/4"				
80		0.0135	1/8"	3/4"				
	0.35	0.0138	3.00mm	19mm				
79		0.0145	1/8"	3/4"				
	0.38	0.0150	5.00mm	19mm				
1/64"		0.0156	3/16"	3/4"				
	0.40	0.0157	5.00mm	22mm				
78		0.0160	3/16"	7/8"				
	0.42	0.0165	5.00mm	22mm				
	0.45	0.0177	5.00mm	22mm				
77		0.0180	3/16"	7/8"				
	0.48	0.0189	5.00mm	22mm				
	0.50	0.0197	5.00mm	22mm				
76		0.0200	3/16"	7/8"				
75		0.0210	1/4"	1"				
	0.55	0.0217	6.00mm	25mm				
74		0.0225	1/4"	1"				
	0.60	0.0236	8mm	29mm				
73		0.0240	5/16"	1 1/8"				
72		0.0250	5/16"	1 1/8"				
	0.65	0.0256	10mm	32mm				
71		0.0260	3/8"	1 1/4"				
	0.70	0.0276	10mm	32mm				
70		0.0280	3/8"	1 1/4"				
69		0.0292	1/2"	1 3/8"				
	0.75	0.0295	13mm	35mm				

Note: Standard screw-machine and taper-length straight-shank twist drills start with 1 mm diam sizes.

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
68		0.0310	1/2"	1 3/8"				
1/32"		0.0312	1/2"	1 3/8"				
	0.80	0.0315	13mm	35mm				
67		0.0320	1/2"	1 3/8"				
66		0.0330	1/2"	1 3/8"				
	0.85	0.0335	16mm	38mm				
65		0.0350	5/8"	1 1/2"				
	0.90	0.0354	16mm	38mm				
64		0.0360	5/8"	1 1/2"				
63		0.0370	5/8"	1 1/2"				
	0.95	0.0374	16mm	38mm				
62		0.0380	5/8"	1 1/2"				
61		0.0390	11/16"	1 5/8"				
	1.00	0.0394	18mm	41mm	13mm	35mm	29mm	76mm
60		0.0400	11/16"	1 5/8"	1/2"	1 3/8"	1 1/8"	2 1/4"
59		0.0410	11/16"	1 5/8"	1/2"	1 3/8"	1 1/8"	2 1/4"
	1.05	0.0413	18mm	41mm	13mm	35mm	29mm	76mm
58		0.0420	11/16"	1 5/8"	1/2"	1 3/8"	1 1/8"	2 1/4"
57		0.0430	3/4"	1 3/4"	1/2"	1 3/8"	1 1/8"	2 1/4"
	1.10	0.0433	19mm	44mm	13mm	35mm	29mm	76mm
	1.15	0.0453	19mm	44mm	13mm	35mm	29mm	76mm
56		0.0465	3/4"	1 3/4"	1/2"	1 3/8"	1 1/8"	2 1/4"
3/64"		0.0469	3/4"	1 3/4"	1/2"	1 3/8"	1 1/8"	2 1/4"
	1.20	0.0472	22mm	48mm	16mm	41mm	29mm	76mm
	1.25	0.0492	22mm	48mm	16mm	41mm	29mm	76mm
	1.30	0.0512	22mm	48mm	16mm	41mm	29mm	76mm
55		0.0520	7/8"	1 7/8"	5/8"	1 5/8"	1 3/4"	3"
	1.35	0.0531	22mm	48mm	16mm	41mm	29mm	76mm
54		0.0550	7/8"	1 7/8"	5/8"	1 5/8"	1 3/4"	3"
	1.40	0.0551	22mm	48mm	16mm	41mm	29mm	76mm
	1.45	0.0571	22mm	48mm	16mm	41mm	29mm	76mm
	1.50	0.0591	22mm	48mm	16mm	41mm	29mm	76mm
53		0.0595	7/8"	1 7/8"	5/8"	1 5/8"	1 3/4"	3"
	1.55	0.0610	22mm	48mm	16mm	41mm	29mm	76mm
1/16"		0.0625	7/8"	1 7/8"	5/8"	1 5/8"	1 3/4"	3"
	1.60	0.0630	22mm	48mm	18mm	43mm	51mm	95mm
52		0.0635	7/8"	1 7/8"	11/16"	1 11/16"	2"	3 3/4"
	1.65	0.0650	25mm	51mm	18mm	43mm	51mm	95mm
	1.70	0.0669	25mm	51mm	18mm	43mm	51mm	95mm
51		0.0670	1"	2"	11/16"	1 11/16"	2"	3 3/4"
	1.75	0.0689	25mm	51mm	18mm	43mm	51mm	95mm
50		0.0700	1"	2"	11/16"	1 11/16"	2"	3 3/4"
	1.80	0.0709	25mm	51mm	18mm	43mm	51mm	95mm
	1.85	0.0728	25mm	51mm	18mm	43mm	51mm	95mm
49		0.0730	1"	2"	11/16"	1 11/16"	2"	3 3/4"
	1.90	0.0748	25mm	51mm	18mm	43mm	51mm	95mm
48		0.0760	1"	2"	11/16"	1 11/16"	2"	3 3/4"
	1.95	0.0768	25mm	51mm	18mm	43mm	51mm	95mm
5/64"		0.0781	1"	2"	11/16"	1 11/16"	2"	3 3/4"
47		0.0785	1"	2"	11/16"	1 11/16"	2 1/4"	4 1/4"
	2.00	0.0787	29mm	54mm	18mm	43mm	57mm	108mm
	2.05	0.0807	29mm	54mm	19mm	44mm	57mm	108mm
46		0.0810	1 1/8"	2 1/8"	3/4"	1 3/4"	2 1/4"	4 1/4"

CHAPTER 9

TWIST DRILLS

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
45		0.0820	1 1/8"	2 1/8"	3/4"	1 3/4"	2 1/4"	4 1/4"
	2.10	0.0827	29mm	54mm	19mm	44mm	57mm	108mm
	2.15	0.0846	29mm	54mm	19mm	44mm	57mm	108mm
44		0.0860	1 1/8"	2 1/8"	3/4"	1 3/4"	2 1/4"	4 1/4"
	2.20	0.0866	32mm	57mm	19mm	44mm	57mm	108mm
	2.25	0.0886	32mm	57mm	19mm	44mm	57mm	108mm
43		0.0890	1 1/4"	2 1/4"	3/4"	1 3/4"	2 1/4"	4 1/4"
	2.30	0.0906	32mm	57mm	19mm	44mm	57mm	108mm
	2.35	0.0925	32mm	57mm	19mm	44mm	57mm	108mm
42		0.0935	1 1/4"	2 1/4"	3/4"	1 3/4"	2 1/4"	4 1/4"
3/32"		0.0938	1 1/4"	2 1/4"	3/4"	1 3/4"	2 1/4"	4 1/4"
	2.40	0.0945	35mm	60mm	21mm	46mm	64mm	117mm
41		0.0960	1 3/8"	2 3/8"	13/16"	1 13/16"	2 1/2"	4 5/8"
	2.45	0.0965	35mm	60mm	21mm	46mm	64mm	117mm
40		0.0980	1 3/8"	2 3/8"	13/16"	1 13/16"	2 1/2"	4 5/8"
	2.50	0.0984	35mm	60mm	21mm	46mm	64mm	117mm
39		0.0995	1 3/8"	2 3/8"	13/16"	1 13/16"	2 1/2"	4 3/8"
38		0.1015	1 7/16"	2 1/2"	13/16"	1 13/16"	2 1/2"	4 5/8"
	2.60	0.1024	37mm	64mm	21mm	46mm	64mm	117mm
37		0.1040	1 7/16"	2 1/2"	13/16"	1 13/16"	2 1/2"	4 5/8"
	2.70	0.1063	37mm	64mm	21mm	46mm	64mm	117mm
36		0.1065	1 7/16"	2 1/2"	13/16"	1 13/16"	2 1/2"	4 5/8"
7/64"		0.1094	1 1/2"	2 5/8"	13/16"	1 13/16"	2 1/2"	4 5/8"
35		0.1100	1 1/2"	2 5/8"	7/8"	1 7/8"	2 3/4"	5 1/8"
	2.80	0.1102	38mm	67mm	22mm	48mm	70mm	130mm
34		0.1110	1 1/2"	2 5/8"	7/8"	1 7/8"	2 3/4"	5 1/8"
33		0.1130	1 1/2"	2 5/8"	7/8"	1 7/8"	2 3/4"	5 1/8"
	2.90	0.1142	41mm	70mm	22mm	48mm	70mm	130mm
32		0.1160	1 5/8"	2 3/4"	7/8"	1 7/8"	2 3/4"	5 1/8"
	3.00	0.1181	41mm	70mm	22mm	48mm	70mm	130mm
31		0.1200	1 5/8"	2 3/4"	7/8"	1 7/8"	2 3/4"	5 1/8"
	3.10	0.1220	41mm	70mm	22mm	48mm	70mm	130mm
1/8"		0.1250	1 5/8"	2 3/4"	7/8"	1 7/8"	2 3/4"	5 1/8"
	3.20	0.1260	41mm	70mm	24mm	49mm	76mm	137mm
30		0.1285	1 5/8"	2 3/4"	15/16"	1 15/16"	3"	5 3/8"
	3.30	0.1299	45mm	73mm	24mm	49mm	76mm	137mm
	3.40	0.1339	45mm	73mm	24mm	49mm	76mm	137mm
29		0.1360	1 3/4"	2 7/8"	15/16"	1 15/16"	3"	5 3/8"
	3.50	0.1378	45mm	73mm	24mm	49mm	76mm	137mm
28		0.1405	1 3/4"	2 7/8"	15/16"	1 15/16"	3"	5 3/8"
9/64"		0.1406	1 3/4"	2 7/8"	15/16"	1 15/16"	3"	5 3/8"
	3.60	0.1417	48mm	76mm	25mm	52mm	76mm	137mm
27		0.1440	1 7/8"	3"	1"	2 1/16"	3"	5 3/8"
	3.70	0.1457	48mm	76mm	25mm	52mm	76mm	137mm
26		0.1470	1 7/8"	3"	1"	2 1/16"	3"	5 3/8"
25		0.1495	1 7/8"	3"	1"	2 1/16"	3"	5 3/8"
	3.80	0.1496	51mm	79mm	25mm	52mm	76mm	137mm
24		0.1520	2"	3 1/8"	1"	2 1/16"	3"	5 3/8"
	3.90	0.1535	51mm	79mm	25mm	52mm	76mm	137mm
23		0.1540	2"	3 1/8"	1"	2 1/16"	3"	5 3/8"
5/32"		0.1562	2"	3 1/8"	1"	2 1/16"	3"	5 3/8"
22		0.1570	2"	3 1/8"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
	4.00	0.1575	54mm	83mm	27mm	54mm	86mm	146mm

TWIST DRILLS

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
21		0.1590	2 1/8"	3 1/4"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
20		0.1610	2 1/8"	3 1/4"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
	4.10	0.1614	54mm	83mm	27mm	54mm	86mm	146mm
	4.20	0.1654	54mm	83mm	27mm	54mm	86mm	146mm
19		0.1660	2 1/8"	3 1/4"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
	4.30	0.1693	54mm	83mm	27mm	54mm	86mm	146mm
18		0.1695	2 1/8"	3 1/4"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
11/64"		0.1719	2 1/8"	3 1/4"	1 1/16"	2 1/8"	3 3/8"	5 3/4"
17		0.1730	2 3/16"	3 3/8"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
	4.40	0.1732	56mm	86mm	29mm	56mm	86mm	146mm
16		0.1770	2 3/16"	3 3/8"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
	4.50	0.1772	56mm	86mm	29mm	56mm	86mm	146mm
15		0.1800	2 3/16"	3 3/8"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
	4.60	0.1811	56mm	86mm	29mm	56mm	86mm	146mm
14		0.1820	2 3/16"	3 3/8"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
13		0.1850	2 5/16"	3 1/2"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
	4.70	0.1850	59mm	89mm	29mm	56mm	86mm	146mm
3/16"		0.1875	2 5/16"	3 1/2"	1 1/8"	2 3/16"	3 3/8"	5 3/4"
12		0.1890	2 5/16"	3 1/2"	1 3/16"	2 1/4"	3 5/8"	6"
	4.80	0.1890	59mm	89mm	30mm	57mm	92mm	152mm
11		0.1910	2 5/16"	3 1/2"	1 3/16"	2 1/4"	3 5/8"	6"
	4.90	0.1929	62mm	92mm	30mm	57mm	92mm	152mm
10		0.1935	2 7/16"	3 5/8"	1 3/16"	2 1/4"	3 5/8"	6"
9		0.1960	2 7/16"	3 5/8"	1 3/16"	2 1/4"	3 5/8"	6"
	5.00	0.1969	62mm	92mm	30mm	57mm	92mm	152mm
8		0.1990	2 7/16"	3 5/8"	1 3/16"	2 1/4"	3 5/8"	6"
	5.10	0.2008	62mm	92mm	30mm	57mm	92mm	152mm
7		0.2010	2 7/16"	3 5/8"	1 3/16"	2 1/4"	3 5/8"	6"
13/64"		0.2031	2 7/16"	3 5/8"	1 3/16"	2 1/4"	3 5/8"	6"
6		0.2040	2 1/2"	3 3/4"	1 1/4"	2 3/8"	3 5/8"	6"
	5.20	0.2047	64mm	95mm	32mm	60mm	92mm	152mm
5		0.2055	2 1/2"	3 3/4"	1 1/4"	2 3/8"	3 5/8"	6"
	5.30	0.2087	64mm	95mm	32mm	60mm	92mm	152mm
4		0.2090	2 1/2"	3 3/4"	1 1/4"	2 3/8"	3 5/8"	6"
	5.40	0.2126	64mm	95mm	32mm	60mm	92mm	152mm
3		0.2130	2 1/2"	3 3/4"	1 1/4"	2 3/8"	3 5/8"	6"
	5.50	0.2165	64mm	95mm	32mm	60mm	92mm	152mm
7/32"		0.2188	2 1/2"	3 3/4"	1 1/4"	2 3/8"	3 5/8"	6"
	5.60	0.2205	67mm	98mm	33mm	62mm	95mm	156mm
2		0.2210	2 5/8"	3 7/8"	1 5/16"	2 7/16"	3 3/4"	6 1/8"
	5.70	0.2244	67mm	98mm	33mm	62mm	95mm	156mm
1		0.2280	2 5/8"	3 7/8"	1 5/16"	2 7/16"	3 3/4"	6 1/8"
	5.80	0.2283	67mm	98mm	33mm	62mm	95mm	156mm
	5.90	0.2323	67mm	98mm	33mm	62mm	95mm	156mm
A		0.2340	2 5/8"	3 7/8"	1 5/16"	2 7/16"	3 3/4"	6 1/8"
15/64"		0.2344	2 5/8"	3 7/8"	1 5/16"	2 7/16"	3 3/4"	6 1/8"
	6.00	0.2362	70mm	102mm	35mm	64mm	95mm	156mm
B		0.2380	2 3/4"	4"	1 3/8"	2 1/2"	3 3/4"	6 1/8"
	6.10	0.2402	70mm	102mm	35mm	64mm	95mm	156mm
C		0.2420	2 3/4"	4"	1 3/8"	2 1/2"	3 3/4"	6 1/8"
	6.20	0.2441	70mm	102mm	35mm	64mm	95mm	156mm
D		0.2460	2 3/4"	4"	1 3/8"	2 1/2"	3 3/4"	6 1/8"
	6.30	0.2480	70mm	102mm	35mm	64mm	95mm	156mm

CHAPTER 9

TWIST DRILLS

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
1/4"/E		0.2500	2 3/4"	4"	1 3/8"	2 1/2"	3 3/4"	6 1/8"
	6.40	0.2520	73mm	105mm	37mm	67mm	98mm	159mm
	6.50	0.2559	73mm	105mm	37mm	67mm	98mm	159mm
F		0.2570	2 7/8"	4 1/8"	1 7/16"	2 5/8"	3 7/8"	6 1/4"
	6.60	0.2598	73mm	105mm	37mm	67mm		
G		0.2610	2 7/8"	4 1/8"	1 7/16"	2 5/8"	3 7/8"	6 1/4"
	6.70	0.2638	73mm	105mm	37mm	67mm		
17/64"		0.2656	2 7/8"	4 1/8"	1 7/16"	2 5/8"	3 7/8"	6 1/4"
H		0.2660	2 7/8"	4 1/8"	1 1/2"	2 11/16"	3 7/8"	6 1/4"
	6.80	0.2677	73mm	105mm	38mm	68mm	98mm	159mm
	6.90	0.2717	73mm	105mm	38mm	68mm		
I		0.2720	2 7/8"	4 1/8"	1 1/2"	2 11/16"	3 7/8"	6 1/4"
	7.00	0.2756	73mm	105mm	38mm	68mm	98mm	159mm
J		0.2770	2 7/8"	4 1/8"	1 1/2"	2 11/16"	3 7/8"	6 1/4"
	7.10	0.2795	75mm	108mm	38mm	68mm		
K		0.2810	2 15/16"	4 1/4"	1 1/2"	2 11/16"	3 7/8"	6 1/4"
9/32"		0.2812	2 15/16"	4 1/4"	1 1/2"	2 11/16"	3 7/8"	6 1/4"
	7.20	0.2835	75mm	108mm	40mm	70mm	102mm	162mm
	7.30	0.2874	75mm	108mm	40mm	70mm		
L		0.2900	2 15/16"	4 1/4"	1 9/16"	2 3/4"	4"	6 3/8"
	7.40	0.2913	78mm	111mm	40mm	70mm		
M		0.2950	3 1/16"	4 3/8"	1 9/16"	2 3/4"	4"	6 3/8"
	7.50	0.2953	78mm	111mm	40mm	70mm	102mm	162mm
19/64"		0.2969	3 1/16"	4 3/8"	1 9/16"	2 3/4"	4"	6 3/8"
	7.60	0.2992	78mm	111mm	41mm	71mm		
N		0.3020	3 1/16"	4 3/8"	1 5/8"	2 13/16"	4"	6 3/8"
	7.70	0.3031	81mm	114mm	41mm	71mm		
	7.80	0.3071	81mm	114mm	41mm	71mm	102mm	162mm
	7.90	0.3110	81mm	114mm	41mm	71mm		
5/16"		0.3125	3 3/16"	4 1/2"	1 5/8"	2 13/16"	4"	6 3/8"
	8.00	0.3150	3 3/16"	114mm	43mm	75mm	105mm	165mm
O		0.3160	3 3/16"	4 1/2"	1 11/16"	2 15/16"	4 1/8"	6 1/2"
	8.10	0.3189	81mm	114mm	43mm	75mm		
	8.20	0.3228	81mm	114mm	43mm	75mm	105mm	165mm
P		0.3230	3 5/16"	4 5/8"	1 11/16"	2 15/16"	4 1/8"	6 1/2"
	8.30	0.3268	81mm	114mm	43mm	75mm		
21/64"		0.3281	3 5/16"	4 5/8"	1 11/16"	2 15/16"	4 1/8"	6 1/2"
	8.40	0.3307	87mm	121mm	43mm	76mm		
Q		0.3320	3 7/16"	4 3/4"	1 11/16"	3"	4 1/8"	6 1/2"
	8.50	0.3346	87mm	121mm	43mm	76mm	105mm	165mm
	8.60	0.3386	87mm	121mm	43mm	76mm		
R		0.3390	3 7/16"	4 3/4"	1 11/16"	3"	4 1/8"	6 1/2"
	8.70	0.3425	87mm	121mm	43mm	76mm		
11/32"		0.3438	3 7/16"	4 3/4"	1 11/16"	3"	4 1/8"	6 1/2"
	8.80	0.3465	89mm	124mm	44mm	78mm	108mm	171mm
S		0.3480	3 1/2"	4 7/8"	1 3/4"	3 1/16"	4 1/4"	6 3/4"
	8.90	0.3504	89mm	124mm	44mm	78mm		
	9.00	0.3543	89mm	124mm	44mm	78mm	108mm	171mm
T		0.3580	3 1/2"	4 7/8"	1 3/4"	3 1/16"	4 1/4"	6 3/4"
	9.10	0.3583	89mm	124mm	44mm	78mm		
23/64"		0.3594	3 1/2"	4 7/8"	1 3/4"	3 1/16"	4 1/4"	6 3/4"
	9.20	0.3622	92mm	127mm	46mm	79mm	108mm	171mm
	9.30	0.3661	92mm	127mm	46mm	79mm		

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
U		0.3680	3 5/8"	5"	1 13/16"	3 1/8"	4 1/4"	6 3/4"
	9.40	0.3701	92mm	127mm	46mm	79mm		
	9.50	0.3740	92mm	127mm	46mm	79mm	108mm	171mm
3/8"		0.3750	3 5/8"	5"	1 13/16"	3 1/8"	4 1/4"	6 3/4"
V		0.3770	3 5/8"	5"	1 7/8"	3 1/4"	4 3/8"	7"
	9.60	0.3780	95mm	130mm	48mm	83mm		
	9.70	0.3819	95mm	130mm	48mm	83mm		
	9.80	0.3858	95mm	130mm	48mm	83mm	111mm	178mm
W		0.3860	3 3/4"	5 1/8"	1 7/8"	3 1/4"	4 3/8"	7"
	9.90	0.3898	95mm	130mm	48mm	83mm		
25/64"		0.3906	3 3/4"	5 1/8"	1 7/8"	3 1/4"	4 3/8"	7"
	10.00	0.3937	95mm	130mm	49mm	84mm	111mm	178mm
X		0.3970	3 3/4"	5 1/8"	1 15/16"	3 5/16"	4 3/8"	7"
	10.20	0.4016	98mm	133mm	49mm	84mm	111mm	178mm
Y		0.4040	3 7/8"	5 1/4"	1 15/16"	3 5/16"	4 3/8"	7"
	10.30	0.4055	98mm	133mm			111mm	178mm
13/32"		0.4062	3 7/8"	5 1/4"	1 15/16"	3 5/16"	4 3/8"	7"
Z		0.4130	3 7/8"	5 1/4"	2"	3 3/8"	4 5/8"	7 1/4"
	10.50	0.4134	100mm	137mm	51mm	86mm	117mm	184mm
27/64"		0.4219	3 15/16"	5 3/8"	2"	3 3/8"	4 5/8"	7 1/4"
	10.80	0.4252	103mm	140mm	52mm	87mm	117mm	184mm
	11.00	0.4331	103mm	140mm	52mm	87mm	117mm	184mm
7/16"		0.4375	4 1/16"	5 1/2"	2 1/16"	3 7/16"	4 5/8"	7 1/4"
	11.20	0.4409	106mm	143mm	54mm	90mm	121mm	190mm
	11.50	0.4528	106mm	143mm	54mm	90mm	121mm	190mm
29/64"		0.4531	4 3/16"	5 5/8"	2 1/8"	3 9/16"	4 3/4"	7 1/2"
	11.80	0.4646	110mm	146mm	54mm	92mm	121mm	190mm
15/32"		0.4688	5 5/16"	5 3/4"	2 1/8"	3 5/8"	4 3/4"	7 1/2"
	12.00	0.4724	111mm	150mm	56mm	94mm	121mm	197mm
	12.20	0.4803	111mm	150mm	56mm	94mm	121mm	197mm
31/64"		0.4844	4 3/8"	5 7/8"	2 3/16"	3 11/16"	4 3/4"	7 3/4"
	12.50	0.4921	114mm	152mm	57mm	95mm	121mm	197mm
1/2"		0.5000	4 1/2"	6"	2 1/4"	3 3/4"	4 3/4"	7 3/4"
	13.00	0.5118	122mm	168mm	60mm	98mm	121mm	203mm
33/64"		0.5156	4 13/16"	6 5/8"	2 3/8"	3 7/8"	4 3/4"	8"
17/32"		0.5312	4 13/16"	6 5/8"	2 3/8"	3 7/8"	4 3/4"	8"
	13.50	0.5315	122mm	168mm			124mm	210mm
35/64"		0.5469	4 13/16"	6 5/8"	2 1/2"	4"	4 7/8"	8 1/4"
	14.00	0.5512	122mm	168mm			124mm	210mm
9/16"		0.5625	4 13/16"	6 5/8"	2 1/2"	4"	4 7/8"	8 1/4"
	14.50	0.5709	122mm	168mm			124mm	222mm
37/64"		0.5781	4 13/16"	6 5/8"	2 5/8"	4 1/8"	4 7/8"	8 3/4"
	15.00	0.5906	132mm	181mm			124mm	222mm
19/32"		0.5938	5 3/16"	7 1/8"	2 5/8"	4 1/8"	4 7/8"	8 3/4"
39/64"		0.6094	5 3/16"	7 1/8"	2 3/4"	4 1/4"	4 7/8"	8 3/4"
	15.50	0.6102	132mm	181mm			124mm	222mm
5/8"		0.6250	5 3/16"	7 1/8"	2 3/4"	4 1/4"	4 7/8"	8 3/4"
	16.00	0.6299	132mm	181mm			130mm	229mm
41/64"		0.6406	5 3/16"	7 1/8"	2 7/8"	4 1/2"	5 1/8"	9"
	16.50	0.6496	132mm	181mm			130mm	229mm
21/32"		0.6562	5 3/16"	7 1/8"	2 7/8"	4 1/2"	5 1/8"	9"
	17.00	0.6693	143mm	194mm			137mm	235mm
43/64"		0.6719	5 5/8"	7 5/8"	2 7/8"	4 5/8"	5 3/8"	9 1/4"

CHAPTER 9

TWIST DRILLS

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
11/16"		0.6875	5 5/8"	7 5/8"	2 7/8"	4 5/8"	5 3/8"	9 1/4"
	17.50	0.6890	143mm	194mm			143mm	241mm
45/64"		0.7031			3"	4 3/4"	5 5/8"	9 1/2"
	18.00	0.7087					143mm	241mm
23/32"		0.7188			3"	4 3/4"	5 5/8"	9 1/2"
	18.50	0.7283					149mm	248mm
47/64"		0.7344			3 1/8"	5"	5 7/8"	9 3/4"
	19.00	0.7480					149mm	248mm
3/4"		0.7500			3 1/8"	5"	5 7/8"	9 3/4"
49/64"		0.7656			3 1/4"	5 1/8"	6"	9 7/8"
	19.50	0.7677					152mm	251mm
25/32"		0.7812			3 1/4"	5 1/8"	6"	9 7/8"
	20.00	0.7874					156mm	254mm
51/64"		0.7969			3 3/8"	5 1/4"	6 1/8"	10"
	20.50	0.8071					156mm	254mm
13/16"		0.8125			3 3/8"	5 1/4"	6 1/8"	10"
	21.00	0.8268					156mm	254mm
53/64"		0.8281			3 1/2"	5 3/8"	6 1/8"	10"
27/32"		0.8438			3 1/2"	5 3/8"	6 1/8"	10"
	21.50	0.8465					156mm	254mm
55/64"		0.8594			3 1/2"	5 1/2"	6 1/8"	10"
	22.00	0.8661					156mm	254mm
7/8"		0.8750			3 1/2"	5 1/2"	6 1/8"	10"
	22.50	0.8858					156mm	254mm
57/64"		0.8906			3 5/8"	5 5/8"	6 1/8"	10"
	23.00	0.9055					156mm	254mm
29/32"		0.9062			3 5/8"	5 5/8"	6 1/8"	10"
59/64"		0.9219			3 3/4"	5 3/4"	6 1/8"	10 3/4"
	23.50	0.9252					156mm	273mm
15/16"		0.9375			3 3/4"	5 3/4"	6 1/8"	10 3/4"
	24.00	0.9449					162mm	279mm
61/64"		0.9531			3 7/8"	5 7/8"	6 3/8"	11"
	24.50	0.9646					162mm	279mm
31/32"		0.9688			3 7/8"	5 7/8"	6 3/8"	11"
	25.00	0.9843					162mm	279mm
63/64"		0.9844			4"	6"	6 3/8"	11"
1"		1.0000			4"	6"	6 3/8"	11"
	25.50	1.0039					165mm	283mm
1 1/64"		1.0156					6 1/2"	11 1/8"
	26.00	1.0236					165mm	283mm
1 1/32"		1.0312					6 1/2"	11 1/8"
	26.50	1.0433					168mm	286mm
1 3/64"		1.0469					6 5/8"	11 1/4"
1 1/16"		1.0625			4"	6 1/4"	6 5/8"	11 1/4"
	27.00	1.0630					175mm	292mm
1 5/64"		1.0781					6 7/8"	11 1/2"
	27.50	1.0827					175mm	292mm
1 3/32"		1.0938					6 7/8"	11 1/2"
	28.00	1.1024					181mm	298mm
1 7/64"		1.1094					7 1/8"	11 3/4"
	28.50	1.1220					181mm	298mm
1 1/8"		1.1250			4"	6 3/8"	7 1/8"	11 3/4"
1 9/64"		1.1406					7 1/4"	11 7/8"
	29.00	1.1417					184mm	302mm

TABLE 9-6—Continued

Diameter			Jobbers Length		Screw-Machine Length		Taper Length	
Fractional, Number (wire gage), & Letter	Metric, mm	Decimal equivalent, inches	Flute length	Overall length	Flute length	Overall length	Flute length	Overall length
1 5/32"		1.1562					7 1/4"	11 7/8"
	29.50	1.1614					187mm	305mm
1 11/64"		1.1719					7 3/8"	12"
	30.00	1.1811					187mm	305mm
1 3/16"		1.1875			4 1/4"	6 5/8"	7 3/8"	12"
	30.50	1.2008					190mm	308mm
1 13/64"		1.2031					7 1/2"	12 1/8"
1 7/32"		1.2188					7 1/2"	12 1/8"
	31.00	1.2205					200mm	318mm
1 15/64"		1.2344					7 7/8"	12 1/2"
	31.50	1.2402					200mm	318mm
1 1/4"		1.2500			4 3/8"	6 3/4"	7 7/8"	12 1/2"
	32.00	1.2598					216mm	359mm
	32.50	1.2795					216mm	359mm
1 9/32"		1.2812					8 1/2"	14 1/8"
	33.00	1.2992					219mm	362mm
1 5/16"		1.3125			4 3/8"	7"	8 5/8"	14 1/4"
	33.50	1.3189					222mm	365mm
	34.00	1.3386					222mm	365mm
1 11/32"		1.3438					8 3/4"	14 3/8"
	34.50	1.3583					225mm	368mm
1 3/8"		1.3750			4 1/2"	7 1/8"	8 7/8"	14 1/2"
	35.00	1.3780					229mm	371mm
	35.50	1.3976					229mm	371mm
1 13/32"		1.4062					9"	14 5/8"
	36.00	1.4173					232mm	375mm
	36.50	1.4370					232mm	375mm
1 7/16"		1.4375			4 3/4"	7 3/8"	9 1/8"	14 3/4"
	37.00	1.4567					235mm	378mm
1 15/32"		1.4688					9 1/4"	14 7/8"
	37.50	1.4764					238mm	381mm
	38.00	1.4961					238mm	381mm
1 1/2"		1.5000			4 7/8"	7 1/2"	9 3/8"	15"
1 9/16"		1.5625			4 7/8"	7 3/4"	9 5/8"	15 1/4"
1 5/8"		1.6250			4 7/8"	7 3/4"	9 7/8"	15 5/8"
1 11/16"		1.6875			5 1/8"	8"		
1 3/4"		1.7500			5 1/8"	8"	10 1/2"	16 1/4"
1 13/16"		1.8125			5 3/8"	8 1/4"		
1 7/8"		1.8750			5 3/8"	8 1/4"		
1 15/16"		1.9375			5 5/8"	8 1/2"		
2"		2.0000			5 5/8"	8 1/2"		

(Cleveland Twist Drill Co.)

good rigidity, thus reducing runout. Larger size center drills are sometimes used for starting operations when drilling is done on screw machines and other machine tools to increase the location accuracies of holes to be produced.

When the design of the workpiece permits, the countersink portion of the center-drilled hole should be 0.003-0.006" (0.08-0.15 mm) larger in diameter than the finished hole size. The included angle of the center drill is sometimes made less than the included angle of the drill that follows.

Spotting and centering drills. These straight-shank twist drills are designed to produce accurate and true centers. They have short flute and overall lengths, and no body clearance, which allows them to be chucked close to the cutting point for

accurate hole starting or workpiece centering. A constant web thickness eliminates the need for thinning these drills when repointing.

Holes that are to be drilled and chamfered in workpieces having a hard or scaled surface can be produced more economically by spot drilling (to form the chamfer) prior to drilling. With this method, the spot drill cuts only a short time through stock that has a detrimental effect on tool life, while the hole drill cuts for a longer time through easier-to-machine material. Processing in this way allows the drills to be rapidly advanced closer to the workpiece, thus decreasing the feed stroke and allowing a decrease in the feed rate to increase tool life.

CHAPTER 9

TWIST DRILLS

Left-hand drills. These tools are of the same basic design as general-purpose drills, but they are made for left-hand cutting. They are commonly used in screw machines in which tapping operations require left-hand cutting tools.

Double-margin drills. These tools are often used when holes must be straighter and to closer tolerances with respect to size. They have a second pair of margins at the trailing edges of the lands, about midway between the primary margins (see Fig. 9-21), thus providing added bearing in bushings and holes with four guide surfaces. This construction also helps to eliminate the effects of errors in point grinding. Double margins are also used on some step drills for producing accurate holes when guide bushings cannot be used.

Circuit-board drills. These solid carbide tools are used extensively to drill numerous small-diameter holes in printed circuit boards. The boards generally consist of an insulating epoxy-resin, glass-based material coated with a thin layer of conductive copper. The abrasive nature of the material drilled,

smallness of the hole sizes, and high-speed operation of the machines used make carbide drills essential. Most circuit-board drills have a 35° helix angle, four-facet point, tapered web, and back taper. They are available in fractional, number, and metric sizes, with 1/8" (3.2 mm) diam straight shanks and 1 1/2" (38 mm) overall lengths.

Chipbreaker drills. These tools have flute modifications or other design features to curl and break the chips produced in most materials in which long stringy chips are normally produced, thus minimizing clogging problems. Most of various chipbreaker designs involve a flute configuration that forces long chips to curl and bend sufficiently so that small broken chips are produced.

Transverse sections through several chipbreaker drills are illustrated in Fig. 9-22. Such flute designs are built into the drills and no additional notches or grooves are required when regrounding. For some applications, particularly with larger size twist drills, chip curling grooves are ground in the rake faces of

TABLE 9-7
General Dimensions of Three and Four-Flute Core Drills,
Fractional Sizes Only

Drill Diameter			Straight Shank							
			Three Flute				Four Flute			
			Flute Length		Overall Length		Flute Length		Overall Length	
Fraction	in.	mm	in.	mm	in.	mm	in.	mm	in.	mm
1/4	0.2500	6.350	3 3/4	95	6 1/8	156	---	---	---	---
9/32	0.2812	7.142	3 7/8	98	6 1/4	159	---	---	---	---
5/16	0.3125	7.938	4	102	6 3/8	162	---	---	---	---
11/32	0.3438	8.733	4 1/8	105	6 1/2	165	---	---	---	---
3/8	0.3750	9.525	4 1/8	105	6 3/4	171	---	---	---	---
13/32	0.4062	10.317	4 3/8	111	7	178	---	---	---	---
7/16	0.4375	11.112	4 5/8	117	7 1/4	184	---	---	---	---
15/32	0.4688	11.908	4 3/4	121	7 1/2	190	---	---	---	---
1/2	0.5000	12.700	4 3/4	121	7 3/4	197	4 3/4	121	7 3/4	197
17/32	0.5312	13.492	4 3/4	121	8	203	4 3/4	121	8	203
9/16	0.5625	14.288	4 7/8	124	8 1/4	210	4 7/8	124	8 1/4	210
19/32	0.5938	15.083	4 7/8	124	8 3/4	222	4 7/8	124	8 3/4	222
5/8	0.6250	15.875	4 7/8	124	8 3/4	222	4 7/8	124	8 3/4	222
21/32	0.6562	16.667	5 1/8	130	9	229	5 1/8	130	9	229
11/16	0.6875	17.462	5 3/8	137	9 1/4	235	5 3/8	137	9 1/4	235
23/32	0.7188	18.258	---	---	---	---	5 5/8	143	9 1/2	241
3/4	0.7500	19.050	5 7/8	149	9 3/4	248	5 7/8	149	9 3/4	248
25/32	0.7812	19.842	---	---	---	---	6	152	9 7/8	251
13/16	0.8125	20.638	---	---	---	---	6 1/8	156	10	254
27/32	0.8438	21.433	---	---	---	---	6 1/8	156	10	254
7/8	0.8750	22.225	---	---	---	---	6 1/8	156	10	254
29/32	0.9062	23.017	---	---	---	---	6 1/8	156	10	254
15/16	0.9375	23.812	---	---	---	---	6 1/8	156	10 3/4	273
31/32	0.9688	24.608	---	---	---	---	6 3/8	162	11	279
1	1.0000	25.400	---	---	---	---	6 3/8	162	11	279
1 1/32	1.0312	26.192	---	---	---	---	6 1/2	165	11 1/8	283
1 1/16	1.0625	26.988	---	---	---	---	6 5/8	168	11 1/4	286
1 3/32	1.0938	27.783	---	---	---	---	6 7/8	175	11 1/2	292
1 1/8	1.1250	28.575	---	---	---	---	7 1/8	181	11 3/4	298
1 1/4	1.2500	31.750	---	---	---	---	7 7/8	200	12 1/2	318

TABLE 9-7—Continued

Straight Shank												
Drill Diameter			Three Flute				Four Flute					
			Morse Taper No.	Flute Length in.	Flute Length mm	Overall Length in.	Overall Length mm	Morse Taper No.	Flute Length in.	Flute Length mm	Overall Length in.	Overall Length mm
Fraction	in.	mm	No.	in.	mm	in.	mm	No.	in.	mm	in.	mm
1/4	0.2500	6.350	1	2 7/8	73	6 1/8	156	---	---	---	---	---
9/32	0.2812	7.142	1	3	76	6 1/4	159	---	---	---	---	---
5/16	0.3175	7.938	1	3 1/8	79	6 3/8	162	---	---	---	---	---
11/32	0.3438	8.733	1	3 1/4	83	6 1/2	165	---	---	---	---	---
3/8	0.3750	9.525	1	3 1/2	89	6 3/4	171	---	---	---	---	---
13/32	0.4062	10.217	1	3 5/8	92	7	178	---	---	---	---	---
7/16	0.4375	11.112	1	3 7/8	98	7 1/4	184	---	---	---	---	---
15/32	0.4688	11.908	1	4 1/8	105	7 1/2	190	---	---	---	---	---
1/2	0.5000	12.700	2	4 3/8	111	8 1/4	210	2	4 3/8	111	8 1/4	210
17/32	0.5312	13.492	2	4 5/8	117	8 1/2	216	2	4 5/8	117	8 1/2	216
9/16	0.5625	14.288	2	4 7/8	124	8 3/4	222	2	4 7/8	124	8 3/4	222
19/32	0.5938	15.083	2	4 7/8	124	8 3/4	222	2	4 7/8	124	8 3/4	222
5/8	0.6250	15.815	2	4 7/8	124	8 3/4	222	2	4 7/8	124	8 3/4	222
21/32	0.6562	16.668	2	5 1/8	130	9	229	2	5 1/8	130	9	229
11/16	0.6875	17.462	2	5 3/8	137	9 1/4	235	2	5 3/8	137	9 1/4	235
23/32	0.7188	18.258	2	5 5/8	143	9 1/2	241	2	5 5/8	143	9 1/2	241
3/4	0.7500	19.050	2	5 7/8	149	9 3/4	248	2	5 7/8	149	9 3/4	248
25/32	0.7812	19.842	2	6	152	9 7/8	251	2	6	152	9 7/8	251
13/16	0.8125	20.638	3	6 1/8	156	10 3/4	273	3	6 1/8	156	10 3/4	273
27/32	0.8438	21.433	3	6 1/8	156	10 3/4	273	3	6 1/8	156	10 3/4	273
7/8	0.8750	22.225	3	6 1/8	156	10 3/4	273	3	6 1/8	156	10 3/4	273
29/32	0.9062	23.000	3	6 1/8	156	10 3/4	273	3	6 1/8	156	10 3/4	273
15/16	0.9375	23.812	3	6 1/8	156	10 3/4	273	3	6 1/8	156	10 3/4	273
31/32	0.9688	24.608	3	6 3/8	162	11	279	3	6 3/8	162	11	279
1	1.0000	25.400	3	6 3/8	162	11	279	3	6 3/8	162	11	279
1 1/32	1.0312	26.192	3	6 1/2	165	11 1/8	283	3	6 1/2	165	11 1/8	283
1 1/16	1.0625	26.988	3	6 5/8	168	11 1/4	286	3	6 5/8	168	11 1/4	286
1 3/32	1.0938	27.783	4	6 7/8	175	12 1/2	318	4	6 7/8	175	12 1/2	318
1 1/8	1.1250	28.575	4	7 1/8	181	12 3/4	324	4	7 1/8	181	12 3/4	324
1 5/32	1.1562	29.367	4	7 1/4	184	12 7/8	327	4	7 1/4	184	12 7/8	327
1 3/16	1.1875	30.162	4	7 3/8	187	13	330	4	7 3/8	187	13	330
1 7/32	1.2188	30.958	4	7 1/2	190	13 1/8	333	4	7 1/2	190	13 1/8	333
1 1/4	1.2500	31.750	4	7 7/8	200	13 1/2	343	4	7 7/8	200	13 1/2	343
1 9/32	1.2812	32.542	---	---	---	---	---	4	8 1/2	216	14 1/8	359
1 5/16	1.3125	33.338	---	---	---	---	---	4	8 5/8	219	14 1/4	362
1 11/32	1.3438	34.133	---	---	---	---	---	4	8 3/4	222	14 3/8	365
1 3/8	1.3750	34.925	---	---	---	---	---	4	8 7/8	225	14 1/2	368
1 13/32	1.4062	35.717	---	---	---	---	---	4	9	229	14 5/8	371
1 7/16	1.4375	36.512	---	---	---	---	---	4	9 1/8	232	14 3/4	375
1 15/32	1.4683	37.300	---	---	---	---	---	4	9 1/4	235	14 7/8	378
1 1/2	1.5000	38.100	---	---	---	---	---	4	9 3/8	238	15	381
1 17/32	1.5312	38.892	---	---	---	---	---	5	9 3/8	238	16 3/8	416
1 9/16	1.5675	39.688	---	---	---	---	---	5	9 5/8	244	16 5/8	422
1 19/32	1.5938	40.483	---	---	---	---	---	5	9 7/8	251	16 7/8	429
1 5/8	1.6250	41.275	---	---	---	---	---	5	10	254	17	432
1 21/32	1.6562	42.067	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 11/16	1.6875	42.862	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 23/32	1.7188	43.658	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 3/4	1.7500	44.450	---	---	---	---	---	5	10 1/8	257	17 1/8	435

CHAPTER 9

TWIST DRILLS

TABLE 9-7—Continued

Straight Shank												
Drill Diameter			Three Flute				Four Flute					
			Morse Taper No.	Flute Length in.	Flute Length mm	Overall Length in.	Overall Length mm	Morse Taper No.	Flute Length in.	Flute Length mm	Overall Length in.	Overall Length mm
Fraction	in.	mm										
1 25/32	1.7812	45.262	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 13/16	1.8125	46.038	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 27/32	1.0000	46.833	---	---	---	---	---	5	10 1/8	257	17 1/8	435
1 7/8	1.8750	47.625	---	---	---	---	---	5	10 3/8	264	17 3/8	441
1 29/32	1.9062	48.417	---	---	---	---	---	5	10 3/8	264	17 3/8	441
1 15/16	1.9375	49.212	---	---	---	---	---	5	10 3/8	264	17 3/8	441
1 31/32	1.9688	50.008	---	---	---	---	---	5	10 3/8	264	17 3/8	441
2	2.0000	50.800	---	---	---	---	---	5	10 3/8	264	17 3/8	441
2 1/8	2.1250	53.975	---	---	---	---	---	5	10 1/4	260	17 3/8	441
2 1/4	2.2500	57.150	---	---	---	---	---	5	10 1/8	257	17 3/8	441
2 3/8	2.3750	60.325	---	---	---	---	---	5	10 1/8	257	17 3/8	441
2 1/2	2.5000	63.500	---	---	---	---	---	5	11 1/4	286	18 3/4	476

(ANSI Standard B94.11 M-1979, published by ASME)

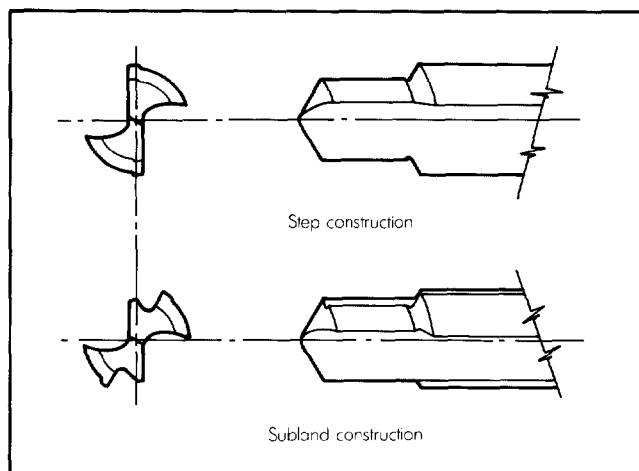


Fig. 9-17 Two types of construction used for multidiameter drills. (National Twist Drill)

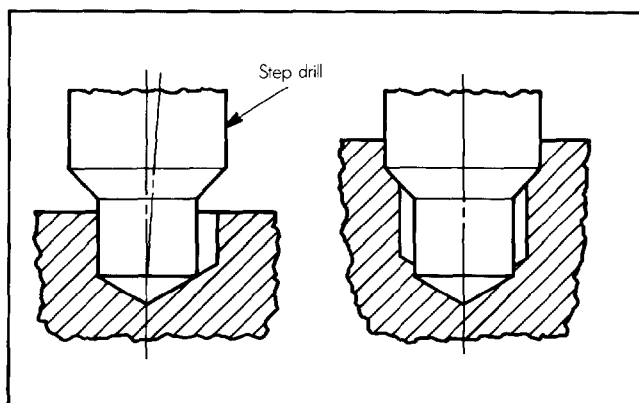


Fig. 9-18 When the smaller diameter pilot of a step drill enters a hole, it may drill oversized (left). When a larger diameter drill body enters the hole, it centers the pilot and produces a hole to close tolerance.

the cutting lips, or chip splitting grooves are ground in either the clearance or rake faces. Chipbreaker drills sometimes require more torque than standard drills, and there is a minimum feed rate below which they do not operate effectively.

Automotive series drills. These tools are straight-shank twist drills of general-purpose construction. They have the same flute and overall lengths as regular jobbers-length drills and are furnished with tang drive. They are also available in a long series with taper lengths and tang drive.

Crankshaft drills. These tools are designed for drilling holes in crankshafts, but are also used to produce holes deeper than five or six times the drill diameter in any tough or hard material. They have heavy webs, a higher than normal helix angle, a split or crankshaft point, and a surface treatment for increased abrasion resistance.

Cotter-pin drills. These are heavy-duty jobbers-length drills designed for drilling cotter-pin holes in bolts, steering knuckles, pins, and similar parts, as well as cross holes in the heads of bolts. Heavy construction resists the high strains from applications such as these and makes them suitable for drilling some of the hard alloy steels, including the harder stainless steels. Such drills are generally surface treated.

Rail drills. These heavy-duty tools, originally designed for drilling high-manganese railroad rails, are now being used to drill a variety of hard, tough, high tensile strength and high temperature resistant materials. They are made with a low helix angle, heavy web, special web thinning for ease of penetration, and short flute and overall lengths, and are surface treated.

Pipe-flange twist drills. These tools, with special geometry and surface treatment, are designed to produce accurate holes in pipe flanges, valves, and fittings made from iron castings.

Drills for plastics. These tools are designed with wide flutes and a low helix angle for use in drilling many plastics, fibrous materials, asbestos, and hard rubber. They have a special surface treatment to resist the abrasive action of these materials.

Aircraft drills. Tools used extensively by the aircraft and aerospace industries include extension drills. These tools have a flute construction conforming to National Aerospace Standard

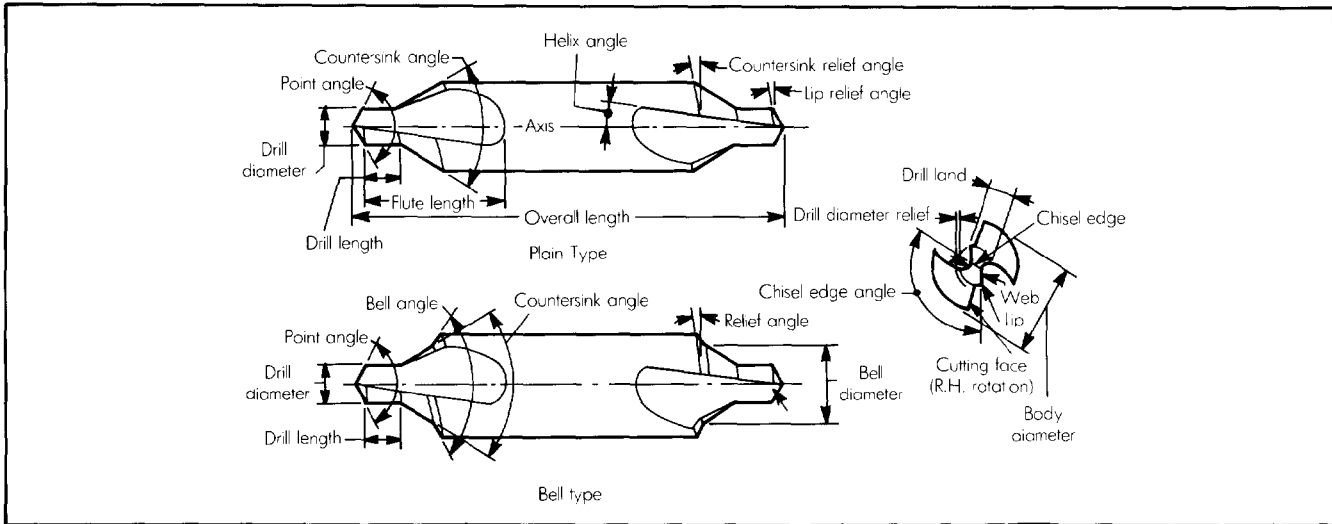


Fig. 9-19 Identifying features of plain and bell types of combined drill and countersink. (ANSI B94.11M-1979, published by ASME)

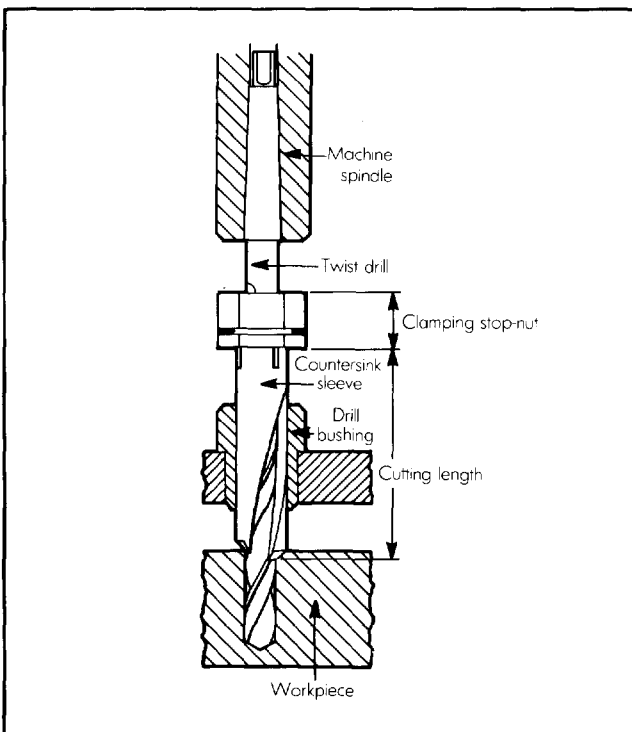


Fig. 9-20 Two-piece step drill consisting of a separate countersink sleeve and a standard drill. (Guhring, Inc.)

(NAS) 907 prepared by the Aerospace Industries Association, spring-tempered shanks to prevent permanent bending, and overall lengths of 6 or 12" (152 or 305 mm). They are available with either 118° or 135° split or offset drill points. These drills are used principally for drilling holes in inaccessible places that cannot be reached with regular length drills; they are generally operated in portable drilling units.

Straight-shank, jobbers-length aircraft drills conforming to NAS 907 are available in two types. Type A drills, with 118° split points, are used for drilling aluminum, mild steels,

magnesium, and other soft materials. Type B drills, with 135° split points, are of heavier design and are surface treated for drilling hard steels, stainless steels, titanium, and other hard materials.

Short-flute aircraft drills with 118° points are desirable for tough, portable applications in titanium, heat-treated steels, and aluminum. These drills are surface treated and have no body clearance. Type C drills, with 135° split points, are made in screw-machine lengths for use in machines or portable units to drill hard, tough sheet metals of the heat-resistant stainless and titanium alloy types.

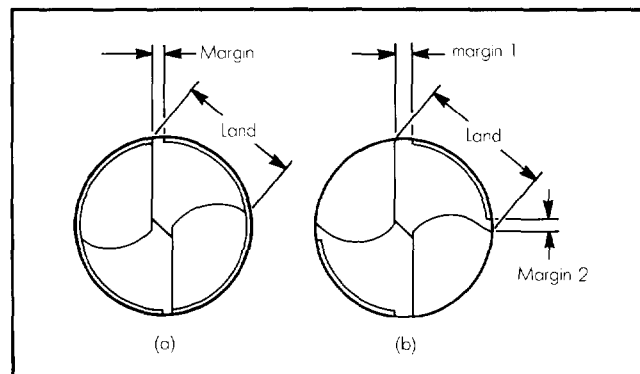


Fig. 9-21 (a) Conventional twist drill; (b) double-margin drill. (National Twist Drill)

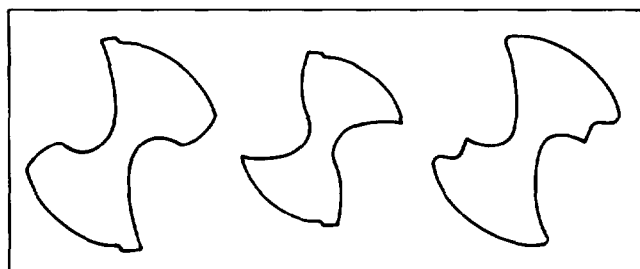


Fig. 9-22 Transverse sections through several types of chipbreaker drills. (National Twist Drill)

CHAPTER 9

TWIST DRILLS

Tools for portable drilling. These drills, which can be held in the chucks of portable electric or air drilling units, are available with 1/4" (6.35 mm) or 1/2" (12.7 mm) diam shanks. Flute and overall lengths are shorter than those for general-purpose drills to provide increased rigidity. Body or skin drills, also often used in portable drilling units for producing holes in sheet metal, are of similar construction.

Sheeters drills are also of similar design, but they have heavier webs, 135° offset points, and shanks of the same diameters as the fluted portions of the drills. They are used to drill holes in corrugated or flat metal sheeting or panels for building erection, for drilling truck and automobile bodies, and for other general-purpose portable drilling.

Threaded-shank drills. These tools are used with angular drives and flexible shafts for working in close quarters in which conventional types of portable drills would be too bulky.

Rotabroach cutters. These tools (see Fig. 9-23), made by Hougen Manufacturing Co., are difficult to classify, but they do compete with conventional twist drills except in the production of blind holes. They resemble heavy-duty hole saws, as well as trepanning tools, in that they produce slugs of solid material rather than just chips. Standard tools are available to produce through holes to 3" (76 mm) diam in materials to 3" thick, and each cutter has from 7 to 22 teeth, depending upon diameter. Special tools have been built that are 7" (178 mm) diam and cut through holes 10" (254 mm) deep. The introduction of a cutting fluid through the machine spindle or an inducer, a slug ejection system, and a rigid setup are recommended when using these tools.

An important advantage of Rotabroach cutters is that they require less thrust force (only about 10% for smaller diameter tools) and power than twist drills of comparable size when operated at the same feed rate. This is because there is no dead-center area like the chisel edge region of a twist drill. The cutters can be operated at higher feed rates, thus increasing metal removal rates, when using the same horsepower required for twist drilling. Other advantages include close tolerances and smooth finishes.

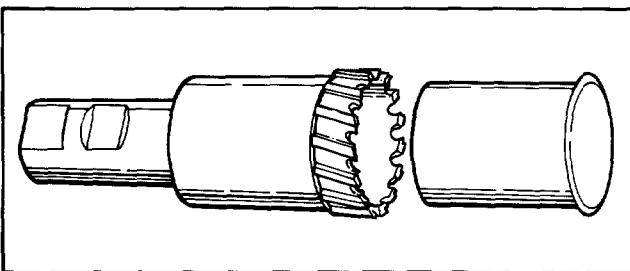


Fig. 9-23 Rotabroach cutter which requires less thrust force than twist drill. Slug produced is seen at right. (Hougen Manufacturing Co.)

Special twist drills. It is always good practice, in the interest of economy, to use standard twist drills whenever possible. However, despite the wide variety of types and sizes commercially available, special twist drills are often needed for many applications.

When special drills are to be specified, information covering the details shown in Fig. 9-24 should be supplied to the drill manufacturers. Suitable allowance should be specified for resharpener purposes and also for clearance for the spindle above the drill bushings. If a particular style of flute construction is desired, it should be specified by reference to a standard drill

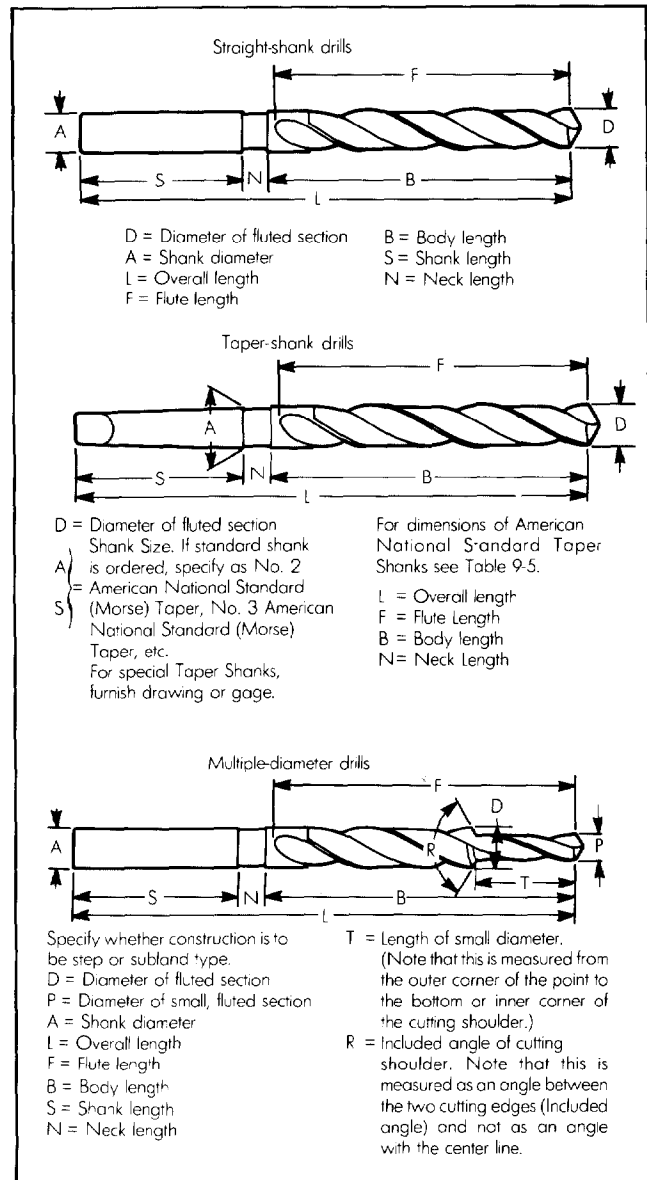


Fig. 9-24 Detailed information that should be supplied when ordering special twist drills. (Cleveland Twist Drill Co.)

having the required flute style. If the tang of the required drill is special, the thickness and length should be stated.

When extra-length drills are to be ordered, the type of material to be drilled, depth of hole, whether drilling is to be done in a vertical or horizontal position, and whether the feed will be intermittent or with only occasional withdrawals should be specified. Drills should be as short as possible for maximum life.

Twist Drill Geometries

Efficient drilling of the wide variety of materials encountered in industry today requires many different drill designs and geometries. Many holes can be drilled satisfactorily with standard off-the-shelf twist drills, but no one drill is best for all applications. Variations in drills being used, while maintaining

required strength and sufficient room for easy chip ejection, include different drill points, lip relief and clearance angles, and flute construction (helix angle, web thickness, and web thinning).

Various degrees of included-point angle, with proper lip relief, and specific types of web thinning are used to perform the following:

1. Control the formation of the chips produced.
2. Control the size and shape of the chips.
3. Control chip flow along the flutes.
4. Increase the strength of the cutting lips.
5. Reduce the rate of cutting lip wear.
6. Reduce the thrust required for drilling.
7. Control the hole size and quality.
8. Control the size and amount of burrs produced.
9. Reduce the amount of heat generated.
10. Permit variations in cutting speed and feed rate for more efficient drilling.

Drill points. Since drill points form the cutting edges, their geometries are critical to tool performance. There is a variety of point styles being used today; some of the more common ones are described in this section. Proper selection, control, and use of drill points can result in substantial savings in drilling costs.

Single-angle points. Standard twist drills having conventional points with a 118° included angle are the most commonly used because they provide satisfactory results in drilling a wide variety of materials. The cutting lips on these drills are essentially straight lines, with the heel side of each land a smooth curve (see Fig. 9-25, *a*).

A possible limitation to the use of this conventional point is that its straight chisel edge contributes to wandering of the point, often making it necessary to first use a center drill for improved hole accuracy. Also, the sharp corners tend to break down more rapidly than some other geometries available, and there is more of a tendency to produce burrs on breakthrough. As a result, drills with this type point are generally best suited for applications for which close tolerances are not required.

As the hardness of the workpiece material decreases, improved drill performance can be achieved by reducing the included angle of the drill point to 60° – 90° . Drills having these more acute point angles produce thinner chips for a given feed rate and are commonly used, with low-helix flutes, for

producing holes in soft plastics and nonferrous materials. Drills having points with a 90° included angle are also used occasionally for drilling soft cast irons and certain woods.

Similarly, as the hardness of the workpiece material or depth of hole increases, the included angle of the drill point is increased to 135° – 140° . These larger point angles produce thicker and narrower chips for a given feed rate. Drills with these flatter points are generally used to produce holes in harder, tougher materials, and they usually minimize burring. It is especially important to use guide bushings with drill points having higher angles because there is a tendency for the points to skid or walk on the workpiece surfaces when starting holes.

Double-angle points. Twist drills with double-angle points (see Fig. 9-25, *b*) are generated by first grinding a larger included angle (118° or 135°) and then a smaller included angle (typically 90°) on the corners. This provides the effect of chamfers and reduces abrasive wear on the corners.

Initial applications for this style point were in drilling medium and hard cast irons as well as other very abrasive materials to reduce corner wear on the drills. More recent applications include improving hole sizes and finishes and drilling very hard materials to reduce chipping of the corners of the lips. Twist drills with double-angle points are often used for the same applications as drills with rounded-edge (radiused lip) points, discussed later in this section.

Reduced-rake points. A common and easily applied point variation is the flattened cutting lip. Both cutting edges are flattened on their flute faces, called dubbing, from the cutting lip corners to the chisel edge, as illustrated in Fig. 9-25, *c*. This type of point reduces the effective axial rake to 0 – 5° positive, causing a pushing or plowing of metal rather than a shearing action. Reduced shearing action is an effective method of preventing the tools from digging in when drilling is performed on materials with low tensile strengths such as many types of brass, bronze, and some of the harder acrylic plastics such as Plexiglas. Reducing the rake also strengthens the cutting lips, and this type of point is often used in operations in which chipping of the lips has been a problem.

Four and six-facet points. The geometry of a four-facet point (see Fig. 9-25, *d*) is generated by grinding flat primary relief (10 – 18°) and secondary clearance angles (25 – 35°) on the end of each flute. The width of the primary relief flat is equal to one-half the web thickness, resulting in four facets on the end of

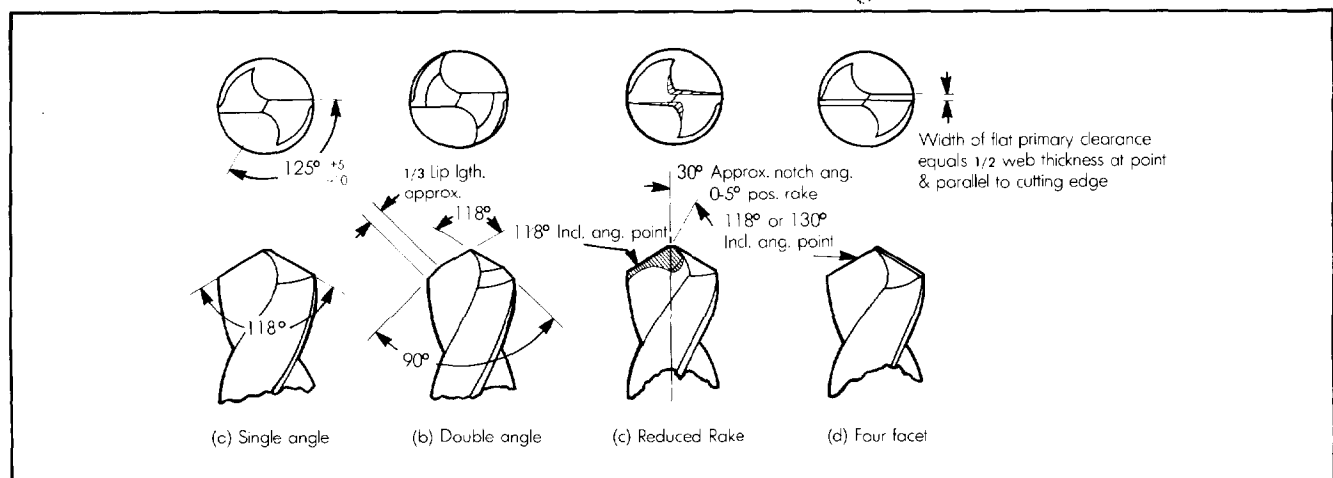


Fig. 9-25 Four types of drill point geometries used for different applications. (Cleveland Twist Drill Co.)

CHAPTER 9

TWIST DRILLS

the drill which subtend at a point on the drill axis and entirely remove the chisel edge. Six-facet points are produced by adding two cutting edges at the web of four-facet points.

Since these points are exactly in the middle of the drills, the tools are self-centering and accurate and straight holes can be produced. They also require less power and thrust and permit increased feed rates. Drills with these points, however, are subject to more wear on their margins, and they cannot be modified to suit drilling of various materials. Another disadvantage is the cost of resharpener with a special machine.

Four and six-facet points have found their greatest use for solid carbide drills used to produce holes in printed circuit board materials such as fiberglass-epoxy. The points may also be used on small-diameter HSS drills that do not lend themselves to normal point-splitting techniques.

Split-points. This type of point, also called a crankshaft point, was originally developed for use on drills designed for producing small-diameter, deep holes in automotive crankshafts. Since then it has gained widespread use for drilling a wide variety of hard and soft materials. Heavy-duty types with thicker webs are used for drilling stainless steels, titanium, tough alloys, and high temperature resistant alloys. Drills with this type of point are also employed extensively for applications in which guide bushings cannot be used, as well as for portable drilling applications.

In generating split points on drills, the clearance face of each cutting edge is given a sharp (55° typical) secondary relief to the center of the chisel edge (see Fig. 9-26), thus creating a secondary cutting lip on the opposite cutting edge. The angle between these lip segments acts as a chipbreaker when drilling is done on many materials, producing smaller chips that are readily ejected through the flutes. More importantly, however, the additional cutting edges produced and the reduction in width of the original chisel edge reduces thrust requirements (typically 25-30% compared to conventional 118° points) and improves the centering capability. A disadvantage is the need for a point-splitting grinding machine.

Helical (spiral) points. This type of point is generated by reducing the drill point from a chisel edge to a helical (spiral) point, as illustrated in Fig. 9-27. This produces an S-shaped chisel with a radiused crown effect which has its highest point at the center of the drill axis. This S-shaped chisel creates a continuous cutting edge extending from margin to margin across the web.

Advantages of drills with a helical point include a self-centering capability and some reduction of thrust. Their use also results in better hole geometry and improved hole size.

A possible disadvantage of this type of point is that burrs are sometimes produced at hole breakthrough. Also, the S-shaped chisel is weaker than straight chisel points, resulting in faster dulling when hard materials are drilled. Special machines are required to grind these points.

Rounded-edge (radiused-lip) points. These points are generated by grinding a blended, rounded edge (radiused corner or lip) on conventional points (see Fig. 9-28). Points such as these provide a continuously varying point angle, with the lips and margins blended by a smooth curve. Because the drill cuts on long, curved lips, there is less load per unit area and less heat generated. Since the corner is eliminated, margin wear is reduced. Breakthrough burrs are eliminated, and tool life can be lengthened compared to conventionally pointed drills when cast iron is drilled. Feed rates can also be increased because of the improved heat dissipation.

Twist drills with rounded-edge points are used when drill life is most important. Drills with these points are not self-centering and are best applied where guide bushings are used. When used on NC machines, prior center drilling is required. The time required for center drilling, however, may be more than offset by longer tool life. A possible limitation is that special grinding machines are required to produce these points. Also, when steel is drilled, these points cut closer to size, which may reduce drill life compared to that possible with conventional points because of greater corner and margin wear.

Combined helical/rounded-edge points. Drill grinding

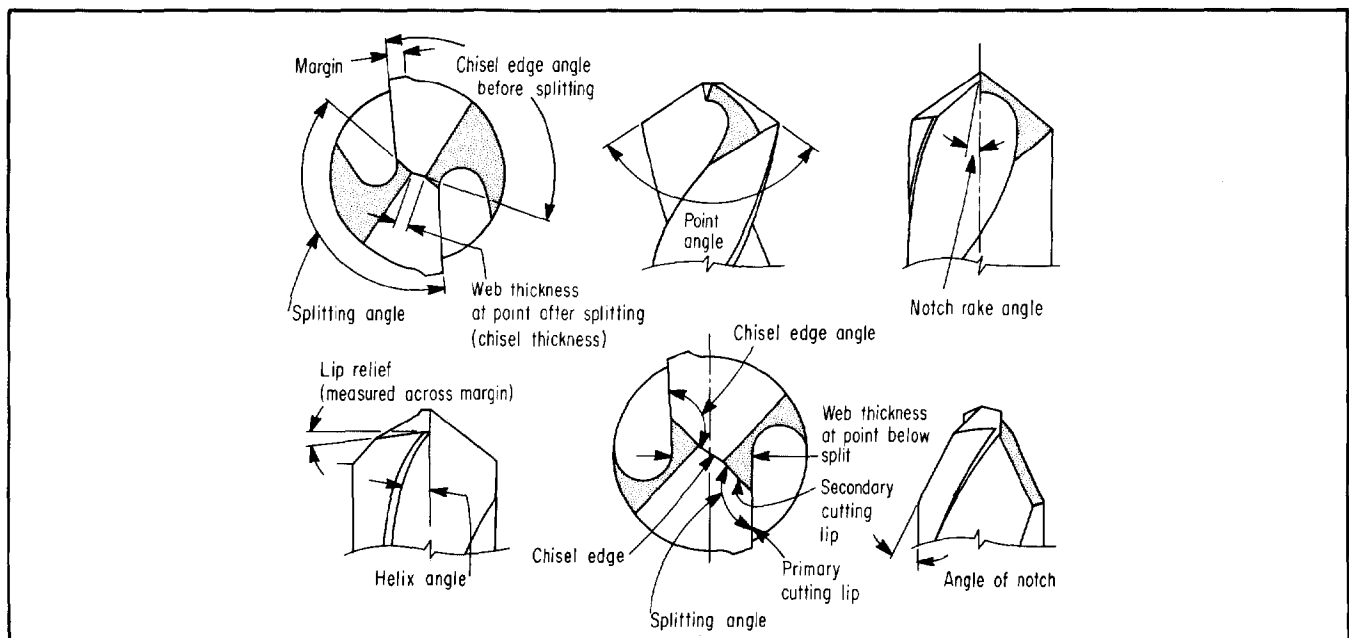


Fig. 9-26 Geometry of split-point twist drill.

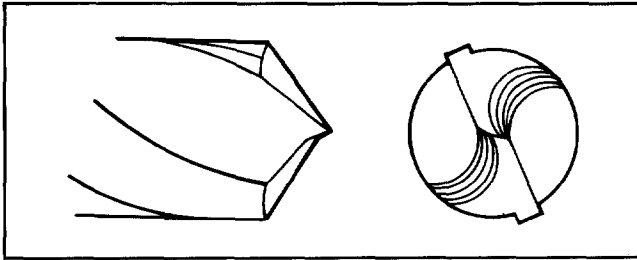


Fig. 9-27 Helical (spiral) point which has an S-shaped point rather than a straight-line chisel edge.

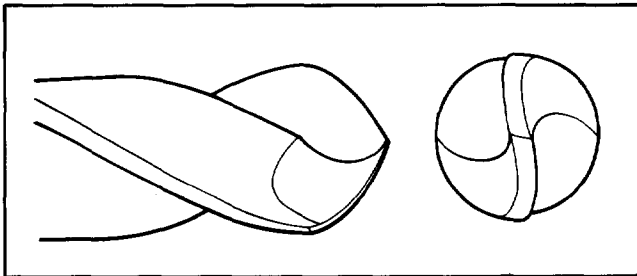


Fig. 9-28 Rounded-edge point which has lips and margins blended by smooth curves.

machines are available that combine features of both the helical and rounded-edge points. The point produced (see Fig. 9-29) provides the self-centering capability of helical points and the long life, burr-free breakthrough, and higher feed capacity of rounded-edge points. These features make the drills capable of producing accurate holes on NC machines without the need for prior center drilling.

Lip relief. The lip relief angle used on a drill is also important. With inadequate lip relief, a drill will not cut freely; excessive relief will shorten the drill life. It is even more important that the relief angle on each lip be equal and that the relief surfaces be in close axial relationship to each other. The amount of hole oversize produced increases with an increase in relative lip height.

The amount of relief depends primarily upon the drill diameter, cutting edge strength, and the material to be drilled. Suggested lip clearance angles at the peripheries of general-purpose drills with 118° included angle points are presented in Table 9-8. For drilling harder materials, the angle is often reduced; for softer and nonferrous materials, it is generally increased.

Higher relief angles generally provide best results with light feeds and low-strength, nonferrous materials. Some of the plastics and cast irons also require higher relief angles because of their abrasiveness. With heavy feeds, as well as harder workpiece materials, reduced relief angles provide cutting edges with additional support to withstand the higher cutting loads.

Web thickness. Because the chisel edges are noncutting portions of drill points, the webs of twist drills are made as thin as possible, consistent with adequate structural strength. The approximate web thicknesses near the points of standard drills, expressed as a percentage of the drill diameters, are presented in Table 9-9. Heavy-duty drills have thicker webs (about twice the thickness of standard general-purpose drills) and often have narrower flutes to increase torsional stiffness.

Most drills are manufactured with webs which increase in thickness toward their shanks. Resharpening of a worn drill

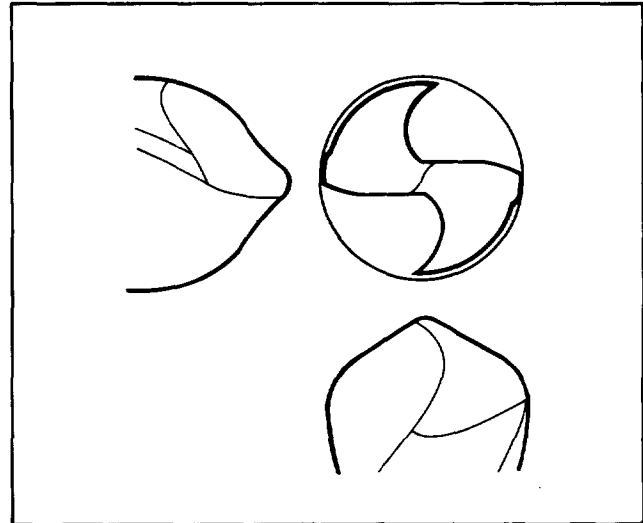


Fig. 9-29 Twist drill point combines features of helical and rounded-edge points. (Giddings & Lewis-Bickford Machine Co.)

TABLE 9-8
Suggested Lip Clearance Angles on Points for
General-Purpose Twist Drills with 118° Included Angle Points

Drill Diam Range	Lip Relief Angle, degrees
Nos. 80 to 75 (0.0135-0.0210")	25-29
0.55 to 0.8 mm	24-28
No. 67 (0.0320")-1.55 mm	22-26
1/16" to No. 48 (0.0625-0.0760")	21-25
5/64" to No. 42 (0.0781-0.0935")	20-24
3/32" to No. 36 (0.0938-0.1065")	18-22
7/64" to No. 31 (0.1094-0.1200")	17-21
1/8" to No. 13 (0.1250-0.1850")	15-19
3/16" to D (0.1875-0.246")	14-18
1/4" to M (0.2500-0.295")	13-17
19/64 to 11/32" (0.2969-0.3438")	12-16
S to W (0.348-0.386")	11-15
25/64 to 29/64" (0.3906-0.4531")	10-14
15/32 to 3/4" (0.4688-0.7500")	9-13
49/64 to 1" (0.7656-1.0000")	7-11
Over 1" (1.0000")	6-8

TABLE 9-9
Approximate Web Thickness of Standard
Twist Drills, Percentage of Drill Diameter

Drill Size, Number and Fractional, Inches	Web Thickness Percentage of Drill Diameter
Nos. 80 to 66 (0.0135-0.0330)	30
Nos. 65 to 56 (0.0350-0.0465)	25
No. 55 to 3/16 (0.0520-0.1875)	20
3/16 to 1/4 (0.1875-0.2500)	17
1/4 to 5/8 (0.2500-0.6250)	15
5/8 to 1 3/8 (0.6250-1.3750)	13
1 3/8 to 2 3/8 (1.3750-2.3750)	12
Over 2 3/8 (2.3750)	11

TWIST DRILLS

shortens the drill and increases the web thickness and chisel edge length (see Fig. 9-30). This results in increased thrust requirements, additional heat generation, and shorter drill life unless the web is thinned to its original thickness. Heavy-duty drills generally require thinning before they are used. The web thickness on some deep-hole drills, however, does not increase; the thickness is the same at the chisel end as it is at the end of the flutes and is called a parallel web.

Thrust force on a twist drill is more sensitive to changes in web thickness than is the drill torque, as can be seen by comparing the two graphs in Fig. 9-31. These graphs show how drill torque and thrust are influenced by changes in chisel edge length, which is proportional to web thickness. They were calculated from test results with 1/2" (12.7 mm) diam drills fed at the rate of 0.010 ipr (0.25 mm/rev) in SAE 3245 steel having a hardness of 200 Brinell. In addition to total torque and thrust requirements, these graphs also show the approximate contributions of drill web and lip regions to the total.

Most of the drill torque results from the outer portions of the drill lips because this is where most of the material removal occurs. For a drill of regular proportions, only about 15% of the torque comes from the web. With a drill of regular design, about 50% of the total thrust force is caused by the web. If the web thickness is doubled, the thrust force is increased by more than 60%; then, about 75% of the total thrust is caused by the web.

Web thinning. Several types of web thinning are commonly used. The type shown in Fig. 9-32, *a*, is perhaps the most common. Length *A* is usually one-half to three-fourths the length of the cutting lip. In this type of thinning, as well as in all others, it is important that the thinning cut extend far enough up the flute so that an abrupt wedge is not formed at the extreme point. The distance of the thinned cut varies with the amount of thinning required, but an average of one-fourth to one-half the drill diameter is usually satisfactory.

Sometimes it is advisable to extend the thinning out to the extreme edge in order to change the shape of the chip. In this type of thinning (see Fig. 9-32, *b*), a positive effective rake is

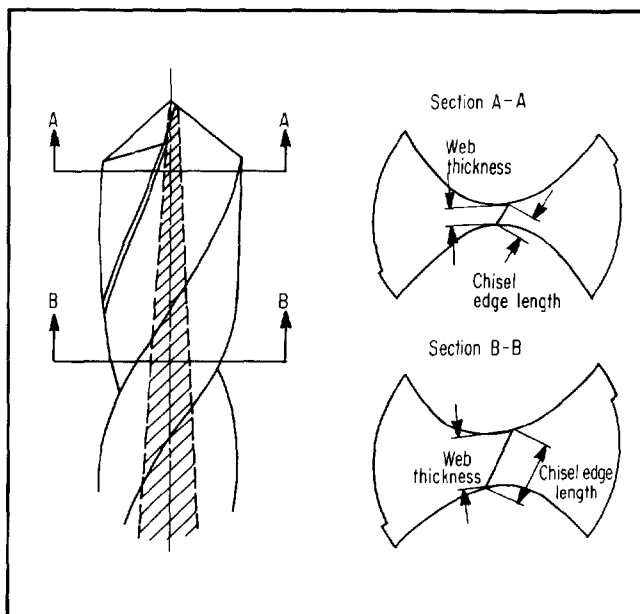


Fig. 9-30 Resharpener of twist drill progressively increases the web thickness and chisel edge length.

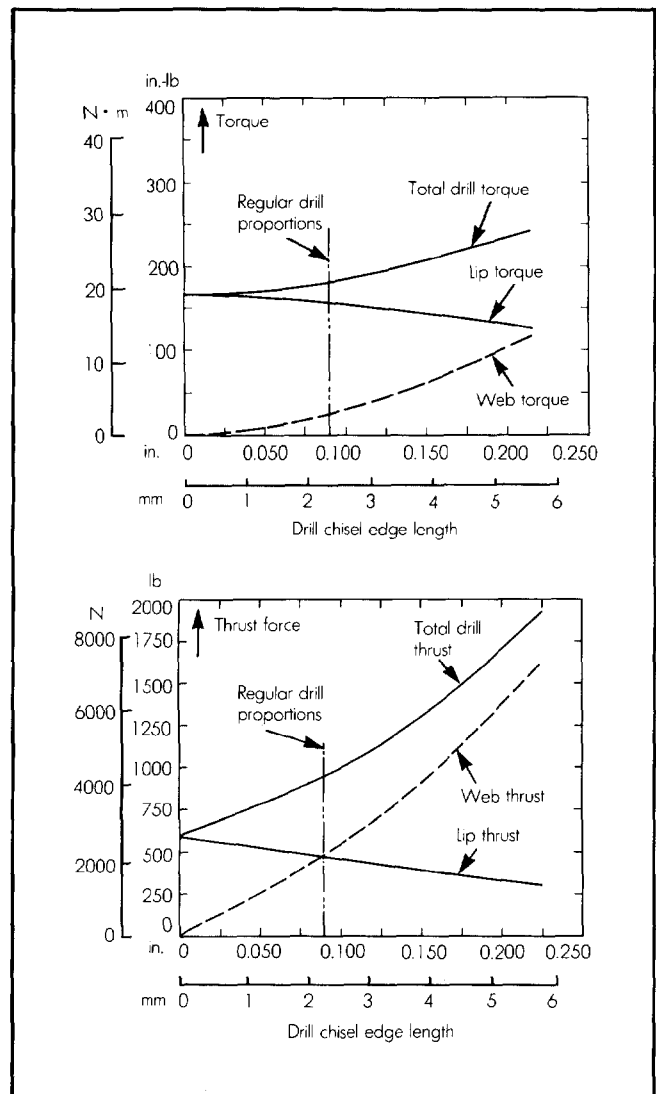


Fig. 9-31 (a) Effect of chisel edge length on torque; (b) thrust force (bottom graph). (National Twist Drill)

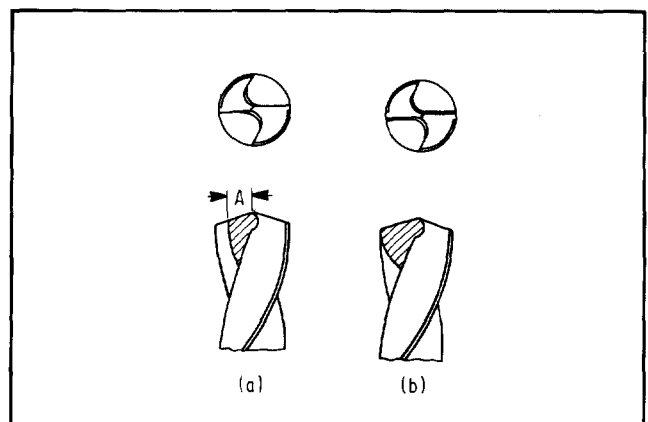


Fig. 9-32 Two types of web thinning: (a) at chisel edge and (b) undercut thinned point.

maintained the full length of the cutting edge. A third type of thinning often used results in the split or crankshaft point described previously.

Some manufacturers offer self-thinned webs on jobbers-length twist drills. These drills have a web which is straight (uniformly thick) for one-third or more of the flute lengths and then tapers toward the shank.

Helix angles. The helix angle on standard twist drills generally ranges from 25-33°. High-helix (fast-spiral) drills with helix angles of 35-40° and low-helix (slow-spiral) drills with helix angles of 15-20° are also commercially available for drilling certain materials and special applications, as discussed previously in the section on types of drills.

Although used in Europe for many years, a relatively recent introduction to the United States is the parabolic flute twist drill that provides more open flutes for improved chip removal and cutting fluid flow while permitting heavy webs for high torsional strength. Combined with webs of constant thickness, they have proven to be ideal for deep-hole drilling in cast irons, mild steels, and aluminum alloys. A twist drill with parabolic flutes and a split point is illustrated in Fig. 9-33. For some deep-hole applications (with depth-to-diameter ratios to 12:1 or more), speeds and feeds to 100% greater than conventional drills have been reported, often eliminating the need for woodpeckering.

COOLANT-FED TWIST DRILLS

Coolant-fed twist drills having means for directing coolant (fluid, gas, or mist) to the cutting edges offer many advantages for certain drilling applications. Oil-hole drills have been used for many years to produce deep holes. They were originally developed for applications such as horizontal or inverted drilling in which point cooling and/or chip ejection presented problems.

Now, improved types of coolant-fed twist drills are being used increasingly for a wide variety of more common drilling applications, including shallow holes, in which higher penetration rates or longer tool life are desired. They are employed extensively in machining centers, multispindle bar and chucking machines, screw machines, turret lathes, and other machine tools for producing holes shallower than those that are usually drilled by self-guiding deep-hole drills (discussed later in this chapter). Small-diameter, coolant-fed twist drills are also used extensively for portable drilling applications in the aircraft industry.

Advantages of Coolant-Fed Tooling

In many drilling applications, coolant is misdirected and wasted. As a result, benefits are minimal. With coolant-fed tooling, however, the coolant is directed close to the cutting edge, thereby substantially reducing friction and temperatures at the drill/workpiece interface. This reduces wear and

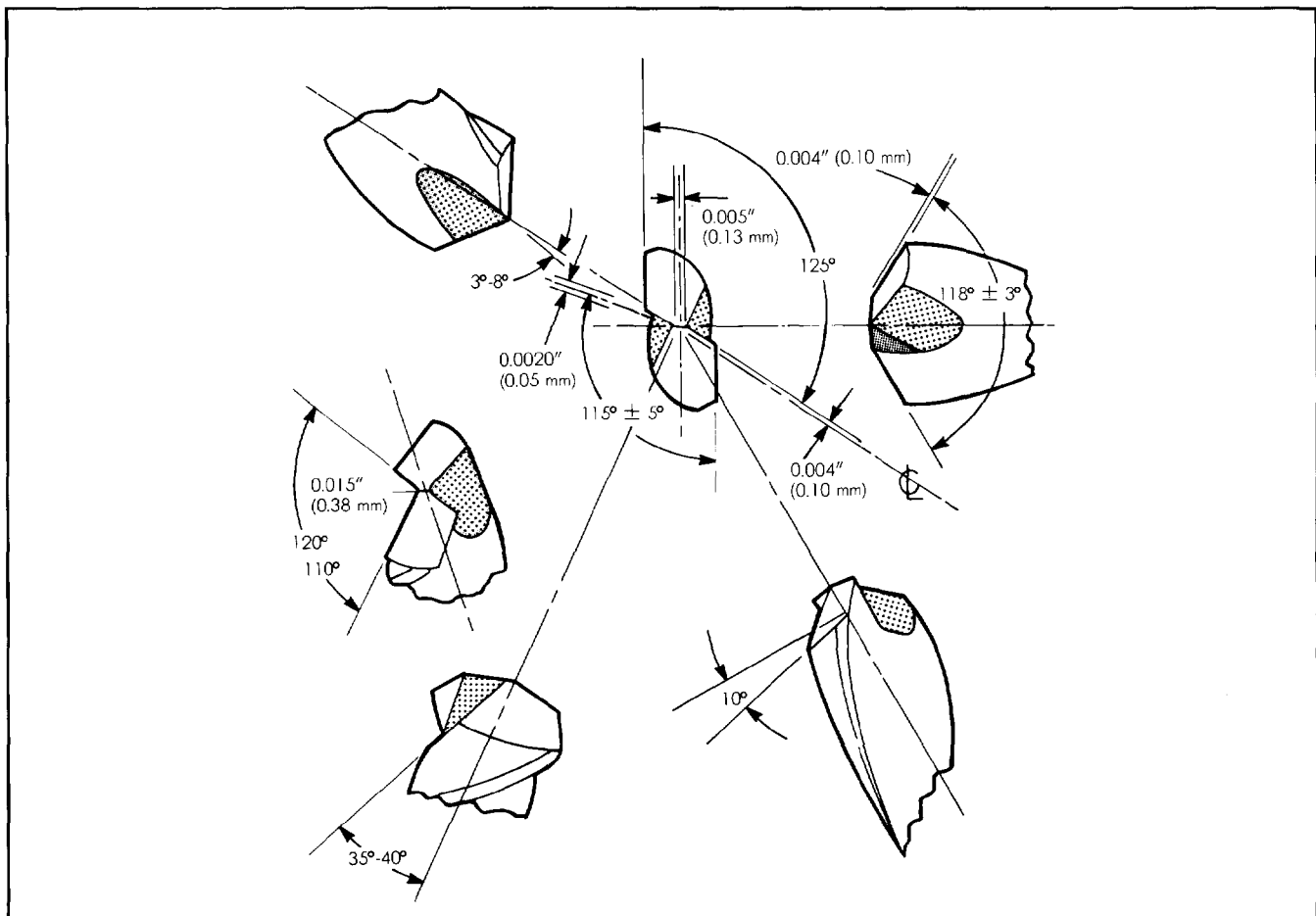


Fig. 9-33 Twist drill with parabolic flutes and a split point. (Bendix Industrial Tools Div.)

CHAPTER 9

COOLANT-FED TWIST DRILLS

lengthens drill life, resulting in reduced costs per hole drilled because of less downtime for tool changes and regrinds.

Depending upon the application, faster cutting speeds and feed rates can generally be used, thus increasing the metal removal rate and productivity. Results of one drilling study using two solid and two coolant-fed drills showed that the coolant-fed drills were capable of considerably higher penetration rates (see Fig. 9-34). This plot of drill life as a function of penetration rate was made for drilling 4" (102 mm) deep through holes in SAE 1018 steel, having a hardness of 127 Bhn, for two varieties of solid 7/8" (22 mm) diam twist drills and two varieties of 7/8" coolant-fed twist drills. The life of the solid drills for this single test ranged from over 350" (8890 mm) when drilled at a penetration rate of 6 ipm (152 mm/min) to less than 50" (1270 mm) when drilled at 12 ipm (305 mm/min), using

flood coolant. By nature of comparison, coolant-fed drill D produced over 400" (10 160 mm) at a penetration rate of 11 ipm (279 mm/min), and coolant-fed drill C, more than 400" at 23 ipm (584 mm/min).

The high-helix, coolant-fed drill, C, is clearly the best style for use in this rather mild, normally stringy chip material and effectively more than triples the penetration rate for this application. A coolant flow of 2.5 gpm (9.5 L/min) was provided through these drills, obtained by using a pressure pump operating at 70 psi (483 kPa). For setting up any coolant-fed drill applications, common practice is to start with speeds 10% higher and feeds 25% higher than those established for solid twist drills.

Productivity is also improved by eliminating the need for interrupting the drilling cycle to withdraw the tool to clear chips

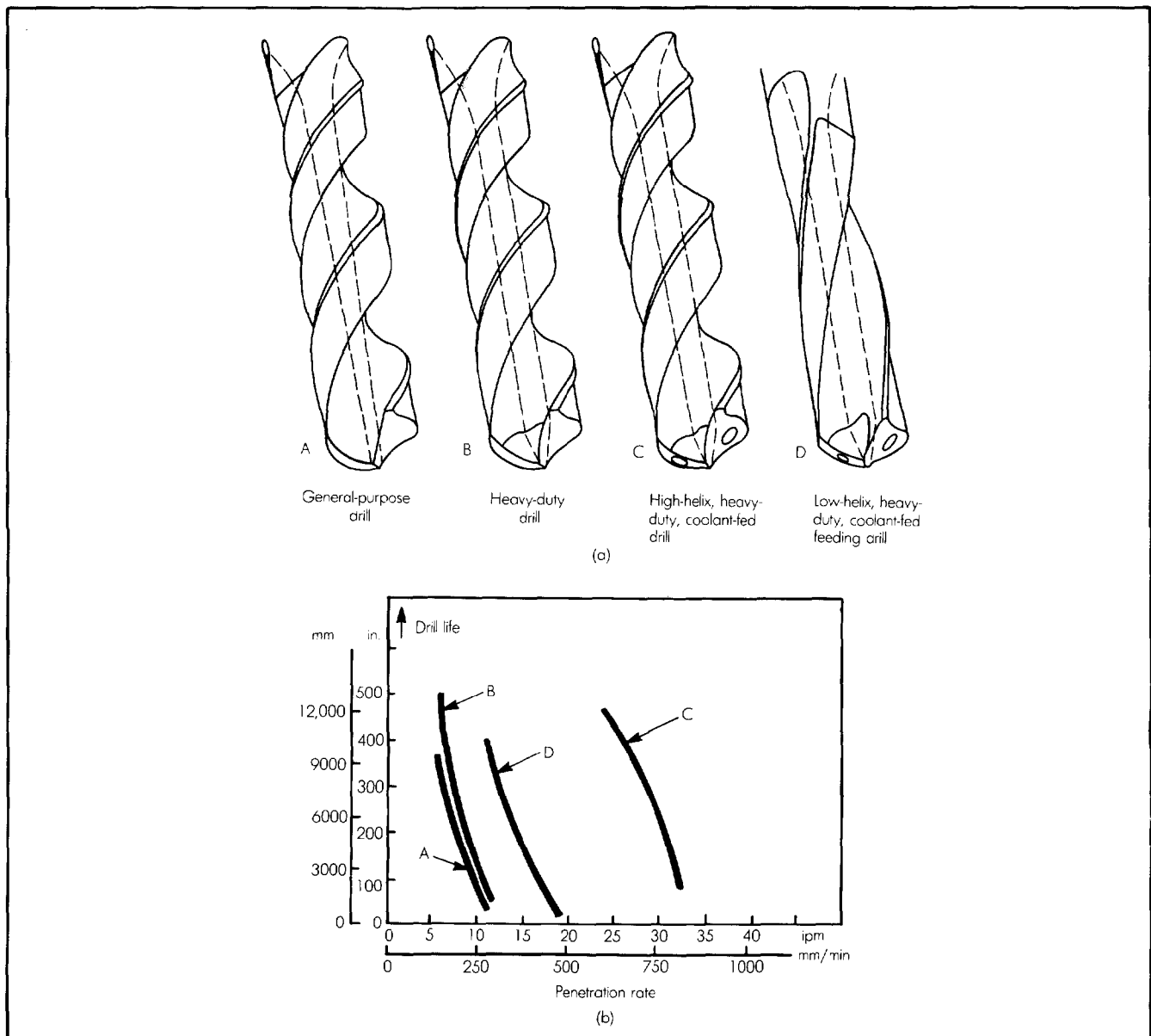


Fig. 9-34 (a) Two solid and two coolant-fed drills used in study to show (b) increased penetration rates possible with coolant-fed drills.

COOLANT-FED TWIST DRILLS

(woodpeckering). Coolant-fed drills generally improve chip removal. Consequently, heat transfer to the chips, tool, and workpiece is minimized; buildup on the drill cutting edges is reduced; and holes with improved accuracy and smoother finishes are produced. Air pollution from smoke or mist may also be minimized because of reduced heat and friction.

For some applications in the aircraft, aerospace, and nuclear industries, the use of coolant-fed drills has improved surface integrity by eliminating tearing, checking, and cracking of workpiece materials.

Limitations of Coolant-Fed Twist Drills

The higher initial cost of coolant-fed drills and related equipment is often an unjustified deterrent to the more extensive use of these tools. Comparative studies frequently show that increased productivity, longer tool life, and other advantages often make these tools economically desirable. It is the cost per hole, not the tool cost, that is most important.

Longer setup time may be a problem in justifying coolant-fed tools for some short-run applications. While some machines are equipped with hollow spindles to supply coolant to these tools, other machines may require some alteration to provide coolant piping, splash shielding, and other accessories. Another possible limitation is that the machine to be used may not have adequate power to take advantage of the higher feeds and speeds possible with coolant-fed drills.

Types of Coolant-Fed Drills

There are several styles of coolant-fed twist drills. One style has a tube secured in a groove machined in each land to deliver coolant to the cutting lips. Another, more common, type has holes extending internally through each of the two lands.

Coolant-fed twist drills are available with either straight or taper shanks, in a wide range of sizes, 1/8 to 3" (3.2 to 76 mm) diam, and in extra-long lengths. Medium to heavy-duty drills with constant (parallel) web construction are generally best for most applications. A fairly heavy web construction, about 30% of the drill diameter, is generally needed for strength because of the increased feed rates often used with these tools.

Drills with a low helix angle of about 14-22° are widely used for stationary tooling, horizontal drilling applications, and cast iron drilling. The low helix angle provides shorter paths for the coolant and chips, but requires higher torque and thrust forces. Drills with a medium-high helix angle of about 30-34° are generally recommended for vertical drilling. They usually outperform low-helix drills in soft materials and in most applications when the tools are rotating.

The points of coolant-fed twist drills, generally having a standard 118° included angle, should be notch thinned to the proper geometry for the material to be drilled so that thrust requirements can be minimized. A web thickness at the chisel point of 5-10% of the drill diameter, depending upon drill size, is generally recommended.

Slight modifications of the point geometry may be necessary in drilling some materials to ensure the formation of short, tightly curled, figure six shaped chips that will flow easily through the drill flutes. Chip width can be reduced and thickness increased by using a higher included-point angle (such as 135°), which produces shorter cutting lips and more desirable chips. A corner chamfer may be helpful for some applications. The shortest possible drill should always be used for the hole depth required to increase rigidity and improve accuracy.

High-speed steel drills are satisfactory for most coolant-fed

drilling applications. Carbide-tipped drills are generally more economical for producing a large number of holes in cast iron or other abrasive materials and often provide longer life when hard or tough materials are being drilled.

Coolant Supply

Straight-shank, coolant-fed twist drills, normally used in horizontal, nonrotating applications, are held in either a solid toolholder or a collet-style drillholder. Drill sizes over 1/2" (12.7 mm) diam have the end of the shank drilled and tapped for use with a coolant inlet hose fitting when applicable. Sizes below 1/2" generally have a shallow counterbored hole encompassing the two coolant holes for ease of entry. In many holders, this end face of the drill shank is used for coolant sealing. The ends of straight-shank drills should be squared accurately for proper sealing.

Taper-shank drills, normally used in rotating spindle applications, are furnished with a midshank crosshole feed, or an inducer feed, or they are end feeding. The standard crosshole feed is used in conjunction with a coolant-inducer, socket-style holder similar to the one shown in Fig. 9-35, or with a machine toolholder which has internal coolant passages. For machines which can supply coolant through their hollow spindles, the drills have the coolant holes extending through their tang ends.

Inducer-type, coolant-fed twist drills incorporate coolant inlets on the neck portion of the drill, ahead of the shank. A feed gland, held in place by retaining rings, eliminates the need for a coolant-inducer/socket component. This decreases the projection length of the drill point from the machine spindle, and thus increases setup rigidity and tool life.

Depending upon the material to be drilled, the drill diameter, hole depth, and other factors, it is preferable to use coolant volumes from 3-10 or more gpm (11-38 L/min), with pressures from 50-300 psi (345-2068 kPa), for coolant-fed drills. In some instances, however, lower pressure sump pumps may provide suitable performance. Good results are being obtained with both constant pressure and pulsating coolant systems, discussed later in this chapter under the subject of "Operating Parameters."

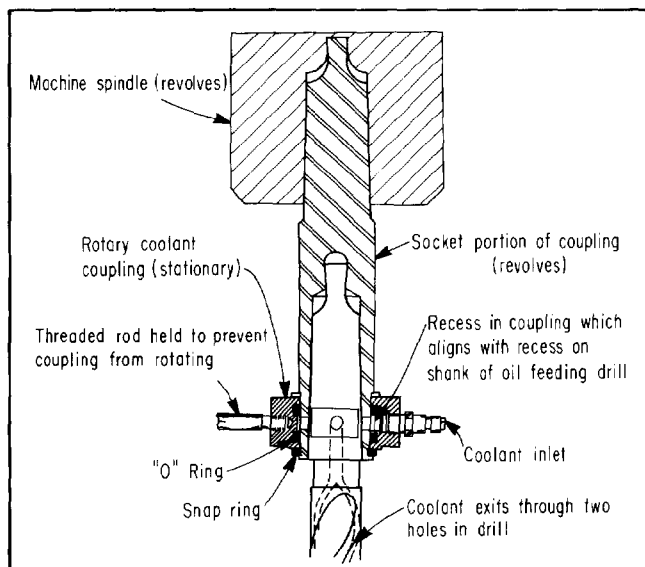


Fig. 9-35 Coolant-inducer, socket-style holder for coolant-fed twist drills.

CHAPTER 9

FLAT, HALF-ROUND, AND STRAIGHT-FLUTE DRILLS

FLAT, HALF-ROUND, AND STRAIGHT-FLUTE DRILLS

All holes produced by the drilling process are not cut with twist drills. For example, the flat, half-round, and straight-flute drills discussed in this section, while not used as extensively as twist drills, have definite advantages for certain applications. Other nontwist drills discussed in subsequent sections of this chapter are indexable-insert, spade, and deep and small-hole drills.

Flat Drills

These tools derive their name from the shape of the drill body, which is flat rather than the normal round configuration (see Fig. 9-36). They are produced by grinding tapered opposite flats on the drill body. The flats are usually not parallel, which creates a web that is thicker at the shank. A slight back taper of 0.001" (0.03 mm) or less is ground on the body.

Flat drills have a low productive capacity, but because of their simple design and low cost, they are occasionally used for low-volume applications in drilling hard forgings and castings. With spear points, flat drills are also used to produce holes in glass and tile. Standard flat drills, made from high-speed steel or carbide, are available from 3/32 to 1/2" (2.4 to 13 mm) diam. For small-hole drilling, they are generally made from carbide only in diameters from 0.001 to 3/32" (0.03 to 2.4 mm).

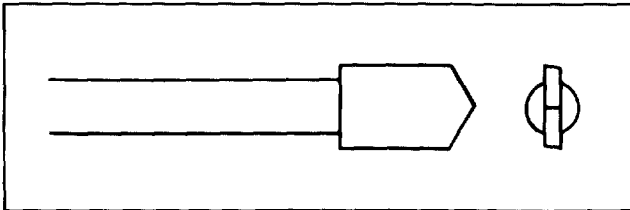


Fig. 9-36 Flat drill of the type sometimes used to produce holes in hardened steel, glass, and tile.

Half-Round Drills

Half-round drills are round rods with about half of their diameters machined or ground away to form the drill bodies (see Fig. 9-37). The drills have a conical point that is offset to provide radial relief for a single cutting lip. The points can be ground either like a conventional twist drill with radial relief or a gundrill (discussed later in this section), depending upon the rigidity of the setup. It is important that the apex of the cone be exactly on the centerline of the drill. Half-round drills are available in fractional, number (wire gage), letter, and metric sizes from about 0.003-1" (0.08-25.4 mm) diam and are made of high-speed steel, of carbide, or with carbide tips.

Half-round drills with precise conical points can sometimes start holes accurately without the use of guide bushings, especially when the tools are short and rigid, and have little tendency to wander. As a result, they are used extensively for screw machine applications and on some NC machines. For some applications, holes are started with short, rigid center drills and completed with half-round drills. These drills are particularly suitable for use in brass and, in some cases, aluminum alloys and diecast materials. Because of the support they provide in the holes being drilled, they are capable of maintaining close tolerances and producing smooth finishes, sometimes eliminating the need for subsequent reaming. Chip-ejection limitations, however, restrict their deep-hole drilling capabilities.

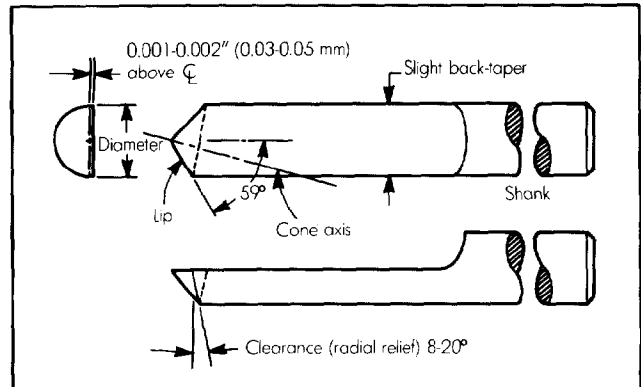


Fig. 9-37 Half-round drill which has a conical point that is offset to provide radial relief for the single cutting lip.¹

Straight-Flute Drills

These tools are designed to produce short chips and have limited applications for drilling brass, copper alloys, and other soft, nonferrous materials, particularly in horizontal drilling applications. The straight flutes eliminate the tendency for the drills to run ahead and grab or pull in the material; because of this, they are desirable for thin sheet metal work. The drills are commercially available in fractional and number sizes with short (jobbers) lengths and straight or taper shanks.

Heavy-duty, carbide-tipped, straight-flute drills are specifically made for drilling steels having a hardness over $R_C 50$ and for producing shallow holes in brittle materials. These tools, sometimes called die drills, have heavy lands and shallow flutes. They are not suitable for drilling soft steels from which long chips are produced.

Coolant-fed, carbide-tipped drills with two straight flutes are also available. Some are made from crimped tubing and others are made from solid HSS bars. Some drill manufacturers offer replaceable tip tools having two straight flutes with HSS or carbide bits being held in place with retaining screws instead of being brazed. Coolant is supplied by tubes brazed into grooves along the lands or through holes in the drill body.

INDEXABLE-INSERT DRILLS

The most important recent advance in drilling technology was the development during the early 1970's of drills with indexable carbide inserts. These tools can produce relatively shallow holes from the solid at faster rates and lower cost than HSS twist drills in many applications.

Studies have shown that about 60% of all drilling applications in industry are considered to be short holes having depths up to about three times their diameters. Many of these holes can be drilled with indexable-insert drills. Others are not practical to produce with these tools because the holes are too small in diameter or because inadequate machines (with respect to speed, power, and rigidity) are employed.

Advantages of Insert Drills

Major advantages of indexable-insert drills are increased productivity, reduced costs, and better versatility.

Increased productivity. The use of carbide inserts brings drilling close to the machining rates possible with turning and milling. The higher cutting speeds possible permit holes to be drilled substantially faster than HSS twist drills and even faster than carbide spade drills (discussed next in this section). The

INDEXABLE-INSERT DRILLS

cycle time on transfer machines and other high-production applications is often determined by the capabilities of the HSS drills used. Now, with the faster penetration rates of indexable-insert drills often matching the rates of other operations, cycle times can generally be reduced. Potential productivity is also increased because the almost flat lead angle of indexable-insert drills results in a shorter feed stroke before cutting, compared to twist or spade drills that have point angles.

Reduced costs. The use of low-cost inserts with multiple cutting edges eliminates regrinding costs. The multiple cutting edges available provide savings from not having to replace the entire tool. Indexing the inserts does not change their positions and the tool length, thus any tool resetting costs are eliminated.

Versatility. Indexable-insert drills can be used as nonrotating tools for applications on lathes or other machines or as rotating tools on drilling machines, machining centers, and other machine tools. The machines used, however, must be rigid, be in good condition, and have ample speed and power capabilities. Some of the tools have the capability, when used on suitable machines, to perform boring as well as drilling operations. For example, mounted on the cross slide of an NC lathe, some tools can be moved radially outward to drill holes larger than the tool diameter or make a boring pass, thus improving the accuracy and finish of the hole.

Other tools can perform turning, contouring, and facing operations, as well as multiple operations such as drilling, chamfering, and spotfacing. Figure 9-38 illustrates the drilling, boring, facing, and turning operations performed in producing bushings, 4.331" (111.01 mm) diam x 2.756" (70.00 mm) long, from solid (no prehole) 1021 steel.

Limitations of Indexable-Insert Drills

The smallest diameter hole that can be produced with indexable-insert drills now commercially available is 5/8" (16 mm). Maximum drilling depth is generally two to three

times the hole diameter; some tools, however, are drilling to four times the diameter, and a few specials to depths over five times the diameter. Indexable-insert drills are not precision hole producing tools, and subsequent operations may be required for improved accuracies and smoother finishes. Small pilot holes are not useful and can be detrimental, and the tools cannot be used to enlarge existing holes.

While indexable-insert drills require less thrust than twist drills because they have no webs or chisel edges, they do require more power because of increased metal removal rates. Rigid machines in good condition, with adequate speed and power capabilities, and cutting fluid under pressure are necessary to take full advantage of the productive capabilities of these tools. Horsepower requirements increase proportionately with the drill diameter. Bench, upright, and radial drilling machines are generally not suitable for use with these tools because they lack sufficient speed, power, or rigidity. Safety guards are required on any machine used for drilling through holes with the workpiece rotating and the drill stationary because slugs produced can be thrown outward at high velocity.

Indexable-insert drills can be used to drill many materials, as is discussed later in this section, but most are not suitable for laminated or stacked materials. This is because the discs or slugs produced would be pressed into or welded to the next layer of material and because increased pressures can damage the inserts and possibly the drill body. The surfaces of workpieces to be drilled should preferably be flat. When using negative-rake inserts, convex surfaces can present problems and concave surfaces are not generally recommended because they might throw the drill out of balance. Angular surfaces rising more than 0.040" (1 mm) in a 2" (51 mm) distance and interrupted cuts are also generally not recommended for use with most of these tools; there are, however, successful applications with angular starting surfaces and interrupted cuts when positive-rake inserts are used.

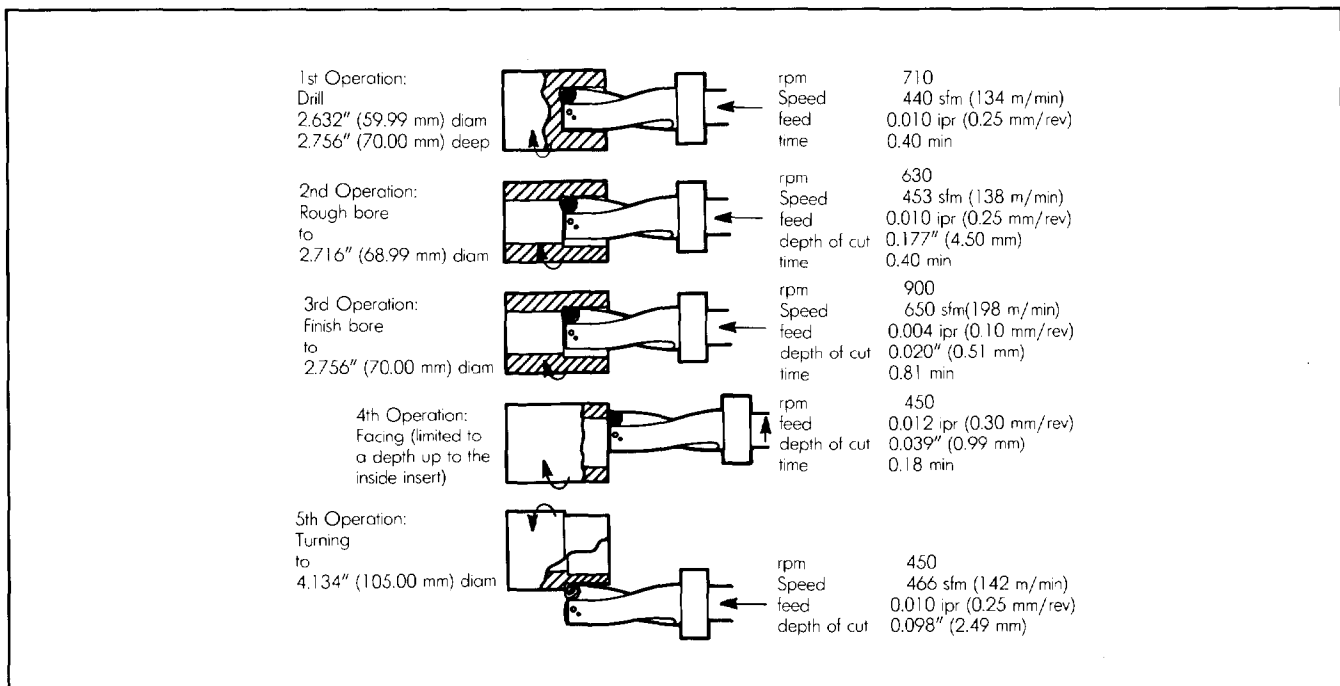


Fig. 9-38 Versatility of some indexable-insert tools permit drilling, boring, facing, and turning. (R. B. Tool Co., Inc.)

INDEXABLE-INSERT DRILLS

Tool Designs

Indexable-insert drills of slightly different design are available from various tool manufacturers. Most consist of a hardened alloy steel body, with straight or taper shank, which is held by an adapter attached to the machine spindle. Some tools are of one-piece construction and others two-piece, with one of the two pieces being an interchangeable cutting head or nosepiece.

The drills have straight or helical flutes or grooves, and internal coolant holes. Flute or groove design is critical—sufficient space must be provided for the rapid removal of a large volume of chips, but an adequate body cross section must be maintained for strength and rigidity. Coolant enters the tools, usually through tapped openings, and reaches the cutting zones from orifices near the inserts. Chips and coolant exit through the external flutes or grooves on the tools. In applications in which the tool rotates, an inducer or coolant collar (as with other coolant-fed tools) is required unless coolant is supplied through a hollow spindle on the machine.

At the cutting end of each flute or groove, recessed pockets are provided to locate the indexable inserts. Depending upon the drill diameter and design of the tool, one to four inserts are generally used. A few drills, using toolholding cartridges, hold as many as six inserts. Some have a centrally located insert, positioned slightly ahead of the others, for self-centering purposes. The inserts are mounted in positions and at attitudes to counteract each other's lateral cutting forces, thus minimizing side loads. This is necessary because the tools are not guided by the holes being drilled and guide bushings are not used. Several designs of indexable-insert drills are illustrated in Fig. 9-39.

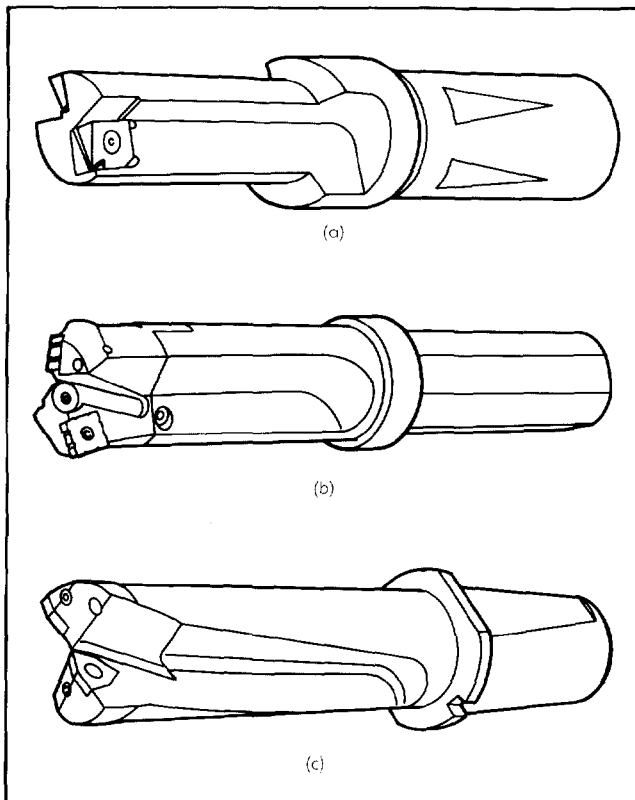


Fig. 9-39 Several designs of indexable-insert drills using carbide inserts: (a) TRW Tools Div.; (b) Valenite Div., Valeron Corp.; (c) Cutting Tool Div., Ingersoll Milling Machine Co.

Size Range of Indexable-Insert Drills

Hole diameters that can be produced with indexable-insert drills vary with the tools available from different manufacturers. A common range offered by some is 3/4 to 3" (19 to 76 mm) diam. The smallest tool commercially available drills holes 5/8" (16 mm) diam, and the largest, 5" (127 mm) diam.

Inserts Used

Round, square, triangular, diamond, trigon, parallelogram, hexagon, and octagon-shaped carbide inserts are used by different tool manufacturers. The geometry of a typical trigon insert used for indexable-insert drills is illustrated in Fig. 9-40. The 84° included angle provides high strength, the 156° V-angle between adjacent cutting edges eliminates the need for precentering nonflat surfaces, and the 12° positive rake angle contributes to long life. The inserts may or may not be the same size, depending upon the drill design and diameter. Some inserts have dimples, grooves, or special geometries for chipbreaking purposes.

Geometry and positioning of the inserts are important to the performance and efficiency of the drills. Continuing improvements are being made in insert geometries to assure constant chip control. Depending upon the insert shape and application, some drills are equipped with negative-rake inserts, requiring negative-rake placement. Others are designed for positive-rake inserts that are placed to provide either a neutral or positive axial rake angle. Positive geometries generally develop less cutting and axial thrust forces and require less horsepower than negative geometries. Effective rake in a negative insert, however, can be higher and can develop lower forces than a positive-rake insert. Replaceable anvils under the inserts can help prevent damage to the tools in case of insert breakage.

The grade of carbide used for the inserts varies with the application. Classification C-5 is used extensively for drilling many steels; C-2 is used for cast irons and nonferrous metals. Coated inserts are also being widely employed because they permit higher cutting speeds, to 1000 sfm (305 m/min) or more, and are less prone to the forming of built-up edges. Inserts made from different grades of carbide are sometimes used in the same tool, depending upon insert position and cutting speed.

Inserts are held in the drill pockets by screws or cam locking pins, eliminating the need for clamps which would obstruct chip flow. It is important that the screws or pins be securely tightened, but not overtightened. Loose inserts will cause chatter and possible tool breakage. Some indexable-insert

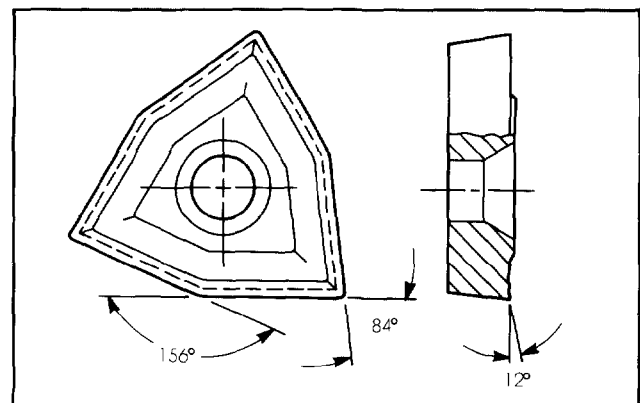


Fig. 9-40 Geometry of a trigon insert used for indexable-insert drills. (Cutting Tool Div., Ingersoll Milling Machine Co.)

drills, especially larger diameter tools, are equipped with cartridges that reduce costs in case of insert failures because they provide protection for the main body of the tool.

Materials Drilled and Applications

Indexable-insert drills are being used primarily for producing holes in steels and irons. More ductile materials like aluminum and copper are also drilled with these tools, but chip ejection may be a problem for some applications. When soft, ductile and gummy materials are being drilled, chip control can be a problem. The thicker chips produced with neutral or negative-rake inserts tend to pack in the flutes or grooves. The tools are not suitable for drilling soft materials such as rubber and plastics.

These tools are especially advantageous for medium and high-production applications on NC and transfer machines. They are also being used, however, for many low-production applications on manual machines, lathes, and other machine tools, providing the machines have the required power, speed, and rigidity.

In one low-volume application involving the production of 5" (127 mm) diam x 26" (660 mm) deep holes in large forgings, cycle time was reduced from 2 hours and 10 minutes with HSS twist drills to 18 minutes with indexable-insert drills. This large indexable-insert drill was designed with cartridges holding six inserts.

SPADE DRILLS

Spade drills consist of a toolholder and an interchangeable blade. A dull blade can be replaced on the machine, as on indexable throwaway tools, without the necessity of refinding size, resetting stops, breaking down setups, and increasing or decreasing the length of a drilling setup. Because of this feature, spade drills can easily be preset for use on automatic and NC machine tools. These inserted-blade tools, also now available with indexable double-edge blades, are a type of flat end-cutting drill used to produce large diameter holes. Spade drilling is done with either the tool or workpiece rotating.

Advantages of Spade Drills

The primary advantage of spade drills lies in the diameter range of standard stocked tools. A few manufacturers stock blades below 1" (25 mm) diam, usually down to 5/8" (16 mm), but the standard range of stocked tooling is 1-6" (25-152 mm), with some manufacturers listing standard blades to 15" (381 mm) or more in diameter. When compared with twist drills of comparable diameters, spade drills offer greater rigidity and lower initial cost, but chip removal is more troublesome on vertical applications. Spade drills of large diameters can be used when trepanning is impractical because of a blind hole or lack of high-pressure coolant equipment.

Another advantage is that they are available in larger diameters than twist drills and can drill deeper holes. Depth-to-diameter ratios have exceeded 120:1 in some horizontal applications of spade drills before the drills needed to be reground. For vertical drilling, however, the depth-to-diameter ratio can seldom exceed 10:1 and is even less for smaller diameter tools because of chip ejection problems.

Limitations of the Tools

Spade drills are not precision tools and should not be used for finishing operations requiring tolerances less than about ± 0.010 " (0.25 mm); subsequent finishing of the holes may be

required. Multidiameter blades, however, provide close tolerances with respect to the concentricity of different diameters and radii or chamfers.

The production of holes with spade drills requires high torque and thrust forces, as discussed later in this chapter under "Operating Parameters." Thrust requirements are sometimes more than for comparable sizes of twist drills. As a result, rigid and powerful machines are required for spade drilling.

Work surfaces that are cylindrical, spherical, or sloping and that have rough surfaces can create problems when spade drills are used, as can be the case with twist drills. Fragile workpieces are also generally difficult to drill with these tools.

Applications of Spade Drills

The majority of spade drills are used for drilling from the solid. Blades for core drilling, counterboring, flat bottoming, and special second-operation work are available for the same holders (see Fig. 9-41). They are generally employed for holes 1" (25 mm) or more in diameter.

Spade drills are popular for toolroom and low-production operations because of the low initial investment required for obtaining a wide range of sizes. For medium or high-production requirements, their primary advantages are the ease with which dull tools can be changed without disturbing the setup, and their rigidity, which permits high feed rates.

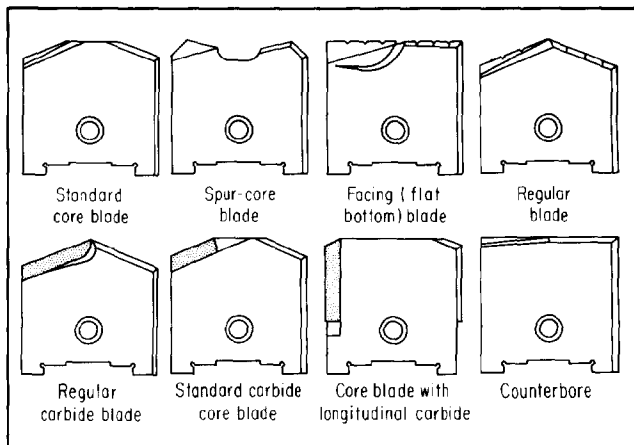


Fig. 9-41 Various blade shapes used on spade drills.

Blades for Spade Drills

Standards for spade drill blades and holders are specified in ANSI Standard B94.49-1975, published by ASME. Nomenclature for spade drill blades is presented in Fig. 9-42. Chip splitters, sometimes called chipbreaker grooves, are notches or grooves on the front lip relief surface for splitting the chips into segments for easier removal from the workpiece. They are always staggered in location from one lip to the other. Locating ears are projections on the back of the spade drill blade, beyond the seating pads, which locate and center the blade in the holder. The locating slot is the space between the locating ears that centralizes the blade in the holder.

Types of blades. Spade drill blades have been accepted by industry manufactured to two different types, the two basically differing from each other with respect to hole location and size. Some suppliers of spade drill blades manufacture them with a keyhole design to encompass both types. Others furnish them with one large hole to encompass both hole locations.

SPADE DRILLS

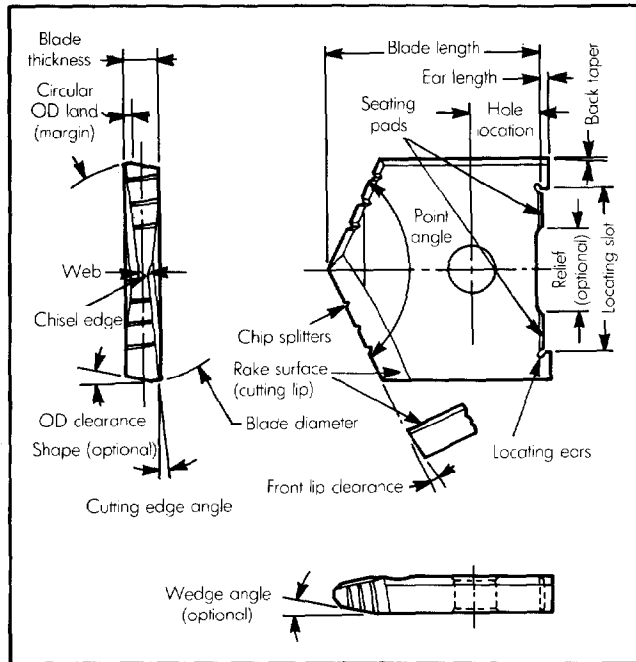


Fig. 9-42 Standard nomenclature for spade drill blades.
(ANSI Standard B94.49-1975, published by ASME)

Blade sizes. Standard spade drill blades are grouped into eight series, designated by letters A through H, having different widths, lengths, hole sizes and diameters, and locator dimensions for each series. Blade diameter ranges for the eight series are:

1. Series A: 1 to 1 1/4" (25 to 32 mm).
2. Series B: 1 17/64 to 1 1/2" (32 to 38 mm).
3. Series C: 1 33/64 to 2" (38.5 to 51 mm).
4. Series D: 2 1/64 to 2 1/2" (51 to 63.5 mm).
5. Series E: 2 3/64 to 3" (65 to 76 mm).
6. Series F: 3 1/64 to 3 1/2" (77 to 89 mm).
7. Series G: 3 33/64 to 4" (89 to 102 mm).
8. Series H: 4 1/64 to 6" (102 to 152 mm).

Many other standard sizes are available from manufacturers of spade drill blades.

Special blades are also built to specifications. Important dimensions that should be included on blade specifications are shown in Fig. 9-43. Specifying further details usually results in higher costs and delayed deliveries. Special blades with greater

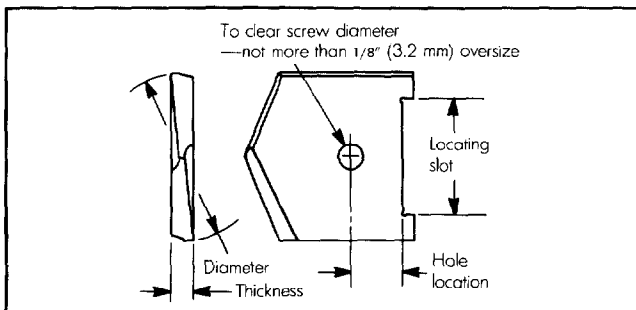


Fig. 9-43 Important dimensions that should be specified for special blades.

length-to-diameter ratios, as in piloted multidiameter blades, often have two holes and are used in special holders having two screws.

Blade materials. A comprehensive discussion of various cutting tool materials is presented in Chapter 3, "Cutting Tool Materials." The most common cutting tool materials used in spade drill blades are Types M2, M3, and M4 high-speed steels. Toughness is an important factor in selecting the blade material, but a compromise between toughness and abrasion resistance generally gives the best results. Premium grades of HSS such as T15 are also employed extensively.

High-speed steels made by powder metallurgy processes are being increasingly applied for spade drill blades. Advantages of these materials include a uniform structure with fine carbide particles and no segregation, faster and more uniform response to heat treatment, improved toughness and tool life, and superior grindability.

Blades for spade drills are also available made from cast cobalt-based alloys and carbides. Both solid carbide and carbide-tipped blades are used. Carbide-tipped blades must be operated at high cutting speeds and with light feed rates to avoid premature wear and crushing at the chisel edge. Solid carbide blades permit faster penetration rates, provide longer life, and are often preferred for drilling very hard or abrasive materials, as well as for some soft materials (except aluminum and magnesium) and for low-carbon or low-alloy steels. Rigid setup conditions are essential for the use of carbide blades.

Blade geometry. The geometry of blades for spade drills resemble that of the cutting ends for twist drills with several exceptions.

Point angle. The included angle of points on spade drills generally varies from 118-135°, with 130° being standard. The point angle cannot be changed to any significant degree without losing much of the blade length. As compared with the 118° point angle of a twist drill, the larger angle shortens the cutting edge, which increases the depth of cut and chip thickness, as well as the cutting force for a given feed. This seeming disadvantage is negated by the lower speeds at which spade drills are operated.

Thicker chips are necessary in spade drilling to create positive curling and breaking. Long stringy chips do not flush from the hole, and they cause packing and breakage. Smaller included angles, 75-90°, however, are sometimes used for drilling certain materials. Split or four-facet points can be ground on the spade drill points to minimize wandering. A secondary point angle or corner chamfer is also used sometimes to lengthen life and permit the use of faster cutting speeds.

Clearance (relief) face. The clearance or relief face, which is generally notched for chip splitting and breaking, is typically ground with a lip relief angle of 6-8°. This angle is increased to 10-12° on core drills, on drills with point angles under 90°, and on drills used for drilling aluminum and other easily machined materials. The thrust requirements on large-diameter drills can also be reduced by increasing the clearance angle and/or by relieving the point. Angles that are more than 12°, however, create a weak cutting edge and should only be used to drill very soft metals. Angles less than 5° generally cause higher thrust requirements and generate excessive heat, thus shortening tool life.

Rake angles. The rake angle on blades ground with a flat top-rake surface is 12°; on those ground with a cylindrical chip curler, it ranges from 12 to nearly 30° at the outer corner and from nearly zero to about 10° adjacent to the chisel edge. Since it is sometimes difficult to duplicate in regrinding, the rake

angle must be considered a variable except on blades with a flat top rake surface. With these, rake angles are sometimes reduced for maximum heat dissipation, for abnormally high feed rates, for hard or brittle-abrasive materials, for materials which produce stringy chips, and for carbide blades. Rake angles higher than 12° are sometimes useful for applications with very light feed rates, softer materials, and materials which give a brittle-stiff or tightly curled chip. Higher rake angles reduce cutting forces and produce less heat from chip-tool friction, but the overriding consideration must always be the achieving of a well-curved chip that flushes readily from the hole.

Indexable blades. Double-edge blades of the throwaway type eliminate the need for regrinding. When one edge becomes dull, the blade is indexed 180° so that the second edge can be used. When the second edge becomes dull, the blade is discarded since regrinding it is uneconomical. It is claimed that these blades can be operated with an increase in feed rate of 15-20%, compared to single-edge blades. Cutting-edge geometry is similar to that of conventional spade drill blades except for the two cutting edges (see Fig. 9-44). The V-shape of the two cutting lips serves as a locator for seating the blade.

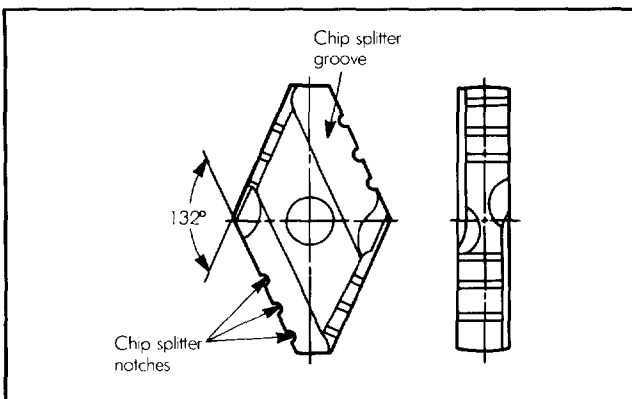


Fig. 9-44 Indexable blade for spade drill which has chip splitter grooves and notches.

Holders for Spade Drills

The nomenclature for spade drill holders is presented in Fig. 9-45. As with the blades, holders have been standardized in two types and grouped into eight series, designated by letters A through H, to accept blades of different widths and diameter ranges. Only eight holders are necessary for the nominal range of 1-6" (25-152 mm) diam. Other holders are available to hold larger diameter and wider blades.

Standard holders are available with straight or taper shanks. Four lengths are standard with most manufacturers: stub for starting and spotting; short for shallow holes; long (or conventional length), which are roughly equivalent to taper-length twist drills; and extra long. Special holders can be made to any length required. Many holders have internal holes for delivery of cutting fluid to the cutting edges.

When support is essential for a long, relatively flexible holder, such as during operations involving locating by a bushing remote from the work surface, breaking through a series of ribs, or line-drilling walls of a box casting or weldment, wear strips on the holder body that have been ground to about 0.005" (0.13 mm) under the blade diameter should be used. Wear strips are usually made from tool steel, nonferrous bearing material, or cast alloy. Wear strips interfere with chip

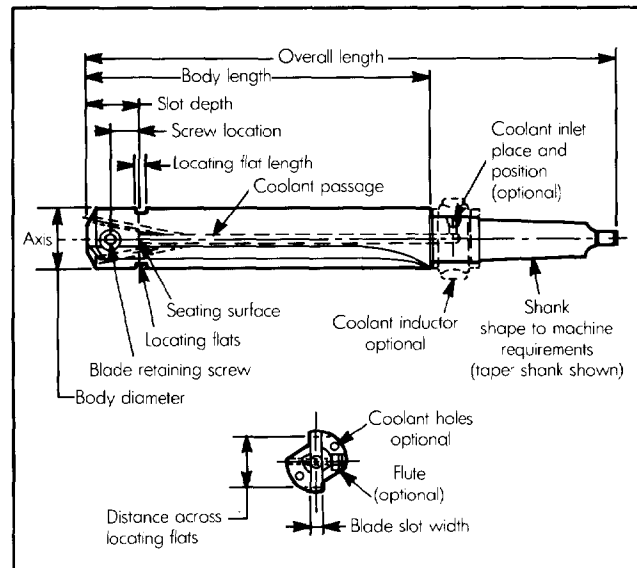


Fig. 9-45 Standard nomenclature for spade drill holders. (ANSI Standard B94.49-1975, published by ASME)

ejection and require frequent regrinding, so they should be avoided unless necessary. The use of wear strips limits the use of the holder to the diameter at which the wear strips are ground, but in most cases the holder can be used without wear strips for other applications. Holder strength is usually not reduced enough to limit such applications. A spade drill holder with wear strips is shown in view *a* of Fig. 9-46, with other styles of spade drills in view *b*.

One tool manufacturer uses wear studs instead of wear strips to provide a four-point contact during the drilling operation (see Fig. 9-47). The wear studs are interchangeable for each blade size within the holder range. Chip flow is not impeded by the use of wear studs. Blade holders provided with pockets for wear studs can be used without the studs for normal drilling applications when breakthrough and rigidity are not a problem.

Holders can be altered for special applications and to perform multiple operations at a single machining position. Figure 9-48 illustrates an indexable-insert cartridge bolted to a pocket in a holder to perform drilling and chamfering in the same operation.

HIGH-PRESSURE COOLANT DRILLS

High-pressure coolant drills are used to produce deeper and/or more precise holes, as well as shallow holes to close tolerances in one pass. They differ from the coolant-fed drills previously discussed in that they are self-guiding, more costly, operate with higher pressure cutting fluids, and are generally used in specialized machines. They can drill much deeper holes than the high-helix, crankshaft, parabolic flute, coolant-hole, and spade drills already described. The major types of high-pressure coolant drills are gundrills, multiple-lip drills, and trepanning tools.

Gundrills

Gun-type tools, or gundrills and gunbores, are single-lip, self-guiding, pressure-coolant tools. Gun-type tools are classified in two basic categories: (1) external chip removal and (2) internal chip removal.

HIGH-PRESSURE COOLANT DRILLS

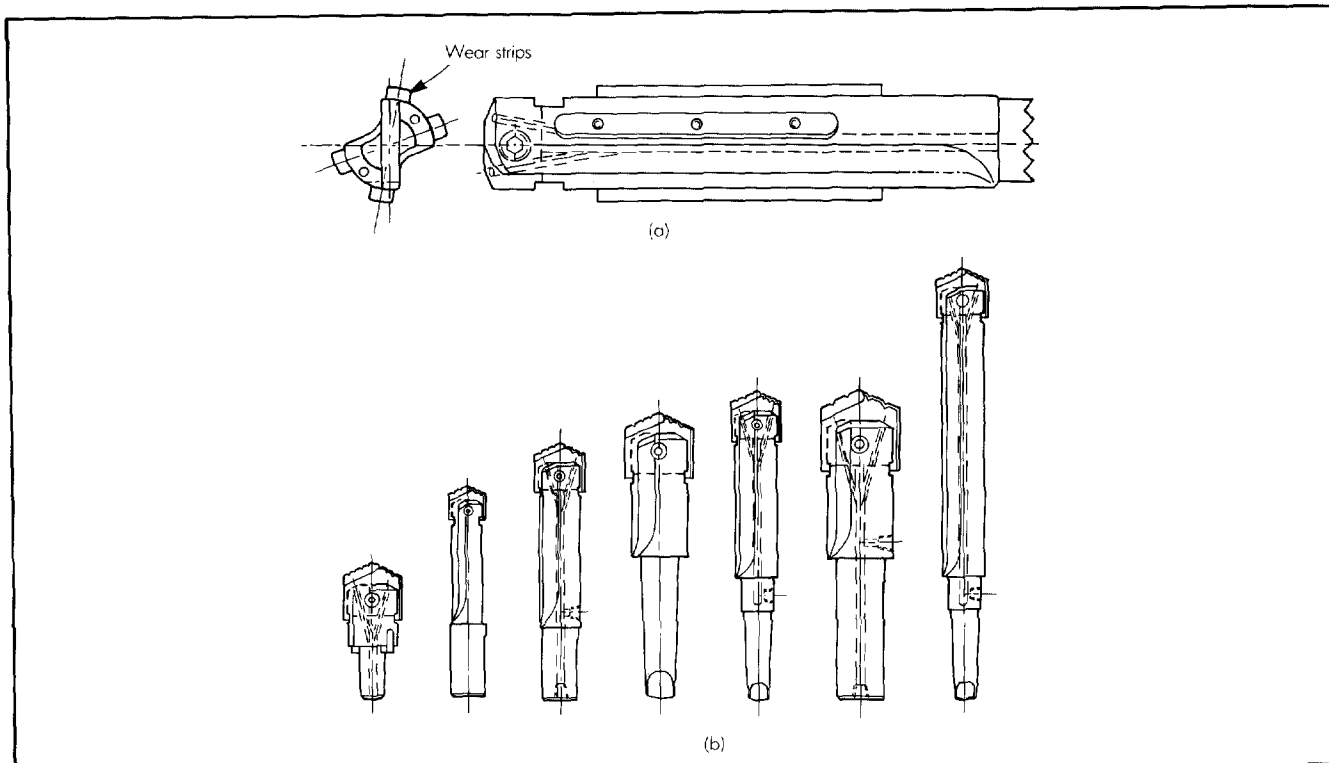


Fig. 9-46 (a) spade drill with wear strips; and (b) other styles of spade drills.

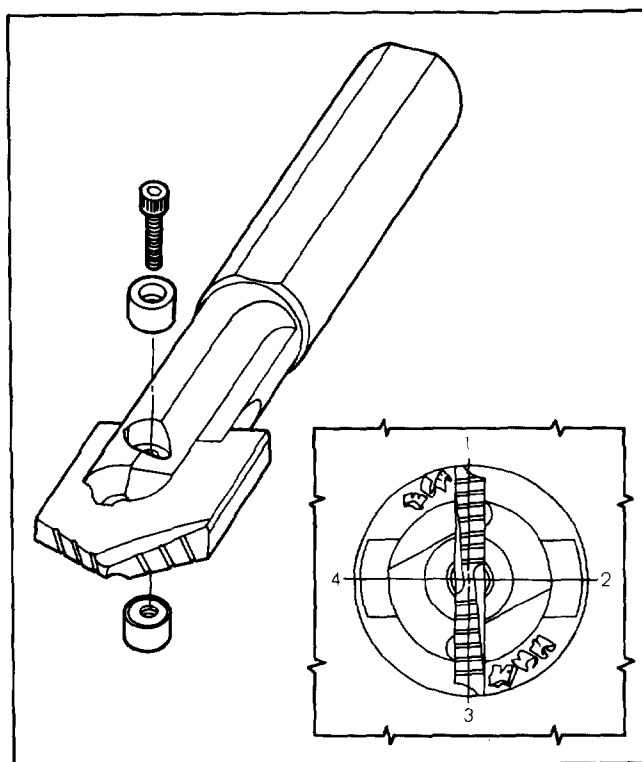


Fig. 9-47 Use of wear studs instead of wear strips, providing four-point contact during drilling, as shown at right. (Erickson Div., Kennametal Inc.)

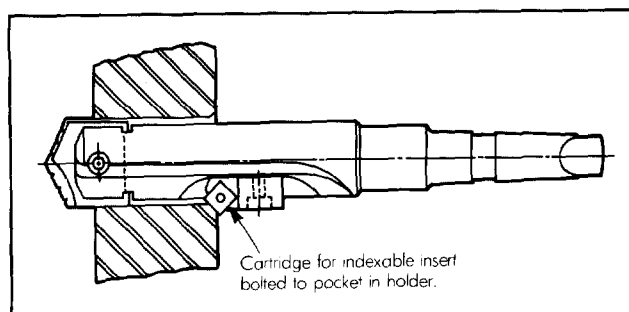


Fig. 9-48 Indexable insert cartridge mounted in holder to perform drilling and chamfering in same operation. (Union/Butterfield Div., Litton Industrial Products)

External-chip-removal gundrill. An external-chip-removal gundrill (see Fig. 9-49) includes a tip with a single V-shaped cutting edge and a coolant hole, a single-fluted tubular shank, and a driver suitably designed for either a spindle socket or a stationary socket. The cutting edge usually cuts through center on one side of the hole, leaving no material to be extruded as with the web of a twist drill. The seldom-used pin-cutting gundrill does not cut to center, but leaves a central pin. This type is limited to use in through holes because it is difficult to remove the pin from a blind hole.

The tip of a gundrill is solid carbide or a carbide insert in a steel body. Solid carbide tips are preferred to eliminate thermal-expansion differences and possible cracking of carbide, but they are generally available only up to 1.500" (38.10 mm) diam. The oil hole through the tip is approximately one-fourth the tip diameter. The tip is attached to a steel shank by brazing or by

HIGH-PRESSURE COOLANT DRILLS

some detachable method. The shank, with the cross section shown in Fig. 9-49, channels high-pressure coolant through the inside, and the chips and coolant exit through the V-flute.

The drill is held in the spindle by a cylindrical driver and is usually retained by setscrews. Other means of retaining the gundrill are used to accomplish quick change and length presetting. Recommended driver sizes and overall lengths are given in Table 9-10. Of course, any length or driver size is available, but it is most economical to stay with standards.

Gundrills are manufactured to a diameter $+0.0000$ to 0.0002 " (0.005 mm) with back-taper of 0.0006 - 0.0008 in./in. (or mm/mm) of tip. This combination of back-taper and diametral tolerance may produce a 0.001 " (0.03 mm) reduction in hole size over the life of the tool as it is resharpened depending upon how it is pointed.

Gundrills are manufactured with different contours (see Fig. 9-50) for application to various materials. The standard contour can be applied to all materials, with special contours required only for precision (under 0.001 ", 0.03 mm) in soft materials such as aluminum or for use of the gundrill as a gunbore.

Gundrills are applied when other hole-making tools would require special handling (i.e., woodpeckering feed) or secondary qualifying operations (i.e., reaming, honing, straightening).

They are economically used for any length hole, from less than 2 to more than 125 diameters deep. The selection of gundrilling over more conventional drilling methods should be made on an economic basis, taking due consideration of eliminated or reduced finishing, handling, or inspection operations.

Gundrills are available from 0.0750 - 2.0000 " (1.905 - 50.800 mm) diam and larger and in lengths to beyond" (3810 mm). Holes produced by gundrilling are generally within 0.0005 " (0.013 mm), while in production a total diametral tolerance of 0.001 " (0.03 mm) is normal over full tool life. The holes are round and have a good finish, generally better than 80 μ in. (2 μ m) and frequently better than 30 μ in. (0.76 μ m). Although the location of a hole at entry is dependent upon fixturing and part condition, it is usually within 0.001 " (0.03 mm). Except when drilled too near a thin section on one side, a gundrilled hole is straight with runout not exceeding 0.001 in. in./in. (or mm/mm) of depth for the smaller tools. With proper fixturing practices and well-maintained equipment, runout has been held to less than 0.0002 in./in. (0.005 mm/mm) in production situations. Depth of hole is limited only by the equipment and gundrills available. By taking advantage of the low runout, twice the hole depth is possible when the part is drilled from both ends. With gundrilling, one can produce intersecting,

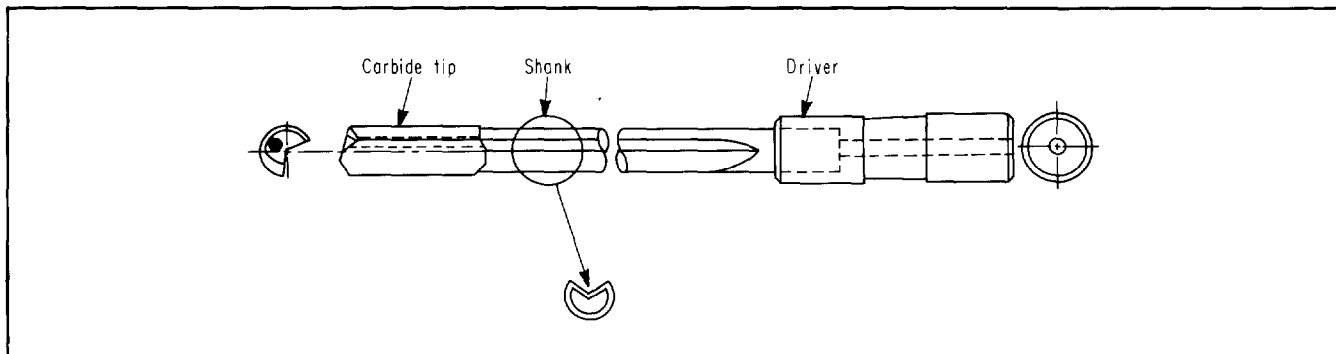


Fig. 9-49 Conventional external-chip-removal gundrill.

TABLE 9-10
Recommended Gundrill Drive Sizes and Overall Lengths*

Drill diam, in. $+0.0000$, -0.0002	Driver size, in. diam x length	Suggested overall lengths, in.
0.0781-0.0882	0.50 x 1.50	6-10-16-22
0.0883-0.1086	0.50 x 1.50	6-10-16-22-28
0.1087-0.1249	0.50 x 1.50	6-10-16-22-28-36
0.1250-0.1350	0.75 x 2.75	10-16-22-28-36
0.1351-0.1561	0.75 x 2.75	10-16-22-28-36-42
0.1562-0.1771	0.75 x 2.75	10-16-22-28-36-42-48
0.1772-0.1999	0.75 x 2.75	10-16-22-28-36-42-48-54
0.2000-0.2199	0.75 x 2.75	10-16-22-28-36-42-48-54-60
0.2200-0.2399	0.75 x 2.75	10-16-22-28-36-42-48-54-60-66
0.2400-0.2599	0.75 x 2.75	10-16-22-28-36-42-48-54-60-66-72
0.2600-0.2799	0.75 x 2.75	10-16-22-28-36-42-48-54-60-66-72-78
0.2800-0.5000	0.75 x 2.75	10-16-22-28-36-42-48-54-60-66-72-78-84
0.5001-0.6399	1.00 x 2.75	16-22-28-36-42-48-54-60-66-72-78-84
0.6400-0.7500	1.00 x 2.75	16-22-28-36-42-48-54-60-66-72-78-84-96-108-120-132-144-150
0.7501-1.0000	1.25 x 2.75	22-28-36-42-48-54-60-66-72-78-84-96-108-120-132-144-150
1.0001-1.2500	1.50 x 2.75	36-42-48-54-60-66-72-78-84-96-108-120-132-144-150
1.2501-2.0000	2.00 x 2.75	36-42-48-54-60-66-72-78-84-96-108-120-132-144-150

* Special lengths and drives available as required.

CHAPTER 9

HIGH-PRESSURE COOLANT DRILLS

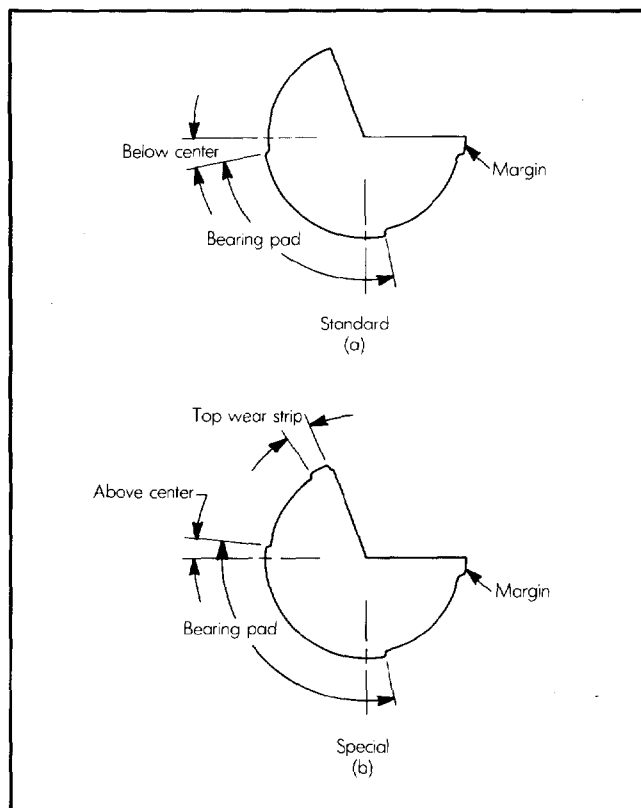


Fig. 9-50 (a) Standard contour and (b) special contour for gundrills.

crossing, and overlapping holes, as illustrated in Fig. 9-51. Almost any material can be drilled economically, from wood or plastic to titanium or Inconel.

Table 9-11 indicates the maximum unsupported length, which is commonly called the whip length, for gundrilling various materials. This is the maximum length of shank that can be run without support at the indicated speed. Any longer unsupported length will result in ballooning or whip of the shank, which causes poor hole quality or gundrill failure.

Many nose geometries have been developed over the years, but the most common is the general-purpose geometry shown in Fig. 9-52, *a*. Another common geometry used on soft materials, such as aluminum and brass, is shown in Fig. 9-52, *b*. The geometry in Fig. 9-52, *c*, was developed for gundrilling stacked parts but also yields good results on a variety of other applications. With gundrilling it is possible, depending upon material and diameter, to produce holes directly with radius and spherical bottom forms by using a suitable nose grind. Gundrills can be manufactured to produce two diameters simultaneously.

Gundrills may be used horizontally or vertically with the tool or the work, or both, rotating. For fixturing and economic considerations, most modern gundrilling equipment is horizontal with the tool rotating and advancing (see Fig. 9-53). Because high speeds are used with light feeds on carbide cutting edges, discussed later in this chapter under the subject of "Operating Parameters," it is important that gundrilling machinery be rigid and have uniform feed rates. The combination of feeds and speeds is usually such that equipment specifically designed for gundrilling is necessary to derive

optimum results. High-pressure coolant-supply systems, also discussed later, are required.

As the gundrill enters the workpiece, it must be guided by a bushing which is in intimate, rigid contact with the work. This bushing is mounted to the gundrilling head, usually in the front wall of a chip box which is used to collect coolant and chips exiting from the hole being drilled. The lineup of this bushing and the machine's spindle nose must be maintained accurately over the full range of travel in order to ensure proper results. Intimate bushing contact can be assured only if the workpiece is clamped against it. For moderate-sized parts, yoke clamping direct to the chip box is best (see Fig. 9-54). When gundrilling through holes, the exit should be blocked so that all the coolant and chips continue to go back through the hole into the chip box after breakout.

Holes concentric to each other may be produced through a sequence of gundrilling operations, taking advantage of the starting bushing as a concentricity register. First, the largest hole is gundrilled. Then an extended bushing is inserted which locates at the bottom of the hole (see Fig. 9-55) on the cone generated by the outer cutting edge. This bushing guides the second gundrill, producing holes concentric at their interface.

Gundrilling has been incorporated into high-production transfer equipment in many industries. Automatic equipment, with up to 352 gundrilling spindles on one machine, is being used in production. Numerically controlled gundrilling consistently produces tube sheets with up to 17,000 holes per part.

Internal-chip-removal gundrill. Internal-chip-removal gundrills, Fig. 9-56, are single-lip, self-guiding, pressure-coolant tools and are applied for the same purposes as external-chip-removal gundrills. They are sometimes referred to as hollow gundrills or drillheads and are available from 0.24-4.0" (6.1-102 mm) diam. Depth is limited only by equipment and tools available. The holes produced are round and have good finish with a diametral tolerance from 0.002-0.004" (0.05-0.10 mm). As with other types of gundrills, runout is low and the hole is straight.

Internal-chip-removal tools (or ID exhaust tools) are mounted on a tubular steel shank. The coolant is introduced between the outside of the boring bar and the ID of the hole being produced. Then, the coolant moves past the cutting edges of the tool and discharges with the chips through the ID of the tubular shank. This arrangement of drilling can also be used for ID exhaust trepanning and gun-reaming (counterboring), as illustrated in Fig. 9-57.

A fluid transfer unit (pressure head) is essential for ID exhaust tools to introduce the coolant to the tool, to seal against the workpiece face to contain the coolant, to guide the tool into the work, and to seal the OD of the tubular shank (boring bar) to contain the coolant (see Fig. 9-58). Internal diameter exhaust tools require that the cutting fluid be at high pressure and/or high volume. Normally, coolant induction is near the starting face of the workpiece, directly behind the starting bushing. This requires that the bushing and the machine spindle be perfectly concentric. In addition, the bushing ID must be held to close tolerance, as indicated in Table 9-12.

The tip of the internal-chip-removal gundrill consists of a hollow steel body with an inserted carbide cutting edge and carbide bearing strips. The cutting edge and bearing strips are brazed to the steel body, then ground cylindrically to form the tool diameter. The diameter of the body must be slightly smaller than that of the complete tool to allow passage of the coolant around the tool.

HIGH-PRESSURE COOLANT DRILLS

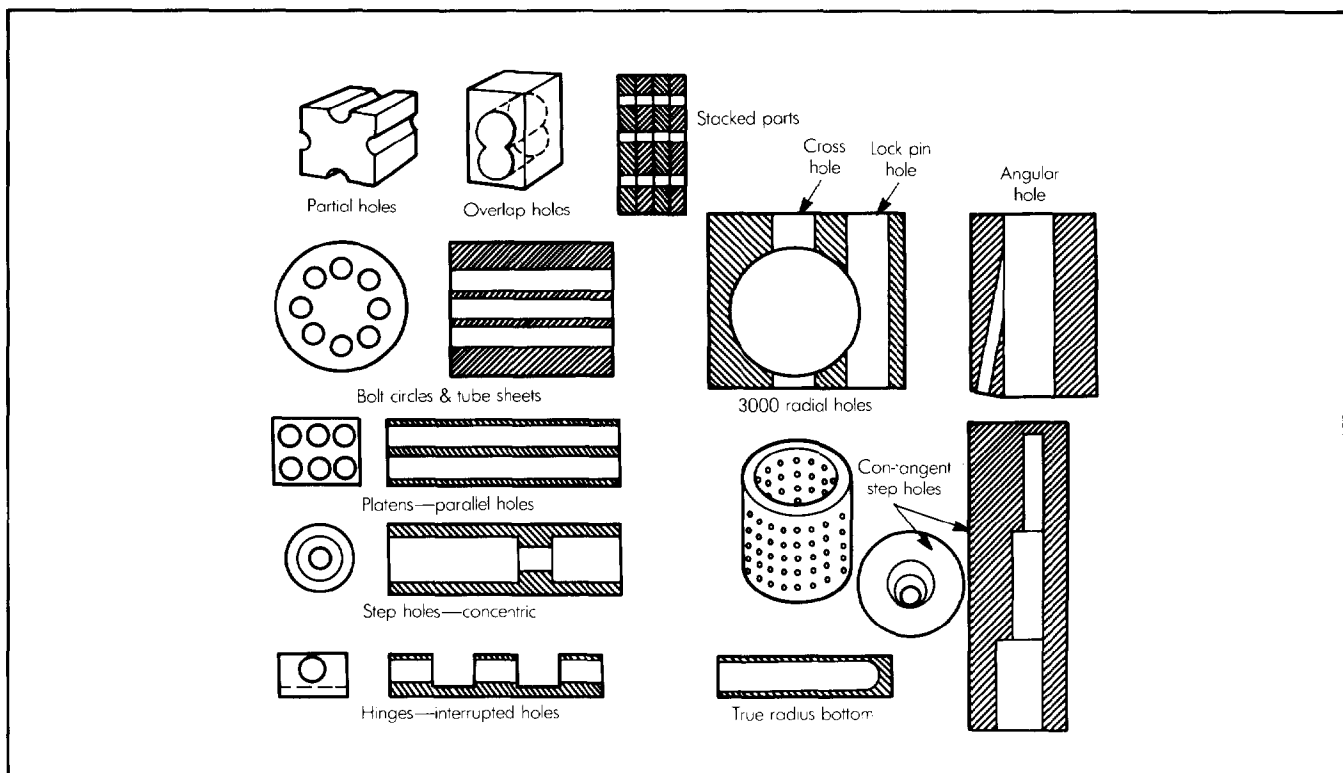


Fig. 9-51 Various types of holes produced by gundrilling.

When the drilling operation begins, the tool (guided by the boring bushing) is aligned centrally by the tool bit and the two carbide guide pads. The guide pads, often of a different carbide grade than the cutting edge, are usually located 180° opposed to the cutting plane and 90° from the cutting edge. The pads may be positioned within a range of a few degrees of these locations, according to the cutting characteristics of the work material and the hole diameter. The 90° pad absorbs most of the cutting force; the 180° pad acts as a gage reference and helps establish the size of the hole.

The carbide pads size and burnish the hole and then utilize it as a pilot bushing for continued penetration. This self-piloting action, using the machined bore for support to oppose the forces generated by the cutting action, continues for the entire hole depth. This prevents deflection of the tool, giving a more accurate diameter and straighter path, and allows the tool to be used at higher feeds and speeds because of the greater rigidity and resistance to chatter or vibration. It should be noted that this principle of machining is also true for ID exhaust trepanning and gun-reaming tools.

The external profile geometry of the guide pads is important, with the leading and trailing ramps always relieved to specific angles. The normally specified 18° angle of the leading ramp serves to trap and force the fluid onto and over the bearing face of the pad; a continuous fluid film is thus maintained between the pad and the wall of the bore being produced. Because this fluid film is important to the operation of the tool, the filtration system must remove all particles which might interfere with the continued coating of the bore with the film.

Cutter geometry is very important, because it pertains to both the actual cutting process and the chipbreaking operation.

Generally, the smaller the tool, the more significant the tool geometry becomes because of the diminished space available for chip ejection.

The nose is designed so that the tool point is offset from the axis of the cutter head (see Fig. 9-59). Near the center cut of the inner angle, a 30° back rake angle is introduced to strengthen the cutting edge in that area. To obtain narrower chips, the cutting edge must be divided into two or three steps (chip separators) depending upon hole diameter and the chipforming characteristics of the work material. A single cutting edge is normally used for drills not exceeding $3/8"$ (9.5 mm) diam, a two-step edge for drills $3/8$ to $1 1/4"$ (9.5 to 32 mm) diam, and a three-step edge for drills over $1 1/4"$ (32 mm) diam. Chipbreakers are required to form C-shaped chips that must be small enough to exit freely through the very restrictive chip mouth and throat. With poor chip-forming materials, a chipbreaker with a small positive rake may be used.

The shanks used for internal-chip-removal gundrills are of tubular construction and hardened to provide wear resistance and strength. Outside diameter tolerance and finish, as well as straightness, are critical factors in the protection of fluid seals and vibration dampeners. Tips under 0.790" (20 mm) diam are sometimes brazed onto the shank; larger sizes are generally mechanically attached.

Multiple-Lip Pressure-Coolant Drills

Multiple-lip pressure-coolant drills are self-piloting tools available as both external and internal-chip-removal types, (see Fig. 9-60). External-chip-removal drills are generally available from $1/4$ to $1 3/4"$ (6.35 to 44.45 mm) diam, while internal-chip-removal types are generally not available below $5/8"$ (16 mm).

TABLE 9-11
Recommended Starting Conditions for Gundrilling

Gundrill Diam	Coolant Pressure psi (kPa)	Low and Medium- Carbon Steel to 200 Bhn			Alloy Steel to 240 Bhn			Tool Steel to 200 Bhn			300 Stainless Steel to 200 Bhn		
		rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)
0.0781	1500 (10 342)	10,000	1.4 (36)	4.5 (114)	10,000	1.6 (41)	4.5 (114)	8560	1.3 (33)	5.2 (132)	9780	1.2 (30)	4.6 (117)
0.0937	1500 (10 342)	10,000	1.8 (46)	5.0 (127)	10,000	2.0 (51)	5.0 (127)	7135	1.4 (36)	5.9 (150)	8150	1.3 (33)	5.5 (140)
0.1250	1500 (10 342)	10,000	2.4 (61)	5.8 (147)	9165	2.5 (63)	6.1 (155)	5345	1.5 (38)	8.0 (203)	6110	1.3 (33)	7.4 (188)
0.1562	1300 (8963)	10,000	3.0 (76)	6.5 (165)	7335	2.5 (63)	7.6 (193)	4280	1.5 (38)	10.0 (254)	4890	1.3 (33)	9.3 (236)
0.1875	1150 (7929)	9170	3.2 (81)	7.5 (190)	6110	2.5 (63)	9.2 (234)	3565	1.4 (36)	12.0 (305)	4075	1.3 (33)	11.2 (284)
0.2187	1050 (7240)	7860	3.1 (79)	8.6 (218)	5240	2.4 (61)	10.6 (269)	3055	1.4 (36)	13.9 (353)	3490	1.2 (30)	13.0 (330)
0.2500	925 (6378)	6875	3.1 (79)	10.0 (254)	4585	2.4 (61)	12.3 (312)	2675	1.4 (36)	16.1 (409)	3055	1.2 (30)	15.0 (381)
0.2812	850 (5861)	6110	3.0 (76)	11.3 (287)	4075	2.3 (58)	13.8 (351)	2375	1.3 (33)	18.1 (460)	2715	1.1 (28)	16.9 (429)
0.3125	775 (5344)	5500	2.9 (74)	12.6 (320)	3665	2.2 (56)	15.4 (391)	2140	1.3 (33)	20.1 (511)	2445	1.1 (28)	18.8 (472)
0.3437	725 (4999)	5000	2.8 (71)	13.8 (351)	3335	2.2 (56)	16.9 (429)	1945	1.3 (33)	22.2 (564)	2220	1.1 (28)	20.7 (526)
0.3750	675 (4654)	4585	2.8 (71)	15.1 (384)	3055	2.1 (53)	18.5 (470)	1780	1.2 (30)	24.2 (615)	2040	1.1 (28)	22.6 (574)
0.4062	625 (4309)	4230	2.7 (69)	16.4 (417)	2820	2.1 (53)	20.0 (508)	1645	1.2 (30)	26.2 (665)	1880	1.0 (25)	24.5 (622)
0.4375	600 (4137)	3930	2.6 (66)	17.6 (447)	2620	2.0 (51)	21.6 (549)	1530	1.2 (30)	28.2 (716)	1745	1.0 (25)	26.4 (671)
0.4687	550 (3792)	3670	2.6 (66)	18.6 (472)	2445	2.0 (51)	23.1 (587)	1425	1.2 (30)	30.3 (770)	1630	1.0 (25)	28.3 (719)
0.5000	525 (3620)	3440	2.5 (63)	20.1 (511)	2290	1.9 (48)	24.7 (627)	1335	1.1 (28)	32.3 (820)	1530	1.0 (25)	30.2 (767)
0.5312	500 (3447)	3235	2.5 (63)	21.4 (544)	2155	1.9 (48)	26.2 (665)	1260	1.1 (28)	34.3 (871)	1440	0.9 (23)	32.1 (815)
0.5625	500 (3447)	3055	2.4 (61)	22.7 (577)	2035	1.9 (48)	27.8 (706)	1190	1.1 (28)	36.3 (922)	1360	0.9 (23)	34.0 (864)
0.5937	475 (3275)	2895	2.4 (61)	24.1 (612)	1930	1.8 (46)	29.5 (749)	1125	1.1 (28)	38.6 (980)	1290	0.9 (23)	36.0 (914)
0.6250	450 (3103)	2750	2.3 (58)	25.3 (643)	1835	1.8 (46)	31.0 (787)	1070	1.0 (25)	40.6 (1031)	1220	0.9 (23)	38.0 (965)
0.6562	450 (2930)	2620	2.3 (58)	26.6 (676)	1745	1.8 (46)	32.6 (828)	1020	1.0 (25)	42.6 (1082)	1165	0.9 (23)	39.9 (1013)
0.6875	425 (2930)	2500	2.2 (56)	27.9 (709)	1665	1.7 (43)	34.1 (866)	970	1.0 (25)	44.7 (1135)	1110	0.9 (23)	41.8 (1062)
0.7187	400 (2758)	2390	2.2 (56)	28.9 (734)	1595	1.7 (43)	35.4 (899)	930	1.0 (25)	46.4 (1179)	1060	0.8 (20)	43.4 (1102)
0.7500	400 (2758)	2290	2.2 (56)	30.4 (772)	1530	1.7 (43)	37.2 (945)	890	1.0 (25)	48.8 (1240)	1020	0.8 (20)	45.5 (1156)

TABLE 9-11—Continued

400 Stainless Steel to 200 Bhn			Gray Cast Iron Bronze			Ductile Cast Iron Aluminum			to 180 Bhn			to 200 Bhn		
rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)	rpm	ipm (mm/min)	in.* (mm)
10,000	1.4 (36)	4.5 (114)	10,000	1.6 (41)	4.5 (114)	10,000	2.5 (63)	4.5 (114)	10,000	3.6 (91)	4.5 (114)	10,000	2.8 (71)	4.5 (114)
10,000	1.8 (46)	5.0 (127)	10,000	2.0 (51)	5.0 (127)	10,000	3.3 (84)	5.0 (127)	10,000	4.9 (124)	5.0 (127)	10,000	3.8 (97)	5.0 (127)
7640	1.8 (46)	6.7 (170)	10,000	2.8 (71)	5.8 (147)	10,000	4.8 (122)	5.8 (147)	10,000	7.1 (180)	5.8 (147)	9165	5.0 (127)	6.1 (155)
6115	1.8 (46)	8.3 (211)	10,000	3.4 (86)	6.5 (165)	10,000	6.0 (152)	6.5 (165)	9780	8.9 (226)	6.6 (168)	7335	5.1 (130)	7.6 (193)
5090	1.8 (46)	10.1 (257)	9170	3.7 (94)	7.5 (190)	10,000	7.1 (180)	7.2 (183)	8150	8.9 (226)	7.9 (201)	6110	5.0 (127)	9.2 (234)
4360	1.7 (43)	11.6 (295)	7860	3.6 (91)	8.6 (218)	10,000	8.2 (208)	7.7 (196)	6985	8.8 (224)	9.2 (234)	5240	4.9 (124)	10.6 (269)
3820	1.7 (43)	13.4 (340)	6875	3.5 (89)	10.0 (254)	10,000	9.1 (231)	8.3 (211)	6110	8.6 (218)	10.6 (269)	4585	4.8 (122)	12.3 (312)
3395	1.7 (43)	15.1 (384)	6110	3.5 (89)	11.3 (287)	10,000	10.1 (257)	8.8 (224)	5435	8.4 (213)	12.0 (305)	4075	4.7 (119)	13.8 (351)
3055	1.6 (41)	16.8 (427)	5500	3.4 (86)	12.6 (320)	9780	10.7 (272)	9.4 (239)	4890	8.3 (211)	13.3 (338)	3665	4.6 (117)	15.4 (391)
2780	1.6 (41)	18.5 (470)	5000	3.3 (84)	13.8 (351)	8890	10.4 (264)	10.4 (264)	4445	8.1 (206)	14.7 (373)	3335	4.5 (114)	16.9 (429)
2545	1.5 (38)	20.2 (513)	4585	3.2 (81)	15.1 (384)	8150	10.2 (259)	11.3 (287)	4075	7.9 (201)	16.0 (406)	3055	4.4 (112)	18.5 (470)
2350	1.5 (38)	21.9 (556)	4230	3.2 (81)	16.4 (417)	7520	10.0 (254)	12.3 (312)	3760	7.8 (198)	17.3 (439)	2820	4.3 (109)	20.0 (508)
2180	1.5 (38)	23.7 (602)	3930	3.0 (76)	17.6 (447)	6895	9.8 (249)	13.2 (335)	3490	7.6 (193)	18.7 (475)	2620	4.2 (107)	21.6 (549)
2035	1.4 (36)	25.3 (643)	3670	3.0 (76)	18.9 (480)	6520	9.6 (244)	14.2 (361)	3260	7.5 (190)	20.0 (508)	2445	4.2 (107)	23.1 (587)
1910	1.4 (36)	27.0 (686)	3440	2.9 (74)	20.1 (511)	6110	9.4 (239)	15.1 (384)	3055	7.3 (185)	21.4 (544)	2290	4.1 (104)	24.7 (627)
1800	1.4 (36)	28.7 (729)	3235	2.9 (74)	21.4 (544)	5750	9.2 (234)	16.1 (409)	2875	7.2 (183)	22.7 (577)	2155	4.0 (102)	26.2 (665)
1700	1.3 (33)	30.4 (772)	3055	2.8 (71)	22.7 (577)	5430	9.0 (229)	17.0 (432)	2715	7.1 (180)	24.1 (612)	2035	3.9 (99)	27.8 (706)
1610	1.3 (33)	32.3 (820)	2895	2.8 (71)	24.1 (612)	5145	8.9 (226)	18.0 (457)	2575	6.9 (175)	25.5 (648)	1930	3.9 (99)	29.5 (749)
1530	1.3 (33)	33.9 (861)	2750	2.7 (69)	25.3 (643)	4890	8.7 (221)	19.0 (483)	2445	6.8 (173)	26.9 (683)	1835	3.8 (97)	31.0 (787)
1455	1.3 (33)	35.7 (907)	2620	2.6 (66)	26.6 (676)	4655	8.6 (218)	19.9 (505)	2330	6.7 (170)	28.2 (716)	1745	3.7 (94)	32.6 (828)
1390	1.3 (33)	37.4 (950)	2500	2.6 (66)	27.9 (709)	4445	8.5 (216)	20.9 (531)	2220	6.6 (168)	29.6 (752)	1665	3.7 (94)	34.1 (866)
1330	1.2 (30)	38.8 (986)	2390	2.6 (66)	28.9 (734)	4250	8.3 (211)	21.7 (51)	2125	6.5 (165)	30.7 (780)	1595	3.6 (91)	35.4 (899)
1275	1.2 (30)	40.7 (1034)	2290	2.6 (66)	30.4 (772)	4075	8.2 (208)	22.8 (579)	2040	6.4 (163)	32.2 (818)	1530	3.6 (91)	37.2 (945)

* Maximum allowable unsupported gundrill shank length. Length in excess of these figures will cause shank to whip (balloon). (Eldorado Tool & Mfg. Corp.)

HIGH-PRESSURE COOLANT DRILLS

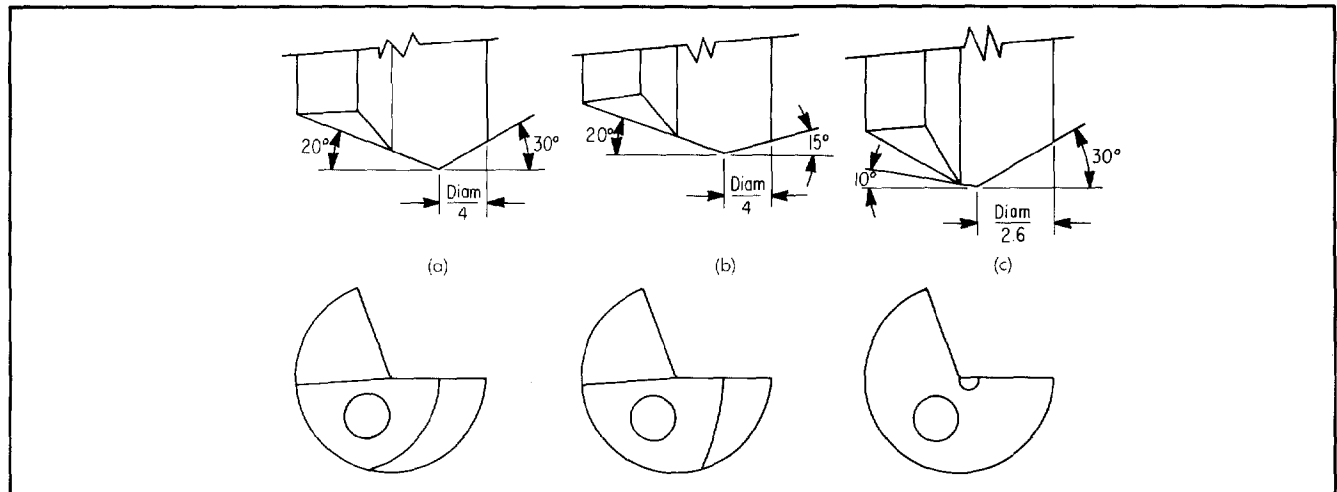


Fig. 9-52 Common nose grinds used for gundrills.

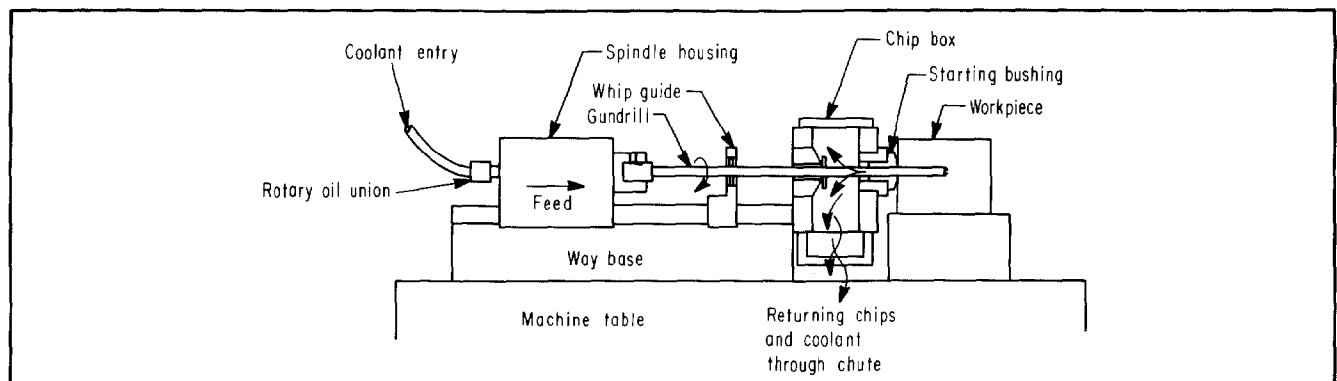


Fig. 9-53 Setup for gundrilling with the tool fed horizontally.

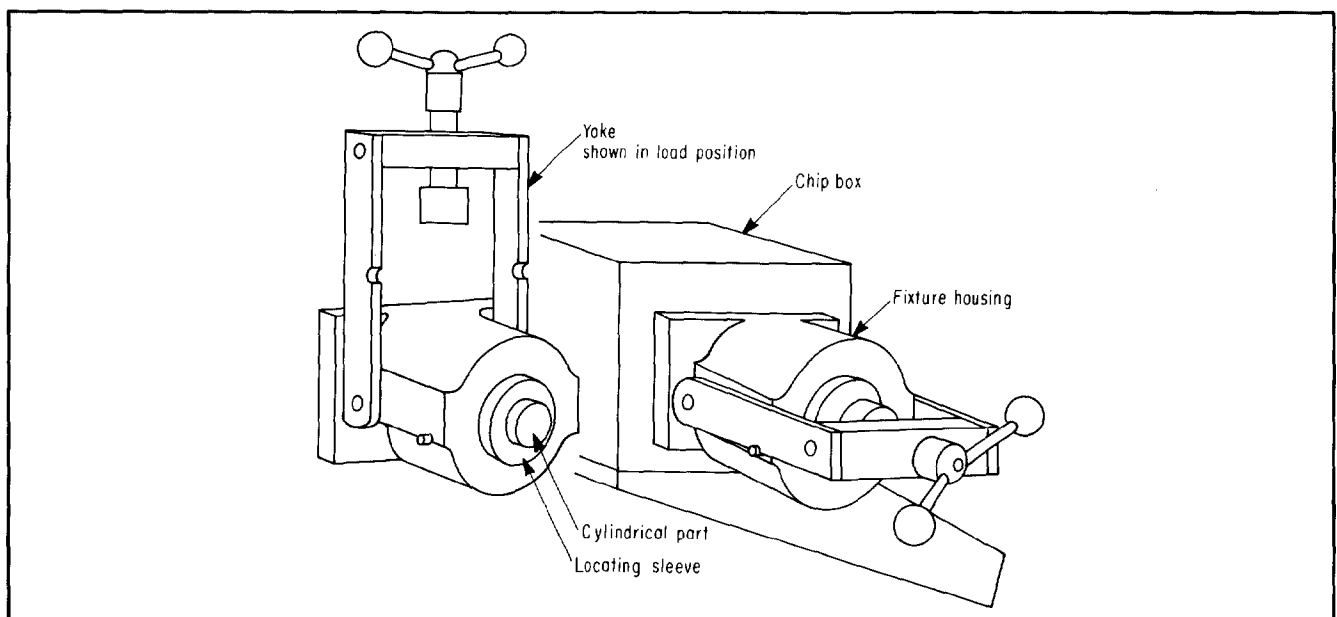


Fig. 9-54 Fixture with yoke-type end clamp, mounted on chip box, can be used for any shape workpiece with appropriate locating sleeve.

HIGH-PRESSURE COOLANT DRILLS

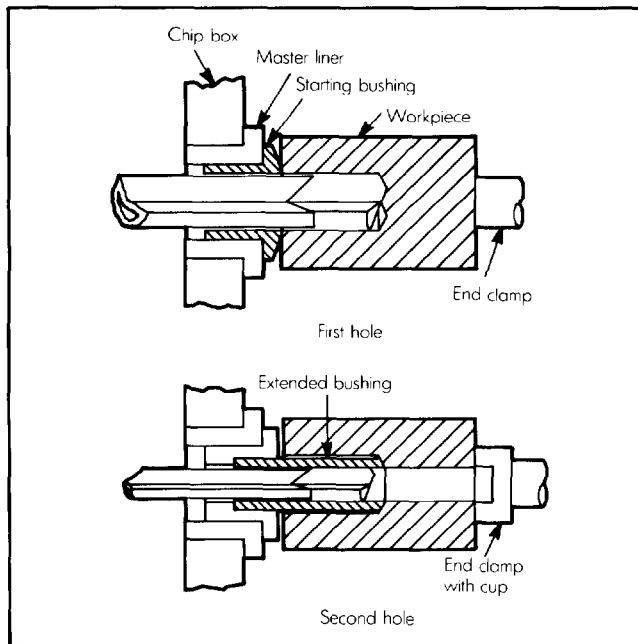


Fig. 9-55 Application of extended bushing to produce concentric holes in gundrilling.

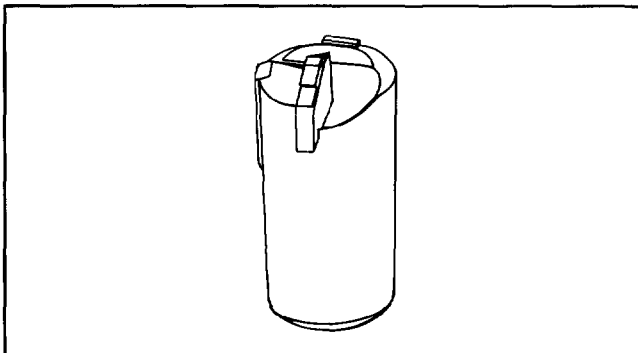


Fig. 9-56 Internal-chip-removal gundrill tip.

Depth of hole is limited only by the equipment and tools available; however, the asymmetrical shank of the external-type drill does not allow for high feed rates in hole depths exceeding 10 x drill diameter. This is because of the low torsional resistance and resulting windup of the shank.

Multiple cutting edges are used to provide higher penetration rates as compared to those possible with single-lip gundrills. Recommended feeds and speeds, as well as pressurized coolant requirements for both types of tools are discussed later in this chapter under "Operating Parameters." Initial cost and regrinding cost for external-chip-removal gundrills are about double that of single-lip, external-chip-removal gundrills. The cutting forces on the multiple lips tend to counteract (balance) each other, leaving only relatively small forces to be taken by the support pads. Cutting-edge and support-pad life, therefore, tend to be greater than that obtainable with single-lip gundrills.

Nose grinding must be precise for regrindable types of drills, and brazing assembly must be extremely accurate for throw-away type tools. Critical geometric errors can cause mismatch

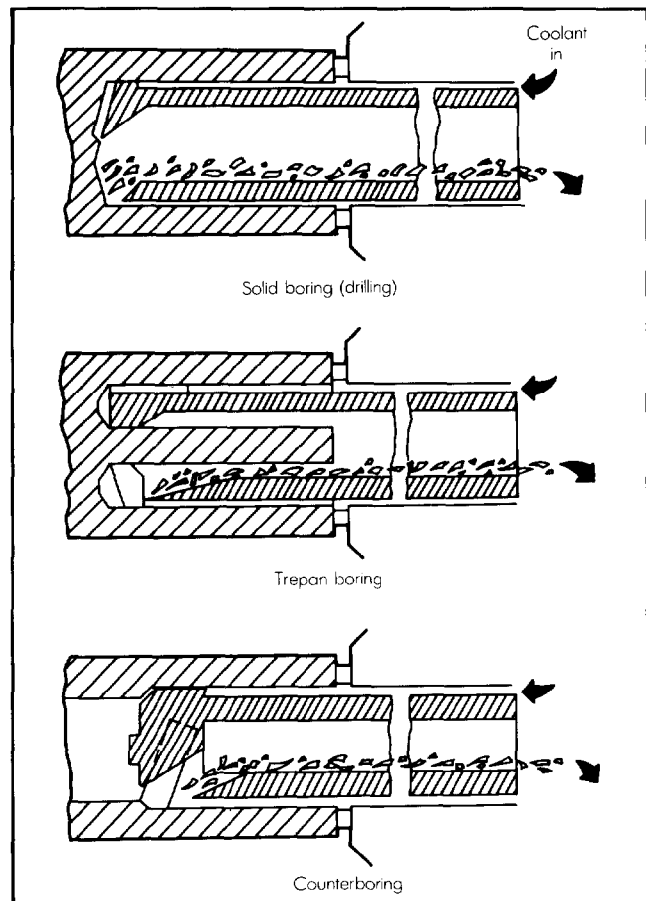


Fig. 9-57 Simplified diagram of pressure head and tool showing coolant and chip path.

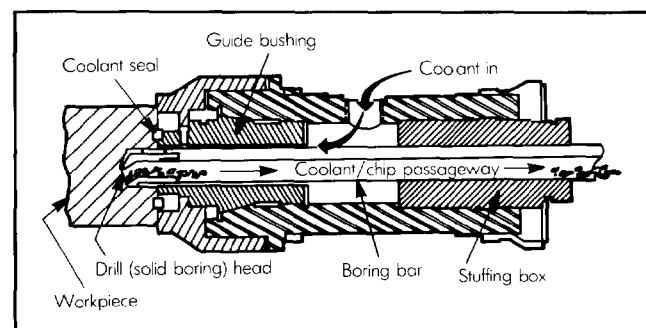


Fig. 9-58 Fluid transfer unit showing path of coolant.

of the cutting edges, resulting in poor hole quality and high tool mortality due to the imbalance of the cutting forces. Both internal and external-chip-removal drills have limited areas for chip removal. Therefore, these tools nearly always require chipbreakers and separators, along with precise control of nose geometry. Figure 9-61 illustrates features of the multiple-lip internal-chip-removal drill which affect efficiency and hole quality. Some exotic and extremely low-carbon steels cannot be successfully drilled with these tools because proper chip control cannot be attained even with special complex geometry.

CHAPTER 9

HIGH-PRESSURE COOLANT DRILLS

TABLE 9-12
Bushing Wear-Limit Ranges for
Internal-Chip-Removal Gundrills

Tool diam, in. (mm)	Wear-Limit Oversize, in. (mm)
3/4 to 2 (19-51)	0.0002-0.0004 (0.005-0.010)
2 to 8 (51-203)	0.0003-0.0006 (0.008-0.015)
8 (203) and larger	0.0004-0.0008 (0.010-0.020)

Multiple-lip tools are available using both HSS and carbide cutting edges and wear pads. A good starting point when using multiple-lip external-chip-removal drills with carbide tips is to use a speed two and one-half times that found best for HSS drills on the same operation, and about three-fourths of the feed per revolution. The reduction in feed and increase in speed, compared to HSS drills, will yield about twice the penetration rate, but it is not uncommon to be able to achieve penetration rates up to five times those of HSS drills.

External-chip-removal multiple-lip drills do not require starter drill bushings if the length-to-diameter ratio of the hole does not exceed 4:1. Length-to-diameter ratios greater than 6:1 require the use of a close-tolerance drill bushing or pilot hole in the workpiece to start the drill. Drill bushings are generally always used with multiple-lip internal-chip-removal drills.

For drilling holes larger than 2 1/2" (63.5 mm) multiple-lip internal-chip-removal drills have been developed with indexable inserts and reversible carbide support pads (see Fig. 9-62). Use

of indexable inserts and reversible pads can provide a substantial reduction in machining costs. They also allow for rapid insert replacement and a selection of a variety of carbide grades. The number of inserts in any drill is always uneven to create a resultant cutting force against the support pads. Chip control can be further enhanced through the use of mechanical chipbreakers and the ability to adapt each insert for a specific chipbreaking role.

Insert cartridges are replaceable and are positioned on opposite sides of the tool centerline to create balanced cutting forces. A feature of one type of these drills is support pads shaped as cylindrical segments and bearing against corresponding pockets located about 90° and 180° from the peripheral insert. The pads are mounted using a setscrew and coil spring, which allow the pads to move in the pocket under pressure and friction forces and thereby assure maximum contact against the wall of the hole being produced.

These drills are generally available in diameters of about 2 1/2" (63.5 mm) and are available in lengths suitable for short-hole and deep-hole drilling up to 100 x drill diameter. Tool diameters from 2 1/2 to 7 1/4" (63.5 to 184 mm) can be used with a double-tube coolant-feed and chip-exhaust arrangement as shown in Fig. 9-63. Tool diameters above 7 1/4" (184 mm) must be used with a single tube and oil pressure head (see Fig. 9-57). Coolant pressure, volume, horsepower, thrust, and torque requirements for these center-cutting tools are discussed later in this chapter. Rigid machines and workholding means are essential to withstand the higher overall power requirements. Guide bushings are required to start the tools into the work.

Trepanning Tools

Trepanning tools are self-guiding pressure-coolant tools

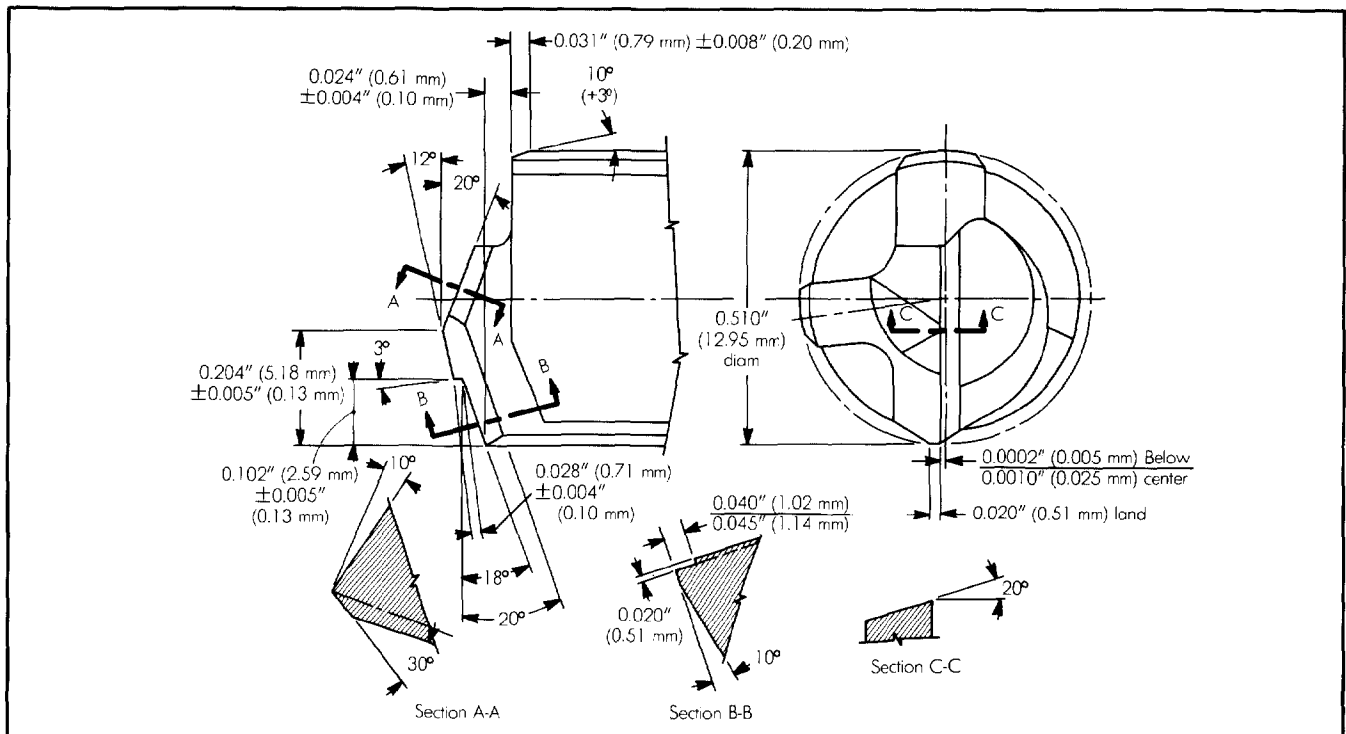


Fig. 9-59 Typical geometry for internal-chip-removal gundrill. (American Heller Corp.)

HIGH-PRESSURE COOLANT DRILLS

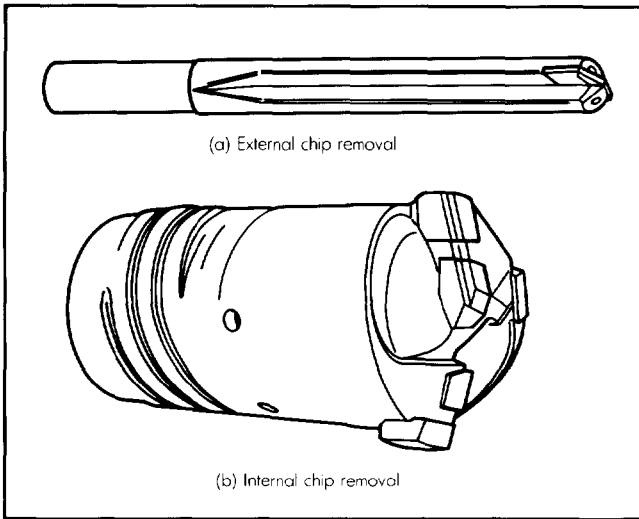


Fig. 9-60 (a) External chip-removal and (b) internal chip-removal, multiple-lip, pressure-coolant drills.

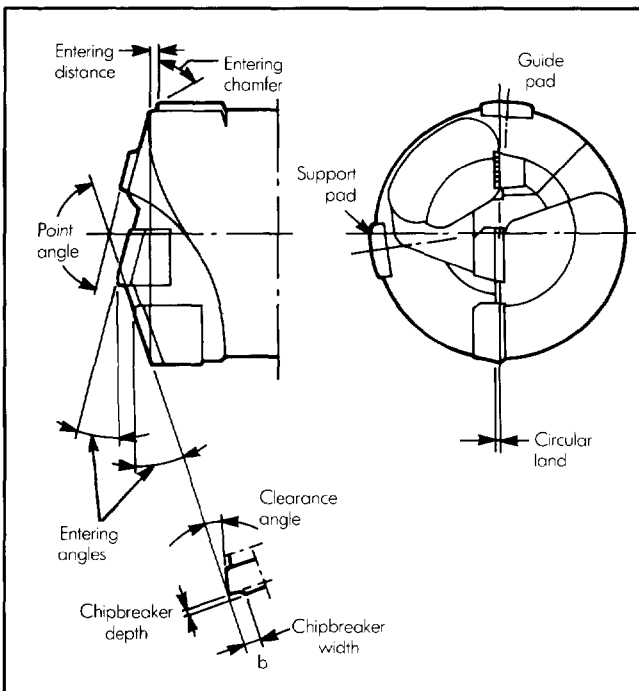


Fig. 9-61 Features of multiple-lip, internal-chip-removal drill that influence efficiency and hole quality. (Sandvik Inc.)

available in both internal and external-chip-removal types. They cut an annular groove leaving a central core (see Fig. 9-64). The most important reason for selecting trepanning tools over center-cutting drills is their reduced power requirements. The solid core is also more desirable than chips from the standpoint of salvage value, which can be extremely important for exotic and precious metals. The cores can be used for metallurgical analysis and test specimens, and in large diameters, can be sold or remachined as bar stock for other jobs.

A variety of designs are available in trepanning tools, including brazed and indexable single and multiple-insert

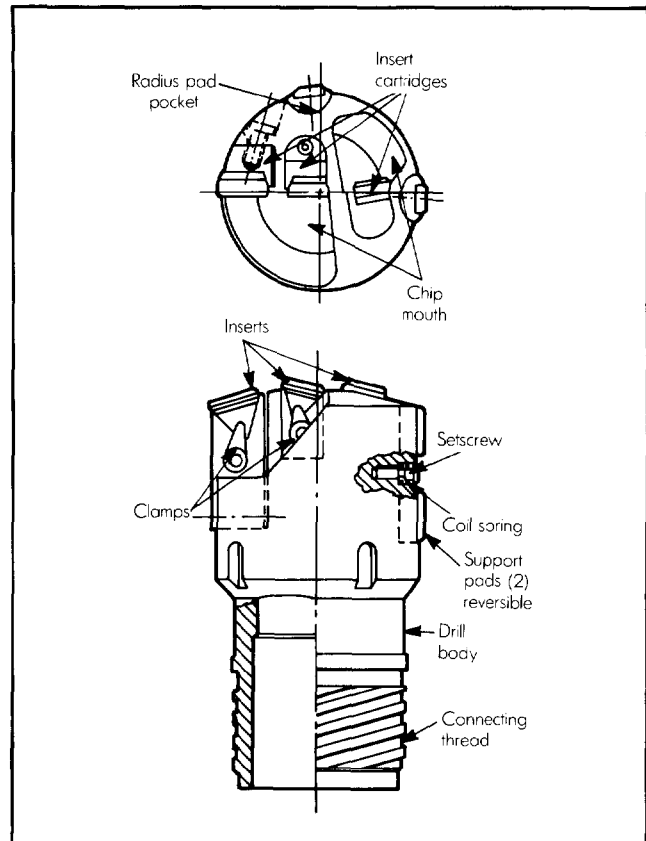


Fig. 9-62 Multiple-lip, pressure-coolant drill with indexable inserts for internal chip removal. (Sandvik Inc.)

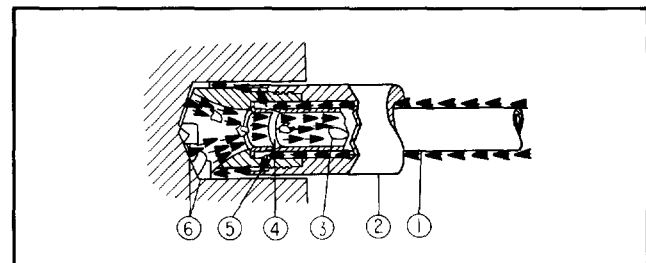


Fig. 9-63 Double-tube coolant-feeding and chip-exhaust drill: (1) inner tube; (2) outer tube; (3) chips and coolant exit, through inner tube; (4) annular nozzle; (5) oil passages to drill point; and (6) chip passages. (Sandvik Inc.)

styles. Single-insert designs require support pads and starting bushings, while some multiple-insert styles can be used without support pads and starting bushings.

Trepanning tools as small as 1 1/2" (38 mm) diam are available, although their most common usage is 2" (51 mm) diam and larger. Trepanning tools which do not have support pads, but depend upon balanced cutting forces provided by multiple inserts, are used for hole depths up to five diameters. Trepanning tools using support pads and started with bushings can be used successfully for depths up to 100 diameters. Diametral accuracy can be $\pm 0.005"$ (0.13 mm) for tools with support pads and $\pm 0.010"$ (0.25 mm) for tools without support pads, depending upon tool and machine setup and tool wear.

CHAPTER 9

HIGH-PRESSURE COOLANT DRILLS

Trepanning tools are generally applied to through holes because of problems with core removal. Special tools have been designed to cut off the core in large blind holes, but their success depends almost entirely upon operator skill. With few exceptions, they have not proven to be cost effective.

The trepanning tool is available with HSS, carbide, and coated carbide cutters. Recommended feeds and speeds, as well as coolant volume and pressure requirements, are discussed later in this chapter. Multiple-insert tools with guide pads (see Fig. 9-65), and an internal-chip-removal capability, can be used at the same starting surface velocity with up to twice the feed per revolution, resulting in a higher effective penetration rate.

Internal-chip-removal tools are preferred whenever the depth-to-diameter ratio exceeds 5:1 due to the resulting higher torsional resistance of the symmetrical, tubular shank. Internal chip removal necessitates the use of an oil pressure head such as the one shown in Fig. 9-66. Internal-chip-removal tools

generally produce smoother surface finishes than external-chip-removal trepanning tools. Both internal and external-chip-removal designs require close control of chip size so that chip evacuation is not interrupted.

Single-cutter trepanning heads commonly use a cutter $3/4"$ (19 mm) wide, which is separated into three cutting edges to produce three independent chips. Each cutting edge has a preformed or ground chipbreaker, as shown in Fig. 9-67. Throwaway-type cutters have generally been found to be more cost effective than regrindable types due to their rather complex cutting geometry.

Multiple-cutter indexable-insert trepanning tools (Fig. 9-68) offer several advantages over single cutter tools. The indexable inserts allow choosing from an extensive assortment of carbide grades with either preformed chipbreakers or mechanical chipbreakers for better chip control. Multiple-cutter designs create more balanced cutting forces than single-cutter designs and correspondingly higher metal removal rates with longer insert and support pad life. Typical insert cartridges are shown in Fig. 9-69. A cylindrically segmented carbide support pad fits into a mating pocket in the trepanning head. The support pad is fixed in the pocket with a setscrew and coil spring to allow the pad to pivot and make maximum contact against the hole wall under friction and cutting forces.

Some external-chip-removal trepanning tools are designed

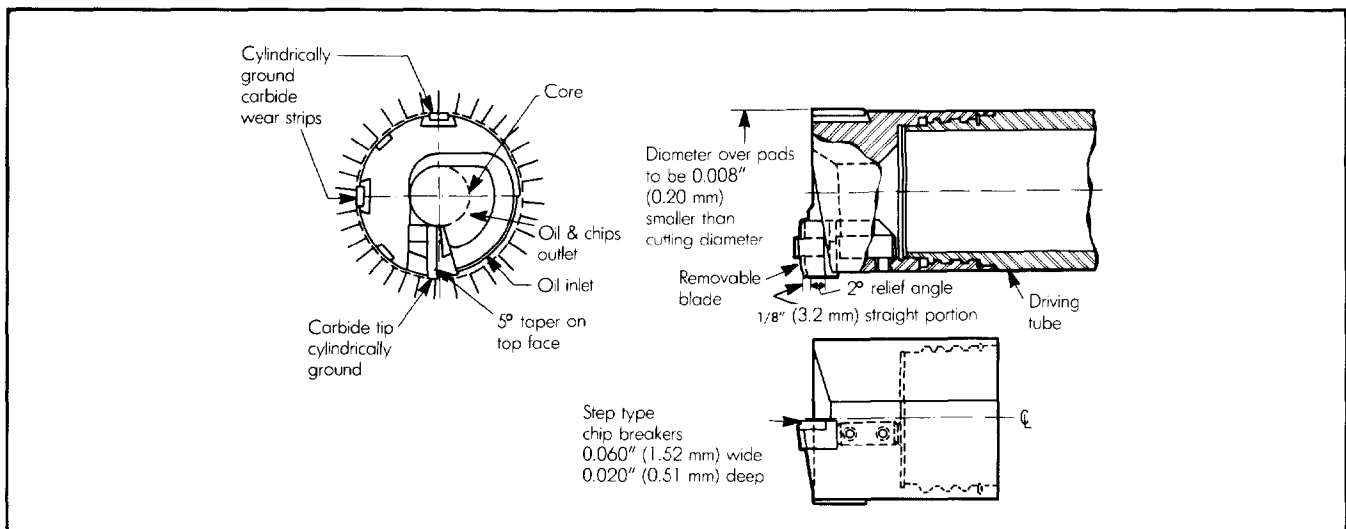


Fig. 9-64 Trepanning tool cuts an annular groove and leaves a central core.

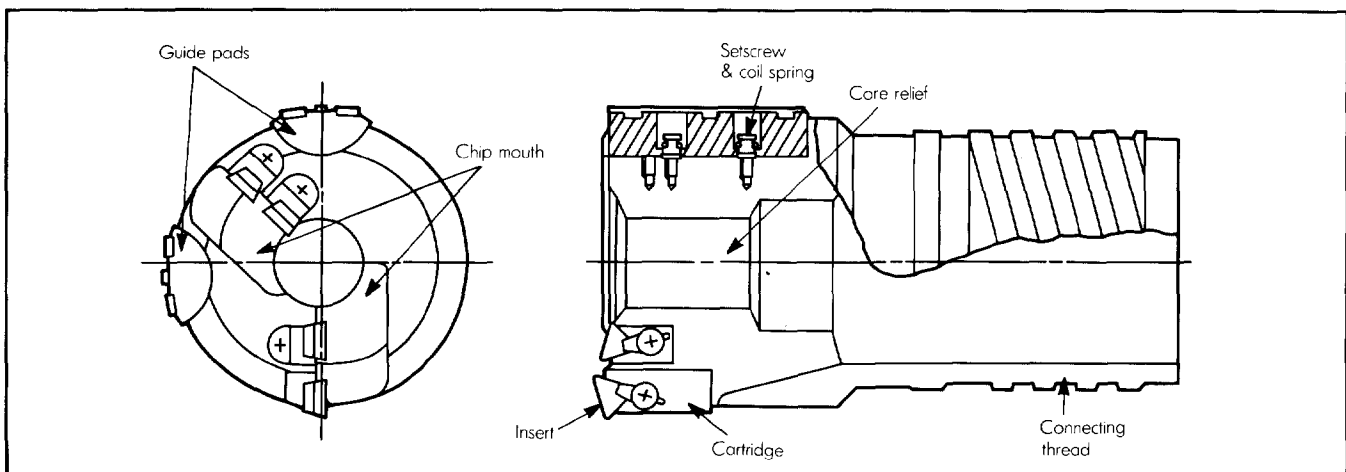


Fig. 9-65 Multiple-insert trepanning tool with guide pads for internal chip removal. (Sandvik Inc.)

HIGH-PRESSURE COOLANT DRILLS

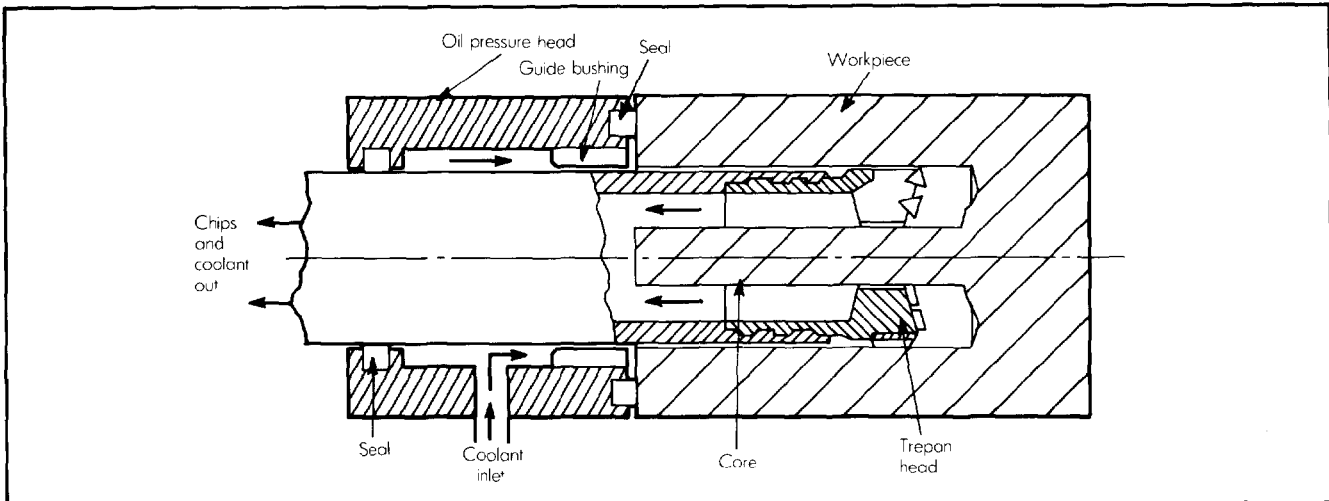


Fig. 9-66 Oil pressure head for internal-chip-removal trepanning tool. (Sandvik Inc.)

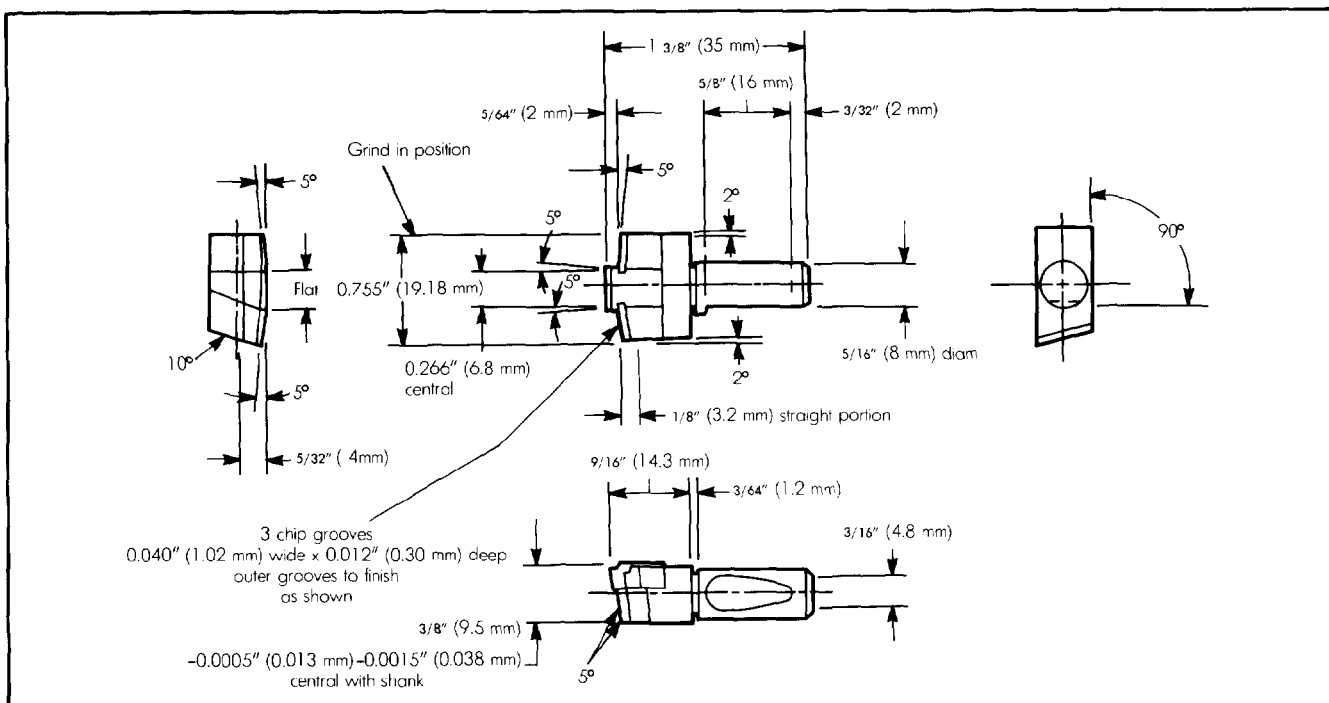


Fig. 9-67 Preformed or ground chipbreakers are provided on each cutting edge of trepanning tool.

for length-to-diameter ratios not exceeding 5:1. They utilize multiple indexable inserts positioned to provide balanced cutting forces. Therefore, they can be used without support pads and starting bushings. The obvious advantage of these tools is that they can be used on conventional, standard machine tools, as well as on advanced special-purpose and automated machines. The balanced cutting forces allow trepanning into irregular surfaces. These trepanning heads can be used as both stationary and rotating tools.

Coolant can be fed to the tool through an adapter (see Fig. 9-70) on rotating tool applications or through the rear of the shank of the tool in the case of stationary tools. Some machine

tools have been designed to feed coolant through their main spindle, so that rotary coolant adapters are unnecessary for rotating tool applications.

Core retention and core extraction devices have been designed for these types of trepanning tools to obtain higher production rates on numerically controlled and other types of automated machine tools.

Thrust forces, horsepower, and coolant volume and pressure, as discussed later, are all relatively low for multiple-cutter trepanning tools with balanced cutting forces. Penetration rates, however, are relatively high due to the constant width of cut that is taken for all sizes of trepanning heads of this type.

SMALL-HOLE DRILLS

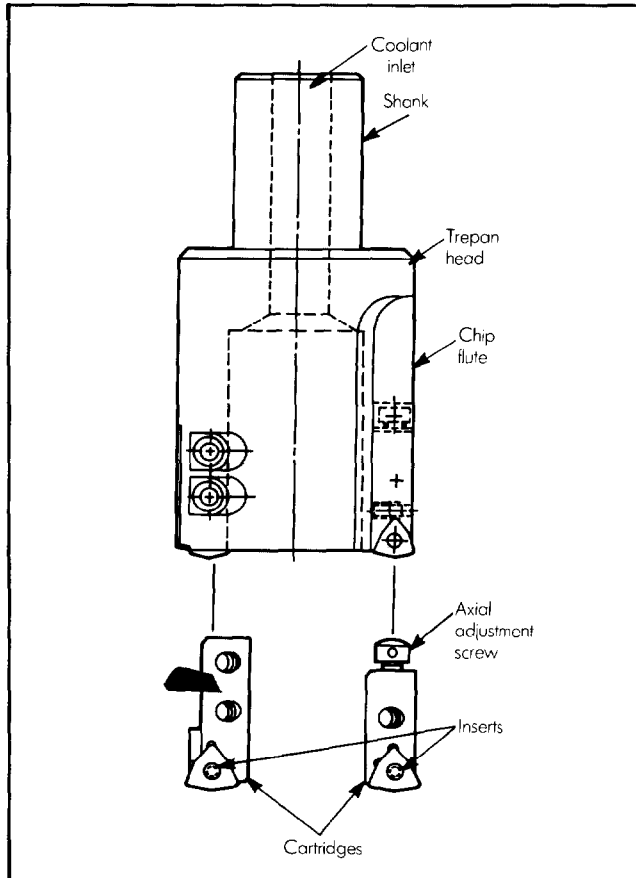


Fig. 9-68 Multiple-insert trepanning tool with external chip removal. (Sandvik Inc.)

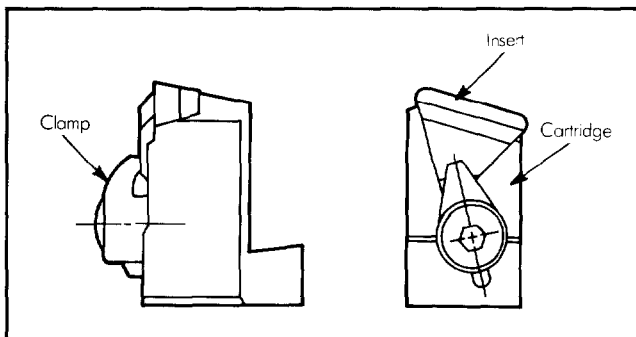


Fig. 9-69 Typical cartridge for indexable-insert-type trepanning tool. (Sandvik Inc.)

SMALL-HOLE DRILLS

A major problem in drilling small holes mechanically (with the production of chips), as compared to nontraditional methods, is compensating for the decreased rigidity of the delicate tools required. Drill rigidity is roughly inversely proportional to the fourth power of the diameter and the force required to deflect the drill is approximately proportional to the cube of the unsupported drill length. Other problems include chip removal, minimizing runout, impact of the drill with the workpiece, and maintaining a uniform feed rate. Also,

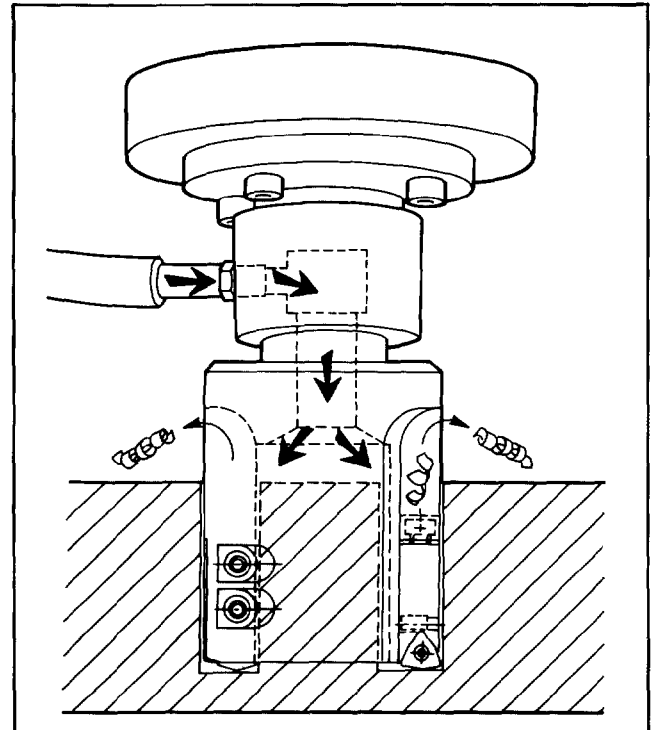


Fig. 9-70 Rotating coolant adapter for external-chip-removal trepanning tool. (Sandvik Inc.)

machinability of the material does not improve with decreasing hole sizes. In fact, the material can be less homogeneous and machinable in the area being drilled because of unalloyed portions, foreign inclusions, work hardening, or other reasons.

Drills Used for Small Holes

The three types of drills used predominately for producing small holes are center, pivot, and straight-shank drills. Others also used include step drills, drills with long shanks, and drills designed for special applications (see Fig. 9-71).

Center drills are the most rigid in common use because of their short flute lengths (usually about four times the diameter) and oversize shank diameters. They should be used whenever the flute length is adequate or when holes are to be started or centered. When used as a hole starter, the diameter of a center drill should be at least as large as the web thickness of the following drill for guidance and minimum deflection.

Pivot drills are probably the most widely used type for producing small holes. They normally have flute lengths from five to seven times their diameters and shanks larger in diameter than their fluted portions. They are available with spiral flutes having 10°, 20°, or 30° helix angles; in flat (spade) versions; in D-flute (half round) styles; and for both left and right-hand cutting.

Straight-shank drills, with flute lengths of about 12 diameters are available in three flute styles: a spiral flute style with helix angles of 10°, 20°, or 30°; a flat or spade flute, two-lip style with a zero helix angle; and a D-flute or half-round style. Since these tools are less rigid than center or pivot drills, they are used primarily to enlarge and increase the depths of holes started with other types of drills. They should generally not be used to start holes.

SMALL-HOLE DRILLS

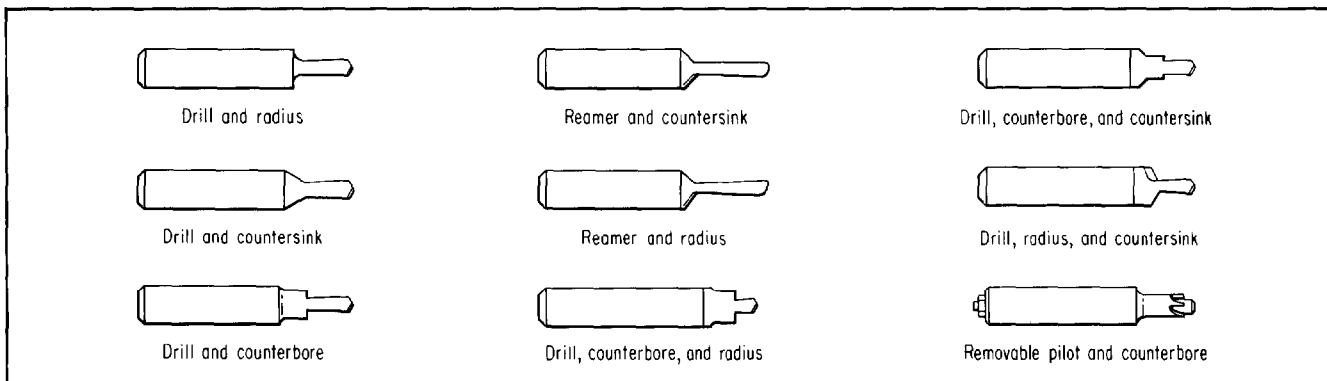


Fig. 9-71 Various tools used to produce small holes.

The webs or center sections of small-diameter drills are proportionately thicker than those of larger diameter drills because it is not practical to manufacture them with the same percentage of web. These thicker webs increase the rigidity of the drills, but they also decrease the length of the cutting lips. Also, thicker webs at the points increase the end pressures required to force the drills into the workpieces.

Materials for Small-Hole Drills

Small drills are available made from carbon and high-speed steels, from solid carbide, and with carbide or diamond inserts. Diamond drills are sometimes preferable for drilling ceramics and refractories. The use of carbide improves the rigidity, and there has been a substantial growth in the use of these drills for small-hole drilling, particularly for printed circuit boards.

Small drills of bimetal construction are available from Von Hollen Tool Co. These tools have a shank extruded from a low-modulus brass, which serves as a torsion bar to absorb shock and torque and as a heat sink and floating holder. An M2 high-speed steel cutting section provides hardness and wear resistance. The fluted HSS portion is adhesively bonded into a hole in the end of the cold-drawn brass shank. Back-tapered flutes minimize heat and wear caused by friction and provide more chip clearance. These drills are available from 0.0075-0.0587" (0.190-1.491 mm) diam, in 0.0004" (0.010 mm) increments, with a size tolerance of ± 0.0002 " (0.005 mm) and flute lengths of seven times the diameters. All shanks are 3/32" (2.4 mm) diameter by 1 1/2" (38 mm) long.

High-speed steel pivot drills are available from one supplier in 0.01 mm increments from 0.05-3 mm, with flute lengths of 0.40-20 mm and shank diameters from 1-3 mm. High-speed steel flat drills are available from 0.04-1.0 mm diam with flute lengths from 0.30-0.70 mm and with 1 mm diameter shanks.

Two-flute pivot drills are available from another firm with an 8° rake angle and 135° point angle for steel and general-purpose drilling, and 15° rake angle and 118° point angle for soft materials. Standard cutting lengths are seven times the diameter, but lengths up to 20 times the diameter are made to special order. While drills with a diameter of 0.0001" (0.003 mm), flute length of 0.0003" (0.008 mm), and shank diameter of 0.040" (1.02 mm) are the smallest standard size, this firm has drilled holes 0.000025" (0.00064 mm) diam.

Drills made by this firm are produced from both tool steels and sintered tungsten carbide. Various grades of tool steel are used for different applications. The most common are AISI Types M33 and 07, with enhanced micrograin structure

treated to a hardness of R_{C65-68} . Carbide drills permit higher stock removal rates and remain sharper longer, but they have lower transverse strength. They are recommended for drilling harder materials, softer grades of ceramic, and production applications.

TOOLHOLDERS FOR DRILLING

Many different devices are used to hold drills and provide the driving connection between the machine spindle and the drill. These include sockets, sleeves, chucks, collets, and other arrangements. Selection of the proper device is critical because drills not properly gripped can be pulled from the driver by friction between the drill and workpiece when the drill is retracted.

Taper-Shank Drills

Drills with taper shanks fit directly into the spindles of drilling machines that have the same size taper as the drills. When the hole in the spindle is larger, a reducing socket or sleeve is used. Short sleeves are preferable to fitted sockets because of the increased rigidity they provide. Drifts are used to remove taper-shank drills from the sockets or sleeves.

Variations of the taper shank are sometimes used for heavy-duty operations. The most common of these is the flatted shank. The socket has a flat in its bore to match that on the drill shank, thus providing a positive drive. For larger diameter drills, a pair of flats is sometimes used. Other special variations employed occasionally include half tangs, squares, keyseats, keyways, grooves, notches, slots, threads, and knurls.

Straight-Shank Drills

Drills with straight shanks are held in several ways, including chucks, sleeves, bushings, and collet-type holders.

Chucks. Three-jaw chucks are the most common holders and drivers for small-diameter, straight-shank drills. They are available in wrench, or key-tightening, and wrenchless (keyless) styles (see Fig. 9-72). These chucks are made with plain or ball bearing construction, and for taper or threaded mounting.

Each chuck holds a range of drill sizes within its capacity. Typical ranges for various chucks are 0 to 1/4", 0 to 3/8", and 0 to 1/2" (6.4, 10, and 13 mm); 1/8 to 5/8" or 1/8 to 3/4" (3 to 16 or 19 mm); and 3/8 to 1" (10 to 25 mm). Small-size keyless chucks have a micrometer graduation on their sleeves to facilitate changing drill sizes. These chucks depend upon friction between the jaws and drill shank for gripping. Care is

CHAPTER 9

TOOLHOLDERS FOR DRILLING

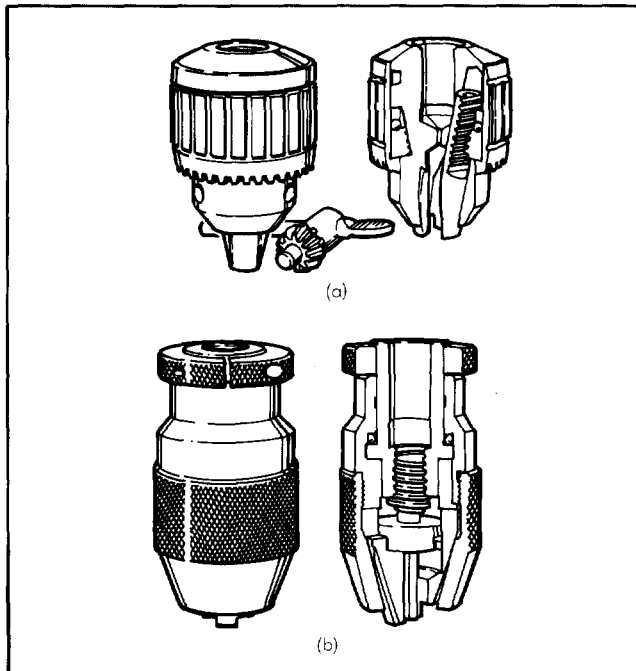


Fig. 9-72 Drill chucks of the (a) key-tightening type and (b) keyless style. (Jacobs Manufacturing Co.)

necessary to assure sufficient tightening to prevent drill slippage and excessive wear.

Sleeves and bushings. Tapered split sleeves are sometimes used to drive small-diameter, straight-shank drills for certain production applications. These sleeves have a taper on their OD to fit the machine spindle or socket and a straight bore to fit the drill shank. Flats or squares on the drill shanks and driving slots in the sleeves are used occasionally, particularly on multispindle machines with close centers.

A disadvantage of these drivers is that a separate sleeve is required for each size drill. Specifications for split sleeve, collet-type drivers for straight-shank drills, reamers, and similar tools are presented in ANSI Standard B94.35-1972, published by ASME.

Straight sleeves are sometimes used to hold larger diameter, straight-shank drills in machines such as turret lathes. Setscrews extending through the sleeves are generally used to hold the drills in place. These devices also have the disadvantage of requiring a different size sleeve for each drill size. The sleeves must fit the drill shanks closely to avoid misalignment when clamping, and the setscrews used often mar the drill shanks.

Collet-type holders. A major advantage of collet-type holders is their capability of gripping drills on their margins or straight shanks. In doing this, the length of the drill projecting from the collet can be varied and kept to a minimum to suit requirements for short-hole, deep-hole, or other drilling applications. Called stubbing, this method increases rigidity by decreasing the length-to-diameter ratios. Resulting benefits can include longer drill life, the capability of using higher feed rates and cutting speeds, reduction in the need for jigs and bushings, possible elimination of center drilling or spotfacing, shorter cycle times, and the ability to use broken drills.

Collets made by Erickson Tool Co., available in both single and double angle designs, are alternately slotted from opposite

ends. Both styles maintain concentricity of 0.0005" (0.013 mm) TIR, measured at the collet face, when assembled in the chuck. With a double angle collet mounted in a chuck (see Fig. 9-73), the collet can flex both transversely and axially to grip the drill margins as well as the back-taper of the drill. A 1/32" (0.8 mm) collapse permits one collet to replace six or seven single-purpose split sleeves. A left-hand backup screw (shown in Fig. 9-73) permits positive tool presetting. Chucks for these collets are available in stub-nose, extended-nose, and close-center types, as well as types suitable for faceplate mounting. Collet-type, full-floating holders are also available. Single angle collets produce high gripping forces to better utilize the power available with modern machines. A slow taper in the body of the chuck, which matches the collet, produces a wedge action as the collet is forced back into the chuck.

Rubber-Flex collets made by the Jacobs Manufacturing Co. consist of hardened steel inserts molded into a synthetic rubber compound. Flexibility of the rubber matrix gives each collet a gripping range of 1/8" (3.2 mm); 11 collets can handle drills from 1/16 to 1 3/8" (1.6 to 35 mm). When mounted on a true spindle, runout measured at the nose of the chuck does not exceed 0.0007" (0.018 mm).

Quick-Change Tooling

Many types of quick-change tooling devices are available to permit changing tools in a few seconds. With the Inmotion system made by Diamond Tool and Die, Inc., the need for auxiliary tools is eliminated and spindle rotation does not have to be stopped. The operator grasps a knurled outer sleeve with one hand and slides it upward to instantly release the tool. When a new tool is inserted and the sleeve is pulled down, the tool is firmly gripped by a toggle-type system with a self-locking taper.

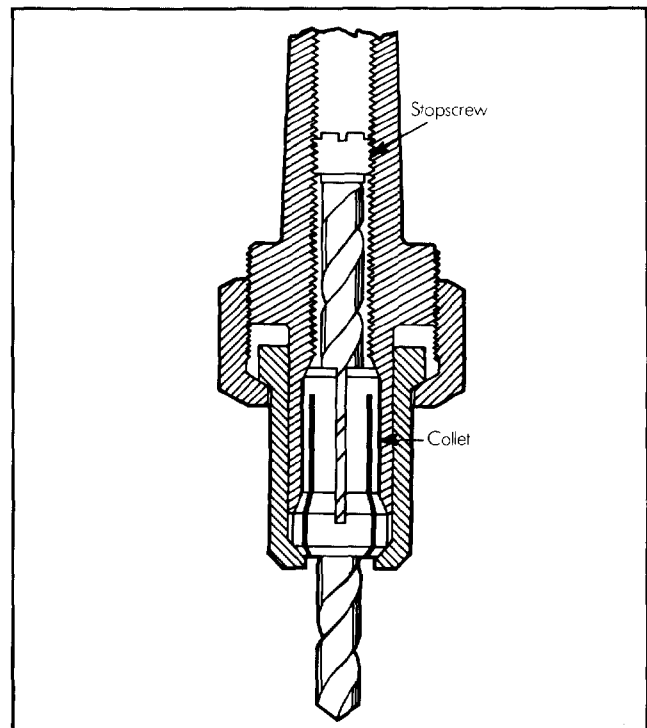


Fig. 9-73 Collet, alternately slotted from opposite ends, which grips drill on its margins. (Erickson Div., Kennametal Inc.)

WORKHOLDING DEVICES FOR DRILLING

Boeing Co. has developed a quick-change chuck system to reduce the time required to change drills in hand-held drill motors, the term commonly used by the aircraft industry for portable power tools. Use of the chucks for pistol grip and angle motors has resulted in an estimated cost savings of over \$500,000 per year at Boeing.

The pistol grip quick-change chuck (see Fig. 9-74) is a direct replacement for the conventional 1/4" (6.4 mm) three-jaw chuck. It is operated by extending a spring-loaded sleeve and inserting a drill with an integral quick-change adapter. The chuck contains three retaining balls which engage in the detent holes of the drill adapter when the sleeve is released. These three balls provide the torque driving mechanism and serve to retain the drill in the chuck. The quick-change adapter is joined to the drill shank by either silver brazing or interference fit.

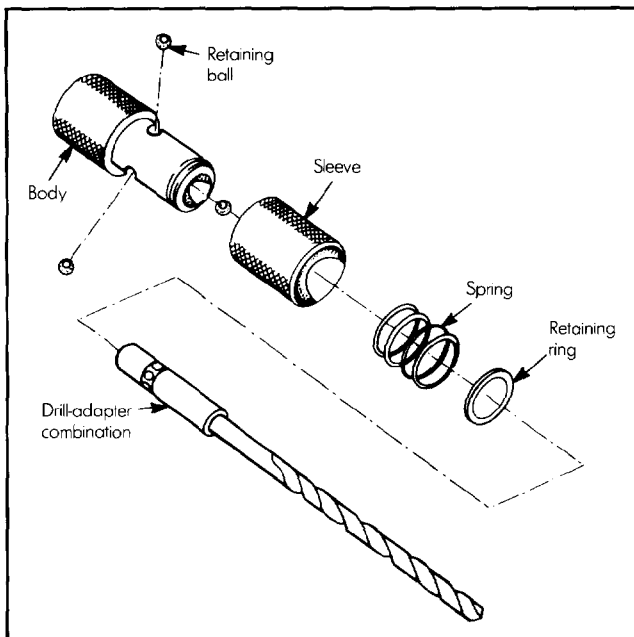


Fig. 9-74 Quick-change chuck for pistol grip drill motors.

Drills can be changed 7 seconds faster with the quick-change chuck than with a conventional key-type chuck. Other advantages include lighter weight, lower cost, the ability to drill in closer quarters, and elimination of damage to drills and workpieces.

The quick-change chuck system for angle drill motors eliminates the need for threaded-shank drills used previously. The operating principle of the angle motor chuck (see Fig. 9-75) is similar to the pistol grip chuck except that the chuck collar is rotated, rather than extended, to open the chuck. The ball detent is identical to the pistol grip version.

This system eliminates the need for wrenches required for threaded-shank drills and, consequently, reduces adapter damage and tool distortion. The use of the angle motor, quick-change chuck saves over 8 seconds per drill change. Lower cost and extended drill life are additional advantages.

WORKHOLDING DEVICES FOR DRILLING

Jigs and fixtures are precision devices used to accurately machine duplicate parts. As a general rule, jigs and fixtures are designed to hold, support, and locate a workpiece while

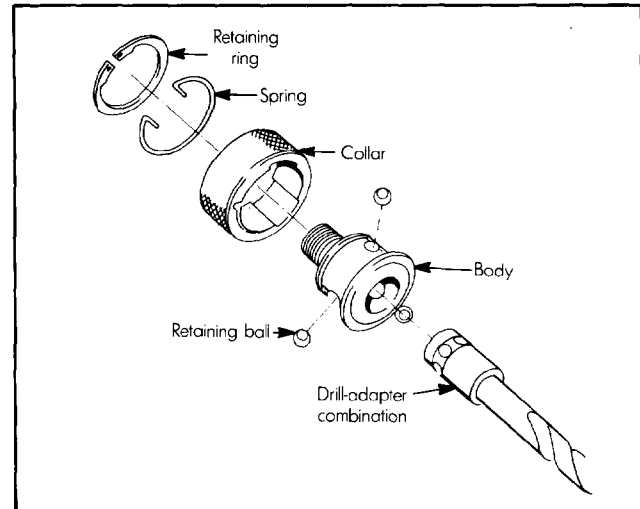


Fig. 9-75 Quick-change chuck for angle drill motors.

controlling the position and alignment of the cutting tools. Good jig and fixture design can minimize tool breakage and the production of unsatisfactory workpieces.

The principle difference between a jig and fixture is the method used to position the cutting tool. Jigs securely hold and locate the workpiece while guiding the cutting tool by means of a bushing or similar device. Fixtures securely hold and locate the workpiece while referencing the position of the cutting tool by means of set blocks and feeler gages.

Although both jigs and fixtures are used for a wide variety of machining operations, the jig is by far the most common type of device for drilling, reaming, and related operations. Regardless of the machining operation, the basic construction and function of a jig are the same for drilling and reaming.

Jigs can be divided into two broad categories: open and closed. Open jigs are generally used for machining parts on a single surface, while closed jigs are used for parts that require machining on two or more surfaces. The terms used to identify these jigs are normally directly related to the basic construction of the tool itself. Typical examples of open and closed jigs are shown in Fig. 9-76.

The specific size, shape, and construction details of any jig are normally determined by the workpiece to be machined. To perform their basic function, however, all jigs must adhere to certain design principles and must include the following elements:

1. Locating devices and work supports.
2. Clamping devices.
3. Tool guidance devices.

Locating and Supporting Principles

To ensure the desired precision in any machining operation, the workpiece must be properly located and supported throughout the entire machining cycle. The terms locating and supporting, as used here, refer to the establishment of the proper relationship between the workpiece and the jig. In most cases, locators and supports serve the same purpose: to establish the desired reference location. Supports are simple locators which are positioned beneath the workpiece to establish the reference position on the vertical axis. Locators, likewise, set the position of the workpiece on both horizontal axes.

WORKHOLDING DEVICES FOR DRILLING

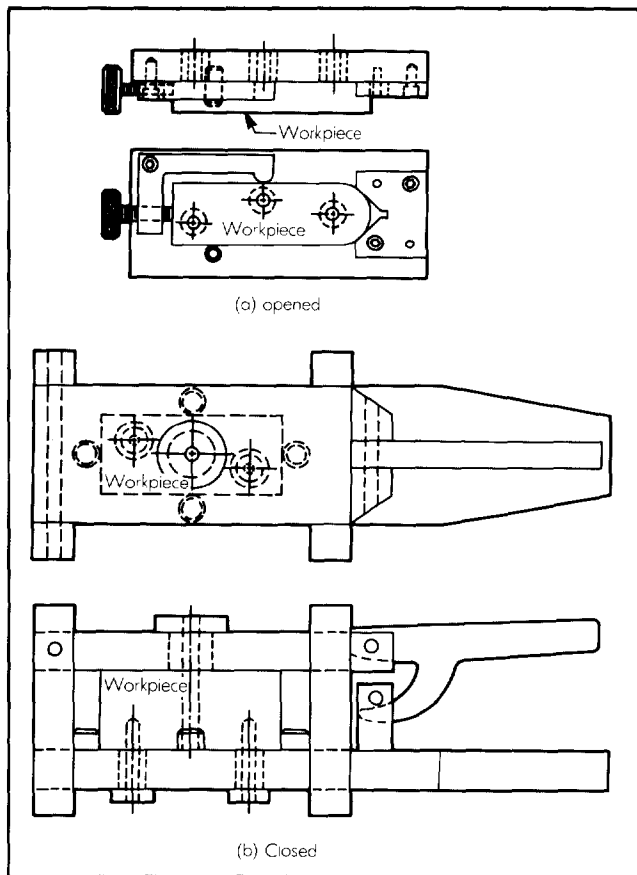


Fig. 9-76 (a) Opened and (b) closed types of jigs.²

Any unrestricted object is free to move in any of 12 directions (see Fig. 9-77). These movements are called degrees of freedom. To completely locate any workpiece, these degrees of freedom must be restricted. This is normally done with a combination of locators and clamps.

The most basic form of workpiece location is the six-point or three-two-one method. In this method, the vertical Z-axis position is established by locating the workpiece on three points (Fig. 9-78, a). The horizontal axes are then located by three more points, two on one plane (Fig. 9-78, b) and one on the other plane (Fig. 9-78, c). This method restricts nine of the twelve degrees of freedom. The remaining three degrees are restricted by the clamping device.

When any locating system is designed for a jig, the following points must be considered to prevent inaccuracies and to assure proper location:

1. All locators should contact the workpiece on a machined surface or designated locating area.
2. The reference surfaces, when not specifically indicated on the print, should be those which are used to dimension the part.
3. Locators should always be positioned as far apart as practical to ensure stability and accuracy.
4. Locators should be positioned to avoid chips and foreign matter. When this is not possible the locators should be relieved to prevent interference when the workpiece is loaded or unloaded (see Fig. 9-79).

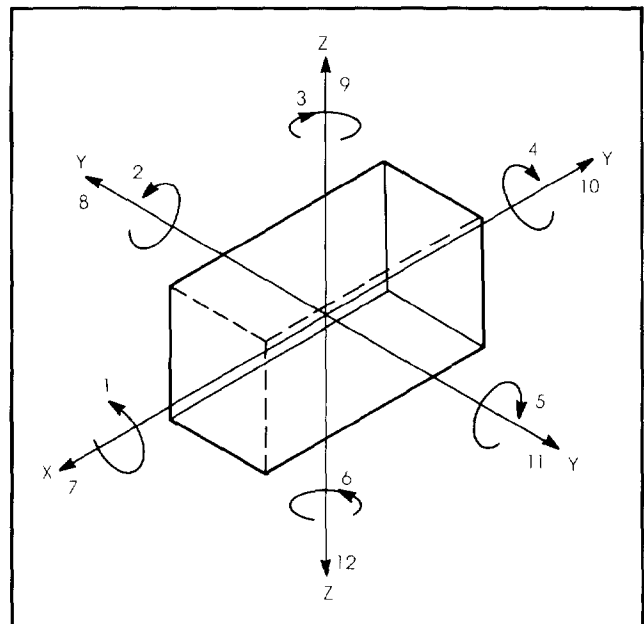


Fig. 9-77 Unrestricted workpieces are free to move in any of 12 directions.²

5. Duplicate locators should always be avoided. Once the reference surface is determined, only that surface should be used to locate the workpiece. Using more than one reference surface to establish the location of the workpiece often results in excessive tolerance stack-up.
6. The location of any workpiece should always be foolproof; that is, the workpiece should only be capable of being loaded into the jig in the correct position. Using a foolproofing pin is one method to prevent improper loading.
7. Locational tolerance should be as liberal as possible. A range of 20-30% of the workpiece tolerance is normally acceptable, and to 50% in some cases. Overly tight tolerance requirements only serve to increase the cost in a disproportionate relationship to the part accuracy.
8. Work supports should always be positioned beneath the area at which the workpiece is to be clamped. Clamping between supports or over an unsupported area can distort or bend the workpiece and may change the location.

Locating Methods

The exact methods selected to locate a part must be determined by several factors. Part size, shape, reference surfaces, and prior machining are all important elements in deciding the methods and locators to be used to position a part.

Locating from a flat surface. There are three types of supports commonly used to locate a flat surface: solid supports, adjustable supports, and equalizing supports (see Fig. 9-80). Solid supports are the least expensive and easiest to use. These locators are normally used when the reference point or datum is a machined surface. Adjustable and equalizing supports are used on rough or uneven surfaces such as a casting. In most cases, adjustable and equalizing supports are used in conjunction with solid supports to compensate for part irregularities. The principle differences between adjustable and

WORKHOLDING DEVICES FOR DRILLING

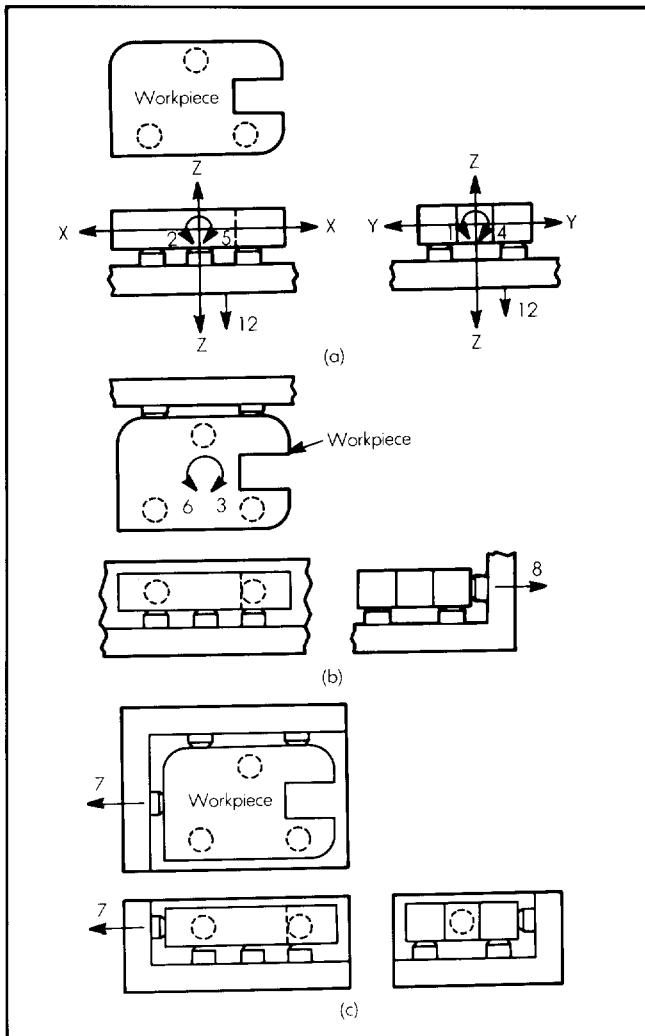


Fig. 9-78 Six-point method of location: (a) 2-axis, (b) horizontal Y-axis, and (c) horizontal X-axis.²

equalizing supports is the number of contact points and the method of adjusting. Adjustable supports normally have a single contact that is manually adjusted, while equalizing supports have two contacts that are adjusted automatically to compensate for part irregularities.

Locating from an internal diameter. Locating from a hole or hole pattern is generally the most efficient and accurate way to locate any workpiece. Several types of locators are specifically designed for this type of part positioning.

When large holes or bores are used, the locator is normally fastened by either screws and dowels or by a mounting shank. Shank-mounted locators are made with either a press fit or threaded shanks. In most cases, the press-fit locators should be used rather than the threaded type. Threaded shank locators are useful in areas in which, because of the construction of the jig, the press-fit type cannot be easily removed.

Pin-type locators and bushings (Fig. 9-81) are used for locating small holes and for aligning tool members. When pin locators are used to align tool members, special alignment bushings are also used. These permit both parts to be replaced when they become worn.

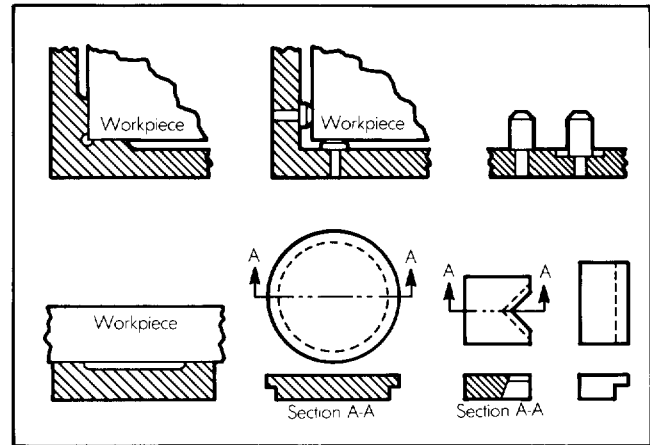


Fig. 9-79 Methods of relieving locators to prevent interference.²

When pin locators are used to directly locate a workpiece, the area of contact is very important. Too much contact causes a part to jam, while too little results in inaccurate location. A range of one-fourth to one-half of the part thickness is generally sufficient to prevent jamming and to ensure proper location.

Another style of locating pin commonly used for jigs is the diamond or relieved pin. These pins are generally used with round locators to prevent radial movement of the workpiece about the round locator. Diamond pins may also be used in pairs to locate a workpiece. Regardless of their configuration, relieved locators all serve the same purpose—to prevent jamming and to speed the loading/unloading of parts.

Locating from an external profile. External profile location is the most common method used to locate workpieces. With this method, the workpiece is located with reference to its outside edges. As with other locators, there are several forms of external profile locators.

The most common type of external profile locator is the fixed-stop locator. These locators are made in many forms and can either be machined into the tool body or installed. One of the more popular types of installed fixed-stop locators is the rest button. The principle advantage of installed locators is their ease of replacement. Unlike machined locators that must be built-up and remachined to correct for wear, installed locators only need to be pressed out and a new locator pressed in when they begin to wear.

Another variation of the fixed-stop locator is the adjustable locator (see Fig. 9-82). An advantage of this type of locator over the fixed-stop type is that since it is adjustable, the position of the locator does not have to be as closely controlled. Any variation in position can be compensated for by the adjustment.

In cases in which the relationship between the hole pattern and the outside edges of the part are not as critical as they are with some castings, sight locators can be used. These locators align the workpiece in an approximate position in the jig. The principle forms of sight locators use scribed lines or milled slots (see Fig. 9-83). A disadvantage of sight locators is that maintaining alignment is difficult when clamping.

The most complete method of locating a workpiece is by nesting (see Fig. 9-84). Nesting locators locate a workpiece by enclosing it in a recess or depression of the exact shape of the part. Since the nest must conform to the exact shape of the workpiece, nesting is also the most expensive method of locating an external profile. In those cases in which the

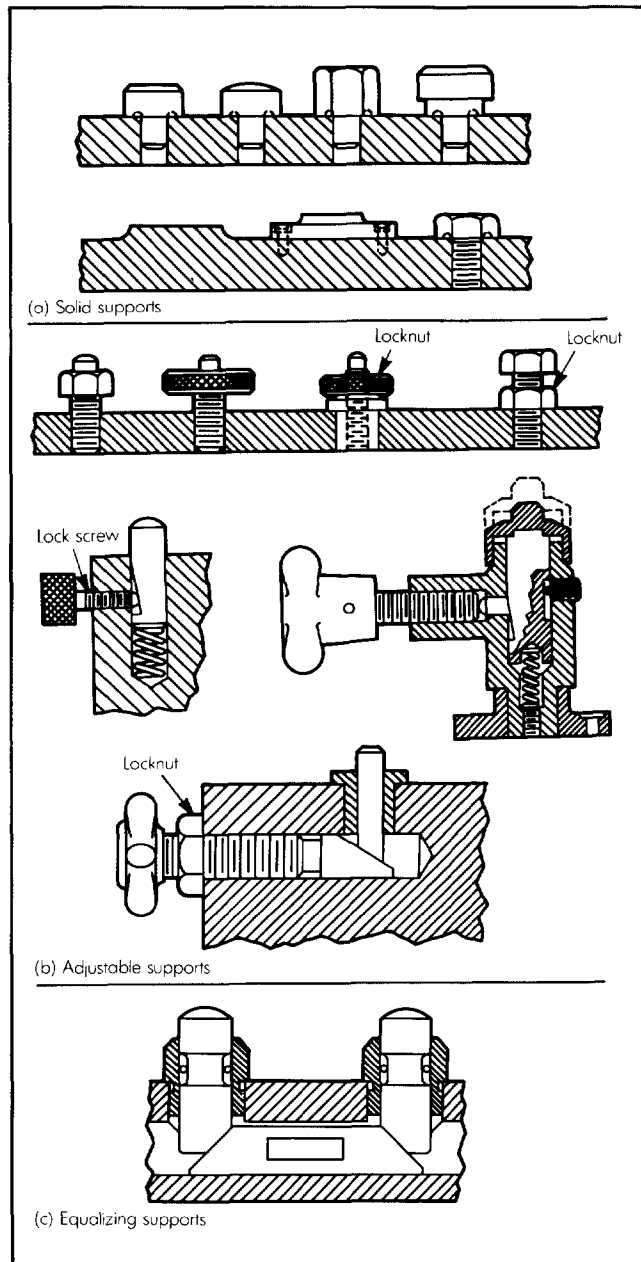


Fig. 9-80 Three types of supports for locating flat surfaces: (a) solid, (b) adjustable, and (c) equalizing.²

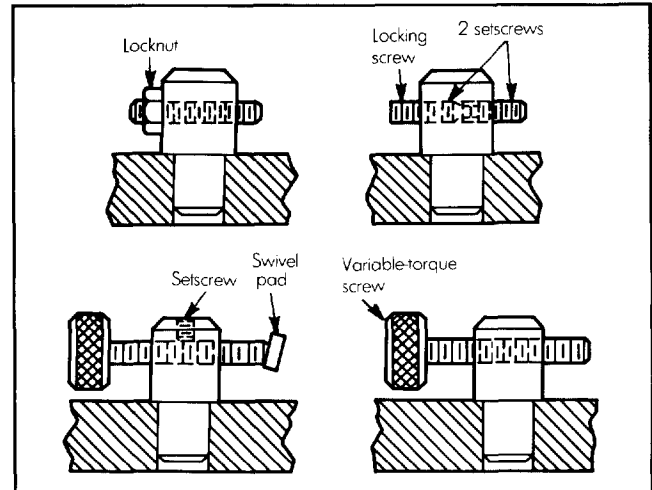


Fig. 9-82 Various types of adjustable locators used on jigs.²

workpiece form cannot be readily machined into the jig to form the nest, cast materials of epoxy or hard urethane resin or low-melt alloys are used to form the nest. Another variation of the nest-type locator is the partial nest. This locator has most of the advantages of the full nest; however, since the area of contact with the workpiece is reduced, the workpiece is easier to load and unload. In either case, when a nest-type locator is used, the outside form of the workpiece must be relatively constant. These locators are normally used on parts that have been machined to their correct outside dimensions.

Burr Clearance and Part Ejection

Any drilling operation produces burrs. To properly perform its design function, a jig must allow clearance for these burrs to prevent the part from jamming in the jig. Two burrs are formed in a drilling operation—the primary burr on the exit side and the secondary burr on the entry side. The secondary burr is normally taken care of by the bushing clearance. The primary burr, however, can sometimes cause problems. In cases in which the workpiece must slide on or off a locator, or in and out of the jig, there must be a burr clearance groove. A burr clearance groove is simply a milled groove slightly deeper than the height of the burr in the area of the jig in which the burr could jam. Burr interference must always be considered, and clearance planned into the jig design.

Part ejection is also an area which must be considered in the design of a jig. Several very simple methods are commonly used to eject parts. While elaborate power ejection methods can be used, a simple lever or spring ejector often performs as well and

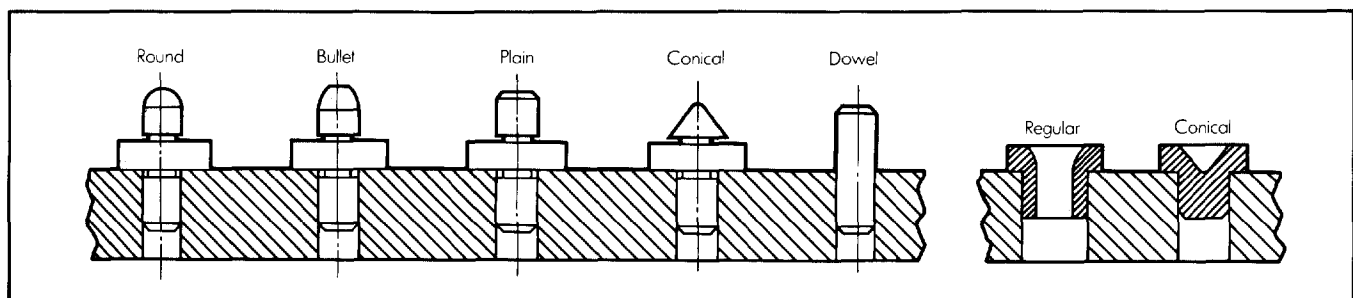


Fig. 9-81 Pin-type locators and bushings for small holes and tool alignments.²

WORKHOLDING DEVICES FOR DRILLING

costs a fraction of the power system. When designing an ejection system, the only thought should be the removal of the workpiece from the jig. Overly elaborate or complicated ejection systems often cost more and are more likely to malfunction than the simple designs.

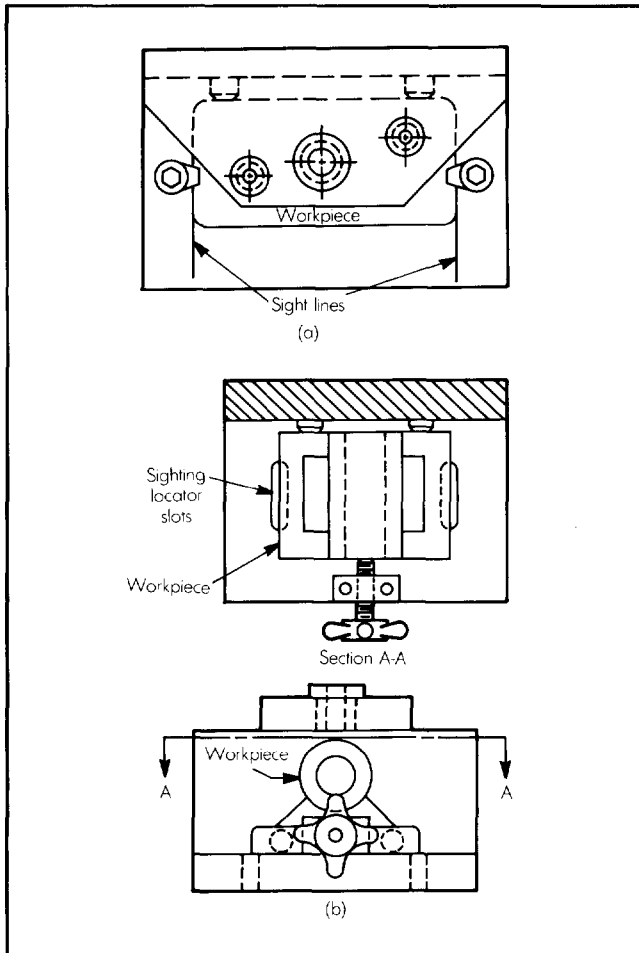


Fig. 9-83 Two types of sight locators: (a) with scribed lines, and (b) with milled slots.²

Workholding and Clamping Principles

Workholding and clamping devices serve only one purpose in any jig—to hold the workpiece securely against the locators and prevent it from being pulled from the jig by the cutting tool during the machining operation. Many times, the clamps that are selected for a particular jig are actually larger than required. To prevent this, and to help select the proper size and type of clamp, an understanding of the tool forces is important.

Forces are generated by the cutting action of the tool. In drilling, these forces originate at the cutting edge of the drill and are dispersed throughout the workpiece in two general directions—downward and radially. In most jig designs, the locators and the tool body resist these forces without any assistance from the clamps. However, once the cutting tool is through the workpiece, another force must be controlled—the natural tendency of the workpiece to climb the flutes of the drill. This is the force that must be restricted or controlled by the clamps. This force represents only a fraction of the force required to drill the hole and should be treated as such when clamping.

In addition to tool forces, the following factors must also be considered when designing or specifying a workholding system:

1. Clamps should always contact the workpiece at its most rigid point. When this is not possible, supports must be provided to prevent the part from bending or distorting.
2. The jig must be designed so that all the cutting forces are directed toward the locators or the tool body. The clamping devices should not be expected to absorb the cutting forces.
3. The workholding device must be positioned so that it does not interfere with the operation of the machine tool or the operation of the jig. Clamps should be accessible so that the operator can use them safely and efficiently.
4. The direction of the clamping force must also be considered with reference to the effect on the workpiece. The clamping force should be directed toward the solid locators and in a manner to keep the workpiece in the jig.

Clamping Methods

Clamping devices are commonly available in a wide range of types. When a clamp is selected for any job, the workpiece size, shape, and machining operations must be considered. Clamping

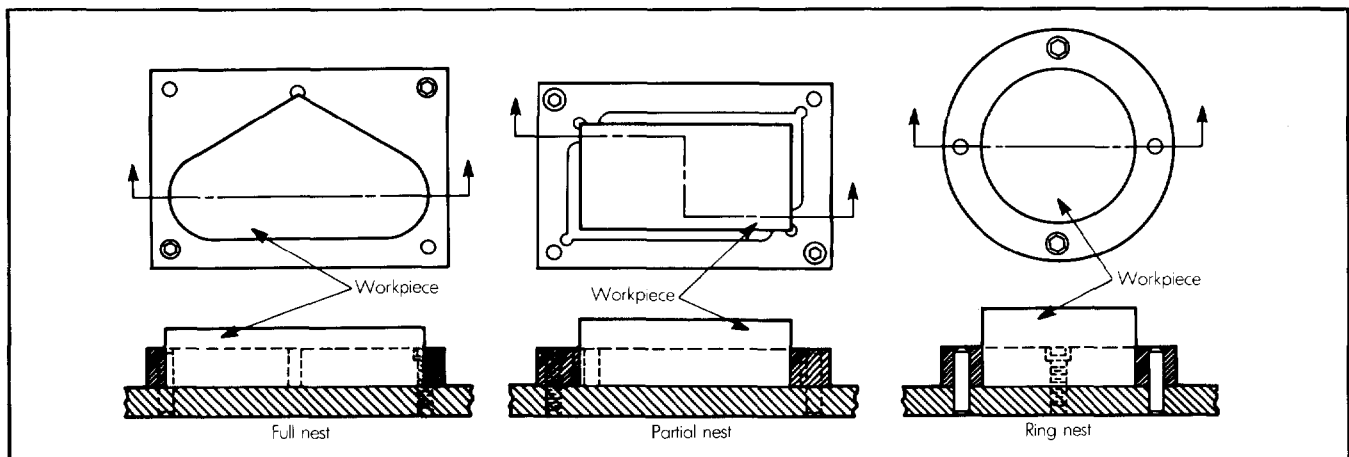


Fig. 9-84 Nesting locators can be of the full, partial, or ring type.²

WORKHOLDING DEVICES FOR DRILLING

speed and force are also important factors in selecting the correct workholder.

Clamping devices normally fall into two general categories—direct and indirect. Direct clamps are those which act directly on the surface of the workpiece to hold it in place. Indirect clamps are those that transfer the holding force through levers or similar devices to apply the required clamping force. Much can be said for both styles of clamps, but the following are a few general principles to consider when selecting either a direct or indirect clamping arrangement:

1. Indirect clamps can generate more force on the workpiece than that exerted by the clamp actuator by simply shifting the fulcrum.
2. Indirect clamps are less affected by the vibration of the machining operation.
3. Indirect clamps can be used to move the position of the clamp actuator away from the machining area, thus allowing safer operation.
4. Direct clamps have fewer parts and simpler operation.
5. Direct clamps normally require less space than indirect clamps.
6. Direct clamps are generally more sensitive to the human touch than indirect clamps.

Regardless of the clamp selected, the major criterion for choosing any clamp should be safety, efficiency, simplicity, speed, ease of operation, and holding force. There are many ways a clamping device can be actuated. Among the most commonly used for jigs are screw threads, cam action, wedges, toggle action, and air or hydraulic cylinders.

Threaded clamps. Screw threads are the least expensive and most widely used type of clamp actuator. The most common variations include strap, swing, and hook clamps (see Fig. 9-85).

Cam action. Cam action clamps are fast acting and available in many different forms. The three most commonly used variations for workholders are the flat eccentric, flat spiral, and cylindrical cam (see Fig. 9-86). Of these, the flat spiral cam is generally the most popular. One note of caution when flat cams are to be used is that they can sometimes be vibrated loose when clamped directly to a workpiece. So, as a general rule of safety, flat cam clamps should only be used with indirect clamping arrangement when heavy vibration is present.

Wedges. Wedge-type workholders use the principle of the inclined plane to exert clamping pressure. The two basic forms of wedges used for workholding devices are the flat wedge and the conical wedge (see Fig. 9-87). Wedges can be made in either self-holding or self-releasing styles. Self-holding wedges normally have an angle of 1-70°, while the self-releasing wedges have an angle over 8°. Conical wedges are normally used in the form of mandrels. The two most commonly used mandrel forms are the solid and the expansion. Solid mandrels have a fixed size and are limited to a specific hole size, while the expansion mandrels are used for a range of hole sizes.

Toggle action. Toggle action clamps are generally available with four basic clamping actions: hold down, pull, squeeze, and straight line (see Fig. 9-88). These clamps are fast acting and have a high ratio of holding force to application force. Due to their construction, toggle action clamps are relatively small and lightweight and should be used only when small loads are to be applied to the workpiece by the cutter.

Power devices. Power clamping devices provide an alternative to manual clamping methods. In general, power clamps offer better control of clamping pressure, less wear on the moving

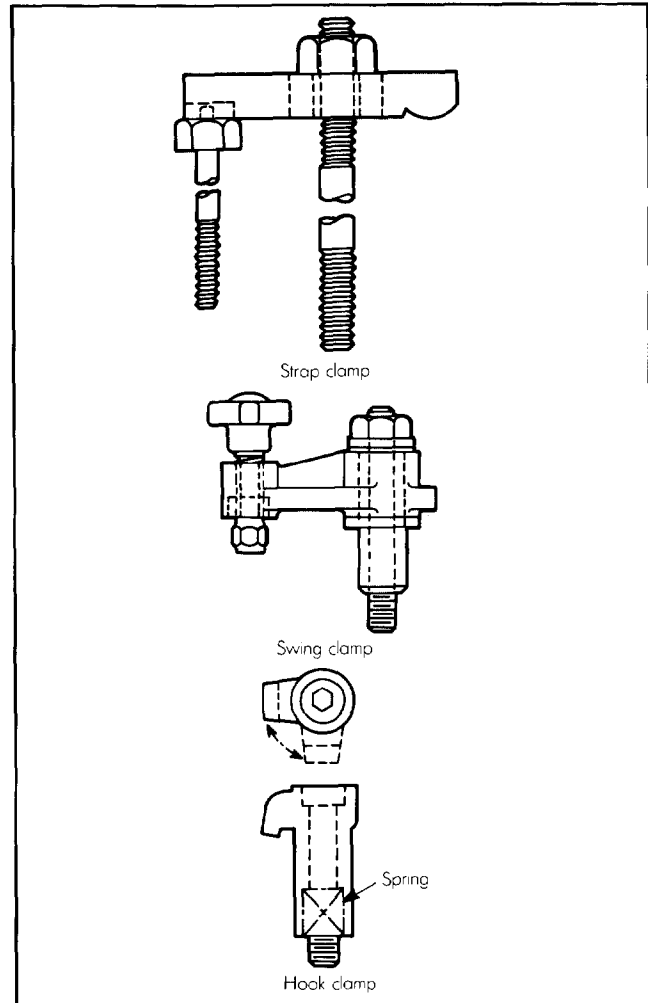


Fig. 9-85 Strap, swing, and hook types of threaded clamps for jigs.²

parts of a clamp, and faster operating cycles. Most power clamping systems operate under hydraulic power, pneumatic power, or combination of pneumatic and hydraulic power. While power clamping systems are more expensive than manual systems, their versatility, speed, and reliability often outweigh their higher initial cost. Typical applications of power clamping devices are shown in Fig. 9-89.

Tool Guiding Principles

The principle method used to guide cutting tools in a jig is the drill bushing. These bushings are commonly used to locate and guide drills and reamers to produce or modify a hole.

A few basic rules which should be followed when the proper drill bushings are to be selected are as follows:

1. The length of the drill bushing must be sufficient to support and guide the drill. Too short a bushing permits the drill to bend, producing out-of-line and oversize holes. Too long a bushing reduces the effective length of the drill and may require the use of a more costly extra-length drill. For drills having a normal helix angle, the length of the bushing should be from one and one-half to two and one-half times the diameter of the drill.

WORKHOLDING DEVICES FOR DRILLING

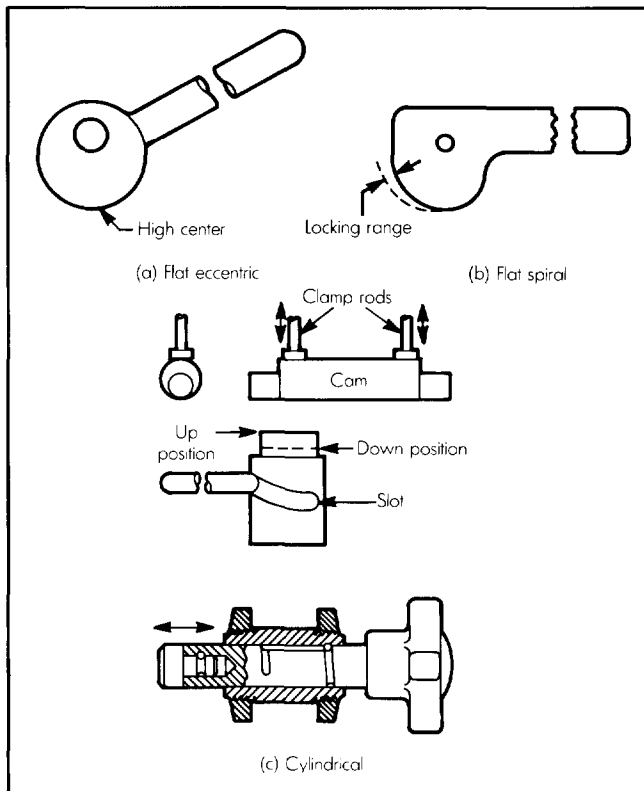


Fig. 9-86 Cam action clamps include (a) flat eccentric, (b) flat spiral, and (c) cylindrical types.²

2. Sufficient clearance should be provided between the bottom of the bushing and the top surface of the workpiece for the removal of chips and to minimize bushing wear (see Fig. 9-90). The recommended clearance for drilling metals such as cast iron, which produce small chips, is one-half the bushing ID. For other metals such as steel, which produce long stringy chips, the clearance should not exceed one and one-half times the bushing ID. An exception to these recommendations is applications requiring maximum precision, in which the bushing is sometimes placed in direct contact with the workpiece. Also, when drilling is performed at an angle relative to the surface of the workpiece, the bushing should be very close to the workpiece. Then, when the drill has penetrated to a depth of about half the drill diameter, the bushing should be retracted to allow chips to escape.
3. The diameters of the bushing bores should provide a minimum clearance of from 0.0005-0.001" (0.013-0.03 mm) for the drills. The larger the bushing diameter, however, the greater the tolerance on its bore.
4. The jig plate should be thick enough to properly hold and support the bushing. In cases in which a very thin jig plate must be used, template bushings can be used to ensure the proper tool support. In cases in which a very thick plate is used, an extended length bushing should be used. Extended length bushings are relieved to maintain the proper contact length.
5. When irregular surfaces must be drilled with close location tolerances, the bushing should conform to the shape of the surface (see Fig. 9-91).

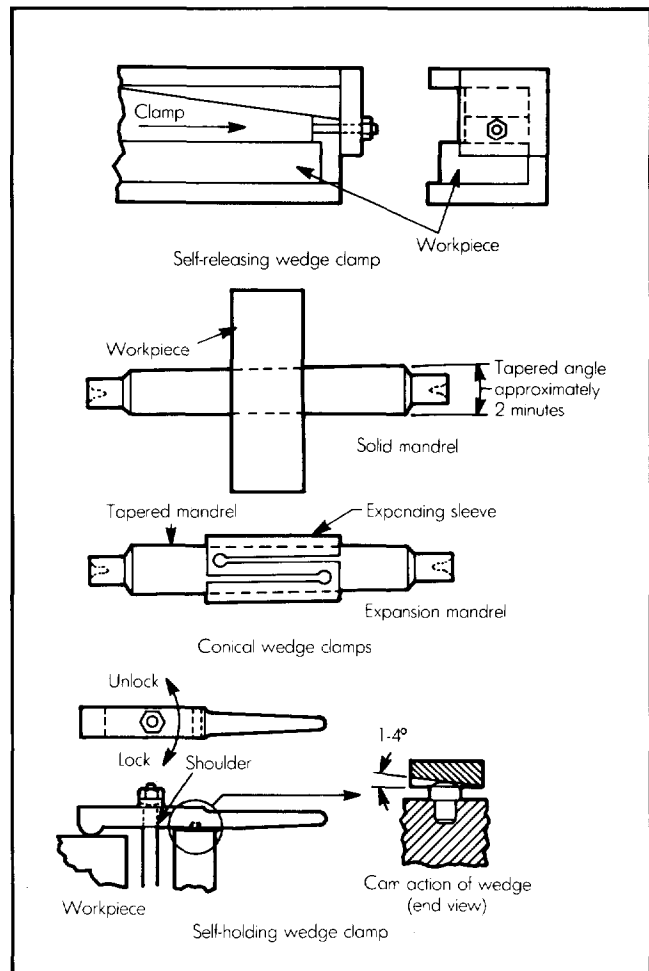


Fig. 9-87 Various types of wedge clamps often used on jigs.

Tool Guiding Methods

To meet their wide and varied applications, drill bushings are available in a range of styles and types. Sizes, types, tolerances, and identification of jig bushings and locking devices used for securing the bushings in the jig or bushing plate are specified in ANSI Standard B94.33-1974, published by ASME.

Press-fit bushings. Press-fit bushings are available in two styles—headed or headless. Two advantages that headless press-fit types offer over headed types are that (1) they can be installed flush with the jig plate surface without the need for counterboring a mounting hole and (2) they can be installed closer together because there is no interference from heads. On the other hand, if space between holes allows, headed bushings offer more resistance to heavy axial loads, which may eventually force a headless bushing out of the jig plate.

Press-fit bushings are designed to be used in applications in which repeated bushing changes are not required. Press-fit bushings are pressed directly into the jig plate and are not normally changed until they wear out. Repeated changes of press-fit bushings can wear the mounting hole and cause inaccuracies.

Liners. Liners are available in either headed or headless styles. Except for their larger, ID tolerances, liners closely resemble press-fit bushings and are used primarily to provide a

CHAPTER 9

WORKHOLDING DEVICES FOR DRILLING

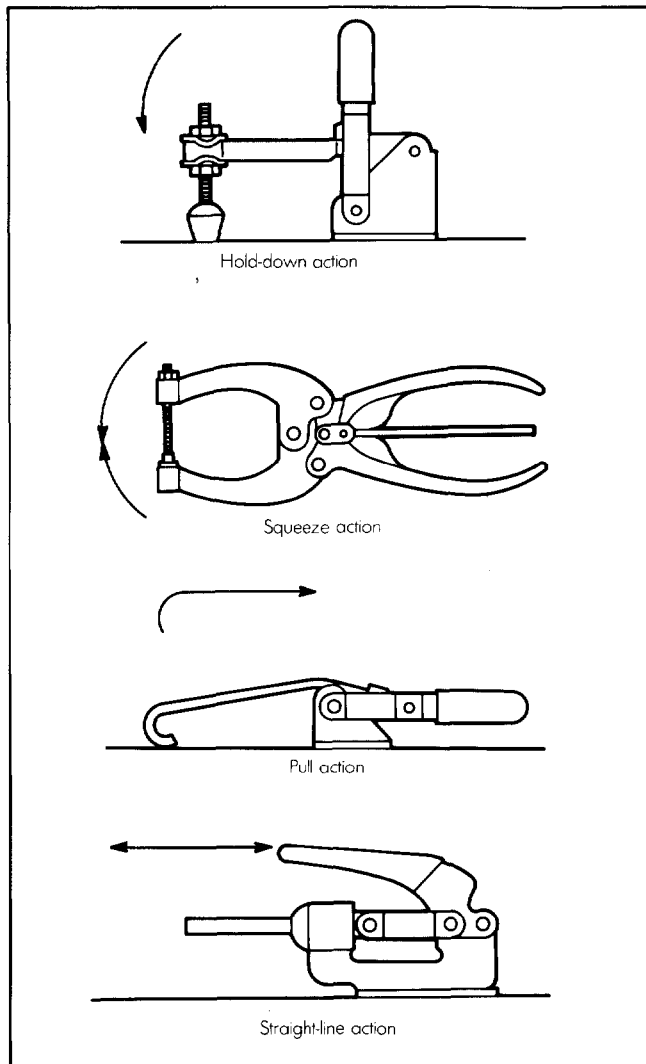


Fig. 9-88 Toggle clamps that provide hold-down, pull, squeeze, and straight-line action.²

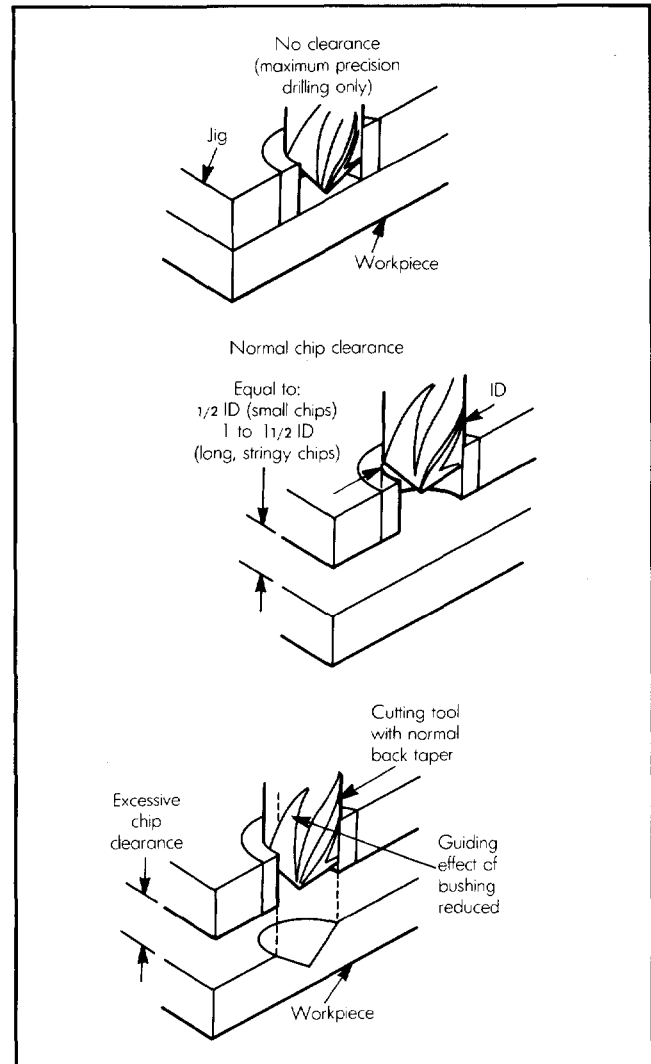


Fig. 9-90 Clearances between bushing and workpiece. Recommended clearance is illustrated in center. (Welch Drill Bushing Co.)

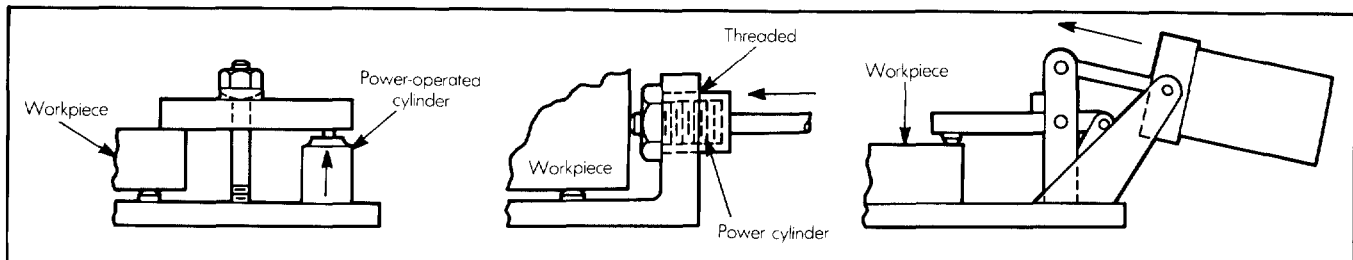


Fig. 9-89 Applications of power clamping devices for workholding jigs.²

hardened surface in the jig plate for repeated changes of renewable bushings.

Renewable bushings. Renewable bushings (Fig. 9-92), are used when the bushings have to be changed frequently because of wear or different-size holes. The two principle types of renewable bushings are fixed and slip, and both are designed to be a slip fit in a liner bushing (installed in the mounting hole in

the jig plate). Fixed renewable bushings are used when the bushings are changed infrequently. These bushings are held in place by a small clamp that must be removed when the bushing is changed. Slip renewable bushings are used when bushing changes are frequent. These bushings are made with a quick-change arrangement that permits them to be changed in a matter of seconds. Slip renewable bushings are mainly used

WORKHOLDING DEVICES FOR DRILLING

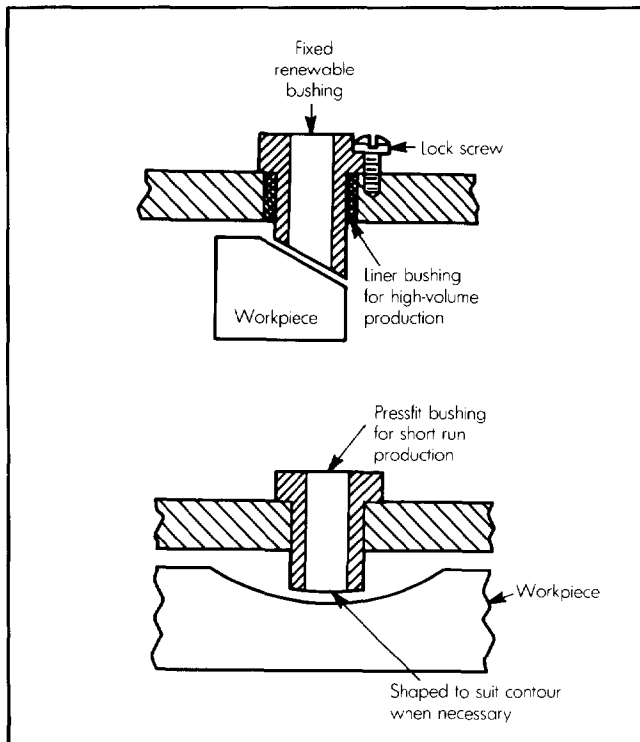


Fig. 9-91 When irregular surfaces are being drilled, the bushing should conform to the surface shape.²

when several operations are to be performed in the same hole, such as drilling and reaming. In this case, the drill bushing would be used to drill the hole, then the bushing would be removed and the reaming bushing installed. After reaming, the drill bushing would be reinstalled for the next hole.

Special purpose bushings. In addition to the standard drill bushings already discussed, several different special-purpose bushings are commonly used for jigs. These include oil-groove bushings, serrated and knurled bushings, and carbide bushings. Oil-groove bushings, having grooves in their bores to distribute cutting fluid, are used for high-speed, continuous machining operations.

Bushings having serrations or knurls on their OD's are used for applications in which the bushing is cast into the jig plate. The most common materials used for this purpose are low-melt alloys and epoxy resins. In addition, some serrated bushings are designed to be pressed into jig plates made from composition board or plywood.

Carbide bushings are available in almost any type and are used for applications in which either long production runs are expected or very abrasive materials are to be machined. These bushings cost more than their high-speed steel counterparts, but often more than make up for this added cost by the extended life they provide.

Bushings are also available with threaded ends for use with portable drilling applications. The threaded end of the bushing is screwed into the nose end of the power tool. Tapered flanges on the tip of the bushing fit under the shoulders of clamps, lockscrews, or lock strips on the jig. This arrangement effectively holds the power tool in proper position and absorbs the thrust and torque of the operation.

Roller and ball bearing, rotary-type, pilot bushings are

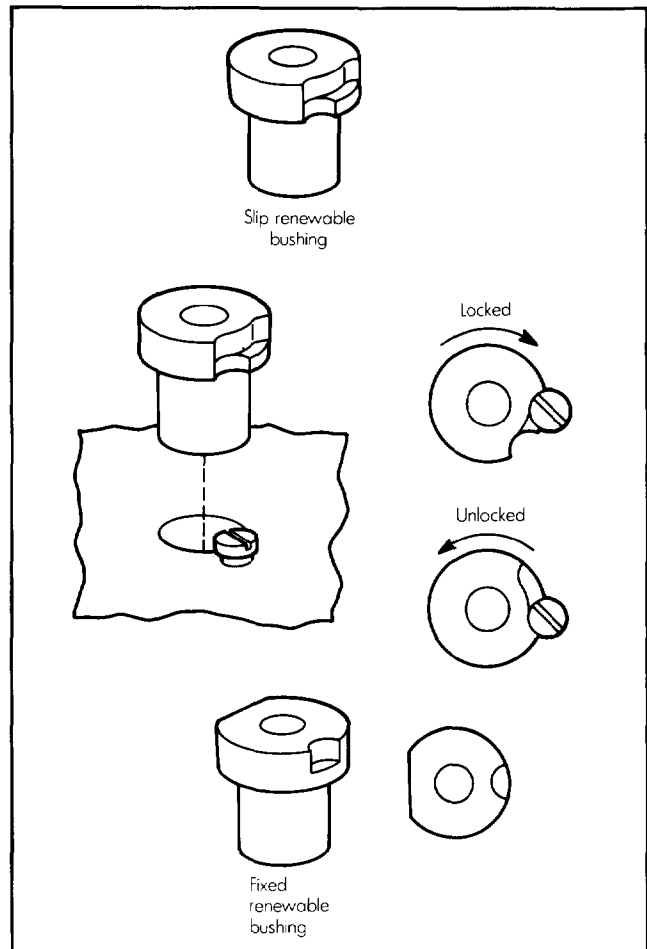


Fig. 9-92 Renewable bushings of the slip and fixed types.²

capable of handling high thrust and radial loads. They maintain close tolerances for long runs and minimize wear. Coolant-fed bushings are sometimes used to facilitate the flow of cutting fluid to the cutting edges for longer tool life.

Traveling bushing plates are often used for multispindle operations. With this arrangement, the bushing plate is attached to the drillhead rather than the workholding fixture and is spring loaded so that the drills can continue to advance after initial contact has been made with the jig or workpiece. Locating pins are generally used to ensure proper alignment of the bushing plate on the workholding fixture.

Check List for Jig Design

The questions raised in Table 9-13 should be carefully considered in designing any jig. Careful attention to these details will result in economical designs with a minimum of problems.

OPERATING PARAMETERS FOR DRILLING

Obtaining maximum economy in the use of drills requires consideration of many factors. One important factor is selecting the proper drill for a specific application. The wide variety of drills available, many of which have been discussed in the preceding sections of this chapter, makes this selection process difficult.

While a drill of almost any design can be used to produce a

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-13
Check List for Jig Design²

Locating and Supporting	
Is the workpiece properly located?	Are machined surfaces used for locating surfaces when possible?
Is the workpiece properly supported?	Can locators be replaced easily?
Will locators allow for tolerance variations in workpieces?	Are adjustable locators used for irregular surfaces?
Are locators properly related to each other?	Has locator contact been kept to a recommended minimum?
Have the most practical locators been used?	Do locators and supports clear the cutting tool?
Are the locators positioned as far apart as practical?	Has the proper tolerance been applied to the locators?
Have relieved (diamond) locators been used where appropriate?	Will ejectors, if used, interfere with the locators?
Are locators positioned or relieved to avoid burrs or chips?	Have locators been designed to allow easy loading and unloading?
Has a foolproofing device been employed?	
Are any locators unnecessary?	
Clamping and Workholding	
Will the clamp securely hold the workpiece?	Can the clamps be operated safely within easy reach of the operator?
Could the clamp bend or distort the workpiece?	Are the clamps self-contained or must wrenches be used?
Are clamps located over supports?	Are the clamps easy to operate?
Does the clamp hold the workpiece against the locators?	Can the clamp compensate for tolerance variations in part size?
Is the tool thrust directed away from the clamps?	Has an equalizing device been used for multiple-clamping tools?
Does the clamp operate quickly?	Can clamp parts be easily replaced when worn?
Is the clamp solid?	If a wrench must be used, will one wrench fit all the clamps?
Will the clamp damage the workpiece surface?	
Could the clamp loosen during use?	
Are the optimum number of clamps used?	
Will the clamp location interfere with cutters, loading, or unloading?	
Tooling	
Will drills, cutters, etc., clear the jig while in use?	Can bushings (fixed) be easily replaced when worn?
Will the jig clear the cutter when loading or unloading?	Are bushings used which are of proper size and length?
Have the cutter sizes been specified?	Are bushings properly relieved to avoid fouling with chips?
Are bushings easily removed and replaced for different operations?	Are standard bushings used wherever possible?
General Design Considerations and Safety	
Will the jig perform with an acceptable degree of repeatability?	Are coolant flow and chips directed away from the operator?
Is the jig easy and safe to load and unload?	Are all sharp edges removed?
Has the human element been considered in the design?	Does the design require the operator to work too closely to revolving tools?
Does the jig require a means to secure it to the machine table?	Is enough leverage allowed for hand-held jigs?
Is the jig easy to clean?	Is there any chance of the clamp loosening or the workpiece being pulled from the jig?
Will coolant freely drain from the jig?	Has every possible detail been studied to protect the operator from injury?
Are loose parts attached with a cable?	

hole in almost any material, lower costs for production applications necessitate use of the correct drill. Variables influencing the selection of the proper drill include the composition, hardness, and surface condition of the material to be drilled; the diameter and depth of the holes to be produced; the accuracy, surface finish, and production requirements; the type and condition of the machine to be used; and the rigidity of the setup. Standard drills should be used whenever possible because of reduced costs, interchangeability, better availability, and proven designs.

Once a drill has been selected, many operating parameters must be established. These include power requirements, cutting speeds and feed rates, and the cutting fluid to be used. Other

important considerations are regrinding, troubleshooting, and safety.

Power Requirements

In order to provide suitable equipment for any drilling operation, it is necessary to determine the torque and thrust required to rotate and feed the drill at the desired rate. Torque and speed set the power requirements; thrust determines the machine rigidity and strength requirements.

Twist drills. With twist drills, thrust force is very sensitive to variations in chisel edge length or web thickness. Web thinning reduces thrust requirements, decreases the cutting temperature, increases drill life, and improves hole geometry.

OPERATING PARAMETERS FOR DRILLING

Research and analysis indicate that drill torque and thrust are functions of the drill diameter, drill chisel edge length, feed per revolution, and workpiece material. The effects of cutting fluids and drill helix angle are relatively minor unless difficulty with chip ejection is encountered.

Reasonable estimates of torque and thrust requirements of sharp twist drills of various sizes and designs can be made from the following formulas:

For torque:

$$M = K_f^{0.8} d^{1.8} A \quad (1)$$

For thrust:

$$T = 2K_f^{0.8} d^{0.8} B + Kd^2 E \quad (2)$$

where:

M = torque, in.-lbf

T = thrust force, lb

K = work-material constant (see Table 9-14)

f = drill feed, ipr

d = drill diameter, in.

A, B, E = Drill design constants (See the following text and Table 9-15.)

For metric usage, the torque in N·m can be obtained by multiplying in.-lbf by 0.113. Thrust force in newtons is determined by multiplying the force in pounds by 4.448.

TABLE 9-14
Work-Material Constants for
Calculating Torque and Thrust

Work Material	K
Steel, 200 Bhn	24,000
Steel, 300 Bhn	31,000
Steel, 400 Bhn	34,000
Most aluminum alloys	7000
Most magnesium alloys	4000
Most brasses	14,000
Leaded brass	7000
Cast iron, 165 Bhn	15,000
Free-machining mild steel, resulfurized	18,000
Austenitic stainless steel (Type 316)	34,000

(National Twist Drill)

The most important drill design feature affecting torque and thrust is the ratio of the chisel edge length, c , to the drill diameter, d . The torque and thrust constants, A , B , and E , in Table 9-15 are based upon the ratio c/d . Because it is often easier to measure the web thickness at the point w , the table also includes approximate values of the ratio w/d . For drills of standard design, $c/d = 0.18$ can be used, and for drills with split points, $c/d = 0.03$ gives a reasonable estimate. To simplify raising the feed, f , and diameter, d , to the powers required in the formulas, values are given in Tables 9-16 and 9-17. If exact values are not found in the tables, the use of the next larger value in the tables give conservative results. Interpolation, however, gives more accurate results. The formulas yield torque and thrust requirements for sharp drills. An extra 30-50% should be provided to allow for dulling.

TABLE 9-15
Torque and Thrust Constants Based upon Ratios c/d or w/d

c/d	Approx. w/d	Torque constant A	Thrust constant B	Thrust constant E
0.03	0.025	1.000	1.100	0.001
0.05	0.045	1.005	1.140	0.003
0.08	0.070	1.015	1.200	0.006
0.10	0.085	1.020	1.235	0.010
0.13	0.110	1.040	1.270	0.017
0.15	0.130	1.080	1.310	0.022
0.18	0.155	1.085	1.355	0.030
0.20	0.175	1.105	1.380	0.040
0.25	0.220	1.155	1.445	0.065
0.30	0.260	1.235	1.500	0.090
0.35	0.300	1.310	1.575	0.120
0.40	0.350	1.395	1.620	0.160

(National Twist Drill)

* c = chisel edge length, in.
 d = drill diam, in.
 w = web thickness, in.

TABLE 9-16
Torque and Thrust Terms Based upon Feed

Feed, f , ipr	$f^{0.8}$	Feed, f , ipr	$f^{0.8}$
0.0005	0.0025	0.012	0.030
0.001	0.004	0.015	0.035
0.002	0.007	0.020	0.045
0.003	0.010	0.025	0.055
0.004	0.012	0.030	0.060
0.005	0.014	0.035	0.070
0.006	0.017	0.040	0.075
0.008	0.020	0.050	0.090
0.010	0.025		

(National Twist Drill)

If the operating speed is known, the torque can be converted to approximate horsepower requirements by means of the following formula:

$$hp = \frac{MS}{62,500} \quad (3)$$

where:

hp = approximate horsepower required

M = torque, in.-lbf

S = drill speed, rpm

or:

$$kW = \frac{MS}{9524} \quad (4)$$

where:

M = torque, N·m

To allow for machine transmission losses, the power (based on dull drills) should be multiplied by about 1.25. Alternatively, power requirements for sharp drills could be multiplied by about 1.7, thus allowing for both dulling and machine losses.

Somewhat quicker calculations of power and thrust require-

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-17
Torque and Thrust Terms
Based Upon Diameter

Diam, d, in.	d ^{0.8}	d ^{1.8}	d ²
0.063	0.110	0.007	0.004
0.094	0.150	0.014	0.009
0.125	0.190	0.025	0.016
0.156	0.225	0.035	0.025
0.188	0.260	0.050	0.035
0.218	0.295	0.065	0.050
0.250	0.330	0.082	0.065
0.281	0.365	0.105	0.080
0.313	0.395	0.125	0.105
0.344	0.425	0.145	0.120
0.375	0.455	0.170	0.140
0.438	0.515	0.225	0.190
0.500	0.575	0.285	0.250
0.563	0.630	0.355	0.315
0.625	0.685	0.430	0.390
0.688	0.740	0.510	0.470
0.750	0.795	0.595	0.565
0.875	0.900	0.785	0.765
1.000	1.000	1.000	1.000
1.125	1.010	1.235	1.270
1.250	1.195	1.495	1.565
1.375	1.290	1.775	1.890
1.500	1.385	2.075	2.250
1.625	1.475	2.400	2.640
1.750	1.565	2.740	3.060
1.875	1.655	3.100	3.520
2.000	1.740	3.480	4.000
2.250	1.915	4.300	5.070
2.500	2.080	5.200	6.250
2.750	2.250	6.190	7.560
3.000	2.410	7.240	9.000
3.500	2.720	9.500	12.250
4.000	3.030	12.100	16.000

(National Twist Drill)

ments can be made for drills of standard design through the use of graphs (Fig. 9-93 and 9-94). These show power and thrust requirements for drills of various sizes operating at various feeds in AISI 1112 steel at 100 rpm. For speeds other than 100 rpm, the power varies in direct proportion to the speed, while thrust force is substantially independent of speed. These values are for sharp drills; extra allowances must be made for drill-dulling and machine-transmission losses. The value given for AISI 1112 steel can be converted to other materials with reasonable accuracy through the use of the conversion factors in Table 9-14.

Indexable-insert drills. Because drills with indexable carbide inserts operate at substantially higher cutting speeds than HSS twist drills, they require more power. Required power depends upon the material to be drilled and increases proportionately with increases in cutting speed, feed rate, and tool diameter. Approximate motor power requirements for indexable-insert drills made by one firm are presented in Table 9-18.

Test results indicating power requirements with relation to various feed rates and drill diameters when holes were produced in steel having a tensile strength of 140,000 psi (966 MPa), with

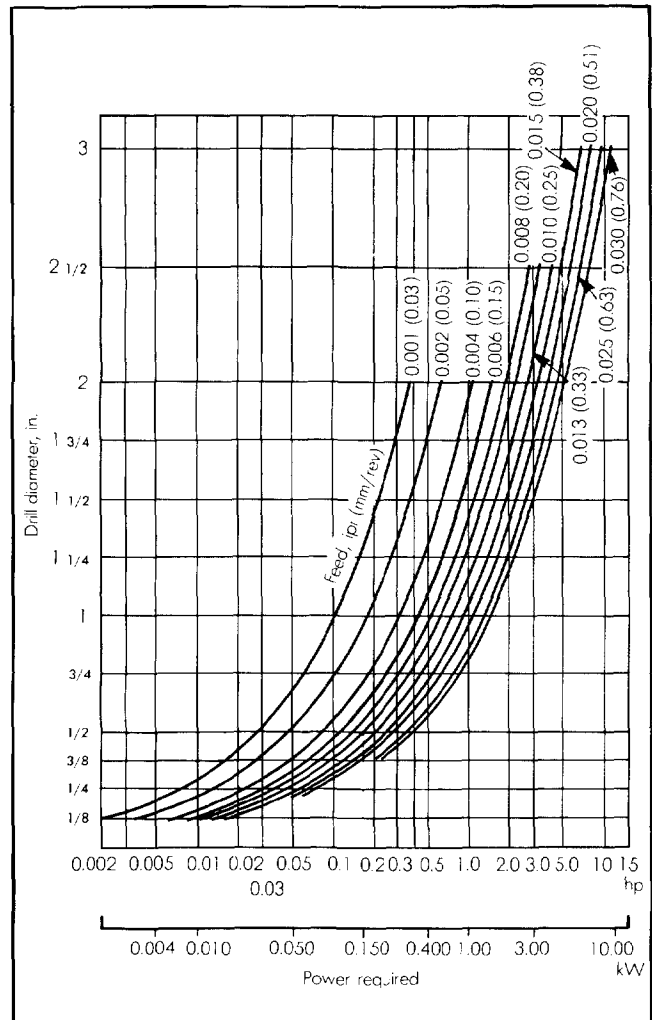


Fig. 9-93 Horsepower required to drill AISI 1112 steel at 100 rpm.³

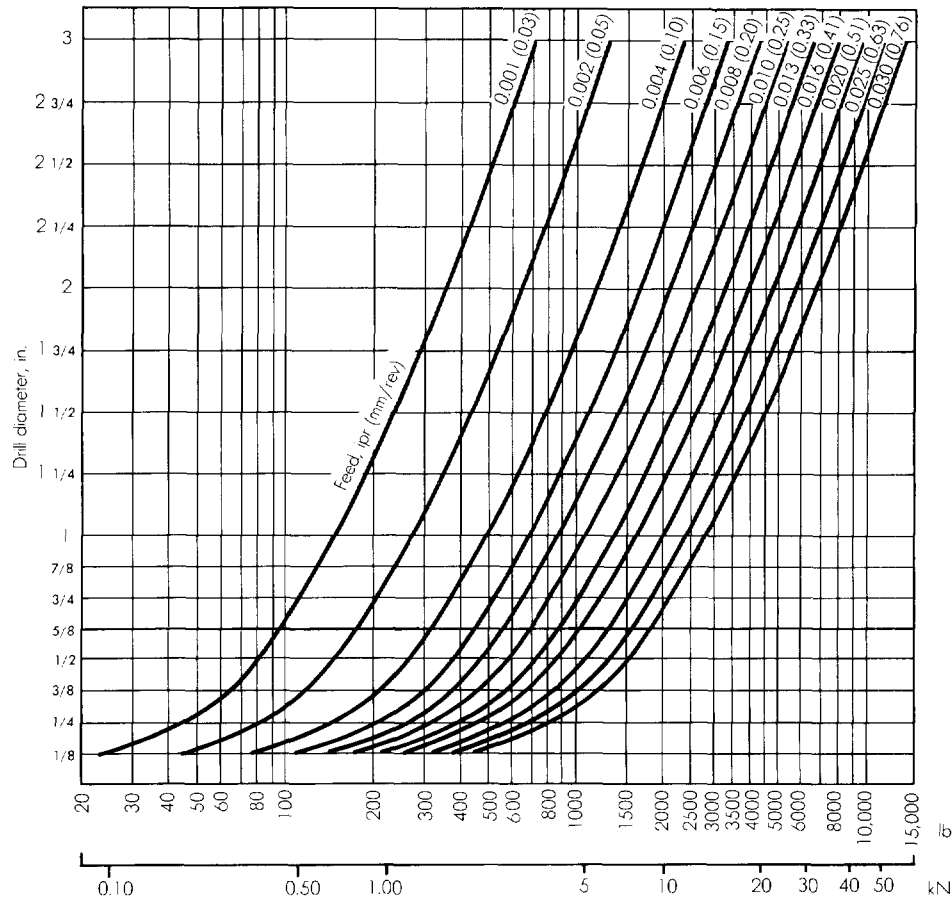
a cutting speed of 330 sfm (100 m/min), are plotted in Fig. 9-95. Because power requirements increase with cutting edge wear, the use of load meters or power monitors is recommended, especially for high-production applications. This can prevent damage to the tool, workpiece, and machine.

Thrust requirements for indexable-insert drills are less—by 30% or more—than HSS twist drills. Indexable-insert drills with a positive geometry require less axial thrust forces than drills with negative geometry. Test results indicating thrust forces with relation to various feed rates and drill diameters when holes were produced in steel having a tensile strength of 150,000 psi (1035 MPa), with a cutting speed of 330 sfm (100 m/min), are plotted in Fig. 9-96. Thrust requirements increase with increases in drill diameter, feed rate, and tensile strength of the material being drilled, but decrease with increases in cutting speed.

Spade drills. Higher thrust forces are required when spade drills are used than when twist drills or certain indexable-insert drills are used. The thrust force for spade drills can be calculated by using the formula:

$$F_t = \frac{3M}{D} \quad (5)$$

OPERATING PARAMETERS FOR DRILLING

Fig. 9-94 Thrust developed to drill AISI 1112 steel at 100 rpm.³

where:

F_t = thrust force, lb
 M = torque, in.-lbf
 D = spade drill diameter, in.

or:

$$F_t = \frac{3000M}{D} \quad (6)$$

where:

F_t = thrust force, N
 M = torque, N·m
 D = spade drill diameter, mm

Torque can be calculated as follows:

$$M = 49,500 D^2 f_r P \quad (7)$$

where:

M = torque, in.-lbf
 D = spade drill diameter, in.
 f_r = feed rate, ipr
 P = unit power (power per in.³ of metal removed per minute), hp

or:

$$M = 7.5 D^2 f_r P \quad (8)$$

where:

M = torque, N·m
 D = drill diameter, mm
 f_r = feed rate, mm/rev
 P = unit power (power per cm³ of metal removed per minute), kW

These formulas for torque are based only on volumetric considerations and do not consider the size effect, which would entail the use of different exponents and constants.

Approximate power requirements for spade drills can be determined by using formulas (3) and (4) for twist drills.

Unit power, torque, and thrust for producing holes in various materials with spade drills of different diameters, using several feed rates, are presented in Table 9-19. The unit powers, P , listed include allowances for a 25% dulling of the drills and a spindle drive efficiency of 90%. For feed rates below 0.010 ipr (0.25 mm/rev), the unit power, P , should be increased by 10%; below 0.0008 ipr (0.20 mm/rev), 20%; and below 0.006 ipr (0.15 mm/rev), 40%.

TABLE 9-18
Approximate Power Requirements for Producing Holes
in Various Materials with Indexable Insert Drills of Different Diameter

	Material Drilled																	
	Cast Irons			Low-Carbon Steels			High-Carbon Steels			High Tensile Strength Steels			Stainless Steels			Titanium Alloys		
Drill Diam., in.	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)
1 1/8	165	0.004	2.0	200	0.003	2.5	200	0.003	3.0	165	0.004	4.5	230	0.003	3.5	100	0.003	1.0
	(50)	(0.10)	(1.5)	(61)	(0.08)	(1.9)	(61)	(0.08)	(2.2)	(50)	(0.10)	(3.4)	(70)	(0.08)	(2.6)	(30.5)	(0.08)	(0.7)
	165	0.008	3.5	200	0.005	3.7	200	0.005	5.0	165	0.005	6.0	230	0.004	4.5	100	0.004	1.5
	(50)	(0.20)	(2.6)	(61)	(0.13)	(2.8)	(61)	(0.13)	(3.7)	(50)	(0.13)	(4.5)	(70)	(0.10)	(3.4)	(30.5)	(0.10)	(1.1)
	300	0.004	3.0	300	0.003	3.5	300	0.003	4.5	250	0.004	7.0	280	0.003	4.0	135	0.003	1.5
	(91)	(0.10)	(2.2)	(91)	(0.08)	(2.6)	(91)	(0.08)	(3.4)	(76)	(0.10)	(5.2)	(85)	(0.08)	(3)	(41)	(0.08)	(1.1)
1 3/8	300	0.008	6.0	300	0.005	5.5	300	0.005	7.0	250	0.005	9.0	280	0.004	5.5	135	0.004	1.5
	(91)	(0.20)	(4.5)	(91)	(0.13)	(4.1)	(91)	(0.13)	(5.2)	(76)	(0.13)	(6.7)	(85)	(0.10)	(4.1)	(41)	(0.10)	(1.1)
	165	0.005	3.0	250	0.004	4.5	250	0.004	6.0	195	0.004	6.5	265	0.004	6.0	100	0.004	1.5
	(50)	(0.13)	(2.2)	(76)	(0.10)	(3.4)	(76)	(0.10)	(4.5)	(59)	(0.10)	(4.8)	(81)	(0.10)	(4.5)	(30.5)	(0.10)	(1.1)
	165	0.010	5.5	250	0.006	6.7	250	0.006	8.5	195	0.006	10.0	265	0.005	7.5	100	0.007	2.5
	(50)	(0.25)	(4.1)	(76)	(0.15)	(5)	(76)	(0.15)	(6.3)	(59)	(0.15)	(7.5)	(81)	(0.13)	(5.6)	(30.5)	(0.18)	(1.9)
1 5/8	300	0.005	5.0	350	0.004	6.5	325	0.004	7.5	300	0.004	10.0	300	0.004	7.0	150	0.004	2.0
	(91)	(0.13)	(3.7)	(107)	(0.10)	(4.8)	(99)	(0.10)	(5.6)	(91)	(0.10)	(7.5)	(91)	(0.10)	(5.2)	(46)	(0.10)	(1.5)
	300	0.010	9.5	350	0.006	9.5	325	0.006	11.0	300	0.006	15.5	300	0.005	8.5	150	0.007	3.5
	(91)	(0.25)	(7.1)	(107)	(0.15)	(7.1)	(99)	(0.15)	(8.2)	(91)	(0.15)	(11.6)	(91)	(0.13)	(6.3)	(46)	(0.18)	(2.6)
	165	0.006	3.6	300	0.005	8.0	300	0.005	10.0	230	0.005	11.5	280	0.004	7.5	115	0.005	2.5
	(50)	(0.15)	(2.7)	(91)	(0.13)	(6)	(91)	(0.13)	(7.5)	(70)	(0.13)	(8.6)	(85)	(0.10)	(5.6)	(35)	(0.13)	(1.9)
2	165	0.012	7.5	300	0.007	11.5	300	0.008	16.0	230	0.007	16.0	280	0.005	9.5	115	0.008	4.0
	(50)	(0.30)	(5.6)	(91)	(0.18)	(8.6)	(91)	(0.20)	(11.9)	(70)	(0.18)	(11.9)	(85)	(0.13)	(7.1)	(35)	(0.20)	(3)
	300	0.006	5.0	425	0.005	11.5	400	0.005	13.5	330	0.005	16.5	345	0.004	9.5	165	0.005	3.5
	(91)	(0.15)	(3.7)	(130)	(0.13)	(8.6)	(122)	(0.13)	(10.1)	(101)	(0.13)	(12.3)	(105)	(0.10)	(7.1)	(50)	(0.13)	(2.6)
	300	0.012	9.5	425	0.007	16.0	400	0.008	21.5	330	0.007	23.0	345	0.005	11.5	165	0.008	5.5
	(91)	(0.30)	(7.1)	(130)	(0.18)	(11.9)	(122)	(0.20)	(16)	(101)	(0.18)	(17.2)	(105)	(0.13)	(8.6)	(50)	(0.20)	(4.1)
2	165	0.006	4.5	300	0.005	10.0	300	0.005	12.5	230	0.005	14.0	280	0.004	9.5	115	0.005	3.0
	(50)	(0.15)	(3.4)	(91)	(0.13)	(7.5)	(91)	(0.13)	(9.3)	(70)	(0.13)	(10.4)	(85)	(0.10)	(7.1)	(35)	(0.13)	(2.2)
	165	0.012	9.0	300	0.007	14.0	300	0.008	20.0	230	0.007	20.0	280	0.005	11.5	115	0.008	4.5
	(50)	(0.30)	(6.7)	(91)	(0.18)	(10.4)	(91)	(0.20)	(14.9)	(70)	(0.18)	(14.9)	(85)	(0.13)	(8.6)	(35)	(0.20)	(3.4)
	300	0.006	8.5	425	0.005	14.0	400	0.005	16.5	330	0.005	20.5	345	0.004	11.5	165	0.005	4.0
	(91)	(0.15)	(6.3)	(130)	(0.13)	(10.4)	(122)	(0.13)	(12.3)	(101)	(0.13)	(15.3)	(105)	(0.10)	(8.6)	(50)	(0.13)	(3)
	300	0.012	16.5	425	0.007	19.5	400	0.008	26.5	330	0.007	28.5	345	0.005	14.5	165	0.008	6.5
	(91)	(0.30)	(12.3)	(130)	(0.18)	(14.5)	(122)	(0.20)	(19.8)	(101)	(0.18)	(21.3)	(105)	(0.13)	(10.8)	(50)	(0.20)	(4.8)

TABLE 9-18—Continued
Approximate Power Requirements for Producing Holes
in Various Materials with Indexable Insert Drills of Different Diameter

	Material Drilled																	
	Cast Irons			Low-Carbon Steels			High-Carbon Steels			High Tensile Strength Steels			Stainless Steels			Titanium Alloys		
Drill Diam., in.	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)	Cutting speed, sfm (m/min)	Feed rate, ipr (mm/rev)	Power required, hp (kW)
2 1/2	165	0.08	7.5	330	0.005	13.5	325	0.005	17.0	265	0.006	24.5	295	0.004	12.5	130	0.006	5.0
	(50)	(0.20)	(5.6)	(101)	(0.13)	(10.1)	(99)	(0.13)	(12.7)	(81)	(0.15)	(18.3)	(90)	(0.10)	(9.3)	(40)	(0.15)	(3.7)
	165	0.014	13.0	330	0.008	21.5	325	0.008	27.0	265	0.008	32.5	295	0.005	15.5	130	0.009	7.5
	(50)	(0.36)	(9.7)	(101)	(0.20)	(16)	(99)	(0.20)	(20.1)	(81)	(0.20)	(24.2)	(90)	(0.13)	(11.6)	(40)	(0.23)	(5.6)
	300	0.008	13.5	490	0.005	20.5	450	0.005	23.5	395	0.006	36.5	395	0.004	16.5	175	0.006	6.5
	(91)	(0.20)	(10.1)	(149)	(0.13)	(15.3)	(137)	(0.13)	(17.5)	(120)	(0.15)	(27.2)	(120)	(0.10)	(12.3)	(53)	(0.15)	(4.8)
3 1/2	300	0.014	24.0	490	0.008	32.5	450	0.008	37.5	395	0.008	49.0	395	0.005	20.5	175	0.009	10.0
	(91)	(0.36)	(17.9)	(149)	(0.20)	(24.2)	(137)	(0.20)	(28)	(120)	(0.20)	(36.6)	(120)	(0.13)	(15.3)	(53)	(0.23)	(7.5)
	165	0.010	13.0	350	0.006	24.0	350	0.006	30.0	265	0.006	34.5	300	0.004	17.5	135	0.006	7.0
	(50)	(0.25)	(9.7)	(107)	(0.15)	(17.9)	(107)	(0.15)	(22.4)	(81)	(0.15)	(25.7)	(91)	(0.10)	(13.1)	(41)	(0.15)	(5.2)
	165	0.015	20.0	350	0.010	40.0	350	0.010	50.0	265	0.009	52.0	300	0.006	28.0	135	0.010	11.5
	(50)	(0.38)	(14.9)	(107)	(0.25)	(29.8)	(107)	(0.25)	(37.3)	(81)	(0.23)	(38.8)	(91)	(0.15)	(20.9)	(41)	(0.25)	(8.6)
4 1/2	300	0.010	23.5	550	0.006	38.00	500	0.006	43.5	400	0.006	52.0	400	0.004	23.0	190	0.006	10.0
	(91)	(0.25)	(17.5)	(168)	(0.15)	(28.3)	(152)	(0.15)	(32.5)	(122)	(0.15)	(38.8)	(122)	(0.10)	(17.2)	(58)	(0.15)	(7.5)
	300	0.015	35.5	550	0.010	63.5	500	0.010	72.0	400	0.009	78.0	400	0.006	34.5	190	0.010	16.5
	(91)	(0.38)	(26.5)	(168)	(0.25)	(47.4)	(152)	(0.25)	(53.7)	(122)	(0.23)	(58.2)	(122)	(0.15)	(25.7)	(58)	(0.25)	(12.3)
	165	0.010	17.0	375	0.007	39.0	350	0.007	46.0	275	0.006	45.5	325	0.003	18.0	135	0.006	9.0
	(50)	(0.25)	(12.7)	(114)	(0.18)	(29.1)	(107)	(0.18)	(34.3)	(84)	(0.15)	(33.9)	(99)	(0.08)	(13.4)	(41)	(0.15)	(6.7)
	165	0.015	25.5	375	0.012	66.5	350	0.012	79.0	275	0.010	75.5	325	0.006	24.0	135	0.010	15.0
	(50)	(0.38)	(19)	(114)	(0.30)	(49.6)	(107)	(0.30)	(58.9)	(84)	(0.25)	(56.3)	(99)	(0.15)	(17.9)	(41)	(0.25)	(11.2)
	300	0.010	30.5	575	0.007	52.5	550	0.007	71.0	425	0.006	71.0	425	0.003	33.5	200	0.006	13.5
	(91)	(0.25)	(22.8)	(175)	(0.18)	(39.2)	(168)	(0.18)	(53)	(130)	(0.15)	(53)	(130)	(0.08)	(25)	(61)	(0.15)	(10.1)
	300	0.015	46.0	575	0.012	105.0	550	0.012	122.0	425	0.010	118.0	425	0.006	50.0	200	0.010	22.5
	(91)	(0.38)	(34.3)	(175)	(0.30)	(78.3)	(168)	(0.30)	(91)	(130)	(0.25)	(88)	(130)	(0.15)	(37.3)	(61)	(0.25)	(16.8)

(Waukesha Cutting Tools, Inc.)

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

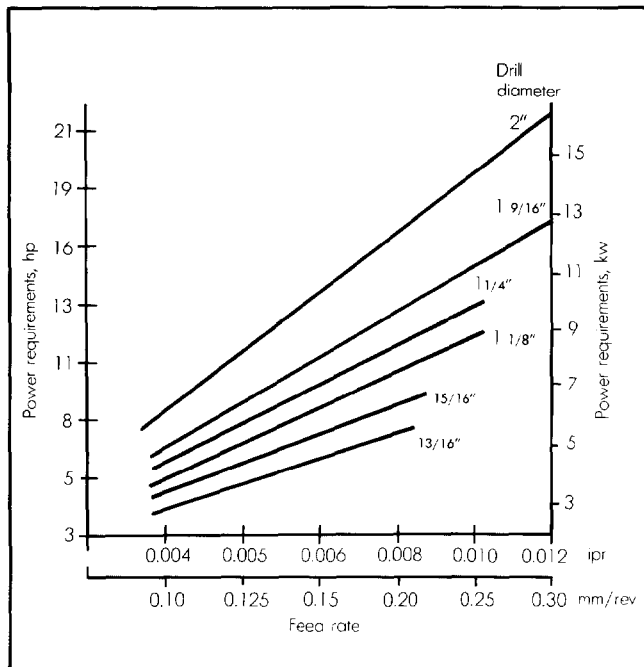


Fig. 9-95 Test results show power requirements using various feed rates and indexable-insert drills of different diameter to produce holes in steel having a tensile strength of 140,000 psi (966 MPa). The cutting speed was 330 sfm (100 m/min). (Waukesha Cutting Tools, Inc.)

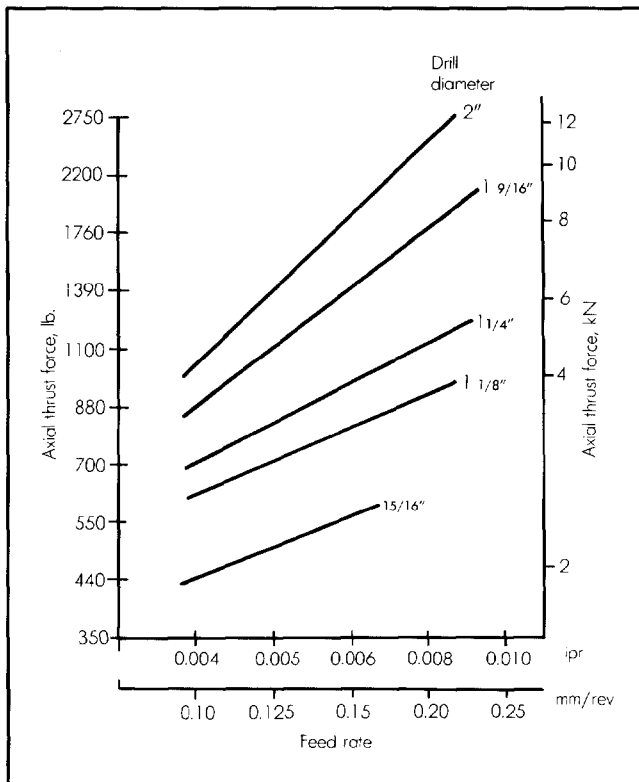


Fig. 9-96 Thrust requirements for indexable-insert drills of different diameters, using various feed rates and a cutting speed of 330 sfm (100 m/min), when producing holes in steel having a tensile strength of 150,000 psi (1035 MPa). (Waukesha Cutting Tools, Inc.)

High-pressure coolant drills. The power required to produce holes in carbon and alloy steels having a hardness of 200-275 Bhn and tensile strengths from 92,000-135,000 psi (634-931 MPa), using multiple-lip, high-pressure coolant drills with internal chip removal, is shown in Fig. 9-97. Thrust and torque requirements for the same tools and workpiece materials are presented in Figs. 9-98 and 9-99.

Power requirements for several applications of multiple-cutter trepanning tools with external chip removal are plotted in Fig. 9-100. The curves in this graph level out for larger diameter tools because a constant depth of cut, 18 mm (0.709"), was used for all tool diameters.

Drilling Speeds and Feeds

Drill speed refers to a drill's peripheral or surface speed in feet per minute (sfm) or meters per minute (m/min). Drill speed is related to spindle speed (rpm) as follows:

$$sfm = 0.0262 d \times rpm \quad (9)$$

where:

$$sfm = \text{drill surface speed, fpm}$$

$$d = \text{drill diameter, in.}$$

or:

$$m/min = 0.00314 d \times rpm \quad (10)$$

where:

$$m/min = \text{drill surface speed, m/min}$$

$$d = \text{drill diameter, millimeters}$$

or:

$$rpm = \frac{3.82 \times sfm}{d} \quad (11)$$

where:

$$d = \text{drill diameter, in.}$$

or:

$$rpm = \frac{3.1831 \times m/min}{d} \quad (12)$$

where:

$$d = \text{drill diameter, mm}$$

Feed rate is expressed as inches per revolution (ipr) or millimeters per revolution (mm/rev), as well as inches per minute (ipm) or millimeters per minute (mm/min).

The parameters of cutting speed and feed rate control metal removal rate, hole quality, and drill life. Any increase in these parameters generally increases metal removal rate, but decreases tool life. While an increase in either speed or feed has an equal effect on metal removal rate, an increase in speed usually has a larger effect in reducing drill life than an increase in feed rate.

As a result, the highest possible feed rate should be used for drilling, with moderate cutting speeds to provide satisfactory holes and economical tool life, balanced against production requirements. Feed and speed selection depend upon many variables, including the composition and hardness of the material to be drilled, depth and diameter of hole, type and condition of the drilling machine, rigidity of the setup, efficiency of the cutting fluid, and requirements for accuracy, surface finish, and production rates.

Twist drills. Specific recommendations for operating parameters must be based on the cutting tool material and the

OPERATING PARAMETERS FOR DRILLING

TABLE 9-19
Unit Power, Torque, and Thrust Values for Spade Drilling Various Materials

Condition of Material	Hardness, Bhn	Drill Diam, in.	Feed Rate, f_r , ipr (mm/rev)	Unit Power, P, hp/in. ³ /min (kW/cm ³ /min)	Torque, M, lb force x in. (N · m)	Thrust, F_t , lb (kN)
Ductile Irons						
Ferritic, annealed	140	1	0.014 (0.36)	0.7	485 (54.8)	1455 (6.47)
	to	2	0.016 (0.41)	(0.03)	2218 (250.6)	3326 (14.79)
	190	3	0.018 (0.46)		5613 (634.3)	5613 (24.97)
		4	0.020 (0.51)		11,088 (1253)	8316 (36.99)
		5	0.022 (0.56)		19,058 (2154)	11,435 (50.86)
Ferritic and Pearlritic, as cast	190	1	0.012 (0.30)	1.2	713 (80.6)	2138 (9.51)
	to	2	0.014 (0.36)	(0.05)	3326 (375.8)	4990 (22.20)
	225	3	0.016 (0.41)		8554 (966.6)	8554 (38.05)
		4	0.018 (0.46)		17,107 (1933)	12,830 (57.07)
		5	0.018 (0.46)		26,730 (3020)	16,038 (71.34)
	225	1	0.012 (0.30)	1.5	891 (100.7)	2673 (11.89)
	to	2	0.014 (0.36)	(0.07)	4158 (469.9)	6237 (27.74)
	260	3	0.016 (0.41)		10,692 (1208)	10,692 (47.56)
		4	0.018 (0.46)		21,384 (2416)	16,038 (71.34)
		5	0.018 (0.46)		33,413 (3776)	20,048 (89.17)
Pearlitic and Martensitic, normalized and tempered	240	1	0.011 (0.28)	1.5	817 (92.3)	2450 (10.90)
	to	2	0.014 (0.36)	(0.07)	4158 (469.9)	6237 (27.74)
	300	3	0.016 (0.41)		10,692 (1208)	10,692 (47.56)
		4	0.018 (0.46)		21,384 (2416)	16,038 (71.34)
		5	0.019 (0.48)		35,269 (3985)	21,161 (94.12)
Martensitic, quenched and tempered	270	1	0.008 (0.20)	1.7	713 (80.6)	2138 (9.51)
	to	2	0.009 (0.23)	(0.08)	3208 (362.5)	4811 (21.40)
	330	3	0.010 (0.25)		7574 (855.9)	7574 (33.69)
		4	0.012 (0.30)		16,157 (1826)	12,118 (53.90)
		5	0.014 (0.36)		29,453 (3328)	17,672 (78.60)
Plain Carbon Steels: 1006-1026						
Hot rolled, normalized	85	1	0.012 (0.30)	1.1	653 (73.8)	1960 (8.72)
	to	2	0.015 (0.38)	(0.05)	3267 (369.2)	4900 (21.79)
	175	3	0.018 (0.46)		8820 (996.7)	8820 (39.23)
		4	0.022 (0.56)		19,166 (2166)	14,375 (63.94)
		5	0.025 (0.63)		34,031 (3845)	20,419 (90.82)
Hot rolled, annealed Cold drawn	125	1	0.012 (0.30)	1.1	653 (73.8)	1960 (8.72)
	to	2	0.015 (0.38)	(0.05)	3267 (369.2)	4900 (21.79)
	175	3	0.018 (0.46)		8820 (996.7)	8820 (39.23)
		4	0.022 (0.56)		19,166 (2166)	14,375 (63.94)
		5	0.025 (0.63)		34,031 (3845)	20,419 (90.82)
	175	1	0.012 (0.30)	1.2	713 (80.6)	2138 (9.51)
	to	2	0.015 (0.38)	(0.05)	3564 (402.7)	5346 (23.78)
	225	3	0.018 (0.46)		9623 (1087.4)	9623 (42.80)
		4	0.022 (0.56)		20,909 (2363)	15,682 (69.75)
		5	0.025 (0.63)		37,125 (4195)	22,275 (99.08)
	225	1	0.010 (0.25)	1.3	643 (72.7)	1930 (8.58)
	to	2	0.014 (0.36)	(0.06)	3604 (407.3)	5405 (24.04)
	275	3	0.016 (0.41)		9266 (1047)	9266 (41.22)
		4	0.020 (0.51)		20,592 (2327)	20,592 (91.59)
		5	0.022 (0.56)		35,393 (3999)	35,393 (157.43)

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-19—Continued

Condition of Material	Hardness, Bhn	Drill Diam, in.	Feed Rate, f_r , ipr (mm/rev)	Unit Power, P, hp/in. ³ /min (kW/cm ³ /min)	Torque, M, lb force x in. (N · m)	Thrust, F _t , lb (kN)
Plain carbon steels: 1027, 1030, 1033-1046, 1049, 1050, 1052, 1055, 1060, 1062, 1064-1066, 1070, 1074, 1078, 1080, 1085, 1088, 1090, and 1095						
Hot rolled, normalized or annealed Cold drawn	125 to 175	1	0.012 (0.30)	1.1 (0.05)	653 (73.8)	1960 (8.72)
		2	0.015 (0.38)		3267 (369.2)	4900 (21.80)
		3	0.018 (0.46)		8820 (996.7)	8820 (39.23)
		4	0.022 (0.56)		19,166 (2166)	14,375 (63.94)
		5	0.025 (0.63)		34,031 (3845)	20,419 (90.82)
	175 to 225	1	0.012 (0.30)	1.2 (0.05)	713 (80.6)	2138 (9.51)
		2	0.015 (0.38)		3564 (402.7)	5346 (23.78)
		3	0.018 (0.46)		9623 (1087.4)	9623 (42.80)
		4	0.022 (0.56)		20,909 (2363)	15,682 (69.75)
		5	0.025 (0.63)		37,125 (4195)	22,275 (99.08)
Hot rolled, normalized or annealed Hot rolled, quenched and tempered Cold drawn	225 to 275	1	0.010 (0.25)	1.3 (0.06)	643 (72.7)	1930 (8.58)
		2	0.014 (0.36)		3604 (407.3)	5405 (24.04)
		3	0.016 (0.41)		9266 (1047)	9266 (41.22)
		4	0.020 (0.51)		20,592 (2327)	15,444 (68.69)
		5	0.022 (0.56)		35,393 (3999)	21,236 (94.46)
	275 to 325	1	0.009 (0.23)	1.4 (0.06)	668 (75.5)	2005 (8.92)
		2	0.012 (0.30)		3326 (375.8)	4990 (22.20)
		3	0.014 (0.36)		8732 (986.7)	8732 (38.84)
		4	0.016 (0.41)		17,741 (2005)	13,306 (59.19)
		5	0.018 (0.46)		31,185 (3524)	18,711 (83.23)
Hot rolled, quenched and tempered	325 to 375	1	0.008 (0.20)	1.4 (0.06)	594 (67.1)	1782 (7.93)
		2	0.010 (0.25)		2772 (313.2)	4158 (18.49)
		3	0.012 (0.30)		7484 (845.7)	7484 (32.29)
		4	0.014 (0.36)		15,523 (1754)	11,642 (51.78)
		5	0.016 (0.41)		27,720 (3132)	16,632 (73.98)
	375 to 425	1	0.007 (0.18)	1.5 (0.07)	624 (70.5)	1871 (8.32)
		2	0.009 (0.23)		3029 (342.3)	4544 (20.21)
		3	0.010 (0.25)		7128 (805.5)	7128 (31.71)
		4	0.012 (0.30)		15,206 (1718)	11,406 (50.73)
		5	0.014 (0.36)		27,720 (3132)	16,632 (73.98)
Free-machining alloy steels, resulfurized: 3140, 4140, 4150, and 8640						
Hot rolled, normalized or annealed Cold drawn	150 to 200	1	0.014 (0.36)	1.1 (0.05)	762 (86.1)	2287 (10.17)
		2	0.016 (0.41)		3485 (393.8)	5227 (23.25)
		3	0.018 (0.46)		8821 (996.8)	8821 (39.24)
		4	0.022 (0.56)		19,166 (2166)	14,375 (63.94)
		5	0.025 (0.63)		34,031 (3846)	20,419 (90.82)
	200 to 250	1	0.014 (0.36)	1.3 (0.06)	901 (101.8)	2703 (12.02)
		2	0.016 (0.41)		3194 (360.9)	4791 (21.31)
		3	0.018 (0.46)		10,425 (1178)	10,425 (46.37)
		4	0.022 (0.56)		22,651 (2560)	16,988 (75.56)
		5	0.025 (0.63)		40,219 (4546)	24,131 (107.33)

OPERATING PARAMETERS FOR DRILLING

TABLE 9-19—Continued

Condition of Material	Hardness, Bhn	Drill Diam, in.	Feed Rate, f_r , ipr (mm/rev)	Unit Power, P, hp/in ³ /min (kW/cm ³ /min)	Torque, M, lb force x in. (N · m)	Thrust, F_t , lb (kN)
Hot rolled, quenched and tempered	275	1	0.012 (0.30)	1.4	832 (94.0)	2495 (11.10)
	to	2	0.014 (0.36)	(0.06)	3881 (438.6)	5821 (25.89)
	325	3	0.016 (0.41)		9979 (1128)	9979 (44.39)
		4	0.018 (0.46)		19,958 (2255)	14,969 (66.58)
		5	0.020 (0.51)		34,650 (3915)	20,790 (92.47)
	325	1	0.010 (0.25)	1.4	693 (78.3)	2079 (9.25)
	to	2	0.012 (0.30)	(0.06)	3326 (375.8)	4990 (22.20)
	375	3	0.014 (0.36)		8732 (986.7)	8732 (38.84)
		4	0.016 (0.41)		17,741 (2005)	13,306 (59.19)
		5	0.018 (0.46)		31,185 (3524)	18,711 (83.23)
	375	1	0.008 (0.20)	1.6	673 (76.0)	2020 (8.98)
	to	2	0.010 (0.25)	(0.07)	3168 (358.0)	4752 (21.14)
	425	3	0.012 (0.30)		8554 (966.6)	8554 (38.05)
		4	0.014 (0.36)		17,741 (2005)	13,306 (59.19)
		5	0.015 (0.38)		29,700 (3356)	17,820 (79.26)
Aluminum alloys, wrought, heat treated: 2011, 2014, 2017, 2018, 2024, 2025, 2117, 2218, 4032, 6053, 6061, 6062, 6063, 6151, 6262, 6464, 7075, 7079, and 7178						
Solution treated and aged	75	1	0.024 (0.61)	0.3	356 (40.2)	1069 (4.75)
	to	2	0.028 (0.71)	(0.01)	1663 (187.9)	2495 (11.10)
	150	3	0.032 (0.81)		4277 (483.3)	4277 (19.02)
	(500 kg)	4	0.036 (0.91)		8554 (966.6)	6415 (28.53)
		5	0.040 (1.02)		14,850 (1678)	8910 (39.63)

(Madison Industries)

material to be drilled. Suggested speeds and feeds that can be used as starting points for producing holes in various materials with twist drills are presented in Table 9-20. Some aircraft manufacturers are performing high-speed drilling at 20,000 rpm (vs. the conventional speed of 6000 rpm) on 7075-T651 aluminum alloy with hand-held, air-driven drilling units.

The method of power feeding must be taken into consideration. A mechanical power feed is the most effective method because it maintains a constant feed per revolution. Non-mechanical power feeds are less effective because they tend to reduce the feed as the thrust increases with drill wear.

Multidiameter drills. Proper speed and feed selection for multiple diameter drills is vital to good performance. The surface speed used should be based upon the largest drill diameter, while the feed rate must be selected based on the smallest drill diameter. This consideration is often ignored in setting job conditions until failure occurs. The same consideration should also be applied to multiple-spindle drilling in which the spindle speeds are all the same.

Core drills. Multiple-flute (three or four-flute) core drills are generally operated at about two-thirds the surface speed of two-flute twist drills in a given material. The feed used is the same per cutting edge as for a conventional two-flute drill. A four-flute core drill would therefore have a feed per revolution twice that of a standard two-flute twist drill.

Producing deep holes with twist drills. When producing holes having depths more than about three to four times the diameters of twist drills, the speeds and feeds should be reduced. Suggested reductions are given in Table 9-21.

Coolant-fed twist drills. A fairly safe starting point for coolant-fed twist drills is an increase of 10% in speed and 25% in feed rate over that recommended for solid twist drills. In many cases, higher increases are possible. As the feed is increased, however, the speed should be increased proportionately less in order not to burn or dull the drill prematurely.

While coolant-fed twist drills are capable of producing holes to depths greater than six diameters, some reduction of feed and speed is generally necessary in deep holes (when hole depth to drill diameter ratios exceeds 4:1). A 10-20% reduction is recommended for ratios of 6:1 to 10:1. Ratios beyond 10:1 may require partial or full retraction of the drill to relieve chips.

Indexable-insert drills. Drills with indexable carbide inserts can generally be operated at higher speeds and feeds than twist drills. Recommended starting points for indexable-insert drills made by one manufacturer are presented in Table 9-22. Optimum speeds and feeds vary depending upon the material to be drilled, rigidity of the machine and setup, and available power.

For applications in which the workpiece surfaces are not perpendicular to the drill axes, it is recommended that a lower starting feed rate be used until the full diameter of the drill is in the cut. The same should be done for interrupted cuts, such as drilling through crossholes or curved surfaces.

Spade drills. When spade drills are used, the speed is often reduced slightly and the feed increased over that used for the same size twist drill. For deep holes, both the speed and feed should be reduced as the depth increases. Many problems in spade drilling can be traced to insufficient feed. This causes

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

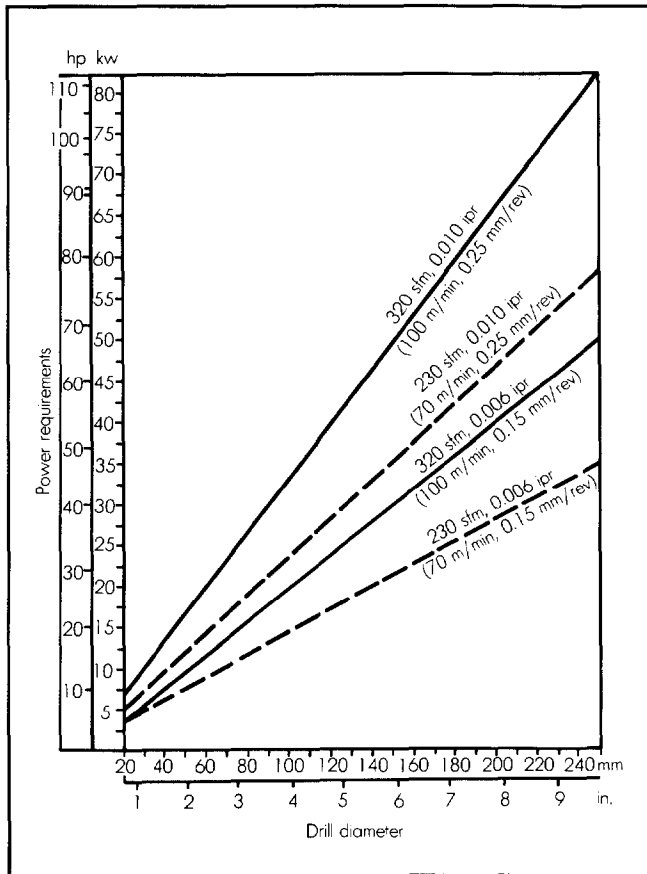


Fig. 9-97 Power requirements for multiple-lip, high-pressure coolant drills with internal chip removal. (Sandvik Inc.)

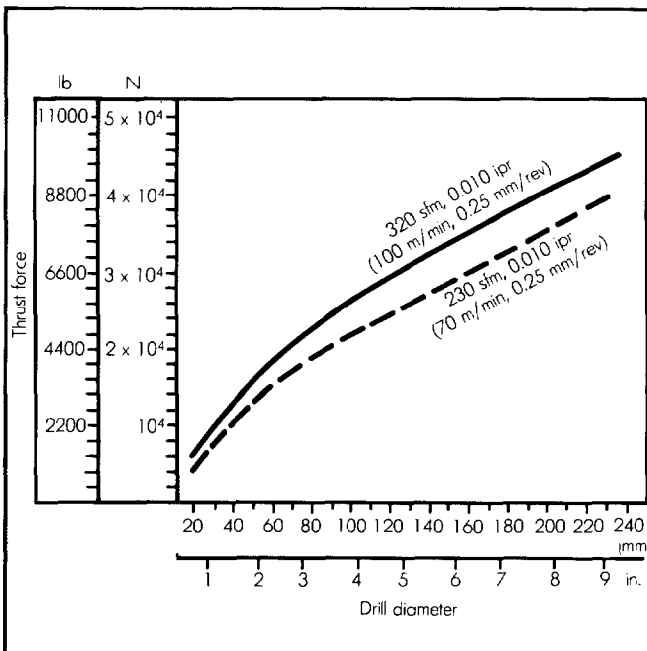


Fig. 9-98 Thrust requirements for multiple-lip, high-pressure coolant drills with internal chip removal. (Sandvik Inc.)

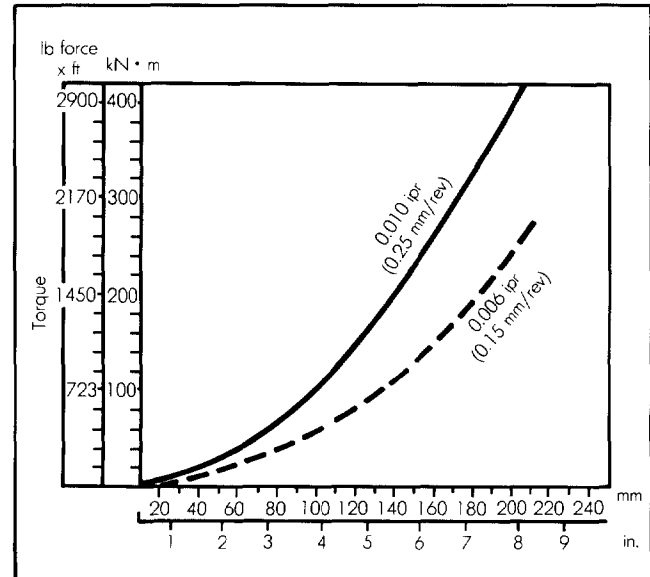


Fig. 9-99 Torque requirements for multiple-lip, high-pressure coolant drills with internal chip removal. (Sandvik Inc.)

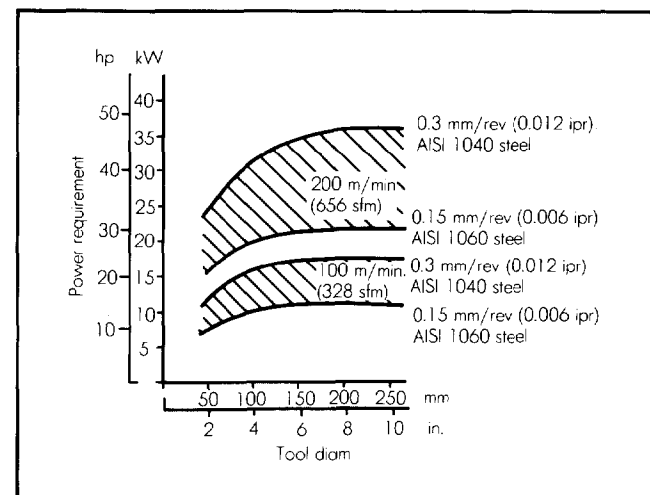


Fig. 9-100 Power requirements for balanced-force, multiple-cutter trepanning tools. (Sandvik Inc.)

vibration and edge chipping, chips which do not flush readily from the hole, and work hardening when drilling some materials.

Some average feeds and speeds for various materials recommended by one manufacturer of spade drills are presented in Table 9-23. The values shown can generally be increased 15-20% when indexable blades are used. For intermediate size drills, the values can be interpolated.

Actual feeds and speeds used depend upon the condition of the material to be drilled and the many variables in the setup. For annealed material, it is suggested that the lowest feed and speed listed be used for initial startup and that first feed, then speed, be increased in increments of about 20% as conditions allow. Hard or very tough materials should be drilled initially with feeds and speeds 30% below those listed; the same method of progressive increase should be used.

Excessive feeds as well as too low a feed rate, can cause undue

OPERATING PARAMETERS FOR DRILLING

chipping near the center and perhaps total failure by breakage. Excessive speeds can be recognized by the temper color. Blue or black temper colors appear at the outside corners and indicate excessive heat, which softens the edge and allows premature flank wear. Flank wear may be allowed to progress to a width of 0.030-0.040" (0.76-1.02 mm) before regrinding.

High-pressure coolant drills. Since the cutting edges of gundrills are carbide, high surface speeds with light feed rates are recommended. Conservative starting conditions that can be used as a guide are presented in Table 9-11. Adjustments of the speed and feed values given should be made after the first few holes are drilled, based on results. This is necessary due to the variables of the material drilled, tools and machine used, rigidity of the setup, and quality of hole desired.

If the hardness of the material to be gundrilled is much higher than the values shown in Table 9-11, only three-fourths of the speed and feed given should be used to start. Allowable whip length may then be increased 10% over the table value. If the unsupported shank length of the gundrill is greater than allowable whip length, whip guides must be used. If whip guides are not available, the drill speed must be reduced or a shorter gundrill used.

Recommended feeds and speeds for multiple-flip, internal-chip-removal gundrills are presented in Table 9-24. The same data for multiple-flip, external-chip-removal drills is given in Table 9-25, and for trepanning tools with internal chip removal, Table 9-26.

Small-hole drills. Perhaps the most significant item concerning operating conditions for small-diameter drills is speed and feed selection. Normally, the conditions existing when problems develop are overspeeding and underfeeding. In too many instances, a job is set up to operate at a given speed, and the feed is then set so that the drill does not fracture. This generally results in definite underfeeding.

Successful drilling with small-diameter drills requires selection of feeds that produce free-flowing chips and not a powder. If a powder is produced, the tool is almost wearing itself through instead of cutting and there is a tendency for the powder to pack and clog. Thus, feed selection for small-diameter drills must be based on chip thickness, not on inches per minute. Selection of a speed based upon the strength and load-carrying capacity of the drill using that feed then results in a successful production operation.

A common misconception is that the cutting speed should remain constant regardless of the drill diameter. One firm specializing in small-hole drilling generally recommends a speed of approximately 2900 rpm for drilling with about 0.020" (0.51 mm) diameter drills, and decreasing speeds for reduced diameter drills. A speed of only 50 rpm was used by this firm to drill holes 0.000025" (0.00064 mm) diameter in Zircaloy tubing.

Slower feed rates and higher drill speeds can help minimize buckling and point walking; however, if the material is abrasive and/or subject to work hardening, these conditions can increase tool wear and may cause breakage. Soft and nonferrous materials can often be drilled at higher speeds. For example, plastics and printed circuit boards (such as copper cladding on glass epoxy) are being drilled at speeds up to 50,000 rpm or more.

Depending upon the material, the feed rate required to start drilling may be more, and is often less, than needed to continue. This may require slight changes in the feed rate during drilling to maintain good chip flow. Also, when a drill breaks through a surface in drilling through holes, a reduction in feed rate is often

necessary to prevent burrs. For these reasons, a controlled, but adjustable, sensitive feed mechanism may be preferable to a positive feed system.

Maximum depths in drilling small holes are generally limited to about three times the hole diameter when the holes are less than 0.0001" (0.003 mm) diam, and range up to 20 or more times the diameter for holes larger than 0.004" (0.10 mm) diam. Deep-hole drilling of small holes generally requires slower speeds and is usually accomplished by gradually increasing the hole diameter and depth, using the most rigid drills possible.

Cutting Fluids for Drilling

As with most other machining operations, the most important characteristics for drilling are the ability of the cutting fluid to cool and lubricate. Lubrication reduces friction between the sliding chips and the drill and also minimizes heat caused by friction between the drill and workpiece. Many drill failures are the result of excessive heat.

Cutting fluids have a direct influence on tool life, built-up edge, and hole finish. Water-based emulsified oil is the most commonly used coolant today for drilling operations. It is suitable for a wide variety of steels and some of the nonferrous materials.

Some firms specializing in drilling small holes have reported good results from using water-soluble oils in conjunction with a mist sprayer operating at a pressure of 25-35 psi (172-241 kPa). The pressurized air serves to remove chips and, together with the cutting fluid, effectively cools the drill.

When drilling high temperature resistant alloys and work hardening grades of stainless steel, tool life can be enhanced with a lubricant of sulfur-based oil or sulfo-chlorinated mineral oil.

A detailed discussion of cutting fluids is presented in Chapter 4 of this volume, "Cutting Fluids and Industrial Lubricants." This discussion includes types of cutting fluids available, selection parameters for various operations, application and filtration methods, and maintenance requirements.

Most abrasive nonmetallic materials such as fiberglass, ceramics, etc., are drilled without a liquid coolant. Two methods for removing the abrasive powdered chips from the hole are a jet of air, or preferably a vacuum.

Coolant-fed twist drills. Coolant pumps on some drilling machines are not satisfactory for the use of coolant-fed tools because they deliver inadequate pressures and flow rates. Depending upon the material to be drilled, the drill diameter, hole depth, and other factors, coolant-fed drills may require volumes from 3-10 or more gpm (11-38 L/min), with pressures from 50-300 psi (345-2068 kPa).

Good results are being obtained with both constant high pressure and pulsating coolant systems. Advocates of pulsating coolant (100-250 pulses per minute) claim that it breaks a vapor barrier (fluid at the hot cutting edges that has vaporized and remains there to block additional fluid from entering), thus permitting fresh coolant to reach the cutting edges. In addition, the pulsating coolant acts as a hydraulic ram to remove chips.

Loss of fluid can be a problem with coolant-fed drills when the drills break out of the holes. The fluid escapes through the initial exit hole, leaving the drill corners uncooled. This can be minimized by supplementing through-the-drill cooling with flood cooling. Flood cooling can be controlled so that it is only actuated just before breakthrough.

Indexable-insert drills. Coolant is particularly critical when using indexable-insert drills because of the high volume of chips produced. Pressure and volume requirements depend upon drill

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-20
Suggested Operating Conditions for Producing Holes
in Various Materials with Twist Drills

Material Drilled	Hardness		Cutting Tool Material*	Peripheral Speed, sfm (m/min)	Feed Rate**	Helix Angle, degrees	Point Angle, degrees	Point Style
	Brinell	Rockwell						
Aluminum and its alloys	45-105	to R _B 62	HSS	350 (107)	Z	32-42	90-118	---
Asbestos	---	---	WC	55 (17)	Y	17-20	90	---
Bakelite	---	---	WC	80 (24)	Y	30-40	90-118	---
Carbon	---	---	HSS	60-70 (18-21)	W	25-35	90-118	---
Copper and its alloys								
High machinability	to 124	R _B 10-70	HSS	200 (61)	Z	15-40	118	---
Low machinability	to 124	R _B 10-70	HSS	70 (21)	Z	0-25	118	---
Fiberglass-epoxy	---	---	WC	650 (198)	0.0025- 0.0050" (0.063- 0.127 mm)	35-40	118-130	4 facet
Glass	---	---	WC	15-25 (4.6-7.6)	Light hand	0	---	Spear
High-temperature alloys								
Cobalt based	180-230	R _B 89-99	HSS-Co	20 (6.1)	W	28-35	118-135	Split
Iron based	180-230	R _B 89-99	HSS-Co	25 (7.6)	X	28-35	118-135	Split
Nickel based	150-300	to R _C 32	HSS-Co	20 (6.1)	W	28-35	118-135	Split
Iron								
Cast (soft)	120-150	to R _B 80	HSS	140-150 (43-46)	Z	20-30	90-118	---
			WC	90-165 (27-50)	Y	14-25	90-118	---
Cast (med. hard)	160-220	R _B 80-97	HSS	80-110 (24-34)	Y	20-30	90-118	---
			WC	90-165 (27-50)	X	14-25	90-118	---
Hard chilled	400	R _C 41	WC	30 (9)	X	0-25	130-140	Notched
Malleable	112-126	to R _B 71	HSS	90-120 (27-37)	Y	20-30	90-118	---
			WC	100-150 (30-46)	X	14-25	118	---
Ductile	190-225	to R _B 98	HSS	60 (18)	Y	20-30	118	---
			WC	80-100 (24-30)	X	14-25	118	---
Magnesium and its alloys	50-90	to R _B 52	HSS	150-400 (46-122)	Z	25-35	118	---
Marble	---	---	WC	15-25 (4.6-7.6)	Light hand	0-10	90-130	---
Plastics	---	---	HSS	100 (30)	Y	15-25	118	---
			WC	100-200 (30-61)	X	15-25	118	---

OPERATING PARAMETERS FOR DRILLING

TABLE 9-20—Continued

Material Drilled	Hardness		Cutting Tool Material*	Peripheral Speed, sfm (m/min)	Feed Rate**	Helix Angle, degrees	Point Angle, degrees	Point Style
	Brinell	Rockwell						
Rubber (hard)	---	---	HSS	100-300 (30-91)	X	10-20	90-118	---
			WC	200-300 (61-91)	W	15-25	118	---
Steel								
Plain carbon								
to .25 C	125-175	R _B 71-88	HSS	80 (24)	Y	25-35	118	---
to .50 C	175-225	R _B 88-98	HSS	65 (20)	Y	25-35	118	---
to .90 C	175-225	R _B 88-98	HSS	55 (17)	Y	25-35	118	---
Alloy steel								
Low C(.12 to .25)	175-225	R _B 88-98	HSS	70 (21)	Z	25-35	118	---
Med C(.30 to .65)	175-225	R _B 88-98	HSS	50-60 (15-18)	X	25-35	118	---
	488+	R _C 50+	WC	75-100 (23-30)	0.0005- 0.0015" (0.013- 0.038 mm)	25-35	135	Notched
Maraging	275-325	R _C 28-35	HSS	55 (17)	Y	25-32	118-135	Split
Stainless								
Austenitic	135-185	R _B 75-90	HSS-Co	55 (17)	X	25-35	118-135	Split
Ferritic	135-185	R _B 75-90	HSS	65 (20)	X	25-35	118-135	Split
Martensitic	135-175	R _B 75-88	HSS-Co	65 (20)	Z	25-35	118-135	Split
Precipitation hardened Tool	150-200	R _B 82-94	HSS-Co	50 (15)	X	25-35	118-135	Split
	196	R _B 94	HSS	60 (18)	Y	25-35	118	---
	241	R _C 24	HSS	50 (15)	Y	25-35	118	---
Titanium								
Pure	110-200	to R _B 94	HSS	100 (30)	X	30-38	135	Split
Alpha and alpha beta	300-360	R _C 31-39	HSS-Co	40 (12)	Y	30-38	135	Split
Beta	275-350	R _C 29-38	HSS-Co	25 (7.6)	W	30-38	135	Split
Zinc alloys	80-100	R _B 41-62	HSS	250 (76)	Z	32-42	118	---

* HSS—High-speed steel; HSS-Co—High-speed steel with cobalt; WC—Tungsten carbide

** Feed Rate, ipr (mm/rev)	Drill Diameter				
	1/8" (3.2)	1/4" (6.4)	1/2" (12.7)	3/4" (19.1)	1" (25.4)
W	0.0015 (0.038)	0.003 (0.08)	0.0035 (0.089)	0.0045 (0.114)	0.005 (0.13)
X	(0.002)	0.0035 (0.089)	0.006 (0.015)	0.085 (0.216)	0.0105 (0.267)
Y	0.003 (0.08)	0.005 (0.13)	0.008 (0.20)	0.0105 (0.267)	0.0125 (0.317)
Z	0.003 (0.08)	0.006 (0.15)	0.010 (0.25)	0.0155 (0.394)	0.0190 (0.483)

(Cleveland Twist Drill Co.)

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-21
Recommended Reductions in Speeds and Feeds
for Producing Deep Holes with Twist Drills

Hole Depth (multiple of drill diam)	Reduction, Percent	
	Speed	Feed
3	10	10
4	20	10
5	30	20
6-8	35-40	20
8-30	40-60	20-60
30-80	60-80	60-90

diameter and hole depth. Large drills require more volume, and deeper holes necessitate higher pressure. General recommendations are in the range of 30-60 psi (207-444 kPa) for pressure and 2-6 gpm (7.6-23 L/min) for volume. Some tool manufacturers recommend even higher pressures, to 130 psi (896 kPa).

Flood application of cutting fluid should only be used for short-hole applications with depth-to-diameter ratios of less than 1:1. When cast iron is drilled, compressed air, or preferably a vacuum, is sometimes used instead of a cutting fluid. If air is used, adequate guarding is essential for protection against flying chips.

Spade drills. Cutting fluid for spade drilling is supplied through the drill holder or by flood application. Volume

TABLE 9-22
Recommended Starting Speeds and Feeds
for Drills with Indexable Carbide Inserts

Material Drilled	Drill Diam, in.	Cutting Speed, sfm (m/min)	Feed Rate, ipr (mm/rev)
Cast irons: nodular, ductile, or malleable	13/16 to 1 1/8	165-300 (50-91)	0.004-0.008 (0.10-0.20)
	1 to 1 3/8	165-300 (50-91)	0.005-0.010 (0.13-0.25)
	1 1/4 to 1 5/8	165-300 (50-91)	0.006-0.012 (0.15-0.30)
	1 1/2 to 2 1/2	165-300 (50-91)	0.008-0.014 (0.20-0.36)
	2 3/8 to 3 1/2	165-300 (50-91)	0.010-0.015 (0.25-0.38)
Steels: 1000 series, such as 1018, 1020, etc.	13/16 to 1 1/8	300-400 (91-122)	0.003-0.005 (0.08-0.13)
	1 to 1 3/8	350-450 (107-137)	0.003-0.006 (0.08-0.15)
	1 1/4 to 1 5/8	400-500 (122-168)	0.004-0.007 (0.10-0.18)
	1 1/2 to 2 1/2	450-600 (137-183)	0.004-0.007 (0.10-0.18)
	2 3/8 to 3 1/2	500-700 (152-213)	0.005-0.009 (0.13-0.23)
Steels: case hardened unalloyed, low carbon	13/16 to 1 1/8	200-300 (61-91)	0.003-0.005 (0.08-0.13)
	1 to 1 3/8	250-350 (76-107)	0.004-0.006 (0.10-0.15)
	1 1/4 to 1 5/8	300-425 (91-130)	0.005-0.007 (0.13-0.18)
	1 1/2 to 2 1/2	330-490 (101-149)	0.005-0.008 (0.13-0.20)
	2 3/8 to 3 1/2	350-550 (107-168)	0.006-0.010 (0.15-0.25)
Steels: high carbon, alloyed, heat treated	13/16 to 1 1/8	200-300 (61-91)	0.003-0.005 (0.08-0.13)
	1 to 1 3/8	250-325 (76-99)	0.004-0.006 (0.10-0.15)
	1 1/4 to 1 5/8	300-400 (91-122)	0.005-0.008 (0.13-0.20)
	1 1/2 to 2 1/2	325-450 (99-137)	0.005-0.008 (0.13-0.20)
	2 3/8 to 3 1/2	350-500 (107-152)	0.006-0.010 (0.15-0.25)

OPERATING PARAMETERS FOR DRILLING

TABLE 9-22—Continued

Material Drilled	Drill Diam, in.	Cutting Speed, sfm (m/min)	Feed Rate, ipr (mm/rev)
Steels: high tensile strength	13/16 to 1 1/8	165-250 (50-76)	0.004-0.005 (0.10-0.13)
	1 to 1 3/8	195-300 (59-91)	0.004-0.005 (0.10-0.13)
	1 1/4 to 1 5/8	230-330 (70-101)	0.005-0.007 (0.13-0.18)
	1 1/2 to 2 1/2	265-390 (81-119)	0.006-0.008 (0.15-0.20)
	2 3/8 to 3 1/2	265-425 (81-130)	0.006-0.009 (0.15-0.23)
Stainless steels	13/16 to 1 1/8	230-280 (70-85)	0.003-0.004 (0.08-0.10)
	1 to 1 3/8	265-300 (81-91)	0.004-0.005 (0.10-0.13)
	1 1/4 to 1 5/8	280-345 (85-105)	0.004-0.005 (0.10-0.13)
	1 1/2 to 2 1/2	295-395 (90-120)	0.004-0.005 (0.10-0.13)
	2 3/8 to 3 1/2	300-400 (91-122)	0.004-0.006 (0.10-0.15)
Titanium alloys	13/16 to 1 1/8	100-135 (30-41)	0.003-0.004 (0.08-0.10)
	1 to 1 3/8	100-150 (30-46)	0.004-0.007 (0.10-0.18)
	1 1/4 to 1 5/8	115-165 (35-50)	0.005-0.008 (0.13-0.20)
	1 1/2 to 2 1/2	130-175 (40-53)	0.006-0.009 (0.15-0.23)
	2 3/8 to 3 1/2	135-190 (41-58)	0.006-0.010 (0.15-0.25)

(Waukesha Cutting Tools, Inc.)

requirements vary from about 9 gpm (34 L/min) for small-diameter drills to 80 gpm (303 L/min) for large drills. Pressures generally vary from 20-200 psi (138-1379 kPa). Deeper holes or vertical bores may require higher volumes and pressures.

If the coolant pressure is not adequate to clear chips from the bores, compressed air can be introduced into the coolant system. By regulating the air pressure or occasionally introducing air into the coolant supply, chips can be cleared from the bore to keep the blade clear. The use of air can be employed for horizontal operations, but is especially helpful for vertical spade drilling applications.

High-pressure coolant drills. Coolant pressures recommended by one gundrill manufacturer for producing holes in various materials are presented in Table 9-11. Pressure and volume requirements for multiple-lip external-chip-removal gundrills are given in Fig. 9-101. The two graphs at the top of this illustration are for deep-hole drilling—producing holes having depth-to-diameter ratios greater than 6:1; the two at the bottom are for short-hole drilling—producing holes with depth-to-diameter ratios less than 6:1.

Pressure and volume requirements for multiple-lip internal-chip-removal drills with single or double tubes are presented in Fig. 9-102. The same requirements for multiple-cutter internal-chip-removal trepanning tools are given in Fig. 9-103.

Drill Reconditioning

Avoiding excessive dulling and proper drill reconditioning are important factors influencing drilling economy, accuracy, and quality.

Drill wear. A drill starts to wear as soon as it is placed in operation. This wear is of several types, each of which has an effect upon drill life. Twist drill wear usually starts as a slight rounding on the cutting lip corners. At the same time, the cutting lips, as well as the chisel edge, begin to wear and form a truly conical surface of narrow width adjacent to these edges. This conical surface has no clearance and tends to rub in the hole rather than cut.

The power and thrust required to force these slightly dulled edges to penetrate the material being cut results in increased heat generation, which in turn causes the cutting edges to wear at an accelerated rate. The increase in wear at the cutting lip corners travels back along the margins, resulting in loss in size. This wear is usually accompanied by some degree of material buildup, also found on the margins.

Minimizing wear. Wear on a drill is not proportional to the number of holes or pieces produced between grinds. After the drill wears to a given level, additional wear occurs at an accelerated rate. Excessive margin wear requires additional shortening of the drill to remove the entire worn section,

TABLE 9-23
Speed and Feed Ranges for Spade Drilling Various Materials

Materials Drilled	Drill Diam, in.									
	1		2		3		4		5	
	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)
Aluminum	1000- 1500	0.015-0.025 (0.38-0.63)	550- 750	0.020-0.030 (0.51-0.76)	350- 500	0.030-0.040 (0.76-1.02)	275- 375	0.035-0.050 (0.89-1.27)	225- 300	0.040-0.060 (1.02-1.52)
Brass:										
lead	400- 700	0.015-0.020 (0.38-0.51)	200- 350	0.020-0.025 (0.51-0.63)	125- 250	0.020-0.030 (0.51-0.76)	100- 200	0.025-0.040 (0.63-1.02)	80- 150	0.040-0.060 (1.02-1.52)
soft	250- 350	0.015-0.020 (0.38-0.51)	125- 175	0.020-0.025 (0.51-0.63)	90- 125	0.020-0.030 (0.51-0.76)	70- 85	0.025-0.040 (0.63-1.02)	50- 70	0.040-0.060 (1.02-1.52)
hard	200- 250	0.012-0.015 (0.30-0.38)	95- 125	0.015-0.020 (0.38-0.51)	60- 80	0.020-0.025 (0.51-0.63)	45- 60	0.020-0.030 (0.51-0.76)	37- 50	0.030-0.050 (0.76-1.27)
Bronze:										
soft	300- 375	0.015-0.020 (0.38-0.51)	150- 200	0.020-0.025 (0.51-0.63)	100- 125	0.020-0.030 (0.51-0.76)	75- 95	0.025-0.040 (0.63-1.02)	60- 75	0.040-0.060 (1.02-1.52)
high-strength	225- 300	0.008-0.012 (0.20-0.30)	100- 150	0.010-0.015 (0.25-0.38)	75- 95	0.015-0.020 (0.38-0.51)	55- 70	0.020-0.025 (0.51-0.63)	45- 55	0.030-0.040 (0.76-1.02)
Cast copper	175- 225	0.005-0.008 (0.13-0.20)	85- 125	0.007-0.010 (0.18-0.25)	60- 80	0.010-0.015 (0.25-0.38)	40- 55	0.015-0.020 (0.38-0.51)	35- 45	0.020-0.025 (0.51-0.63)
Cast iron:										
soft	450- 550	0.015-0.020 (0.38-0.51)	225- 300	0.020-0.025 (0.51-0.63)	150- 200	0.020-0.030 (0.51-0.76)	110- 140	0.025-0.030 (0.63-0.76)	90- 120	0.030-0.040 (0.76-1.02)
medium	250- 375	0.008-0.012 (0.20-0.30)	125- 200	0.010-0.015 (0.25-0.38)	90- 125	0.015-0.020 (0.38-0.51)	65- 95	0.020-0.025 (0.51-0.63)	50- 75	0.030-0.040 (0.76-1.02)
hard	150- 225	0.005-0.008 (0.13-0.20)	75- 110	0.008-0.012 (0.20-0.30)	50- 75	0.010-0.015 (0.25-0.38)	35- 55	0.015-0.020 (0.38-0.51)	30- 45	0.020-0.030 (0.51-0.76)
Cast steel	200- 275	0.006-0.010 (0.15-0.25)	95- 125	0.010-0.015 (0.25-0.38)	65- 90	0.015-0.020 (0.38-0.51)	45- 65	0.020-0.025 (0.51-0.63)	35- 55	0.025-0.030 (0.63-0.76)
Stainless:										
201-302	150- 190	0.005-0.008 (0.13-0.20)	75- 95	0.008-0.012 (0.20-0.30)	50- 65	0.010-0.015 (0.25-0.38)	35- 47	0.015-0.020 (0.38-0.51)	30- 37	0.020-0.030 (0.51-0.76)
303 & 303 Se	225- 275	0.006-0.010 (0.15-0.25)	120- 125	0.010-0.015 (0.25-0.38)	75- 90	0.015-0.020 (0.38-0.51)	55- 65	0.020-0.025 (0.51-0.63)	45- 55	0.025-0.030 (0.63-0.76)
304-316	130- 170	0.005-0.007 (0.13-0.18)	65- 85	0.007-0.010 (0.18-0.25)	45- 55	0.010-0.015 (0.25-0.38)	35- 45	0.015-0.020 (0.38-0.51)	25- 35	0.020-0.025 (0.51-0.63)
316F & 316 Se	275- 350	0.006-0.010 (0.15-0.25)	140- 170	0.010-0.015 (0.25-0.38)	95- 110	0.015-0.020 (0.38-0.51)	70- 85	0.020-0.025 (0.51-0.63)	55- 70	0.025-0.030 (0.63-0.76)
317-414	130- 170	0.005-0.007 (0.13-0.18)	65- 85	0.007-0.010 (0.18-0.25)	45- 55	0.010-0.015 (0.25-0.38)	35- 45	0.015-0.020 (0.38-0.51)	25- 35	0.020-0.025 (0.51-0.63)

TABLE 9-23—Continued

Materials Drilled	Drill Diam, in.									
	1		2		3		4		5	
	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)	Speed, rpm	Feed Rate, ipr (mm/rev)
Stainless (cont.):										
416, 430F	300-	0.006-0.010	150-	0.010-0.015	100-	0.015-0.020	75-	0.020-0.025	60-	0.025-0.030
430 Se	375	(0.15-0.25)	190	(0.25-0.38)	125	(0.38-0.51)	95	(0.51-0.63)	75	(0.63-0.76)
420-430	150-	0.005-0.007	75-	0.007-0.010	50-	0.010-0.015	35-	0.015-0.020	30-	0.020-0.025
	225	(0.13-0.18)	110	(0.18-0.25)	75	(0.25-0.38)	55	(0.38-0.51)	45	(0.51-0.63)
431-446	150-	0.006-0.008	75-	0.008-0.010	50-	0.008-0.015	35-	0.010-0.020	30-	0.015-0.025
	190	(0.15-0.20)	95	(0.20-0.25)	65	(0.20-0.38)	47	(0.25-0.51)	37	(0.38-0.63)
Steels:										
1018-1095	190-	0.005-0.008	95-	0.008-0.012	65-	0.010-0.015	47-	0.015-0.020	37-	0.020-0.030
	275	(0.13-0.20)	140	(0.20-0.30)	95	(0.25-0.38)	70	(0.38-0.51)	55	(0.51-0.76)
B1112 & B1113	300-	0.006-0.010	200-	0.010-0.015	125-	0.015-0.020	95-	0.020-0.025	75-	0.025-0.030
	400	(0.15-0.25)	250	(0.25-0.38)	160	(0.38-0.51)	125	(0.51-0.63)	100	(0.63-0.76)
1117-1213	350-	0.005-0.008	170-	0.008-0.012	110-	0.010-0.015	85-	0.015-0.020	70-	0.020-0.030
	450	(0.13-0.20)	225	(0.20-0.30)	150	(0.25-0.38)	110	(0.38-0.51)	90	(0.51-0.76)
1330-3150	250-	0.005-0.007	125-	0.007-0.010	80-	0.010-0.015	60-	0.015-0.020	50-	0.020-0.025
	290	(0.13-0.18)	140	(0.18-0.25)	95	(0.25-0.38)	70	(0.38-0.51)	55	(0.51-0.63)
4017-4140	290-	0.005-0.007	140-	0.007-0.010	95-	0.010-0.015	70-	0.015-0.020	55-	0.020-0.030
	320	(0.13-0.18)	160	(0.18-0.25)	110	(0.25-0.38)	80	(0.38-0.51)	65	(0.51-0.76)
4145-4640	225-	0.005-0.007	120-	0.007-0.010	75-	0.010-0.015	55-	0.015-0.020	45-	0.020-0.030
	300	(0.13-0.18)	150	(0.18-0.25)	100	(0.25-0.38)	75	(0.38-0.51)	60	(0.51-0.76)
5120-8750	250-	0.005-0.007	125-	0.007-0.010	80-	0.010-0.015	60-	0.015-0.020	50-	0.020-0.030
	320	(0.13-0.18)	160	(0.18-0.25)	110	(0.25-0.38)	80	(0.38-0.51)	65	(0.51-0.76)
52100	200-	0.003-0.005	100-	0.005-0.008	70-	0.006-0.010	50-	0.010-0.015	40-	0.015-0.020
	250	(0.08-0.13)	125	(0.13-0.20)	80	(0.15-0.25)	60	(0.25-0.38)	50	(0.38-0.51)
Tool steels	110-	0.004-0.006	55-	0.006-0.008	37-	0.007-0.010	27-	0.010-0.015	22-	0.015-0.020
	150	(0.10-0.15)	75	(0.15-0.20)	50	(0.18-0.25)	35	(0.25-0.38)	30	(0.38-0.51)
Titanium alloys	110	0.006-0.010	55-	0.010-0.015	37-	0.015-0.020	27-	0.020-0.025	22-	0.025-0.030
	130	(0.15-0.25)	65	(0.25-0.38)	45	(0.38-0.51)	33	(0.51-0.63)	27	(0.63-0.76)
Zinc alloys	750-	0.008-0.012	375	0.010-0.015	250-	0.012-0.017	190-	0.015-0.025	150-	0.025-0.030
	900	(0.20-0.30)	475	(0.25-0.38)	320	(0.30-0.43)	230	(0.38-0.63)	190	(0.63-0.76)

(Madison Industries)

OPERATING PARAMETERS FOR DRILLING

TABLE 9-24
Recommended Speeds and Feeds for Multiple-Lip,
Internal-Chip-Removal Gundrills

Material Drilled	Hardness, Brinell	Cutting Speed, sfm (m/min)	Feed Rate, ipr (mm/rev)
Steels, AISI series:			
1000	150-225	250-300 (76-91)	0.004-0.015 (0.10-0.38)
1100	175-250	250-300 (76-91)	0.004-0.015 (0.10-0.38)
1300	175-250	250-300 (76-91)	0.004-0.015 (0.10-0.38)
2000	135-275	250-300 (76-91)	0.004-0.015 (0.10-0.38)
3000	150-250	250-300 (76-91)	0.004-0.012 (0.10-0.30)
4000	150-225	150-300 (46-91)	0.004-0.012 (0.10-0.30)
5000	150-250	150-300 (46-91)	0.004-0.012 (0.10-0.30)
8000, 9000	175-275	150-300 (46-91)	0.004-0.012 (0.10-0.30)
Stainless steels:			
300 series	135-275	150-300 (46-91)	0.004-0.012 (0.10-0.30)
400 series	135-225	150-300 (46-91)	0.004-0.012 (0.10-0.30)
Cast iron	140-220	150-300 (46-91)	0.004-0.012 (0.10-0.30)
Malleable iron	135-250	250-400 (76-122)	0.006-0.020 (0.15-0.51)
Brass and bronze	75-240	250-400 (76-122)	0.006-0.020 (0.15-0.51)
Aluminum alloys		300-600 (91-183)	0.006-0.020 (0.15-0.51)
Monel metals	200-300	150-300 (46-91)	0.002-0.010 (0.05-0.25)

(Sandvik Inc.)

resulting in fewer holes per drill. Removal of a worn portion by cutting off the drill end can be eliminated when the drills are resharpened in time.

One approach to minimizing excessive wear is to set up a resharpening program to recondition a drill after it has been run a given length of time or has produced a given number of holes. As an example, under certain conditions a 1" (25 mm) diam drill may be able to drill 1000 holes before it is completely dull. At this time, the margins may be worn undersize for a distance of 1" back from the point and the drill must be shortened this amount.

If the drill had been removed from the machine after 750 holes, however, this wear might have been only 1/3" (8.5 mm) and the drill would only need shortening by this amount. In this case, the user could produce 2250 holes per inch of drill.

Three separate steps are required to recondition a dull or worn drill. These are: (1) removal of the worn section, (2) thinning the web, and (3) regrounding the point surfaces.

Removal of worn section. The first principle of proper reconditioning is to remove all of the worn section and the greatest majority of margin buildup. Failure to remove the entire worn section will result in a drill which will not cut

properly due to loss of back-taper and will cause a reduction in tool life. Margin buildup causes binding between the margins and the ID of the hole.

There are no specific drill-life criteria. Three wear areas—lip, chisel edge, and margin—can be observed on twist drills, independently or in combination (see Fig. 9-104), to establish a drill-life criterion. Variations in drill life, however, can be large because of the wide range of operating conditions and differences in the accuracy with which the drills are ground. Generally, the maximum wear land on the lip is about 0.010" (0.25 mm). In order to remove worn, chipped, and burned portions on the ends of twist drills, the minimum recommended amounts to be removed are given in Table 9-27.

Regrounding drill points. After removing the worn portion of the twist drill, it is necessary to reground the surfaces of the point. These two approximately conical surfaces intersect with the faces of the flutes to form the cutting lips. They also intersect with each other to form the chisel edge.

The surface back of these cutting lips must not rub on the work, but must be relieved to permit the edge to penetrate. Without this relief, the drill could not penetrate the metal.

The clearance on the cutting lips of a drill is called lip relief angle and varies with drill size and feed rate. Smaller drills require a larger lip angle, while larger drills utilize smaller angles (see Table 9-8) at the normal feed rates used. In order to maintain maximum strength of the cutting edge, the lip-relief angle should be as small as practical. For maximum drill life, however, the angle should be as high as possible while adequate cutting edge strength is maintained. The optimum angle will also vary with the application. For example, smaller angles may be required to control chatter in some operations.

Lip relief angles between the two cutting edges of ground twist drills may occasionally vary. These variations can cause the side with the larger relief angle to wear faster and to produce a nonconcentric drill loading which increases with drill wear or time.

Point angles for producing holes in various materials are discussed earlier in this chapter under the subject of twist drill geometry. The two cutting lips of a drill should be accurately ground. Regardless of the point angle, the angles of the two cutting lips A_1 and A_2 (Fig. 9-105, *a*) must be equal. Similarly, the length of the two lips L_1 and L_2 should be equal. The point should be symmetrical, and the difference in relative lip height should be kept to an absolute minimum.

The error in centrality of web (Fig. 9-105, *b*) and relative lip height (views *b* and *c*) may cause a decrease in drill performance. The acceptable tolerances in web centrality and relative lip height are presented in Tables 9-28 and 9-29, respectively.

If lips are not ground at the same angle with the drill axis, the drill will be subjected to an abnormal torsional strain because only one lip comes in contact with the workpiece (see Fig. 9-106, *a*). This will cause the drill to dull quickly, may result in breakage, and may produce oversized holes. When the point is ground with equal angles, but has lips of different lengths, the condition shown in Fig. 9-106, *b*, is produced. A drill having cutting lips of different angles and unequal lengths will cause severe operating conditions (Fig. 9-106, *c*).

When several drills of different lengths are used to complete a hole, each successive drill should be ground with a slightly flatter point angle (see Fig. 9-107).

Lengths of the points on twist drills for various point angles can be calculated from the formula given in Table 9-30.

Web thinning. The need for web thinning (on most standard

OPERATING PARAMETERS FOR DRILLING

TABLE 9-25
Recommended Speeds and Feeds for Multiple-Lip, External-Chip-Removal, High-Pressure Coolant Drills

Material Drilled	Cutting Speed, sfm (mm/min)	Drill diam, in. (mm)		
		1/64 to 1/4 (4-6.3)	1/4 to 1/2 (6.3-12.5)	1/2 to 7/8 (12.5-20)
		Feed Rate, ipr (mm/rev)		
Steels:				
carbon	262-410 (80-125)	0.0004-0.0016 (0.010-0.041)	0.0008-0.005 (0.020-0.13)	0.002-0.008 (0.06-0.20)
alloyed	230-328 (70-100)	0.0004-0.0016 (0.010-0.041)	0.0008-0.004 (0.020-0.10)	0.002-0.008 (0.06-0.20)
case hardened	164-262 (50-80)	0.0008-0.0016 (0.020-0.041)	0.0008-0.002 (0.020-0.05)	0.002-0.008 (0.06-0.20)
tool	131-207 (40-63)	0.0004-0.0016 (0.010-0.041)	0.0008-0.002 (0.020-0.05)	0.002-0.008 (0.06-0.20)
stainless:				
ferritic	164-295 (50-90)	0.0004-0.0016 (0.010-0.041)	0.0008-0.004 (0.020-0.10)	0.0016-0.008 (0.041-0.20)
austenitic	164-410 (50-125)	0.0008-0.0016 (0.020-0.041)	0.0008-0.004 (0.020-0.10)	0.0016-0.008 (0.041-0.20)
Cast irons:				
unalloyed and alloyed: less than 250 Bhn	207-328 (63-100)	0.004-0.008 (0.10-0.20)	0.004-0.009 (0.10-0.24)	0.005-0.016 (0.12-0.40)
250-450 Bhn	102-207 (31-63)	0.004-0.008 (0.10-0.20)	0.004-0.009 (0.10-0.24)	0.005-0.016 (0.12-0.40)
malleable:				
less than 220 Bhn	207-328 (63-100)	0.0004-0.008 (0.010-0.20)	0.0016-0.009 (0.041-0.24)	0.005-0.016 (0.12-0.40)
220-320 Bhn	102-207 (31-63)	0.0004-0.008 (0.010-0.20)	0.0008-0.006 (0.020-0.16)	0.004-0.009 (0.10-0.24)
Copper	207-328 (63-100)	0.0004 (0.010)	0.009 (0.24)	0.006-0.024 (0.16-0.62)
Brass	230-427 (70-130)	0.0016 (0.041)	0.009 (0.24)	0.004-0.009 (0.10-0.24)
Aluminum alloys:				
not hard	410-656 (125-200)	0.0004 (0.010)	0.0008-0.012 (0.020-0.30)	0.004-0.020 (0.10-0.50)
cast	262-525 (80-160)	0.0004 (0.010)	0.0008-0.012 (0.020-0.30)	0.004-0.020 (0.10-0.50)

(Sandvik Inc.)

twist drills on which the web thickness increases toward their shanks) as the drills are shortened by removal of their worn sections is discussed in a previous section of this chapter under the subject of twist drill geometries. This discussion includes data on web thicknesses (as a percentage of drill diameters) and on types of web thinning to reduce thrust requirements and heat generation, as well as to increase drill life.

Thinning of webs can be done in a special web-thinning grinder, in a conventional tool-and-cutter grinder, or as a freehand grinding operation. Machine grinding is preferred over hand grinding because of its greater accuracy and reliability. A brief discussion of web thinning grinders is presented in Chapter 11 of this volume, "Grinding." However, regardless of the type of web thinning employed, it is important that it be done evenly with the same amount of stock removed from each cutting edge to ensure web centrality. The grinding wheel should be soft enough to remove this stock without danger of burning the cutting edges. It should always be remembered that chips must form on the drill lips and flow into

the flutes and that the shape of the thinning should be such that it does not interfere with the chip flow.

Machines for grinding drills. Some drills, particularly smaller sizes, are reground by hand (called offhand grinding). This requires considerable experience and skill and never produces consistent accuracy. As a result, machine grinding is recommended whenever practical.

Many grinding machines and fixtures designed specifically for regrounding drills are commercially available. Some of these are discussed in Chapter 11, "Grinding." Special machines with automatic grinding cycles, however, are required to produce helical (spiral) and rounded-edge (radiused lip) points, as well as combination helical/rounded edge points. Some of these machines are cam controlled to produce simultaneous three-axis motion at the drill point for precise generative grinding.

Precision machines and sharp wheels should be used for drill grinding to maintain the required geometry and precision. Some drill grinding machines require the use of cutting fluid; others do not. When cutting fluid is used, an ample and

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

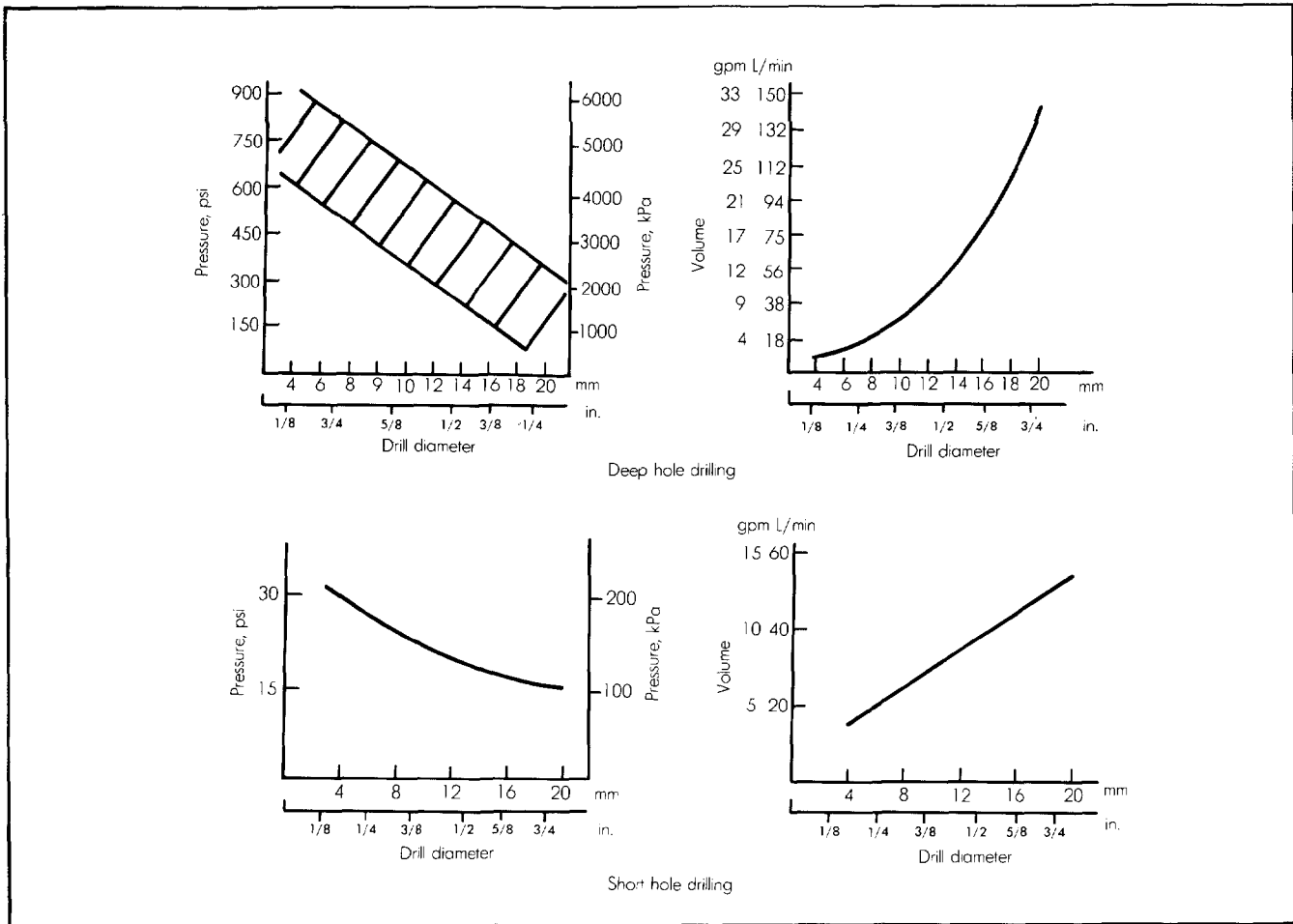


Fig. 9-101 Pressures and volumes for multiple-lip external-chip-removal gundrills. (Sandvik Inc.)

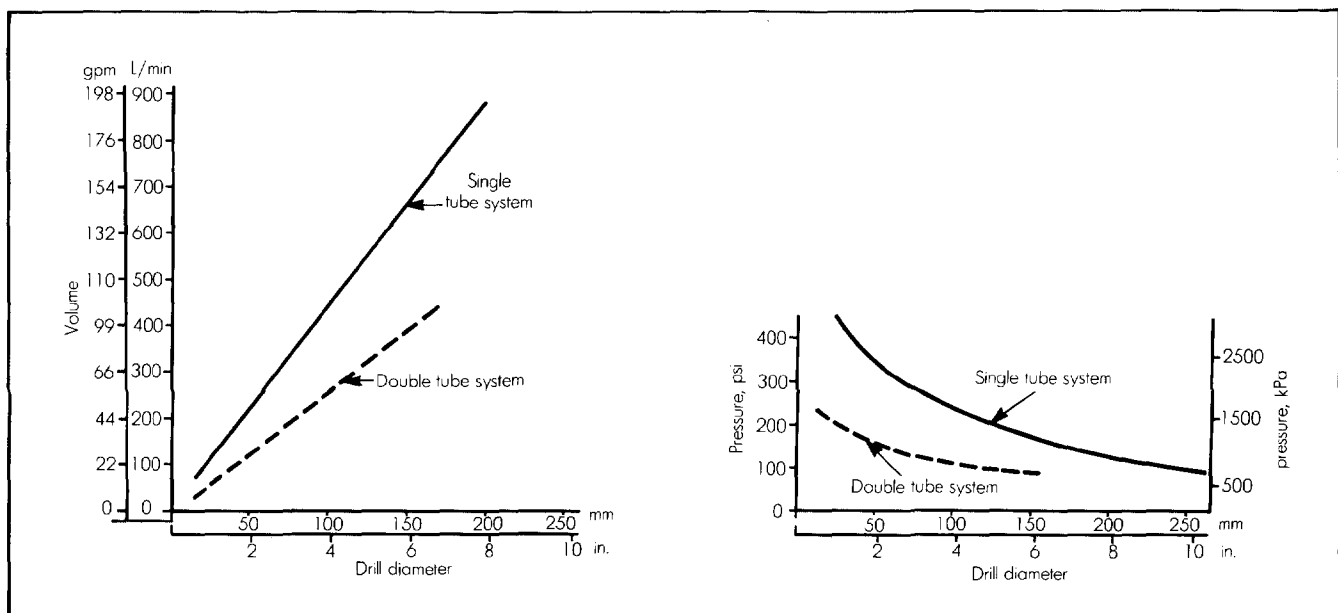


Fig. 9-102 Pressures and volumes for multiple-lip internal-chip-removal drills with single or double tubes. (Sandvik Inc.)

OPERATING PARAMETERS FOR DRILLING

TABLE 9-26
Recommended Speeds and Feeds for Trepanning Tools
with Internal Chip Removal

Tool diam, in.	Speed, rpm, to Provide a Cutting Speed of 300 sfm (91 m/min)*	Feed Rate, ipr (mm/rev)
1.75	650	0.003-0.005 (0.08-0.13)
2.00	575	0.003-0.005 (0.08-0.13)
2.50	460	0.004-0.005 (0.10-0.13)
3.00	380	0.004-0.006 (0.10-0.15)
3.50	325	0.005-0.007 (0.13-0.18)
4.00	285	0.005-0.007 (0.13-0.18)
4.50	250	0.005-0.007 (0.13-0.18)
5.00	230	0.005-0.007 (0.13-0.18)
5.50	210	0.005-0.007 (0.13-0.18)
6.00	180	0.005-0.007 (0.13-0.18)
6.50	170	0.0055-0.0075 (0.140-0.190)
7.00	160	0.006-0.008 (0.15-0.20)
7.50	150	0.006-0.008 (0.15-0.20)
8.00	140	0.006-0.008 (0.15-0.20)
8.50	132	0.006-0.0085 (0.15-0.216)
9.00	126	0.006-0.009 (0.15-0.23)
9.50	120	0.006-0.009 (0.15-0.23)
10.00	112	0.006-0.009 (0.15-0.23)
10.50	108	0.006-0.009 (0.15-0.23)
11.00	102	0.006-0.009 (0.15-0.23)
11.50	100	0.006-0.009 (0.15-0.23)
12.00	95	0.006-0.010 (0.15-0.25)
12.50	90	0.006-0.010 (0.15-0.25)
13.00	88	0.006-0.010 (0.15-0.25)
13.50	85	0.006-0.010 (0.15-0.25)

(American Heller Corp.)

* When drilling the following materials, the cutting speed should be changed as follows:

- Cold-rolled and mild steels, increase speed 10%.
- Heat-treated steels, R_C 30-35, decrease speed 5%.
- Heat-treated steels, R_C 35-40, decrease speed 10-15%.
- Stainless steels, 400 series, decrease speed 10%.
- Stainless steels, 300 series, decrease speed 15%.
- Exotic metal alloys, decrease speed 30-50%.

continuous flow should be directed at the tool/wheel interface. Metallurgical damage from grinding must be avoided. Lack of proper control can produce surface damage of high-speed steel drills and can result in poor tool life as well as in tool failure. Overheating is a prime cause for damage to the cutting edges. If the surfaces of high-speed steel drills are discolored as the result of grinding, the steel is annealed to a depth of about 0.0002" (0.005 mm), resulting in a reduction of hardness. Drills heated by grinding should be cooled in air and never dipped in a liquid, which can cause cracking.

Grinding wheels used. Vitrified-bond, aluminum oxide wheels of medium hardness (I to P grades) and structure (No. 5), with grit sizes of 36-100, are widely used for machine grinding of high-speed steel drills. For carbide drills, diamond wheels (120-180 grit size) or vitrified-bond, silicon-carbide wheels (70-100 grit size, I to L grade, and 8-12 structure) are commonly employed.

Diamond wheels should be used with a cutting fluid if at all

possible. A stream or mist coolant is better than a felt wick. A very satisfactory all-round cutting fluid that keeps the wheel clean and the work cool as well as gives a good finish is a thin solution of soluble oil and water (about 80:1). The diamond resinoid wheel should be dressed occasionally with a piece of lump pumice or a very fine-grit and soft-grade silicon-carbide stick.

Special care is required when grinding carbide tipped drills because the carbide tip and steel shank should be the same diameter and have the same back-taper. The grinding wheel tends to remove more steel than carbide, which can result in the drill's having two diameters and can cause seizing in the drill bushing.

Regrinding spade drill blades. Regrinding the blade of a spade drill is relatively easy when only the clearance face is to be reground. No special machines are necessary to regrind spade drill blades; most regrinding is done on tool and cutter grinders or surface grinders. Fixtures are available from spade drill manufacturers for regrinding operations such as face or end clearance grinding, hook or lip face grinding, and chipbreaker groove grinding. Blades ground with a cylindrical chip curler must have this surface reground at each sharpening to preserve edge geometry. Those ground with a flat top rake need to have this surface reground only after 1/32 to 1/16" (0.8 to 1.6 mm) has been removed from the clearance.

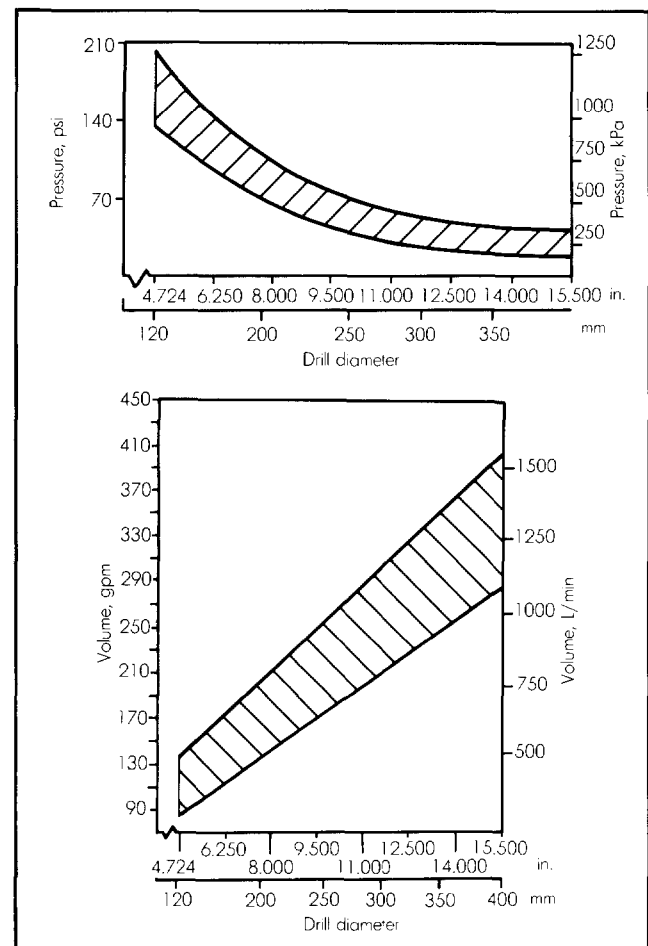


Fig. 9-103 Pressures and volumes for multiple-cutter internal-chip-removal trepanning tools. (Sandvik Inc.)

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-27
Recommended Minimum Amount to be
Removed from Twist Drills
Before Resharpener

Drill diam, in.	Minimum Amount of Drill to be Removed, in. (mm)
0.062-0.125	0.040 (1.02)
0.125-0.219	0.050 (1.27)
0.200-0.562	0.060 (1.52)
0.563 and over	0.100 (2.54)

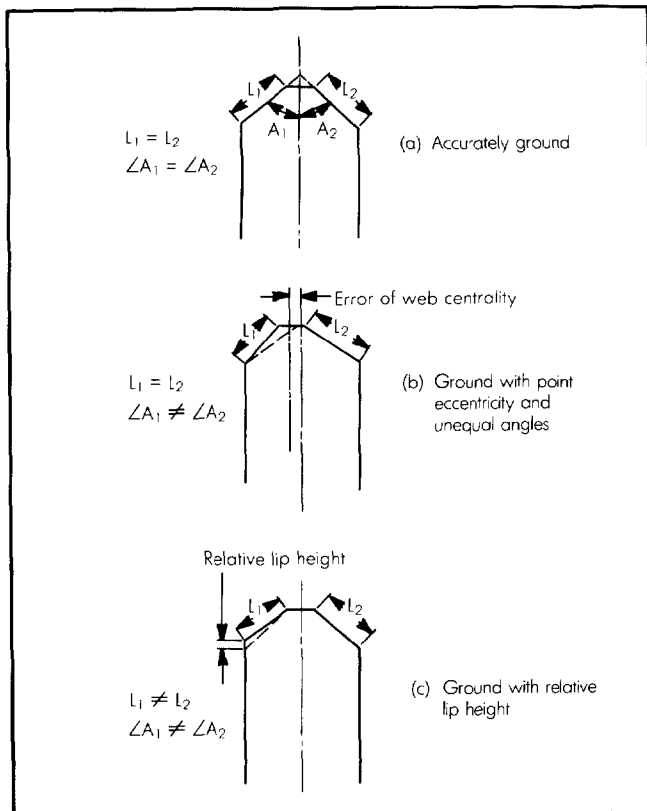


Fig. 9-105 Accurate and inaccurate grinding of points on twist drills.

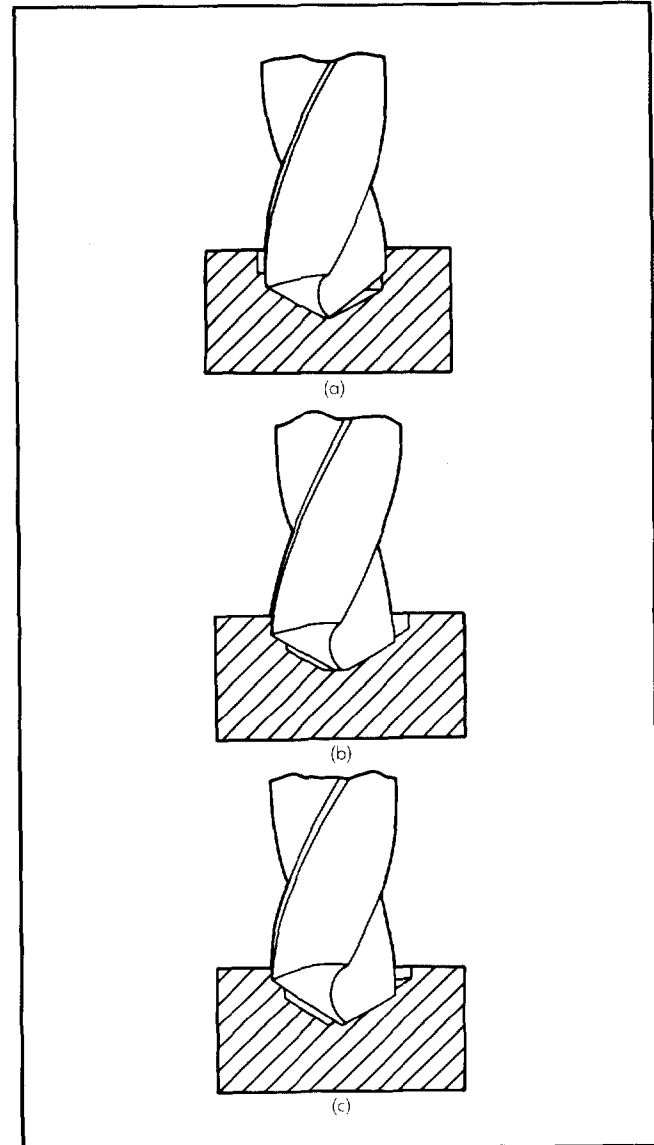


Fig. 9-106 Cutting conditions caused by drills having: (a) lips not ground at the same angle, (b) lips of different length, and (c) lips of different angles and unequal lengths. (National Twist Drill)

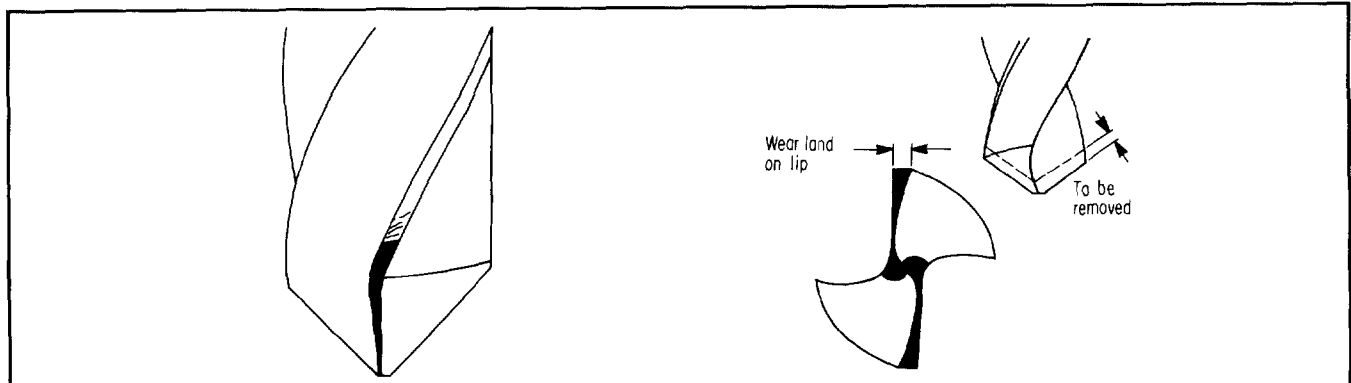


Fig. 9-104 Wear on lip, chisel edge, and margin of a twist drill.

OPERATING PARAMETERS FOR DRILLING

TABLE 9-28
Tolerance Ranges for Web Centrality of Twist Drills*

Drill Diam Range, in.	Tolerance Range for Web Centrality, Total Indicator Variation, in. (mm)
1/16 to 1/8 (0.0625-0.1250)	0.002-0.003 (0.05-0.08)
No. 30 to 1/4 (0.1285-0.2500)	0.003-0.005 (0.08-0.13)
F to 1/2 (0.2570-0.5000)	0.004-0.007 (0.10-0.18)
33/64 to 3/4 (0.5156-0.7500)	0.005-0.009 (0.13-0.23)
49/64 to 1 (0.7656-1.000)	0.006-0.012 (0.15-0.30)
1 1/64 to 1 1/2 (1.0156-1.500)	0.008-0.015 (0.20-0.38)
1 33/64 to 3 1/2 (1.5156-3.500)	0.010-0.018 (0.25-0.46)

* Method of measurement: Rotate the drill in a close-fitting bushing. Record the difference in indicator readings of the web at the point as the drill is indexed 180°.

Suggested dimensions for grinding blades made by one manufacturer of spade drills are presented in Table 9-31. These dimensions have been found compatible with a wide variety of materials and conditions, but some modifications may be necessary for certain applications. For example, if the machine lacks sufficient thrust to allow suitable feeds on large spade drill blades, thrust requirements can be reduced by increasing the clearance angle and/or relieving the point. Clearance angles above 12°, however, create a weak cutting edge and should only be used on very soft metals. Point thinning is usually limited to two-thirds of the original point width and never more than one half.

When very ductile materials are drilled, chips sometimes tend to pack into the chip splitter notches. This problem can sometimes be overcome by regrounding the notches to a shape that may resist packing. Variations of rake angle are also commonly used on spade drills. When very tough or hard materials are drilled, a flat rake angle is sometimes found to improve tool life due to reduced chipping and heat damage to the cutting edge. The deepest portion of the rake angle should never exceed two-thirds of the land width.

When the blades of spade drills are subject to heat damage, the corners of the blades are often clipped by chamfering. This results in thinner chips being produced in the corner areas and in the heat generated being spread over a larger area. The corners are usually ground to an angle of 45°. The chamfer should be kept at least 3/32" (2.4 mm) from the nearest chip

TABLE 9-29
Tolerance Ranges for Relative Lip Heights of Twist Drills*

Drill Diam Range, in.	Tolerance Range for Relative Lip Height, Total Indicator Variation, in. (mm)
1/16 to 1/8 (0.0625-0.1250)	0.001-0.003 (0.03-0.08)
No. 30 to 1/4 (0.1285-0.2500)	0.002-0.004 (0.05-0.10)
F to 1/2 (0.2570-0.5000)	0.002-0.005 (0.05-0.13)
33/64 to 3/4 (0.5156-0.7500)	0.003-0.006 (0.08-0.15)
49/64 to 1 (0.7656-1.000)	0.003-0.007 (0.08-0.18)
1 1/64 to 1 1/2 (1.0156-1.500)	0.005-0.008 (0.13-0.20)
1 33/64 to 3 1/2 (1.5156-3.500)	0.005-0.009 (0.13-0.23)

* Method of measurement: Rotate the drill in a V-block against a back end stop. Measure the cutting lip height variation on a comparator or with an indicator set at a location about 75% of the distance from the center to the periphery of the drill.

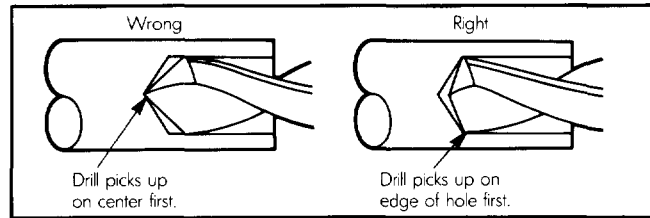


Fig. 9-107 When using more than one drill to complete a hole, successive drills should have a slightly flatter point angle. (Republic Steel Corp.)

splitter notch or should be made wide enough to remove the notch. No chip splitter notches should be used on the corners unless their widths exceed about 1/2" (12.7 mm).

Regrounding high-pressure coolant drills. With a suitable fixture, conventional nose geometries can be ground on gundrills using conventional tool and cutter grinders or surface grinding machines. In regrounding, only enough carbide is removed to eliminate any evidence of wear. This is typically 0.005" (0.13 mm) for small tools and 0.010" (0.25 mm) for larger tools, providing about 100 sharpenings per gundrill.

The amount of stock removed from a gundrill per regrind varies, of course, with the material drilled, hole size, and operating parameters. For the workpiece materials indicated in Table 9-11, an average of 1500" (38 100 mm) per grind can be assumed, giving 150,000" (3 810 000 mm) per tool. When tool layouts are made, allowances must be made for lost lengths due to resharpenings. This is typically 3/8 to 1/2" (9.5 to 12.7 mm) for small tools and 3/4 to 1 (19 to 25 mm) for larger tools.

Resharpening of internal-chip-removal gundrills is more critical, requiring close attention to the many details shown in

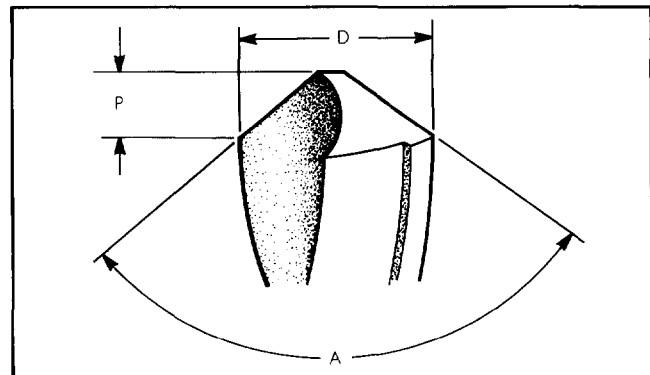


TABLE 9-30
Point Lengths for Various Point Angles on Twist Drills

Included Angle of Drill Point, A, degrees	C
90	0.5
118	0.3
125	0.26
130	0.23
135	0.21
140	0.18
145	0.16
150	0.13

(La Salle Steel Co.)

OPERATING PARAMETERS FOR DRILLING

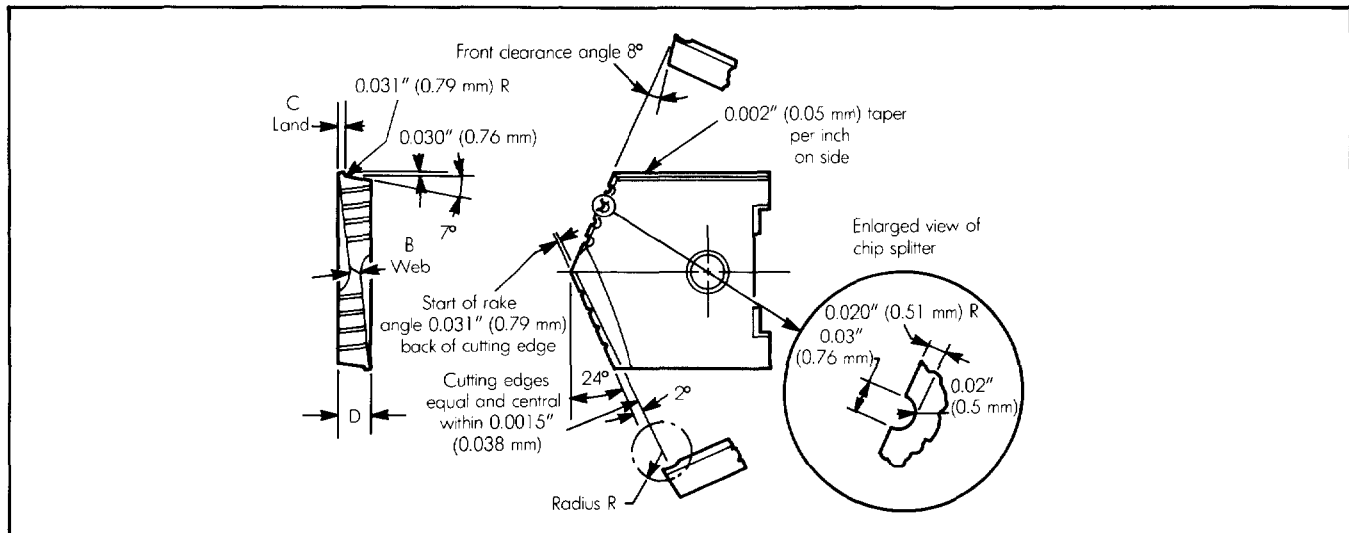


TABLE 9-31
Grinding Dimensions for Spade Drill Blades

Spade Drill Diam Range, in.	Radius R, in. (mm)	Web Thickness B, in. (mm)	Land Width, C, in. (mm)	Blade Width D, in. (mm)
1.0000-1.2500	0.25 (6.3)	0.06 (1.5)	0.06 (1.5)	0.25 (6.3)
1.2501-1.5000	0.31 (7.9)	0.06 (1.5)	0.06 (1.5)	0.28 (7.1)
1.5001-2.0000	0.31 (7.9)	0.08 (2.0)	0.06 (1.5)	0.31 (7.9)
2.0001-2.5000	0.38 (9.7)	0.11 (2.8)	0.09 (2.3)	0.38 (9.7)
2.5001-3.0000	0.38 (9.7)	0.12 (3.0)	0.09 (2.3)	0.44 (11.2)
3.0001-3.5000	0.38 (9.7)	0.16 (4.1)	0.12 (3.0)	0.50 (12.7)
3.5001-4.0000	0.38 (9.7)	0.19 (4.8)	0.12 (3.0)	0.62 (15.7)
4.0001-6.0000	0.50 (12.7)	0.22 (5.6)	0.12 (3.0)	0.69 (17.5)

(Madison Industries)

Fig. 9-108. A single overlarge chip can jam the chip mouth and throat of these tools. Drill life between resharpenings depends largely upon the condition of the chipbreakers, and the reported life of these tools is usually five to seven resharpenings.

Drilling Applications

With the wide variety of materials in which holes must be produced, there are many challenges to the use of drills. There is a practical limit of material hardness and strength beyond which drills are not practical for production applications.

Minimizing chatter. As with all other machining operations, chatter can cause poor performance in drilling. Excessive chatter causes rapid deterioration of drills and reduces hole accuracy and quality.

Chatter can be defined as synchronized vibrations that are set up in the tool, workpiece, machine or as a combination of vibrations in all these elements. The cause is usually lack of rigidity. Chatter can be minimized by using:

1. Rigid machines in good condition having accurate spindles and no worn bearings or loose moving parts.
2. Substantial fixtures that securely clamp and adequately support the workpieces.
3. Properly designed drills. Drill chatter is usually caused by torsional deflection—the longer the drill in proportion to its diameter, the greater the danger of deflection and chatter.

Drilling hard materials. Holes can be drilled in limited quantities in many hard, high-strength materials with certain procedures. Workpieces must be held rigidly in place and be well supported directly under the drill point. Short drills with their webs not thinned excessively and medium lip clearance angles (7-9°) are recommended. Machines used must have ample power and be in good condition. Cutting speeds should be low (20-30 sfm; 6.1-9.1 m/min or less), with medium to heavy feed rates, preferably using a mechanical power feed unit.

Starting holes with carbide drills. Special care should be given to starting holes in any material when using carbide and carbide-tipped drills. Little difficulty is encountered in starting and getting straight holes with these drills when close-fitting, well-aligned guide bushings are used. If bushings are not used, however, guide holes should be produced first, using stub (screw) length drills having a 135-140°, four-facet or split, self-centering point. This allows the OD of the following carbide drill to become secured before cutting starts and can extend tool life.

Drilling stainless steels. Most stainless steels, except the free-machining grades, are generally more difficult to drill than carbon steels and most alloy steels. Work-hardening grades in particular should be drilled continuously with a fairly heavy, uniform feed and moderate speed. Accurate sharpening, with an included point angle of 135-140°, a clearance angle of 6-8°, on larger size drills, and smooth finishes on the cutting edges, is recommended.

OPERATING PARAMETERS FOR DRILLING

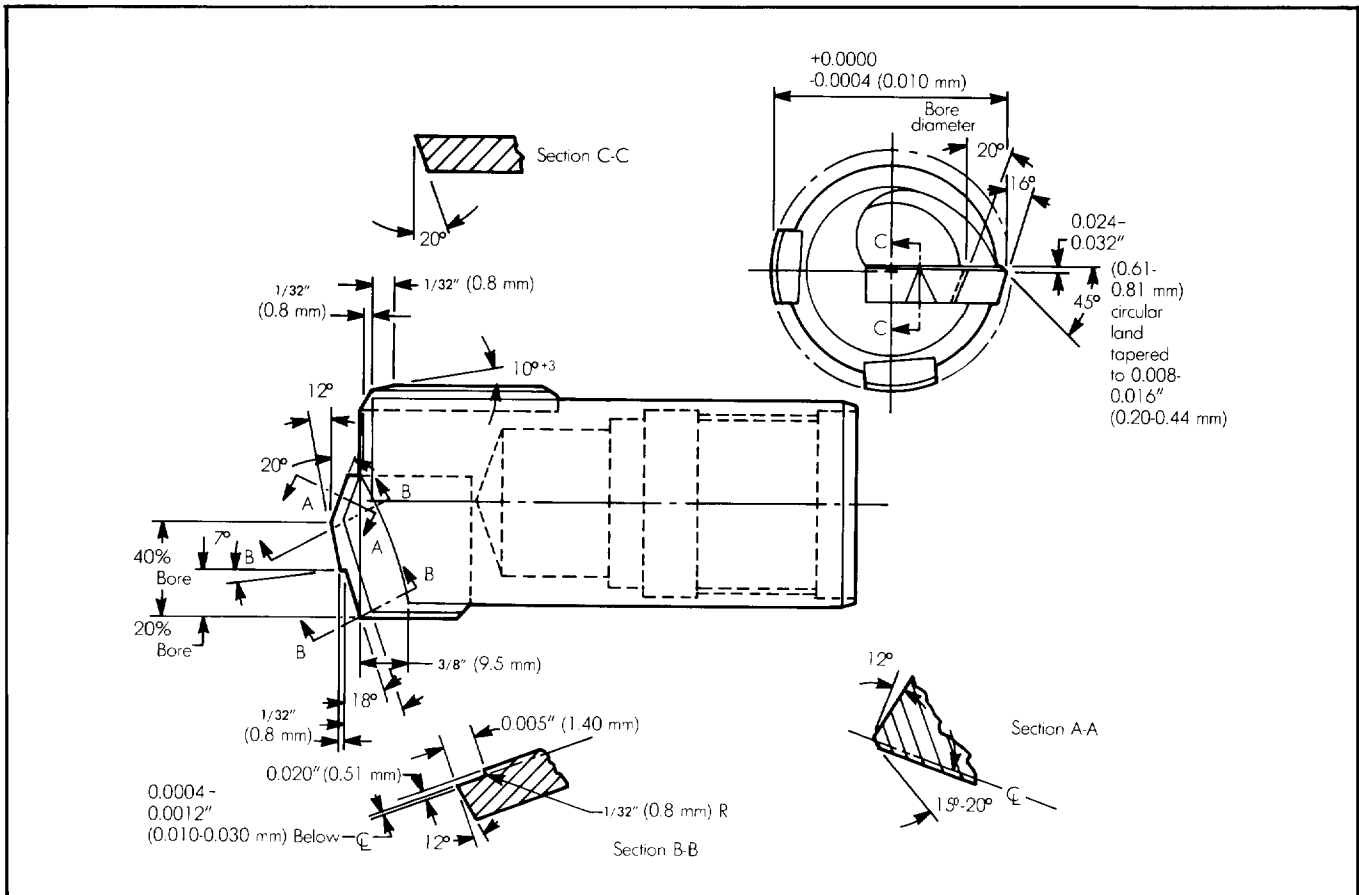


Fig. 9-108 Critical angles and dimensions in resharpening an internal-chip-removal gundrill. (American Heller Corp.)

Drilling armor plate. While this material is hard and tough, it can generally be drilled satisfactorily with standard HSS drills. If the material has been flame cut, areas adjoining the cut are hardened and drilling is difficult. Annealing of these areas is recommended, if possible, before drilling. When this is not practical, low cutting speeds may be necessary. A positive power feed at a medium to heavy, uniform rate is recommended for continuous cutting to prevent work hardening.

Twist drills for producing holes in armor plate should be sharpened with an included point angle of 135-140°, a clearance angle of 6-8° at the periphery, and a chisel point angle of 115-125°. Cutting edges should have as smooth a surface finish as possible and should be free from any grinding wheel burn.

Drilling copper. Problems encountered in drilling copper and its alloys are the long, stringy chips produced because of the gummy nature of the material and the tendency for drills to bind or freeze in the workpieces because of the high coefficient of expansion of the material. These problems can be minimized by using drills of proper design, slower cutting speeds, and higher feed rates. Drills with thin webs, narrow margin widths, increased back-taper, and high relief angles on their points reduce the heat generated in drilling.

Drilling aluminum. Drilling of aluminum and aluminum alloys seldom presents any problems, with the possible exception of some cast alloys and those having a high silicon content. Aluminum alloys having a high silicon content are very abrasive, resulting in rapid tool wear. Soft alloys require drills

having polished flutes to prevent chip packing and material buildup. For deeper holes, twist drills with high helix angles are recommended to minimize chip packing. Thin webs and high lip relief angles should be provided on the drills, and high penetration rates are possible in drilling these materials.

Drilling magnesium and zinc alloys. Fairly high speeds and heavy feeds should be used to drill magnesium and its alloys. These parameters produce large, thick chips, which reduces the fire hazard. The drills must have ample chip space to accommodate the high penetration rates.

Drilling of zinc alloys can cause clogging of the drill flutes because the materials are soft and gummy. Their relatively low melting point and the heat generated in drilling them can also result in welding of the material to the drill flutes. These problems can be minimized by using drills with wide polished flutes, high lip relief angles, thin webs, and narrow margins. Drilling speeds should be conservative and feeds determined by the size and shape of the chips produced and the ease with which they can be ejected.

Drilling titanium. Problems in drilling titanium and titanium alloys can result from the long, thin, curly chips produced. These chips, plus the poor heat conduction of the materials and the heat generated in drilling, tend to cause excessive chip packing and welding to the cutting edges. Successful drilling requires a reduction in the amount of heat generated by using slower speeds, moderate feeds, adequate cutting fluid, sharp and sturdy drills, well-supported workpieces, and rigid machines.

CHAPTER 9

OPERATING PARAMETERS FOR DRILLING

TABLE 9-32
Problems, Causes, and Solutions When Using Twist Drills

Problem	Probable Causes	Possible Solutions
Oversize holes	Dull drill or unequal lip angles and/or lengths Excessive thrust Misalignment Vibration Loose spindle on drilling machine Excessive heat	Regrind or replace drill. Reduce feed rate and/or drill length. Realign setup and/or replace worn bushing. Secure setup. Repair or change machine. Increase drill clearance angle. Use cutting fluid, making sure it reaches drill point. Improve cutting fluid being used. Use coolant-fed drill. Withdraw drill periodically to clear chips.
Hole not straight	Misalignment	Realign and secure fixture. Use drill bushing. Use shorter or self-centering drill to start hole. Repair or change drilling machine. Reduce feed rate. Change speed.
Rough finish in hole	Dull drill or incorrect geometry Excessive thrust Excessive heat Vibration/chatter	Regrind or replace drill. Reduce feed rate. Increase drill clearance angle. Reduce feed rate and/or speed. Increase drill clearance angle. Use cutting fluid, making sure it reaches drill point. Improve cutting fluid being used. Use coolant-fed drill. Secure setup. Repair or change machine.
Burr at entry	Dull drill	Regrind or replace drill.
Burr at exit	Too high a helix angle or too low an included point angle	Change drill or regrind. Use of a backing material is the most effective method of eliminating burrs at breakthrough. Two-diameter step drills usually minimize burring.
Drill will not penetrate workpiece	Dull drill Web too thick Insufficient lip clearance Work too hard	Regrind or replace drill. Thin web or use drill with split or crankshaft point. Regrind or replace drill. Use cutting fluid, making sure it reaches drill point. Improve cutting fluid being used. Improve drill material.
Excessive drill wear, breakdown of outer corners, and/or chipping of cutting lips	Excessive speed or feed Excessive heat Flute packing Workpiece material has hard spots or poor machinability Vibration/chatter	Reduce speed first, then feed rate if required. Use cutting fluid, making sure it reaches drill point. Improve cutting fluid being used. Use coolant-fed drill. Improve alignment and bushing fit. Change drill helix angle. Use chipbreaker drill. Withdraw drill from hole periodically. Use coolant-fed drill. Improve drill material. Increase included angle of drill point. Secure setup. Reduce drill length. Repair or change machine.
Broken drills	Misalignment Dull drill or incorrect geometry Worn drill margins Flute packing Excessive feed Loss of feed control at breakthrough Drill slippage in holder Drill or flutes too long Incorrect speed	Realign and secure fixture. Use drill bushing. Regrind or replace drill. Replace drill. Change drill helix angle. Use chipbreaker drill. Polish drill flutes. Withdraw drill from hole periodically. Use coolant-fed drill. Reduce feed rate. Repair or change drilling machine. Secure drill or change holder. Reduce drill or flute length. Increase or decrease speed.

OPERATING PARAMETERS FOR DRILLING

TABLE 9-33
Troubleshooting Chart for Indexable-Insert Drills

Trouble	Source	Solution
Insert chipping or breakage*	Off-center drill due to misalignment	Maintain proper alignment, concentricity, and angularity.
	Improper seating of tool in toolholder or spindle	Check tool shank and socket for nicks and dirt. Check parting line between tool shank and socket with feeler gage. Check if tool is locked tight.
	Deflection due to too much overhang and lack of rigidity	Check with indicator if tool can be moved by hand. Check if tool can be held shorter.
	Improper seating of inserts in pocket	Clean pockets whenever indexing or changing inserts. Check pockets for nicks and burrs. Check if inserts rest completely on pocket bottoms.
	Damaged locking screws or pins	Check head and thread of screws or pins for nicks or burrs. Do not overtighten.
	Improper speeds and feeds	Check recommended rates.
Grooving on back stroke, drill body rubs hole wall, over or undersized holes	Insufficient coolant supply	Check coolant flow.
	Off-center drill	Maintain proper alignment, concentricity, and angularity. Check slug or bottom of hole for center stub.
	Deflection due to lack of rigidity, improper toolholder, or too much overhang	(Same as above under improper seating and deflection.)
Poor surface finish	Vibrations	Check if tool can be held shorter (rigidity). Check machine and setup rigidity. Check seat in spindle or toolholder. Check speeds and feeds.
	Insufficient coolant supply	Increase coolant pressure and flow; is it constant? If flood coolant, make sure coolant reaches inserts at all times.
Rough cutting action (tool rumbles and deflects)	Too much thrust; feed rate too high	Lower feed rate or (and) increase speed.
Very short, thick, corrugated chips	Feed rate too high in relation to cutting speed	Lower feed rate or (and) increase speed.
Long and stringy chips	Feed rate too low in relation to cutting speed	Increase feed rate or (and) decrease speed. If available, use inserts with narrow chipbreaker.
Unable to loosen insert locking screws or pins	Seized threads due to coolant or heat	Apply water and heat-resistant lubricant to threads of screw or pins.

(Waukesha Cutting Tools, Inc.)

* If constant chipping occurs, especially on inner insert, and conditions are at optimum, try an uncoated carbide insert or a grade with higher transverse rupture strength.

OPERATING PARAMETERS FOR DRILLING

Softer titanium alloys may be drilled at speeds in the range of 40-60 sfm (12.2-18.3 m/min), but harder alloys may require speeds as low as 10-15 sfm (3-4.6 m/min). Drills should be sharpened with an included angle of 118-135°, lip relief angles of 8-10° at the periphery, and a chisel edge angle of 125-135°. Smaller drills generally require flatter points. Larger drills are often chamfered at the cutting corners to an included angle of 90-105°, depending upon the point angle.

Drilling plastics. Molded plastics present no drilling problems as long as the drills provide for the good ejection of chips. For drilling small holes, special drills for plastics are available. These drills have wide polished flutes, a thin web, a low helix angle, an included point angle of 60-90°, and a clearance angle at the periphery of about 12°. Drills with high helix angles, however, are better for producing holes in some plastics. For larger holes, a point with a larger included angle (90-120°) can be used and the heel behind the cutting edge may be ground away, leaving a land about 1/16" (1.6 mm) wide, to reduce friction and provide more chip clearance. Drilling of some plastics results in undersize holes, and slightly oversize drills should be used. When a cutting fluid is used, tests should be made to ensure that the plastic will not react with the fluid.

Drilling composites. Composite materials consisting of high-strength fibers (glass, graphite, etc.) in a plastic matrix are very hard and abrasive, and conventional HSS drills are generally not satisfactory. For small holes, solid carbide drills are being used. For larger holes, diamond-impregnated core drills, operated at high speeds, have been found satisfactory.

Drilling miniature holes. Conventional drills and drilling practices, except for slower speeds, are regularly used to produce holes down to 0.015" (0.38 mm) diam or less. When smaller holes are drilled, however, common practice is to start with a center or pivot drill having a flute length not exceeding four times the drill diameter and a diameter at least as large as the web thickness of the following drill. The second tool is usually a pivot drill having a flute length of five to seven times its diameter and a diameter at least equal to the web thickness but somewhat smaller in diameter than the straight-shank drill used to produce the required diameter and depth.

In some cases, the holes are larger than the diameters of the drills because of tool shape, web thickness, and rake and point angles. The longer the chisel edge between the two cutting edges, the larger the hole is in relation to the diameter of the drill. The use of drill guide bushings is controversial. Some authorities recommend proper-fitting bushings to minimize deflection and help prevent walking; others argue that they should be avoided whenever possible because they generally require the use of straight-shank drills, which are the least rigid for starting, and that clearance between the drill and bushing is usually sufficient to permit some buckling.

Step or peck drilling (frequent tool withdrawal) is generally required to clear chips and prevent packing, and to apply lubricant. Some authorities recommend retraction after feeding the drill into the work from about 1/2 to 1 1/2 times the drill diameter, depending upon the chip formation and rate of drill breakage encountered.

Using indexable-insert drills. Proper toolholders are critical for optimum performance of indexable-insert drills. They should be as short as possible to minimize overhang, and the shank and socket must be clean and free of nicks. Concentricity, straight-

ness, and diameters must be maintained to close tolerances. Loose shanks will create chatter and cause insert damage.

When indexable-insert drills are set up, insert pockets should be cleaned and the inserts properly positioned and securely locked. Alignment, angularity, and runout must be carefully checked to prevent deflection, chatter, and the production of oversized, undersized, or tapered holes.

Troubleshooting Drilling Operations

Possible solutions to problems that occur when twist drills are used are presented in Table 9-32. Table 9-33 is a troubleshooting chart specifically for indexable-insert drills. Table 9-34 can be used as a general guide for improvements in gundrilling results. Only one adjustment should be made at one time to ensure a proper understanding of the results obtained.

Safety in Drilling

Safety requirements for the construction, care, and use of drilling, milling, and boring machines are presented in ANSI Standard B11.8-1974. Most modern machines used for drilling have guards, as well as emergency stop buttons, to protect the operators from rotating or sliding machine components, tools, and workpieces.

Chip wraparound or birdnesting can be a danger in drilling, especially on multispindle machines. The use of chipbreaker drills and increased feed rates may eliminate this problem.

Indexable-insert drills in particular require special attention to adequate guards, such as safety shields, because the high cutting speeds can eject chips, cutting fluid, and slugs produced from the work area with high velocities. Also, if the setup is not rigid, the high thrusts developed will create side loading and chatter, which could break the inserts and send fragments flying.

TABLE 9-34
Guide for Improving Gundrilling Results

Chip wrap or packing:	Hole oversize:
Increase coolant pressure	Increase feed rate
Increase outer angle	Decrease coolant pressure
Increase point location	Increase outer angle
Reduce feed rate	Increase point location
	Decrease cutting speed
Corner rounding:	Use different tool contour
Decrease cutting speed	
Built-up edge or crater:	Hole undersize:
Increase cutting speed	Increase speed and coolant pressure
Increase outer angle	Decrease outer angle
Increase dub-off	Decrease point location
Increase coolant pressure	Cool cutting oil
	Decrease feed rate
Poor tool life:	Poor surface finish:
Decrease speed and feed	Increase speed and coolant pressure
Increase feed (for abrasive materials)	Increase outer angle
Increase coolant pressure	Increase point location
Change nose grind	Use different tool contour
	Decrease feed rate

(Eldorado Tool & Manufacturing Corp.)

REAMING

Reaming is a machining process for enlarging, smoothing, and/or accurately sizing existing holes by means of multiedge fluted cutting tools (reamers). As the reamer and/or workpiece are rotated and advanced relative to each other, chips are produced to remove relatively small amounts of material from the hole wall. Reaming may be performed on the same type of machines used for drilling. The machines used for drilling are discussed earlier in this chapter.

Accuracy of the hole and quality of finish produced by reaming depends primarily upon the condition of the starting hole, rigidity of the machine and fixture, correct speeds and feeds, a suitable and properly applied cutting fluid, and precise resharpener of dull tools.

Since stock removal is small and must be uniform in reaming, the starting holes (drilled or otherwise produced) must have relatively good roundness, straightness, and finish. Reamers tend to follow the existing centerline of the hole being reamed, and in limited instances it may be necessary to bore the holes prior to reaming to maintain required tolerances. With the proper conditions and operating parameters, reaming can produce close tolerances and smooth finishes.

PRODUCT DESIGN FOR REAMING

Better reaming usually results when the product is designed to facilitate that operation. When possible, provision should be made for the reamer to pass through the workpiece (Fig. 9-109, *a*). This eliminates the necessity to ream a blind hole. When reaming a blind hole is unavoidable, the depth of cut should be controlled to prevent bottoming, cutting oversize holes, and possibly damaging the reamer (view *b*). It is advisable to avoid operations requiring multidiameter reamers when the diameters are substantially different because the normally different cutting speeds of the different diameters (view *c*) make it difficult to produce true holes and good surfaces. A reamer should enter a hole at right angles to the work surface to permit all teeth to engage for a good start. This is because reaming at an angle makes it difficult and sometimes impossible to turn out good work (view *d*). Plain reaming should not be depended upon to align a series of holes and center them on a common axis. Concentricity and alignment require line reaming (Fig. 9-109, view *e*), which requires that the holes be of equal or progressively smaller diameters to permit entrance and withdrawal of the reamers. Provision must be made for guiding the reamer bar or arbor at both ends.

STOCK REMOVAL ALLOWANCE

In reaming, the tool normally cuts slightly larger than its own diameter, usually in direct proportion to the amount of stock to be removed. For efficient operation, the amount of stock left in the hole for reaming must be sufficient to permit the reamer to cut at all times rather than to burnish the surface. Variations in the amount of stock to be removed can affect the finish size of the hole reamed.

Removal of too much stock by reaming often causes oversize and rough holes. Oversize holes, walls of holes roughened with grooves at or beyond the finished diameter, bellmouthed holes, or out-of-round holes are common causes of reamer failure. In improperly prepared holes the reamer has a tendency to wedge in the hole rather than machine it. The result can be severe reamer wear and possible breakage.

Table 9-35 presents suggested amounts of stock removal that can be used as a guide for machine reaming various sizes of holes with HSS reamers. More stock generally has to be removed from rough or torn surfaces, or irregularly shaped holes. Much less stock—0.001-0.003" (0.003-0.08 mm)—is removed in hand reaming.

The amount of material left in a hole for a carbide-tipped reamer is very important. Insufficient stock removal will result in excessive wear; too much will cause excessive cutting pressures and may result in tool breakage. About 3% of the reamer diameter should be left in the hole for proper results with carbide-tipped reamers.

The amount of stock left for finish reaming with block-type reamers (discussed later in this section) should seldom exceed 0.015" (0.38 mm) on the diameter. More stock can be removed in roughing operations as long as it is not beyond the capacity of the block. Because of their built-up construction, block reamers are necessarily more fragile than solid reamers. (A general discussion of stock removal allowance and positioning accuracy required in machining operations is presented in Chapter 2 of this volume, "Tolerance Control.")

Stock removal for adjustable floating reamers (also discussed later in this section) should be limited to those shown for finishing in Table 9-35. The stock to be removed from holes having diameters larger than 1" (25.4 mm) should be limited to a maximum of 0.015" (0.38 mm). Due to the adjustable construction of these tools, excessive stock removal may result in poor performance and possible tool breakage.

SELECTING A REAMER

Proper selection of a reamer for a specific application from the wide variety of reamers available depends primarily upon the composition and hardness of the workpiece material; hole diameter, configuration, and depth; amount of stock to be removed; product quantity, accuracy, and finish requirements; and cost (initial, maintenance, and salvage value).

Reamers are available for holes as small as 0.005" (0.13 mm) and as large as 6" (152 mm) diam. A few tool manufacturers offer metric reamers as standard.

REAMERS

A reamer is a rotary cutting tool, generally of cylindrical or conical shape, intended for enlarging and finishing holes to accurate dimensions. It is usually equipped with two or more peripheral channels or flutes, either parallel to its axis or in a right or left-hand helix as required. Those with helical flutes provide smooth shear cutting, are less subject to chatter, and produce a better finish. The flutes form cutting teeth and provide channels for removing the chips.

Terminology and Definitions

Nomenclature, definitions, types, sizes, and tolerances for reamers are presented in ANSI Standard B94.2-1977, published by ASME. Some of the definitions in this standard (see also Fig. 9-110) are as follows:

reamer A rotary cutting tool with one or more cutting elements used for enlarging to size and contour a previously formed hole. Its principal support during the cutting action is obtained from the workpiece.

CHAPTER 9

REAMERS

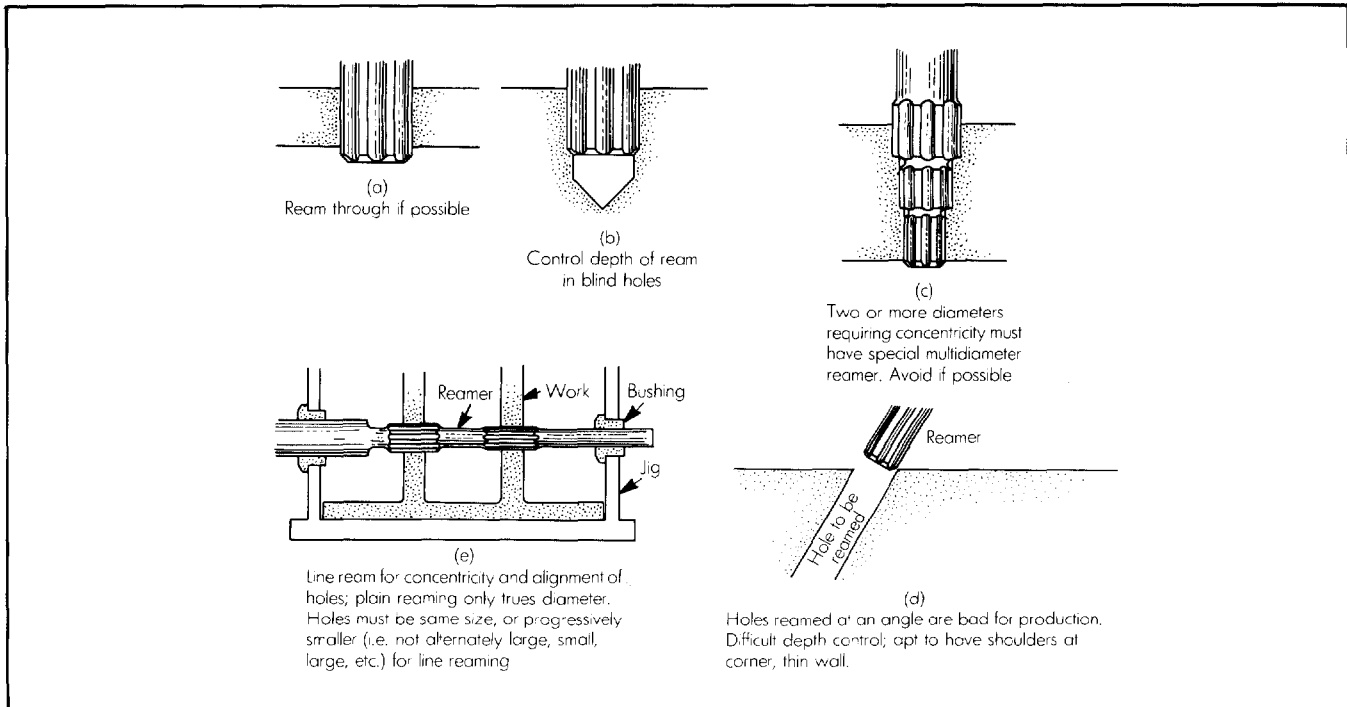


Fig. 9-109 Product design factors relating to reaming.

actual size The actual measured diameter of a reamer, usually slightly larger than the nominal size to allow for wear (see Table 9-36).

angle of taper The included angle of taper on a taper tool or taper shank.

arbor hole The central mounting hole in a shell reamer.

axis The imaginary straight line which forms the longitudinal centerline of a reamer, usually established by rotating the reamer between centers.

back taper A slight decrease in diameter, from front to back in the flute length of reamers.

bevel An unrelieved angular surface of revolution (not to be confused with chamfer).

body The fluted full-diameter portion of a reamer, inclusive of the chamfer, starting taper, and bevel.

chamfer The angular cutting portion at the entering end of a reamer. See also secondary chamfer.

chamfer angle The angle between the axis and the cutting edge of the chamfer measured in an axial plane at the cutting edge.

chamfer length The length of the chamfer measured parallel to the axis at the cutting edge.

chipbreakers Notches or grooves in the cutting edges of some taper reamers designed to break the continuity of the chips.

clearance The space created by the relief behind the cutting edge or margin of a reamer.

core The central portion of a reamer below the flutes which joins the lands.

core diameter The diameter at a given point along the axis of the largest circle which does not project into the flutes.

cutter sweep The section removed by the milling cutter or grinding wheel in entering or leaving a flute.

cutting edge The leading edge of the land in the direction of rotation for cutting.

TABLE 9-35
Suggested Stock Removal for Machine Reaming

Reamer Diam, in.	Stock Removal, in. (mm)	
	Roughing	Finishing
to 1/16	0.003-0.005 (0.08-0.13)	0.002-0.004 (0.05-0.10)
1/16 to 1/8	0.004-0.008 (0.10-0.20)	0.002-0.004 (0.05-0.10)
1/8 to 1/4	0.006-0.012 (0.15-0.30)	0.003-0.005 (0.08-0.13)
1/4 to 3/8	0.008-0.014 (0.20-0.36)	0.004-0.006 (0.10-0.15)
3/8 to 1/2	0.010-0.015 (0.25-0.38)	0.005-0.007 (0.13-0.18)
1/2 to 3/4	0.012-0.018 (0.30-0.46)	0.006-0.008 (0.15-0.20)
3/4 to 1	0.014-0.020 (0.36-0.51)	0.008-0.014 (0.20-0.36)
1 to 1 1/2	0.018-0.025 (0.46-0.63)	0.014-0.020 (0.36-0.51)

cutting face The leading side of the land in the direction of rotation for cutting on which the chip impinges.

external center The pointed end of a reamer. The included angle varies with manufacturing practice.

flutes Longitudinal channels formed in the body of the reamer to provide cutting edges, permit passage of chips, and allow cutting fluid to reach the cutting edges.

angular flute A flute which forms a cutting face lying in a plane intersecting the reamer axis at an angle. It is unlike a helical flute in that it forms a cutting face which lies in a single plane.

helical flute A flute which is formed in a helical path around the axis of a reamer. Also referred to as spiral flute.

spiral flute On a taper reamer a flute of constant lead. In reference to a straight reamer, see the preferred term helical flute.

straight flute A flute which forms a cutting edge lying in an axial plane.

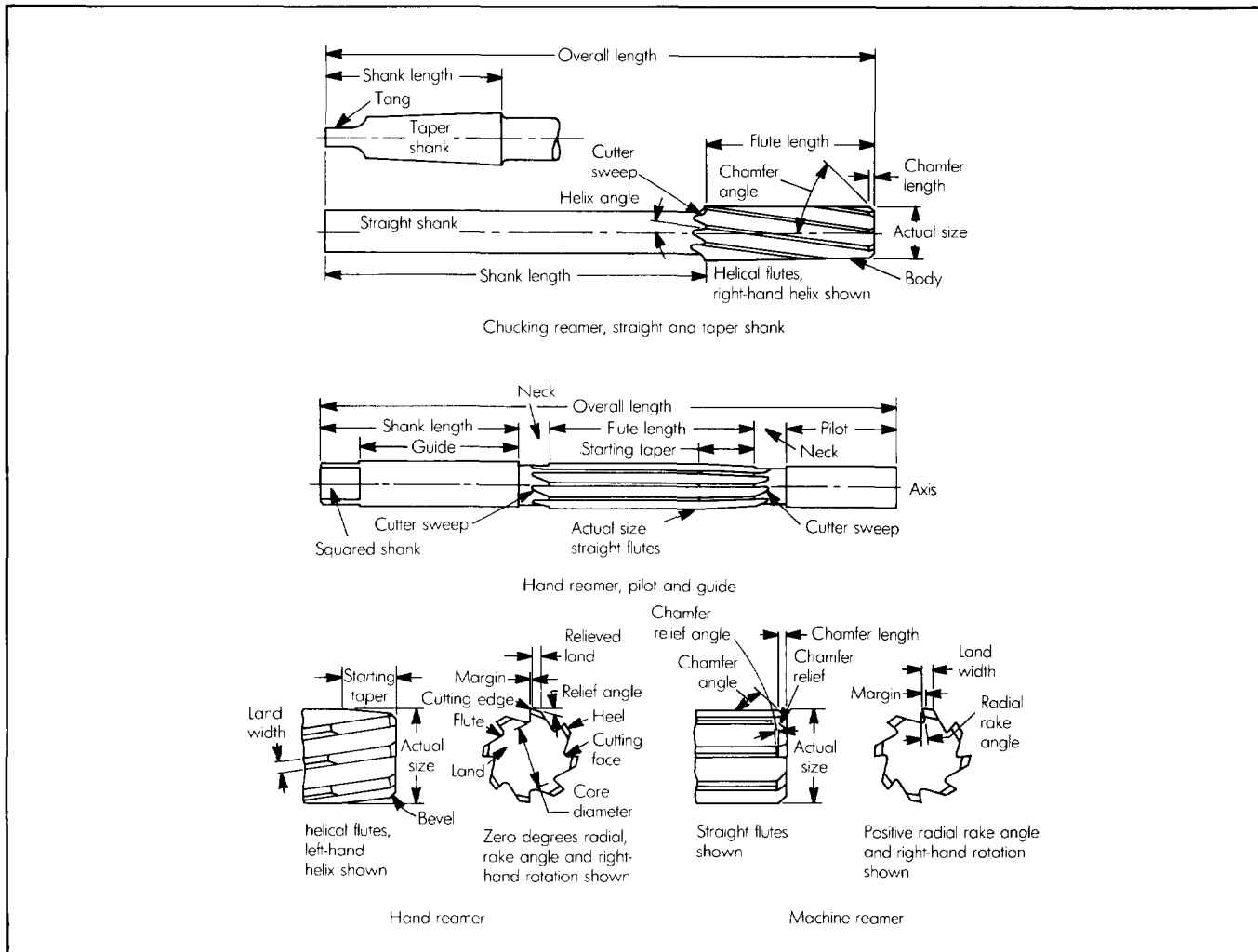


Fig. 9-110 Terms applying to reamers.

flute length The length of the flute not including the cutter sweep.

guide A cylindrical portion following the flute of a reamer to maintain alignment.

helix angle The angle which a helical cutting edge at a given point makes with an axial plane through the same point.

internal center A 60° countersink with clearance at the bottom, in one or both ends of a tool, which establishes the tool axis.

irregular spacing A deliberate variation from uniform spacing of the reamer cutting edges.

land The section of the reamer between adjacent flutes.

land width The distance between the leading edge of the land and the heel measured at a right angle to the leading edge.

lead of flute The axial advance of a helical or spiral cutting edge in one turn around the reamer axis.

length The dimension of any reamer element measured parallel to the reamer axis.

limits The maximum and minimum values designated for a specific element.

margin The unrelieved part of the periphery of the land adjacent to the cutting edge.

neck The section of reduced diameter connecting shank to body, or connecting other portions of the reamer.

nominal size The designated basic size of a reamer.

overall length The extreme length of the complete reamer from end to end, but not including external centers or expansion screws.

periphery The outside circumference of a reamer.

pilot A cylindrical portion preceding the entering end of the reamer body to maintain alignment.

TABLE 9-36
Tolerances on Cutting Diameters for Reamers*

Nominal Reamer Diameter, inch	Tolerance on Cutting Diameter, inch
to 1/4	+0.0001 to 0.0004
17/64 to 1	+0.0001 to 0.0005
over 1	+0.0002 to 0.0006

*These tolerances apply to standard stock reamers and to special reamers where a specific tolerance is not stated.

REAMERS

rake The angular relationship between the cutting face, or a tangent to the cutting face at a given point and a given reference plane or line.

axial rake Applies to angular (not helical or spiral) cutting faces. The angle between a plane containing the cutting face, or tangent to the cutting face at a given point, and the reamer axis.

helical rake Applies to helical cutting faces only (not angular). The angle between a plane tangent to the cutting face at a given point on the cutting edge and the reamer axis.

hook A concave condition of a cutting face. The rake of a hooked cutting face must be determined at a given point.

negative rake Describes a cutting face in rotation whose cutting edge lags the surface of the cutting face.

positive rake Describes a cutting face in rotation whose cutting edge leads the surface of the cutting face.

radial rake angle The angle in a transverse plane between a straight cutting face and a radial line passing through the cutting edge.

relief The result of the removal of tool material behind or adjacent to the cutting edge to provide clearance and prevent rubbing (heel drag).

axial relief The relief measured in the axial direction between a plane perpendicular to the axis and the relieved surface. It can be measured by the amount of indicator drop at a given radius in a given amount of angular rotation.

cam relief The relief from the cutting edge to the heel of the land produced by a cam action.

chamfer relief The axial relief on the chamfer of the reamer.

chamfer relief angle The axial relief angle at the outer corner of the chamfer. It is measured by projection into a plane tangent to the periphery at the outer corner of the chamfer.

eccentric relief A convex relieved surface behind the cutting edge.

flat relief A relieved surface behind the cutting edge which is essentially flat.

primary relief The relief immediately behind the cutting edge or margin. Properly called relief.

radial relief Relief in a radial direction measured in the plane of rotation. It can be measured by the amount of indicator drop at a given radius in a given amount of angular rotation.

relief angle The angle, measured in a transverse plane, between the relieved surface and a plane tangent to the periphery at the cutting edge.

secondary relief An additional relief behind the primary relief.

secondary chamfer A slight relieved chamfer adjacent to and following the initial chamfer on a reamer. See also chamfer.

shank The portion of the reamer by which it is held and driven.

squared shank A cylindrical shank having a driving square on the back end.

starting radius A relieved radius at the entering end of a reamer in place of a chamfer.

starting taper A slight relieved taper on the front end of a reamer.

straight shank A cylindrical shank.

tang The flattened end of a taper shank which fits a slot in the socket.

taper shank A shank made to fit a specified (conical) taper socket.

Clearance and Blade Angles

Cutting done by a reamer (except a hand reamer or taper reamer) is accomplished by the chamfer, which forms a truncated cone on the end of the reamer and carries the cutting edges. The chamfer is shaped so that the reamer will start properly in the hole and the major portion of the cutting is done by the chamfer length. The longitudinal cutting edges should do little or no cutting. The clearance angles in the cutting portions of the reamer are employed to obtain cutting action in the proper places, prevent friction, and clear the chips. Reamers usually embody three types of clearance: chamfer relief, peripheral relief, and longitudinal relief.

Chamfer relief. Chamfer relief is obtained by the clearance angle ground on the cutting ends of the reamer to enable them to penetrate more easily beneath the workpiece surface and produce chips.

Peripheral relief. Peripheral relief is obtained by the clearance angle ground on the lands back of the flute edges to relieve the radial pressure of the reamer flutes against the walls of the hole.

Longitudinal relief. Longitudinal relief or back-taper is the slight taper embodied in standard reamers, which results in a smaller diameter at the shank end than at the cutting end, to prevent the reamer blade from causing side-cutting action due to lack of proper alignment or to dullness of the reamer caused by wear. It is important that reamers used in angular floating holders should embody enough back-taper to prevent interference resulting from improper alignment caused by these types of holders.

Chamfer angle. A chamfer angle usually has a magnitude of about 45° , but it can range from 40° – 50° . Because most of the dulling of reamer blades occurs at the peripheral corners of the chamfer, the maximum effective cutting action continues only as long as the original sharp chamfer angle is maintained. The chamfer must be uniform and concentric with the cutting diameter of the reamer.

Secondary chamfer. Secondary chamfer (Fig. 9-111) sometimes referred to as starting taper or lead, is provided on some reamers. It consists of a chamfer ranging from 1° – 10° which may be located immediately behind the first chamfer. Clearance should be provided for this second taper in a manner that enables it to be brought to a sharp edge. It should not be confused either with the primary chamfer-relief angle or with a similar expression that refers to the starting taper or lead of a hand reamer. The secondary chamfer functions to effect a scraping cut that helps to size and smooth the walls of the hole.

Chamfer on hand reamers for light cutting should be approximately 45° on the entering teeth. This chamfer, in the case of a hand reamer, does practically no cutting and is utilized more as a guide to assist the reamer to enter the hole more readily. A hand reamer should cut on the relieved lead or starting taper located behind the chamfer. The length of the

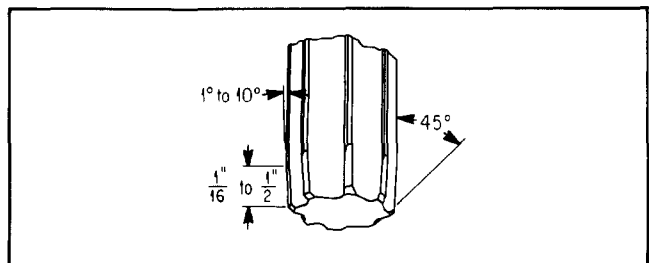


Fig. 9-111 Secondary chamfer is provided on some reamers.

hand-reamer lead section or starting taper customarily is equal to approximately the reamer diameter for sizes of 3/4" (19 mm) and smaller; it decreases to approximately one-half the reamer diameter for sizes ranging from 3/4 to 3" (19 to 76 mm) diam. The diameter of the lead section of the entering end should range from 0.010-0.015" (0.25-0.38 mm) under the nominal reamer diameter.

Hand of Helix and Hand of Cut

The hand of a reamer cut refers to the direction of rotation (right or left) and may be determined by inspection of the chamfer end of the reamer when mounted to make a cut. If the rotary motion of the reamer is counterclockwise, cut is left-hand; if the rotary motion is clockwise, cut is right-hand.

The hand of the helical reamer may be determined by viewing the reamer from either end. If the flutes twist away from the observer in a clockwise direction, the helix is right-hand; if the flutes twist away from the observer in a counterclockwise direction, the helix is left-hand.

Most reamers (Fig. 9-112) are right-hand cutting and may have straight flutes (a), left-hand helix flutes (b), or right-hand helix flutes (c). Straight flutes cost less originally and are easiest to sharpen or recondition. This type of reamer has proved satisfactory for a great many applications. When reamers are being selected for machine tools with play in the gears or with spindles not in the best of condition, it is usually advisable to choose those with left-hand helical flutes. This type of reamer has a negative axial rake and therefore requires more pressure to feed the tool into the work. The additional pressure and the hand of the helix opposite to the hand of the cut reduce the tendency for the reamer to jump ahead when easier cutting is encountered. For machine tools in very good condition, right-hand fluted reamers are satisfactory. They are inherently free cutting and require less power; however, when used on a worn machine, they tend to dig into the workpiece, cause chatter, and produce a rough hole.

Marginal Widths as Related to Peripheral Relief Angles

The purpose of the margin of a reamer land is to size the hole by smoothing the roughness caused by the chamfer and to reduce friction caused by the reamer's rotating in the hole. When a reamer is ground down to a diameter smaller than the original size, the width of the land should be reduced. This will decrease friction and prevent heat buildup and eventual galling. The marginal width and magnitude of a relief angle vary somewhat for particular applications, but the values in Table 9-37 may be used as a basis from which to determine by actual trial the correct values to be used. By properly changing the relief angle, a reamer may be made suitable for use under a great variety of operating conditions.

Kinds of Reamers

Reamers are made in many different forms (Fig. 9-113), including solid and inserted-blade types, adjustable and non-adjustable; they are available for either manual operation (hand reamers) or for machine use (chucking reamers). Materials from which cutting elements of most production reamers are made include high-speed steel and cemented carbides. A comprehensive discussion of cutting tool materials is presented in Chapter 3 of this volume, "Cutting Tool Materials."

Carbide reamers. These tools are being used increasingly because of their longer life, improved accuracy, and resistance

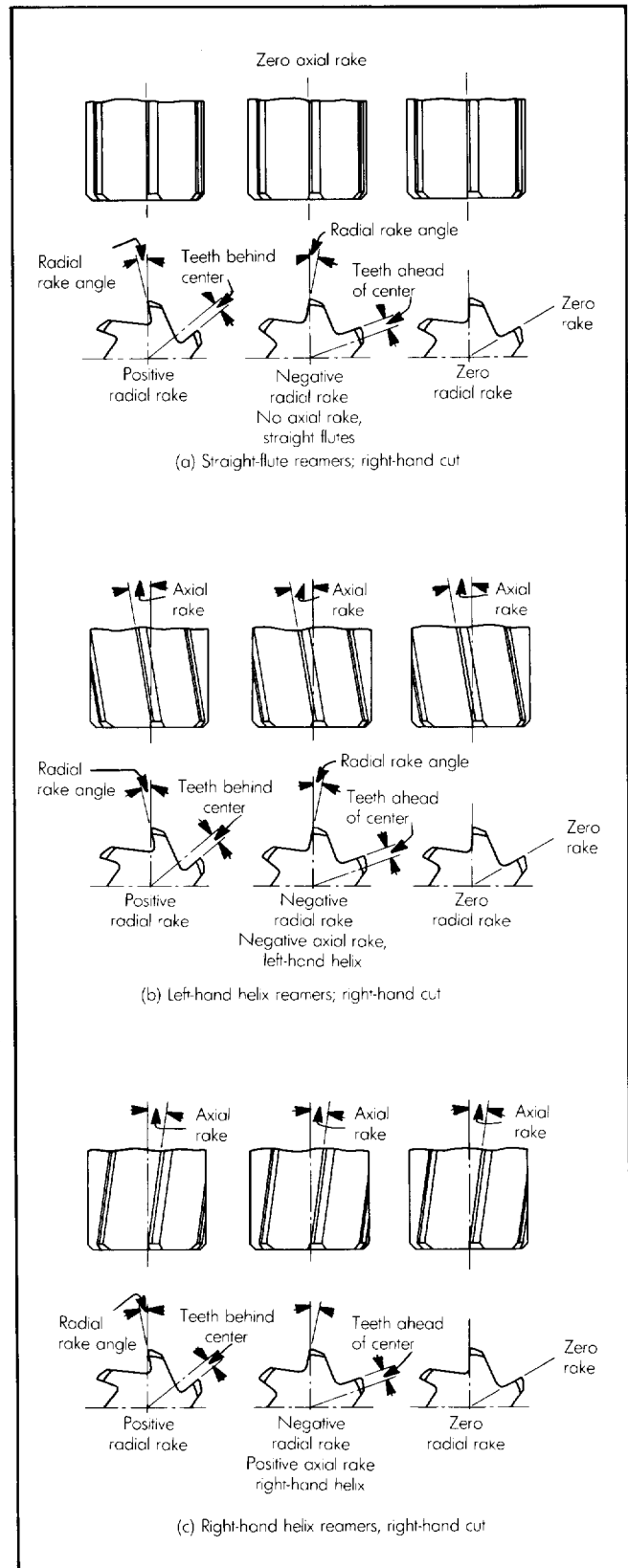


Fig. 9-112 Reamers for right-hand rotation.

CHAPTER 9

REAMERS

TABLE 9-37
Suggested Widths of Margins and Magnitude of Relief Angles for Various Sizes of Reamers

Diam of Reamer, in.	Width of Margin, in. (mm)*	Magnitude of Relief Angle, degrees
Under 1/8**	0.004-0.006** (0.10-0.15)	20-25
1/8 to 1/4	0.006-0.008 (0.15-0.20)	15-20
1/4 to 1/2	0.008-0.010 (0.20-0.25)	11-14
1/2 to 3/4	0.010-0.015 (0.25-0.38)	9-10
3/4 to 1	0.012-0.017 (0.30-0.43)	8-9
1 to 1 1/2	0.014-0.018 (0.36-0.46)	7-8
1 1/2 to 2	0.016-0.022 (0.41-0.56)	7-8
Larger than 2	0.018-0.025 (0.46-0.63)	7-8

* Margin width depends upon material to be reamed.

** The relief angle that must be ground on reamers of very small diameters often depends upon the width of the land, because the heel of the land must always be under the leading edge of the relief angle.

to galling (adherence of metal to the cutting edges), which can cause oversize holes and poor finishes. They are available as solid carbide tools, tipped with carbide on their ends only, and tipped for the full flute lengths. Solid carbide reamers are made in smaller sizes, generally from 1/16 to 3/8" (1.6 to 9.5 mm) diam and in larger sizes on special order.

Carbide-tipped reamers are basically the same in design as HSS reamers. The reamer body, blades or flutes, and tips, however, must be shaped and dimensioned to resist deflection because of the brittleness of the carbide. Practically all carbide-tipped reamers are intended for machine reaming.

Certain angles and clearances are embodied in the cutting elements of carbide-tipped reamers to obtain cutting action in the proper places, prevent friction, and clear the chips. Practically all the cutting is done on the relieved chamfer, which is in the form of a truncated cone on the entering end of the reamer. This chamfer is shaped to start the reamer properly in the hole.

A clearance on the chamfer of 6-8° on the carbide tip and a secondary clearance of 10-15° on the steel body have proved to be satisfactory for most workpiece materials. Under good work conditions and proper alignment of the reamer and the work, a back-taper of 0.0003 in./in. (or mm/mm) of flute length is adequate. Back-taper should be increased to prevent binding of only slightly dull tools when very abrasive materials are being reamed. In general, circular widths may range from 0.007-0.020" (0.18-0.51 mm) depending primarily upon the material to be reamed. Primary relief angles on the carbide tips range from 15° on 1/4" diam tools to 7° on 1 1/2" diam reamers (see Fig. 9-114).

TABLE 9-38
Standard Reamer Dimensions and Number of Flutes for Carbide-tipped Reamers*

Nominal Reamer Size, in.	Overall Length, in.	Flute Length, in.	Shank Diam, in.	Morse Taper No.	Chamfer			No. of Flutes	
					Length	Angle, deg.	Roughing	Standard	Finishing
3/16	4 1/2	1 1/8	11/64	1	1/32	45	3	4	4
1/4	6	1 1/2	15/64	1	1/32	45	3	4	4
5/16	6	1 1/2	9/32	1	1/32	45	3	4	4
3/8	7	1 3/4	5/16	1	1/32	45	3	4	4
7/16	7	1 3/4	3/8	1	1/32	45	3	4	4
1/2	8	2	7/16	1	1/32	45	4	6	6
9/16	8	2	7/16	1	1/32	45	4	6	6
5/8	9	2 1/4	9/16	2	1/32	45	4	6	6
11/16	9	2 1/4	9/16	2	1/32	45	4	6	6
3/4	9 1/2	2 1/2	5/8	2	3/64	45	4	6	6
13/16	9 1/2	2 1/2	5/8	2	3/64	45	4	6	6
7/8	10	2 5/8	3/4	2	3/64	45	4	6	6
15/16	10	2 5/8	3/4	3	3/64	45	4	6	8
1	10	2 3/4	7/8	3	1/16	45	4	8	8
1 1/16	10 1/2	2 3/4	7/8	3	1/16	45	4	8	8
1 1/8	11	2 7/8	7/8	3	1/16	45	4	8	8
1 3/16	11	2 7/8	1	3	1/16	45	4	8	8
1 1/4	11 1/2	3	1	4	1/16	45	6	8	8
1 5/16	11 1/2	3	1	4	1/16	45	6	8	8
1 3/8	12	3 1/4	1	4	5/64	45	6	8	8
1 7/16	12	3 1/4	1 1/4	4	3/32	45	6	8	8
1 1/2	12 1/2	3 1/2	1 1/4	4	3/32	45	6	8	8

* It is always advisable to use standard tools whenever possible, as special tools are manufactured in small quantities, cost more, and are not as readily obtainable.

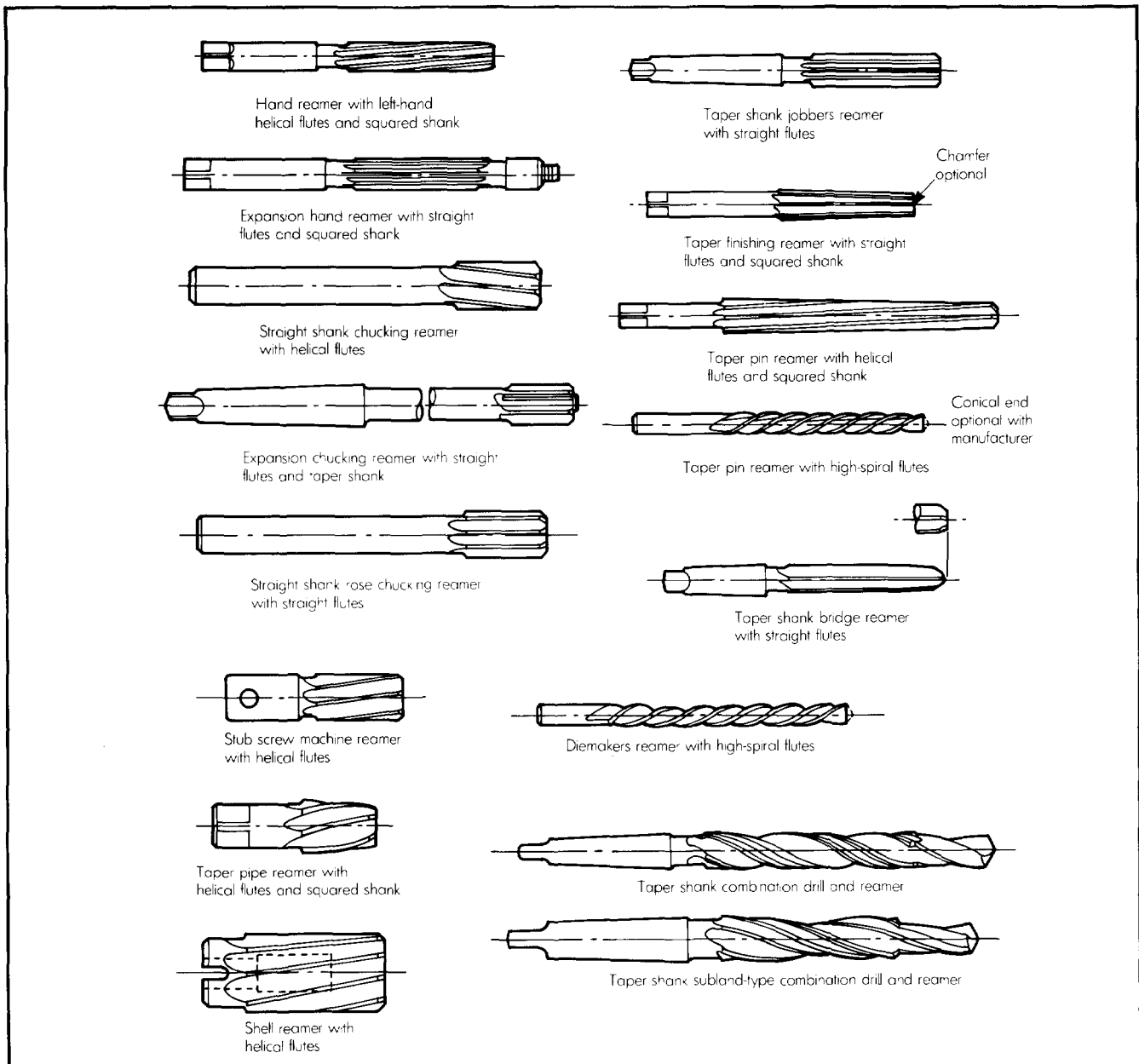


Fig. 9-113 Commercial types of reamers.

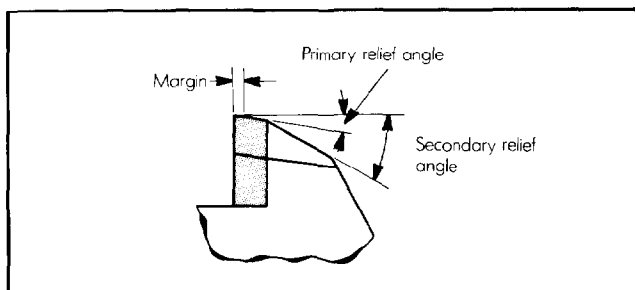


Fig. 9-114 Primary and secondary relief angles for carbide-tipped reamers.

Carbide-tipped reamers used for roughing operations are generally special. They may have either an even or an odd number of flutes, depending upon the reamer size and the application (Table 9-38). Standard and finishing types usually have an even number of flutes. The spacing of the flutes around the reamer body is sometimes uneven to minimize the possibility of chatter. Tools with an uneven number of flutes, however, are difficult to measure. Carbide-tipped reamers may be made with either right or left-hand flutes, or with straight flutes, and may incorporate one, two, or more sections of different diameters for cutting two or more diameters in one operation.

Carbide-tipped reamers in straight-flute design, with strips of tungsten carbide the full length of the flutes, are ideal for

TABLE 9-39
Characteristics of Carbide Reamer Elements Required for Various Workpieces

Material	Cylindrical Land Width, in. (mm)		Lead Angle				Rake Angle Magnitude, degree		Body Clearance, in. (mm)		Helix, degree	
	Roughing	Finishing	Magnitude, degree		Length, in. (mm)		Roughing	Finishing	Roughing	Finishing	Roughing	Finishing
			Roughing	Finishing	Roughing	Finishing						
Free-cutting, abrasive materials:												
Bakelite, hard rubber, cast iron (when using coolant)	0.002- 0.005 (0.05- 0.13)	0.002- 0.005 (0.05- 0.13)	None	2	None	3/16 (4.8)	0-7	0-7	0.0005- 0.001 (0.013- 0.03)	0.0005- 0.001 (0.013- 0.03)	None	None
Free-cutting, nonabrasive materials:												
Cast iron (dry), brass, copper, most die castings, bearing bronze, magne- sium castings, heat- treated cast aluminum	0.015- 0.020 (0.38- 0.51)	0.015- 0.020 (0.38- 0.51)	None	None	None	None	5-7	5-7	0.0002- 0.0004 (0.005- 0.010)	0.0002- 0.0004 (0.005- 0.010)	None	None
Tough, abrasive materials:												
Alloy steel, most cast steels	0.002- 0.005 (0.05- 0.13)	0.002- 0.005 (0.05- 0.13)	None	2	None	3/16 (4.8)	7-10	7-10	0.0005- 0.001 (0.013- 0.03)	0.0005- 0.001 (0.013- 0.03)	None	5-8
Tough, nonabrasive materials:												
Aluminum bronze, free- cutting steel, stainless steel	0.005- 0.010 (0.13- 0.25)	0.005- 0.010 (0.13- 0.25)	None	2	None	3/16 (4.8)	7-10	7-10	0.0002- 0.0005 (0.005- 0.013)	0.0002- 0.0005 (0.005- 0.013)	None	5-8

precision reaming to close tolerances. The wear on the diameter from bushings is minimized, and back-taper can be reduced to a minimum because the danger of pickup on the diameter is reduced.

For best efficiency, carbide-tipped reamers should be used only on machines that are in good condition. Loose or misaligned spindles can cause serious damage to a reamer, and the resulting cut will not conform to the desired size or finish. The magnitude of lead and rake angles, width of land, amount of clearance, and whether straight or spiral flutes should be employed are all related to the kind and characteristics of the material from which the workpiece is made (Table 9-39).

Hand reamers. A hand reamer is generally used to finish or accurately size a previously reamed or bored hole, or it is used for fitting bushings or bearings when only a small amount of metal is removed during the finishing process. Hand reamers are available in either the solid or the expansion type and may have either straight or left-hand helix flutes. The solid type is made from either carbon or high-speed tool steel. Hand reamer blades are usually spaced irregularly around the reamer body.

The amount of expansion possible in an expansion reamer is rather limited and varies with different makes. Breakage of these tools usually is caused by the user's attempting to expand them beyond the intended limits. One reamer manufacturer suggests the limits of expansion in Table 9-40.

TABLE 9-40
Recommended Expansion Limits
for Expansion Reamers

Size Range, in.	Limit of Expansion, in. (mm)
1/4 to 15/32	0.006 (0.15)
1/2 to 31/32	0.010 (0.25)
1 to 1 1/2	0.013 (0.33)
1 9/16 to 2	0.015 (0.38)

Chucking reamers. Chucking reamers are designed primarily for application in turret lathes and screw machines in which in many instances the tool is stationary and the work revolves, but the tools are rotated for some applications. They are available in either straight-flute, right or left-hand, helix-flute forms and may be either of the fluted chucking type or of the rose chucking type (see Fig. 9-115). The essential difference between the rose chucking reamer and the fluted chucking reamer is that the rose type is not relieved on the periphery. Rose reamers also have deeper flutes for increased chip space, permitting greater stock removal.

Straight-flute reamers can embody staggered flutes; i.e., the flutes are irregularly spaced around the circumference of the reamer body. This is no longer standard industry practice, but is sometimes done on coarse-pitch and large-diameter tools to reduce the reamer's tendency to chatter because of slippage or torsional deflection. Chatter may sometimes be minimized by using a reamer having a hand of helix opposed to the hand of cut. Special oversize and undersize reamers are available for press fit and clearance applications by toolmakers.

Stub screw-machine reamers. Reamers used in automatic screw machines are required to function in a limited space, hence their short length. Stub screw-machine reamers are particularly adapted to use in floating holders. Left-hand helix flutes and right-hand rotation for cutting are considered standard; reamers requiring left-hand rotation for cutting and

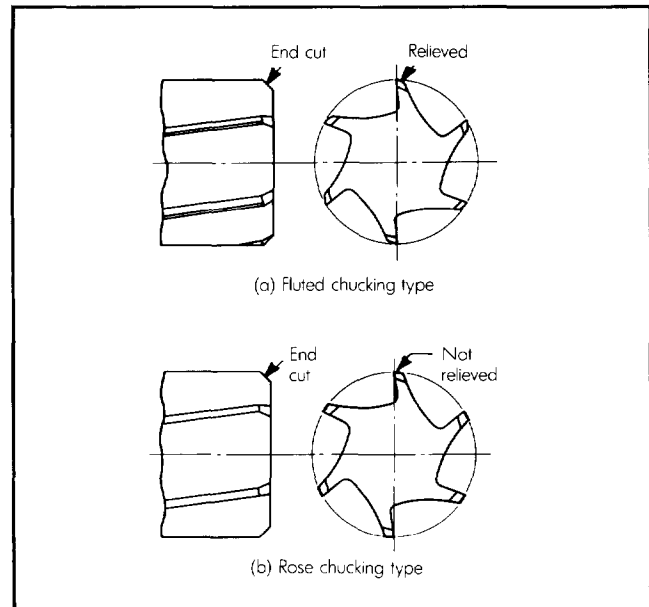


Fig. 9-115 Reamer ends for end cutting (left-hand helix, right-hand cut).

reamers with right-hand helix flutes are considered special types. Stub screw-machine reamers are available in numbered series in consecutive diameter ranges (Table 9-41).

End-cutting reamers. All reamers are considered to be end cutting, but it is often necessary to produce a special angle in a hole or to use the reamer as a counterbore or as a seating tool in a blind hole. In these and similar cases, the reamer end must be shaped to accomplish the purpose (Fig. 9-116).

When reamers are cut to center (second rightmost view, Fig. 9-116), care should be taken to avoid zero flute depth at center, in which case no chips could be removed at or near center. This condition is avoided, in the design shown, by having one land extend about 1/64" (0.4 mm) past center, with sufficient depth of flute to permit chip escape at center. The mating land is thereby automatically shortened, as indicated in section A-A. The 180° end cutting grind (right view) is often preferred.

Jobber's reamers. Taper-shank machine or jobber's reamers usually have straight flutes; their lengths are approximately the same as hand reamers. Some applications require right or left-hand helix styles, but these are considered to be special and manufacturers do not ordinarily stock them.

Shell reamers. Shell reamers, sometimes referred to as hollow reamers, are available with straight or helical flutes and are used on arbors with driving lugs. The holes in shell reamers are ground with a taper to assist in firmly seating the tool on the arbor. Usually, sufficient space is left between the end of the reamer and the shoulder on the arbor so the reamer can be removed without damaging it. Shell reamers are used in turret machines for line reaming of bearings and other similar applications.

Shell reamers are used when the tool is held stationary as the work rotates, when the work is stationary and the tool rotates, and when both tool and work are in motion. Because shell reamers are two-piece assemblies, they lack the rigidity of excessive amounts of stock. Dimensions of driving slots and lugs for straight or taper-shank arbors used with shell reamers are presented in Table 9-42. The

CHAPTER 9

REAMERS

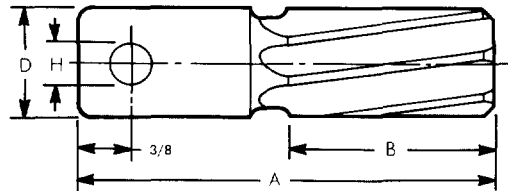


TABLE 9-41
Series Numbers, Dimensions, and Number of Flutes for
Stub Screw Machine Reamers with Helical Flutes*

Series Number	Diameter Range	Dimensions				Number of Flutes
		Length Overall A	Length of Flute B	Diameter of Shank D	Size of Hole H	
00	0.0600 to 0.066 incl.	1 3/4	1/2	1/8	1/16	4
0	0.0661 to 0.074 incl.	1 3/4	1/2	1/8	1/16	4
1	0.0741 to 0.084 incl.	1 3/4	1/2	1/8	1/16	4
2	0.0841 to 0.096 incl.	1 3/4	1/2	1/8	1/16	4
3	0.0961 to 0.126 incl.	2	3/4	1/8	1/16	4
4	0.1261 to 0.158 incl.	2 1/4	1	1/4	3/32	4
5	0.1581 to 0.188 incl.	2 1/4	1	1/4	3/32	4
6	0.1881 to 0.219 incl.	2 1/4	1	1/4	3/32	6
7	0.2191 to 0.251 incl.	2 1/4	1	1/4	3/32	6
8	0.2511 to 0.282 incl.	2 1/4	1	3/8	1/8	6
9	0.2821 to 0.313 incl.	2 1/4	1	3/8	1/8	6
10	0.3131 to 0.344 incl.	2 1/2	1 1/4	3/8	1/8	6
11	0.3441 to 0.376 incl.	2 1/2	1 1/4	3/8	1/8	6
12	0.3761 to 0.407 incl.	2 1/2	1 1/4	1/2	3/16	6
13	0.4071 to 0.439 incl.	2 1/2	1 1/4	1/2	3/16	6
14	0.4391 to 0.470 incl.	2 1/2	1 1/4	1/2	3/16	6
15	0.4701 to 0.505 incl.	2 1/2	1 1/4	1/2	3/16	6
16	0.5051 to 0.567 incl.	3	1 1/2	5/8	1/4	6
17	0.5671 to 0.630 incl.	3	1 1/2	5/8	1/4	6
18	0.6301 to 0.692 incl.	3	1 1/2	5/8	1/4	6
19	0.6921 to 0.755 incl.	3	1 1/2	3/4	5/16	6
20	0.7551 to 0.817 incl.	3	1 1/2	3/4	5/16	8
21	0.8171 to 0.880 incl.	3	1 1/2	3/4	5/16	8
22	0.8801 to 0.942 incl.	3	1 1/2	3/4	5/16	8
23	0.9421 to 1.010 incl.	3	1 1/2	3/4	5/16	8

(ANSI Standard B94.2-1977, published by ASME.)

* All dimensions are given in inches. Tolerances on reamer diameter are +0.0001 to 0.0004 for series 00 to 7 incl. and +0.0001 to 0.0005 for series 8 to 23 incl.; on overall length and length of flute, $\pm 1/16$; and on shank length, -0.0005 to 0.002.

six sizes of arbors accommodate all shell reamers in diameters of 3/4 to 2 1/2" (19 to 63.5 mm). Where applicable, the use of one arbor with several shell reamers may provide economy over the use of individual sizes of standard reamers.

Adjustable reamers. An adjustable reamer (Fig. 9-117) may be used to obtain holes of varying diameters over a limited range. The adjustment must be enough to compensate for wear or to allow the reamer to be reground to its original size. These reamers lack the rigidity of the conventional solid types and

must be used accordingly; removal of excessive amounts of stock with them results in poorly finished holes.

Adjustable floating reamers. These tools, having spring-loaded adjustment dials (see Fig. 9-118), are available for setting size on the machine. The direct-reading, adjustable dial permits setting size in increments of 0.0001" (0.003 mm) on diameter. A radial float of 0.020-0.030" (0.51-0.76 mm) is provided for the HSS or carbide-tipped blades. The blades can be changed without removing the holder from the machine.

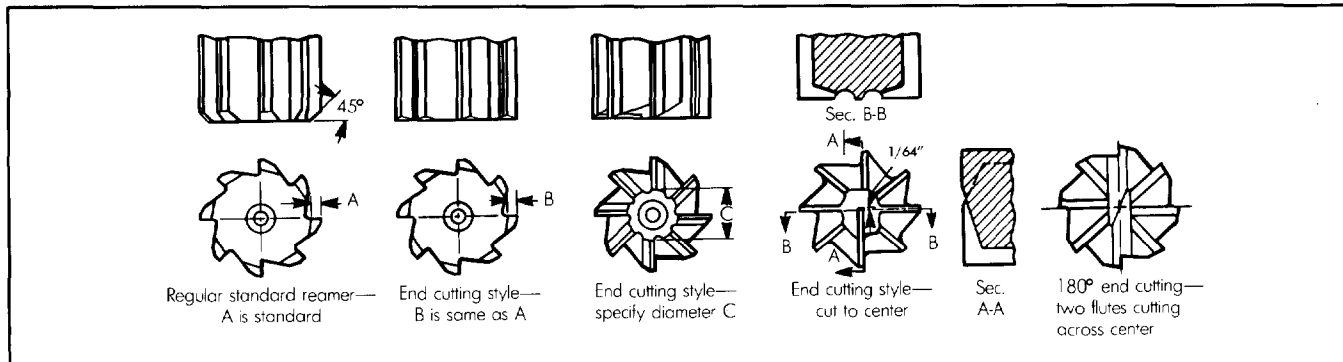


Fig. 9-116 Comparison of regular reamer end with end-cutting types of reamers.

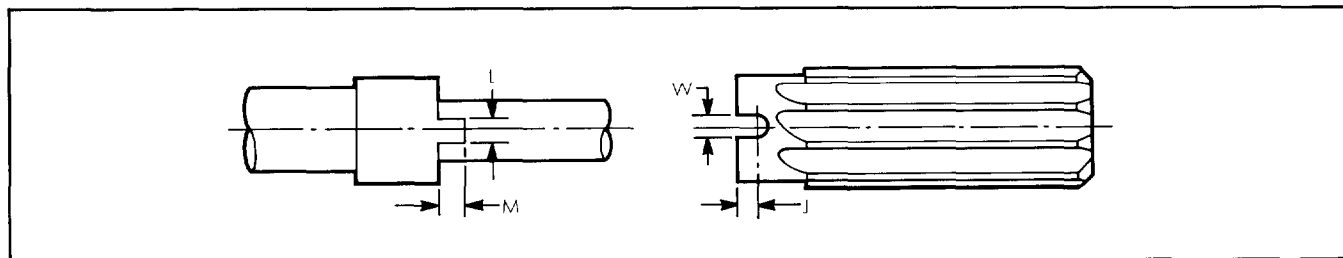


TABLE 9-42
Driving Slots and Lugs for Shell Reamers and Arbors*

Diameter Hole in Reamer at Large End	Number of Arbor	Fitting Reamer Sizes, in.	Dimensions			
			Driving Slot		Lug on Arbor	
			Width W	Depth J	Width L	Depth M
0.375	4	3/4	5/32	3/16	9/64	5/32
0.500	5	13/16 to 1	3/16	1/4	11/64	7/32
0.625	6	1 1/16 to 1 1/4	3/16	1/4	11/64	7/32
0.750	7	1 5/16 to 1 5/8	1/4	5/16	15/64	9/32
1.000	8	1 11/16 to 2	1/4	5/16	15/64	9/32
1.250	9	2 1/16 to 2 1/2	5/16	3/8	19/64	11/32

(ANSI Standard B94.2-1977, published by ASME)

* All dimensions are given in inches. Shell reamer arbor tapers are made to permit a driving fit with the reamer

This type of reamer is recommended only for use in stationary applications.

Combined drills and reamers. Drilling and reaming are usually considered to be separate operations, but in certain applications they may be combined and performed with a single tool with accompanying savings in time and labor. The combined drill and reamer is particularly adapted to drilling and reaming through holes. In order for the reamer to perform properly, the drill must pass completely through the work before the reamer flutes engage the hole. The length of these holes should not exceed the diameter of the drill. Blind holes cannot be satisfactorily processed with this type of tool for two reasons: (1) the reamer should feed faster than the drill feed will permit and (2) the action of the drill may destroy the accuracy of the reamed hole. The drill portion is made from 1 1/2 to 2 times as long as its diameter. The guide bushing should be of proper size to fit the reamer portion and long enough to guide the tool properly when the drill begins to cut. These tools are available

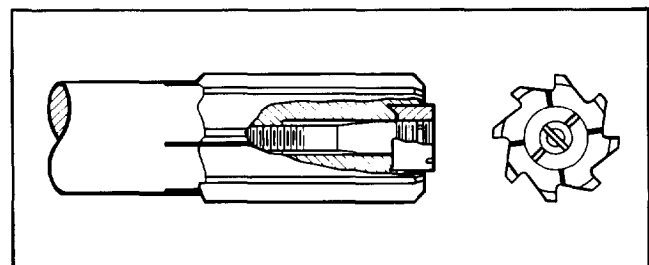


Fig. 9-117 Sectional view of a typical adjustable solid reamer.

in a number of combinations of hand and flute types for both the drill and reamer sections. If a combined drill and reamer is made so that the drill portion is concentric with the reamer portion within very close tolerances, it is not always necessary to drill through and it is possible to drill and ream blind holes to a limited degree.

REAMERS

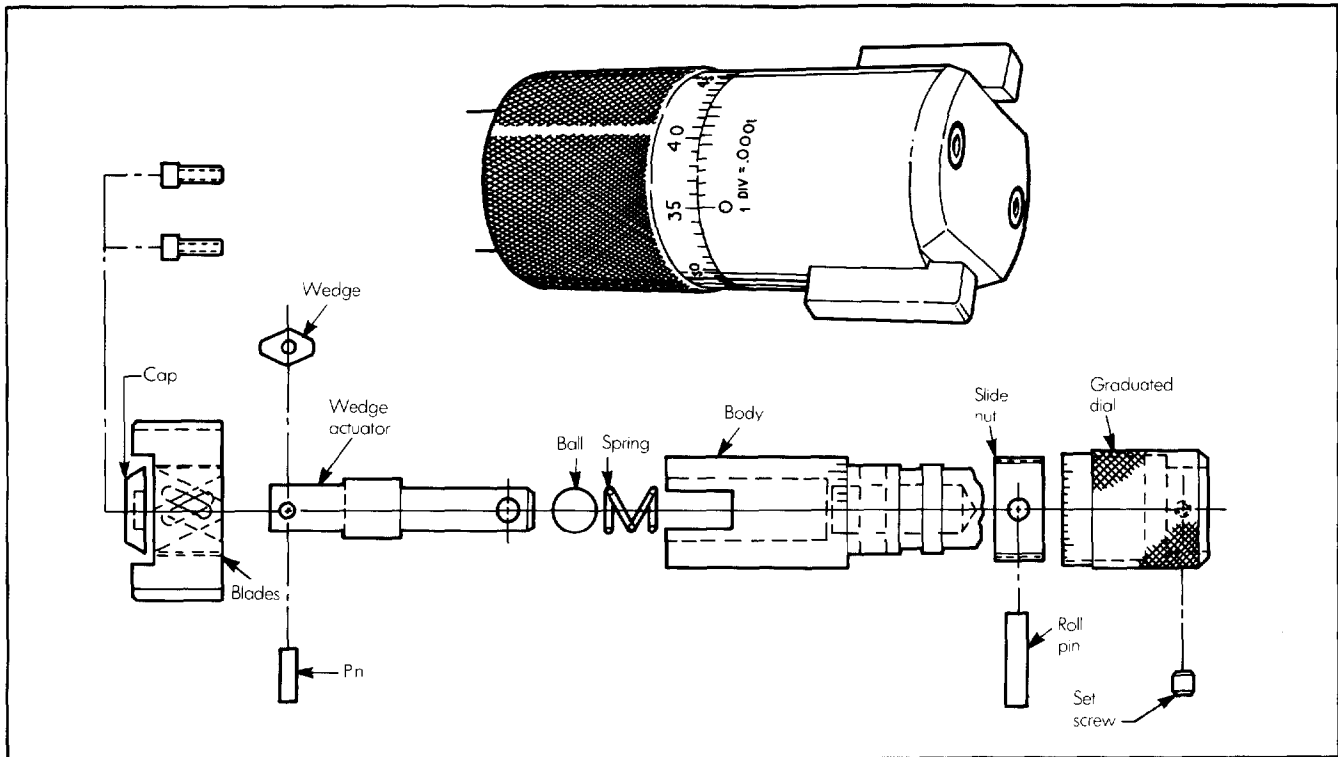


Fig. 9-118 Adjustable floating reamer with micrometer adjusting dial for setting size on machine. (Erickson Div., Kennametal Inc.)

The most common combined drill and reamer is made with the margins of the flutes of the drill section and reamer section coinciding, making it necessary to regrind the tool completely or discard it when the original length of the drill section has been ground away.

The subland combined drill and reamer has a constant-diameter drill margin for the full length of the flute. The reamer flutes, larger in diameter than the drill, are positioned behind the drill margin. When the reamer flutes are reground, the drill margins are not touched. This tool is more expensive to make than the more common combined drill and reamers, but it has much longer life and, in the larger sizes, is more suitable for production work.

A right-hand fluted drill and a straight-fluted reamer are sometimes combined to form a tool that will work to greater precision than other forms of these tools. These tools require a neck between the drill and the reamer sections. For clearing holes with large obstructions in castings, a combination of a right-hand cutting-drilling section with a right-hand flute core-drill section is sometimes used.

Regardless of the type of combined drill and reamer used, it is important to provide enough chip space or chip clearing capability; jammed flutes can be disastrous.

Reamers for tapered holes. The taper reamer, besides being a finishing tool, is a tool for heavy stock removal. At the end of the operation, the cutting edges are engaged in the cut through most of their length. Taper reamers must therefore be constructed as sturdily as possible, cutting edges must be adequately backed up, and flutes must be sufficiently large and have well-rounded bottoms to facilitate chip removal. For machine reaming it has been found that a reamer with few flutes and with cutting edges forming a steep left-hand helix angle

with the axis is the most practical. The number of flutes may vary from two to five in the ordinary range of reamer sizes. The left-hand helix should be at an angle of approximately 45° . A helix angle of greater magnitude than this tends not to improve the quality of the cut but to materially increase the end pressure required to feed the reamer into the work.

Stepped reamer sets may be required for severely tapered holes. A left-hand helix angle reduces the tendency for taper reamers to wedge or lock in the workpiece.

Taper reamers with steep helical angles cannot be used successfully for hand operations because of the end thrust required; but the smoothness of operation in machine reaming outweighs the disadvantages of the necessary end force; the quality of the work is better and longer reamer life is obtained. For hand operation, taper reamers with straight flutes or with not more than 10° of left-hand helix angle have proved to be best under most circumstances. It is important that a taper reamer have sharp edges if it is to cut with the least resistance, even at a slight sacrifice of the taper accuracy. However, a 0.002-0.010" (0.05-0.25 mm) circular land or margin is sometimes beneficial for accuracy and finish, even though higher thrust forces are required.

Taper-pin reamers. Reaming holes for taper pins presents special problems because the holes usually are of small diameters and are comparatively deep. For hand reaming of taper-pin holes, the straight-flute or low-helix-flute type of reamer has found wide acceptance. For machine reaming of taper-pin holes, the helical type of reamer has proved to be more successful than other types; it is more substantial and provides an advantageous cutting action. These reamers are made with left-hand helix of approximately 45° with the axis. The helix is of the opposite hand to the direction of rotation for cutting, and

as a result, the rotary effort (torque) is less, although more end thrust is required. The cutting action somewhat resembles that of a conventional reamer combined with that of a broach. The steep spiral of the helical-type reamer renders it unsuitable for hand operations because of the required high end thrust.

Structural reamers. Structural reamers are frequently referred to by the names of their particular applications; for example, bridge or car reamers. The holes for rivets and bolts in structural steel members, such as I-beams, angles, channels, and plates are usually punched or drilled before the material leaves the shop, and a certain amount of incorrect alignment is usually encountered when the holes are matched up to insert rivets and bolts as the structure is being erected.

The structural or bridge reamer has a sharp taper at the front or entering end to enable it to enter the holes that are not fully aligned. The tapered portion functions to enlarge the holes until the entire body of the reamer can enter and enlarge them to the required size. Although some of this work is accomplished by drill presses in the shop, it more often is done with electric or pneumatic drilling machines at the erection site. This is a very severe operation, as the reamer must perform the combined functions of a reamer and crowbar. Several types of structural reamers of straight and flute styles are in use, each suited to a particular kind of work.

Inserted-blade reamers. Inserted-blade reamers have been developed primarily because of economic considerations. Probably no inserted-blade tool can be quite so rigid, accurate, and dependable as one constructed from a single piece of metal. However, the advantages inherent in solid construction soon are outweighed by the costs involved as sizes increase, and a point is eventually reached at which the construction and use of solid tools are economically prohibitive and built-up construction is required.

Figure 9-119 shows the essential cutting elements of an inserted-blade reamer. The chamfer, *A*, sometimes referred to as the lead, removes excess stock from the walls of the hole, and the cutting point, *B*, sizes the hole to the desired diameter. To prevent the blade from binding or seizing in the hole, clearance, *C*, is provided by tapering the blade behind the cutting point to provide back-taper.

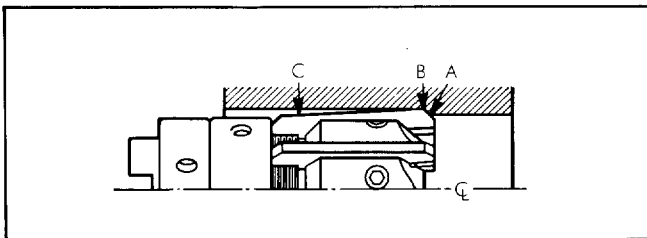


Fig. 9-119 Essential cutting elements of an inserted-blade reamer.

Types and construction. Inserted-blade reamers are commercially available in chucking reamers, shell reamers, and hand reamers. Chucking reamers are regularly furnished in sizes from 5/8 to 3" (16 to 76 mm) diam, shell reamers from 1 1/4 to 6" (32 to 152 mm) diam, and hand reamers from 1 to 3" (25 to 76 mm) diam. Arbors may be furnished with either straight or taper shanks for use with shell reamers, which are available with either straight or taper holes. Straight-hole shell reamers usually are employed for reaming a series of holes in line, for which purpose they are mounted in tandem on a bar. An example is the line reaming of bearings.

An inserted-blade hand reamer ordinarily has a square shank and a pilot sleeve following the blades. Blades in inserted-blade hand reamers usually are furnished without a helix.

Chucking reamers and hand reamers are available in both right and left-hand helix. As a general rule, with conventional right-hand cutting, right-hand helix blades are preferred for roughing and left-hand helix blades for finishing. However, each of these generalities is subject to modification and depends to a great extent upon the conditions prevailing during use. Right-hand helix blades have more of a tendency to dig and pull into the work, especially if conditions are not favorable to extreme rigidity, but they may cut more freely and provide a better finish if conditions are favorable. Left-hand helix blades tend to act as a brake on the reamer and often allow work to be satisfactorily completed on setups which would ordinarily be considered impractical.

The conventional construction of an inserted-blade reamer consists of a body of comparatively soft, tough, workable steel in which blades of a hardened material are positioned. Inserted-blade reamers are comparatively expensive, but cut efficiently. The body is usually constructed from an alloy steel, heat treated to the optimum of hardness, toughness, machinability, and strength necessary to resist both the stresses generated by the cutting action and forces necessary to lock the blades in the body. Blade seats are generally tapered to permit adjustments for wear.

Reamer blades are available in carbon tool steel, high-speed steel, cast alloy, and the various cemented carbides, the latter being furnished either solid or brazed onto steel bodies. Carbon tool steels are nearly obsolete, but are sometimes used in specialized cases, usually in hand or line reamers on soft but greasy metals, such as some of the bronzes and babbitts. In this case the extreme sharpness to which a carbon steel may be ground and honed is the determining factor in its selection.

The majority of reamer blades are made from high-speed steel. The cast alloys are used when a material with a greater production potential than the high-speed steels is required. They are usually classified midway between the high-speed steels and the carbides. The carbides are commonly used for the long-run machining of nonferrous and cast-iron materials.

Methods for holding blades. The assembly must be rigidly locked in place because any displacement of the blade would spoil the work. Of the various locking devices which have been developed, most are modifications of the wedge or screw principle; a number of them are illustrated in Fig. 9-120.

Block reamers. Adjustable block reamers are flat blocks that carry two or more cutting blades and are fitted to slots in suitable bars (see Fig. 9-121). The blades may be made of high-speed steel, cast alloy, or cemented carbide. Block-type equipment may be used generally in the field covered by ordinary reamers and in addition is frequently used to rough bore, semifinish bore, and even face when conditions are favorable. Boring tools are discussed in Chapter 8 of this volume, "Turning and Boring."

For general-purpose use, taper-shank bars for drill presses and straight-shank bars for turret lathes are commercially available. A series of bars may be used in the stations of a turret lathe as needed to perform the required operations on a workpiece. For example, a hole in a cored casting may be rough-bored, semifinish-bored, and reamed, using three bars in three stations of the lathe turret. When not enough stations are available, it is possible to change blocks in one or more of the bars between cuts.

CHAPTER 9

REAMERS

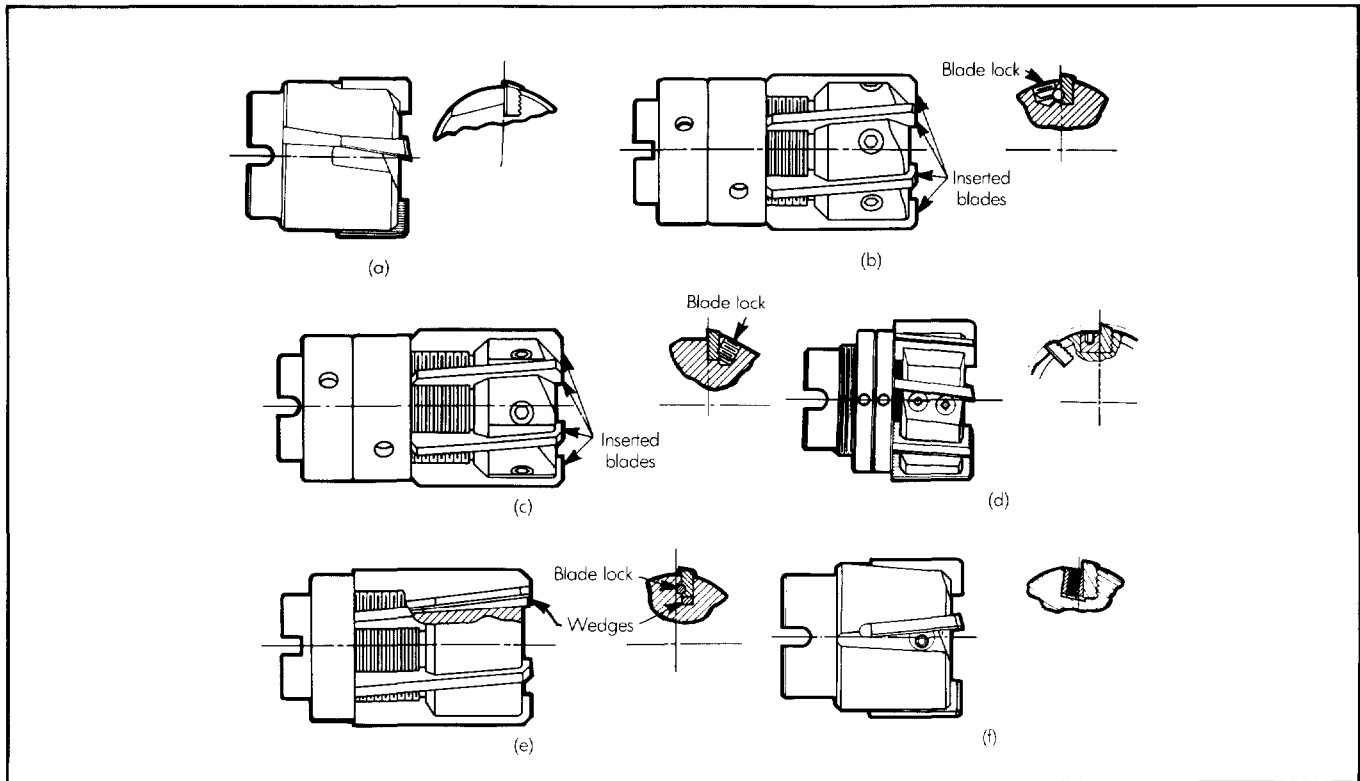


Fig. 9-120 Various methods of holding inserted reamer blades in position in shell reamers: (a) serrated blade wedged in body, (b) pin and screw lock, (c) reversed screw lock, (d) serrated blade and cam lock, (e) wedge and pin lock, and (f) serrated blade and jackscrew lock.

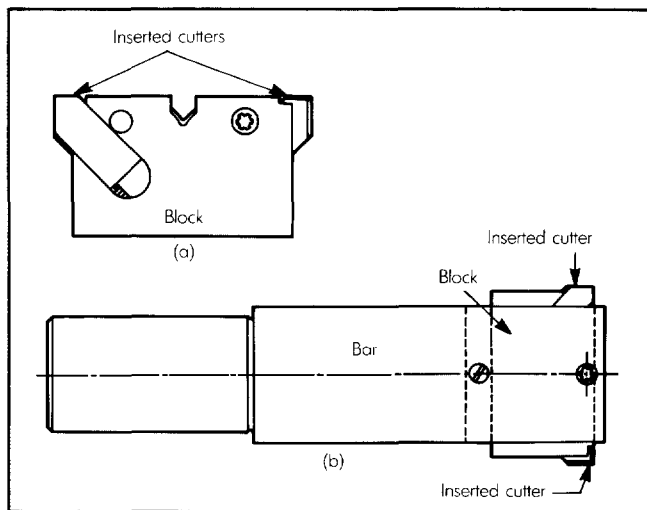


Fig. 9-121 Typical block reamer and bar: (a) block carrying inserted blades; (b) bar with block in place.

Block boring and reaming are particularly adaptable to special bar constructions for use on horizontal boring mills. The inserted-blade block construction adapts itself to all sorts of multiple-diameter in-line boring, counterboring, reaming, and facing operations. Large complicated castings with numerous bores may be handled efficiently with this type of equipment.

Size of blocks. Inserted-blade boring and reaming blocks are commercially available for machining holes from 1-24" (25-610 mm)

diam. Larger tools may be constructed on special order. The blocks range in thickness from 5/16 to 1 1/4" (8 to 32 mm). Each range of blocks considerably overlaps the sizes of those immediately preceding and following it. The thinner blocks are intended for the smaller bore sizes and are mounted on small-diameter bars; larger bars are employed for the thicker blocks.

Block-reamer construction. The block bodies are usually made of alloy steel, heat treated for strength and wearing qualities. Blades incorporate substantially dovetail cross sections and are retained by locking devices similar to those used for inserted-blade machine reamers. Several methods are illustrated in Fig. 9-122. Adjustment screws are provided for advancing the blades for resharpening and for taking the thrust when the reamer is cutting. A key or locking member intended to hold the block in position may also serve to centralize the cutting edges of the blade and hold them equidistant from the bar axis so that both blades will cut equally and produce an accurate bore diameter. Some of the methods used to lock the block in the bar are shown in Fig. 9-123.

A thrust member behind the block, normally retained by a ball-and-spring detent as shown in view a, Fig. 9-123, may be pushed out of the bar by finger pressure when the lock key is loosened slightly. This allows sufficient block movement to clear the lock key for removal from the bar slot. The V-shaped wedge portion of the key enables accurate centralizing when one side of the vee is precision ground until a master block indicates dead central when locked in the bar and is dial indicated on centers.

Withdrawing or loosening the key slightly permits the block to float. This is desirable in most finish-reaming operations with

this type of reamer. It assists in causing the block to cut to exact cutting diameter. However, the block should never float when there are ports or other irregular openings in the bore. Both cutting blades of a floated-block reamer must cut simultaneously; this simultaneous cutting cannot occur when irregular openings are present in the bore. For roughing and semifinishing cuts, the block should be firmly locked in position. A block with centralizing ledges and spring retainers is shown in view *c*, Fig. 9-123. Another method of holding the block in the bar (view *b*) utilizes a tapered hole through which a cone-shaped locking key threaded on the large end is passed. The key is screwed into a locking engagement.

This device provides for contact of the key on one side of the tapered hole in the block, forcing the block into engagement with the thrust plates fastened to each side of the bar at the back end of the slot. The block may be removed from the bar by first removing the key.

Gunbores. These tools, sometimes called gun reamers, were discussed briefly in the preceding section of this chapter on high-pressure coolant drills. Gunbores are related to both boring and reaming because they can be used to remove 0.015" (0.38 mm) or more of material while straightening the hole and improving the surface finish. Gunbores, like gundrills, are single-lip, self-guiding, pressure-coolant tools; they generally follow their own centerline, not the hole being enlarged. Therefore, this tool will not duplicate errors present in the original hole. It is applied in place of more conventional tooling when the replaced process would require peck feed or secondary finishing operations to produce the desired results.

Gunbore designs are based on both external and internal-chip-removal gundrills (see Fig. 9-124). In both categories, there are also designs which flush the chips ahead of the tool, but these designs are limited to use in through holes.

External-chip-removal gunbores. External-chip-removal gunbores, and the related chips-ahead designs, are available in the same sizes and lengths as external-chip-removal gundrills, and the resulting holes have the same characteristics. The external-chip-removal types, under certain conditions, can also remove material to center to form the bottom of a hole. They can also operate as both core drills and counterbores.

Nose grinds are selected for best chip control or to attain a specified bottom form or chamfer. Gunbores can be manufactured to produce two diameters simultaneously. The resulting holes, however, will not be as precise as holes produced in two operations, and the maintenance and tool cost per hole produced is more than double because of the difficulty of regrounding two cutting edges relative to each other.

External-chip-removal gunbores are manufactured in exactly the same way external-chip-removal gundrills are manufactured (see Fig. 9-49; with a special contour, see Fig. 9-50, *b*). Gunbores in a chips-ahead style are manufactured with a ground shank to provide maximum rigidity. The tip is made with only a small flute to provide room for the more complete contour as shown in Fig. 9-125. The gunbore is manufactured with a back-taper with from 0.0004-0.0006 in./in. (or mm/mm) of tip. With a -0.0002" (0.005 mm) manufacturing tolerance, a 0.0008" (0.020 mm) diametral tolerance over the life of the tool can be expected.

The equipment and fixturing required for gunboring are the same as required for gundrilling, except that with gunbores in chips-ahead style, provisions must be made to contain the coolant and chips exiting from the end of the hole (Fig. 9-126).

Internal-chip-removal gunbores. The internal-chip-removal

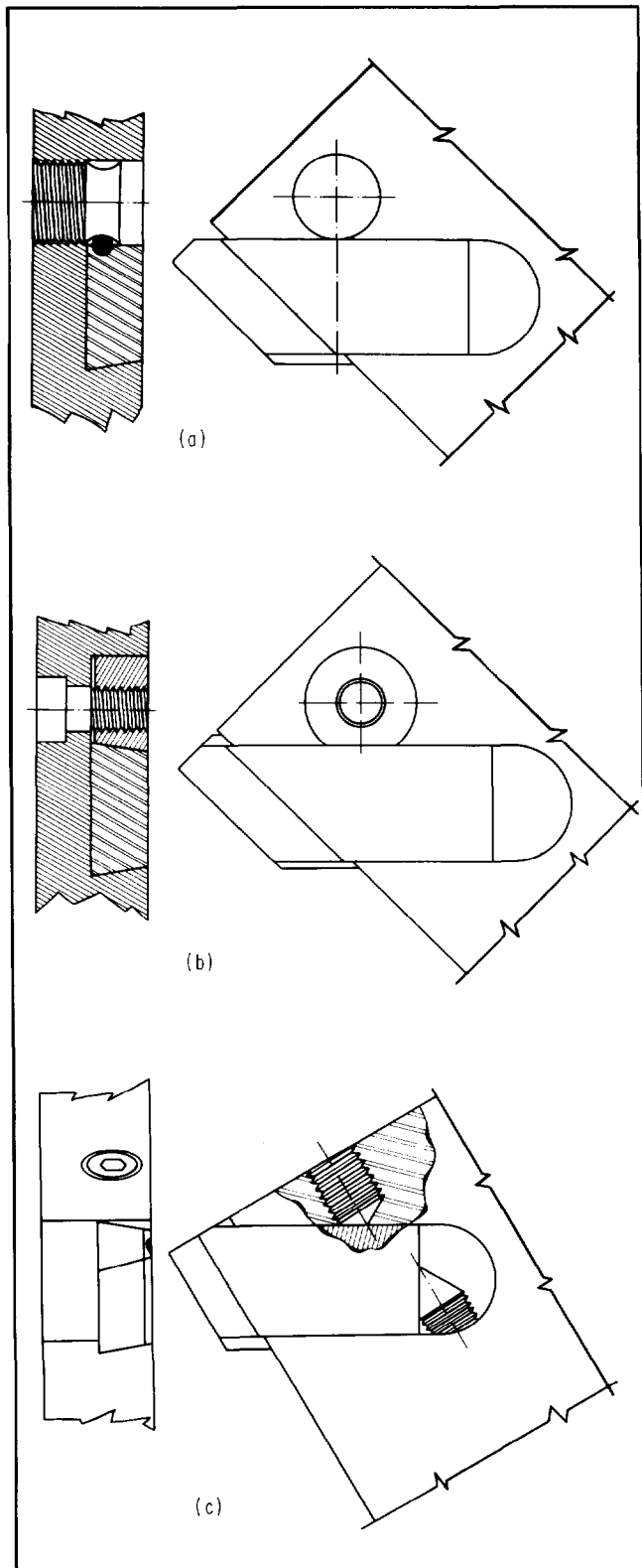


Fig. 9-122 Methods of holding inserted blades in block reamers: (a) screw and pin lock; (b) circular wedge and screw; (c) setscrew lock and adjustment.

REAMERS

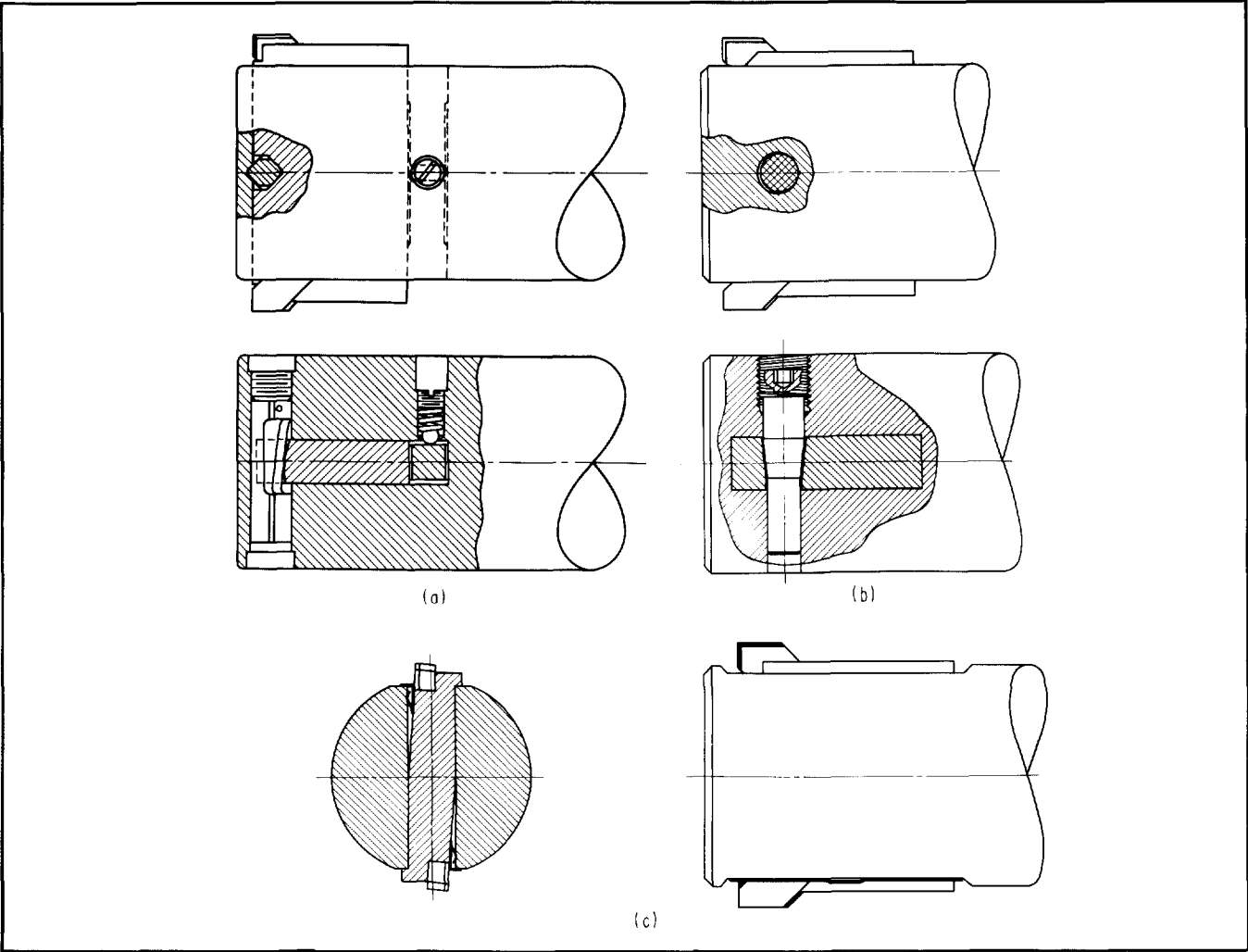


Fig. 9-123 Block-to-bar locking methods: (a) lock with straight-sided wedge and thrust block; (b) block with tapered hole for tapered pin; (c) block with centralizing wedges and spring retainers.

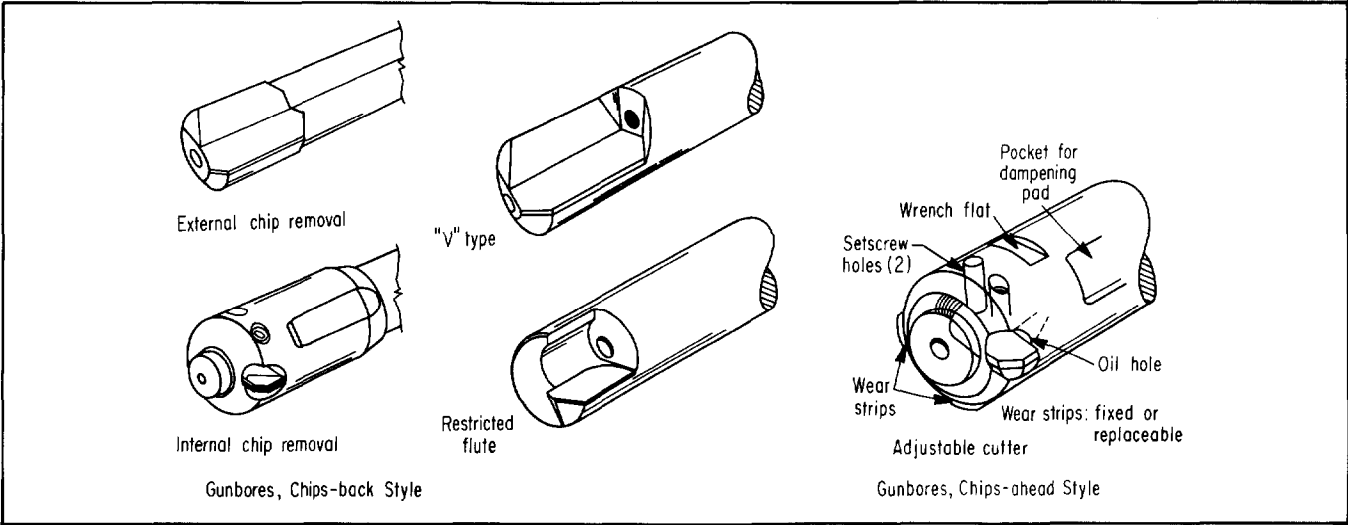


Fig. 9-124 Various designs of gunbores.

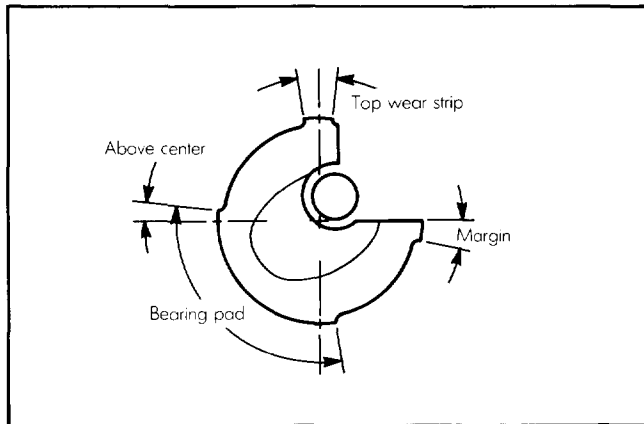


Fig. 9-125 Typical lip contour for chips-ahead gunbore.

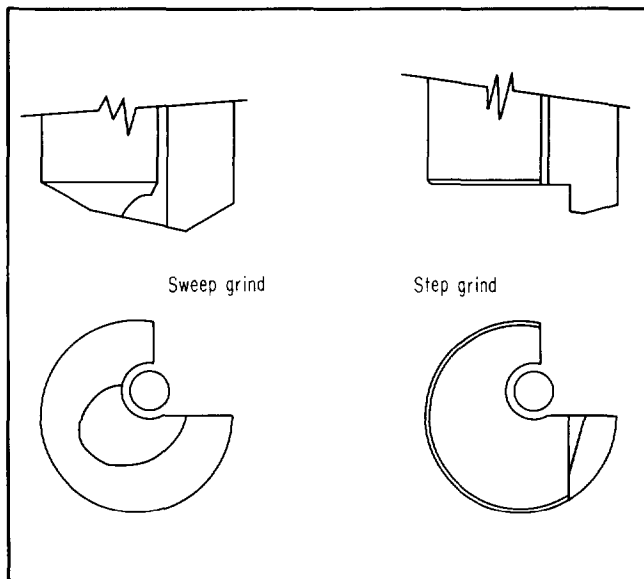


Fig. 9-126 Typical chips-ahead gunbore setups.

gunbore sizes and straightens existing holes, but does not cut to center. The carbide cutting edge may be brazed to the body or may be adjustable for wear. Generally, the tool construction, maintenance, and operation are similar to the internal-chip-removal gundrills described earlier.

Bore reamers. These tools combine boring and reaming in a single operation to minimize problems with respect to hole size, straightness, and finish. Single-point bore reamers, for use in applications for which guide bushings can be used, have a single-point cutting edge on the end of the tool, followed by a reaming section. Multipoint bore reamers are available for applications for which bushings cannot be used.

Coolant-fed reamers. These tools, having means (usually internal passages) for directing coolant to the cutting edges, offer advantages for some applications, particularly when reaming blind holes. In such applications, reduced friction and temperatures at the reamer/workpiece interface decrease wear and lengthen tool life. In some cases, feeds and speeds can be increased and improved accuracies and smoother finishes obtained. The initial cost of coolant-fed reamers is higher, but

increased productivity and improved quality often make them economically desirable.

Special reamers. It is generally more economical to use standard reamers. There are applications, however, for which special reamers are needed. Figure 9-127 presents suggestions for specifying special reamers to tool manufacturers. Suitable allowances should be specified for resharpener purposes and for clearance of the spindle above the guide bushings.

REAMER HOLDERS/DRIVERS

Reamers are commonly held and driven by three-jaw chucks, straight sleeves and setscrews, and for taper shanks, sleeves or sockets. Reamers with adapters for quick-change chucks are used for production applications.

When reamers must guide themselves into previously made holes, they require floating holders to maintain alignment. There are several types of floating holders. Some permit angular float, others permit a parallel (axial) float, and still others permit both angular and parallel float (see Fig. 9-128).

Floating holders have some limitations. If the reamer axis is vertical, floating reamer drives often do a good job of correcting for small amounts of misalignment. When the workpieces rotate, however, as is the case on screw machines, lathes, and some other machine tools, floating holders are sometimes inadequate. This is because relatively large amounts of misalignment are often found on these machines and because the weight of the reamer and holder tend to push the tool into an off-center position.

Some full floating holders, which compensate for both angular and parallel misalignment, are equipped with springs or other components to counterbalance the mass of the holder. A floating holder cannot generally operate both vertically and horizontally and still correct for both angular and parallel misalignment. Application details (vertical or horizontal operation and rotating or stationary tool) should be specified when a floating holder is ordered.

Holder Requirements

Ideally, a floating holder should:

1. Grip the cutting tool reliably at the working end and mount securely and easily into the machine spindle at the opposite end.
2. Transmit cutting torque and shock loads through its mechanism to the machine spindle without damage.
3. Float while cutting forces are present, thus accommodating parallel or angular misalignment, or both. Misalignments of 0.025" (0.63 mm) from the spindle centerline should be allowable.
4. Be durably constructed to withstand normal use (with occasional abuse), while retaining a large percentage of its original efficiency.
5. Have seals and shields to keep cutting fluids and/or chips out of the internal parts, while keeping lubricants inside.
6. Be readily maintainable without the need for special tools or gages for correct reassembly.
7. Be sufficiently low in cost to pay back the purchase price in a year or less of use. Savings are based on reduced setup time, replacement cutters, machine downtime, and rework.
8. Be small in diameter to allow use for close hole spacing requirements.
9. Be short in length to occupy minimum space.

CHAPTER 9

REAMER HOLDERS/DRIVERS

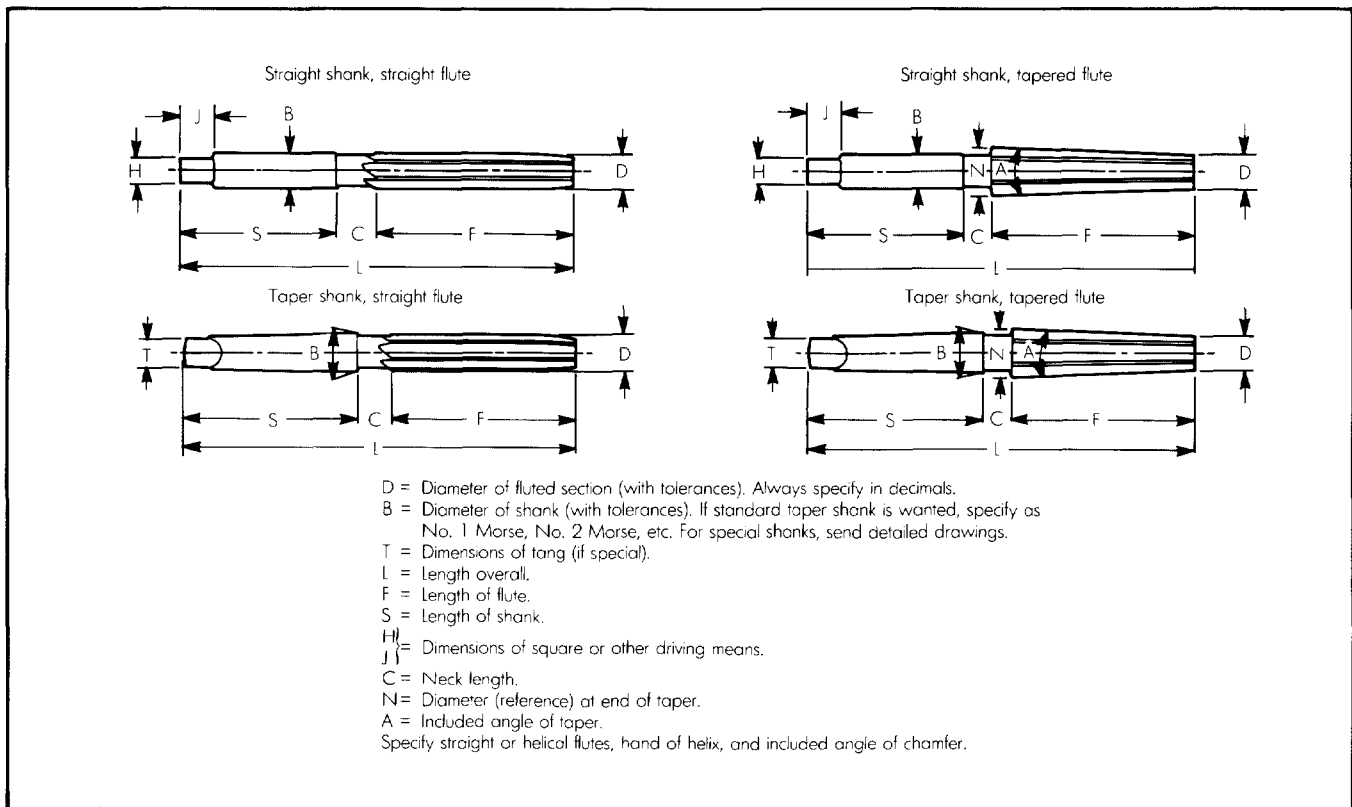


Fig. 9-127 Suggested specifications for special reamers. (National Twist Drill)

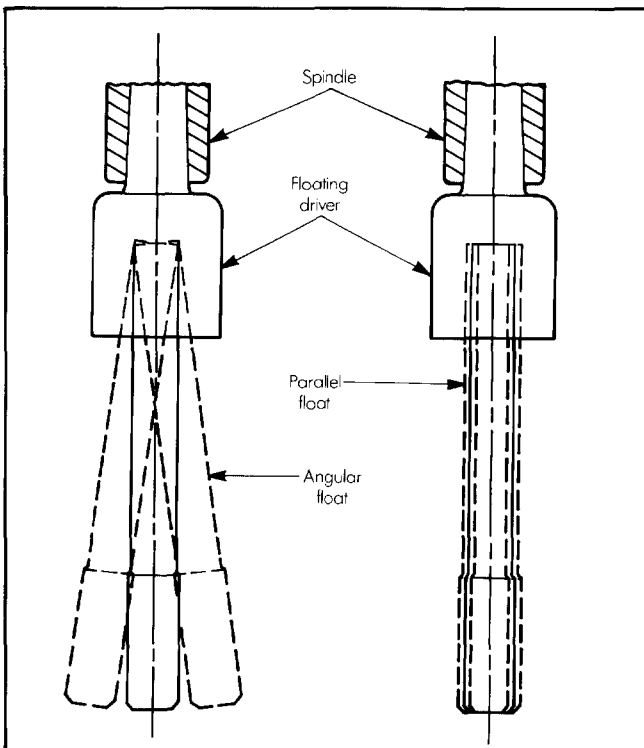


Fig. 9-128 Angular and parallel (axial) float obtained with floating holder/driver for reamers. (Cleveland Twist Drill)

Various Holder Designs

Holders of many different designs are available, but all have internal mechanisms which are essentially specialized couplings. Each design offers certain advantages over the others.

Plain Oldham coupling. The coupling shown in Fig. 9-129, view *a*, is outstanding in terms of low cost, simplicity, and impact load bearing capacity. Its major drawback is the dead spot which occurs for about 17° around each 90° of rotation. This is caused by steel-on-steel frictional characteristics, which can be modified with lubricants or made negligible by the addition of balls or rollers. This, however, increases unit loading which decreases impact loading capacity and durability.

Views *b* and *c* of Fig. 9-129 illustrate variations of the basic Oldham coupling which are intended to reduce friction. This is usually accomplished by the inclusion of antifriction elements (not shown) between the load-bearing components.

Another common variation of the basic four-point coupling (see view *d*) works well until the oscillating balls (which travel only half the misalignment capacity of the holder) wear V-grooves in their travel zone. This results in frictional forces greater than those found in Oldham couplings.

Holders of this type are also available with rollers, but they cannot correct both angular and parallel misalignment. They are primarily designed to correct parallel misalignment.

Gear couplings. As illustrated in Fig. 9-129, view *e*, gear coupling systems distribute driving torque over several teeth in any condition of misalignment. Dead spots are eliminated in this design. Driving torque is not as high as with an Oldham coupling of the same size, but high shock loads are readily absorbed. Long life can be expected due to the multitude of

REAMER HOLDERS/DRIVERS

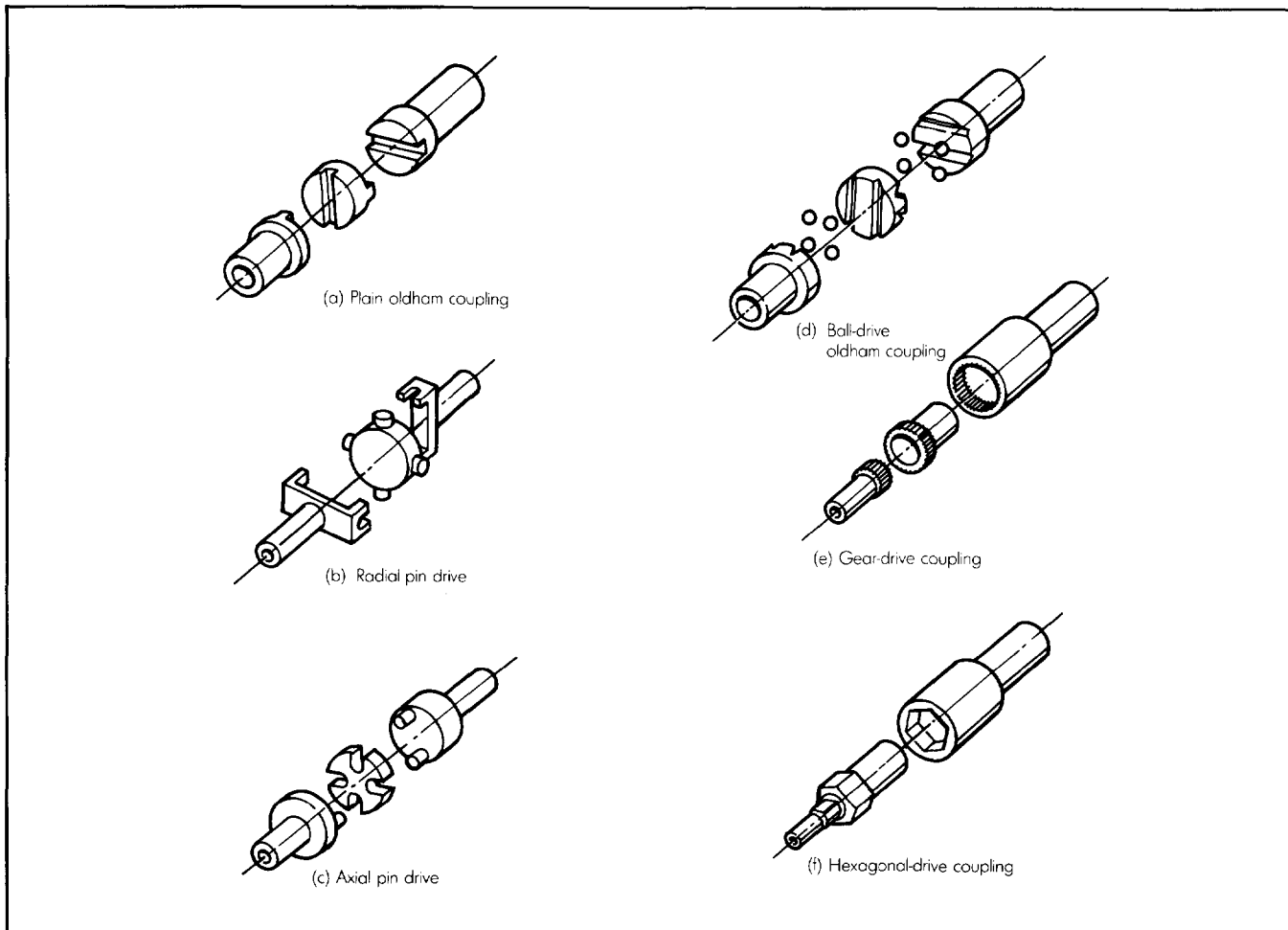


Fig. 9-129 Various types of internal coupling mechanisms used for floating holders. (Industrial Tools Div., Bendix Corp.)

driving elements which provide positive lubrication. Clearances between the telescoping tubular elements may be designed to limit the allowable misalignment. The primary disadvantage of this design is its higher cost.

Hexagonal-drive couplings. A somewhat lower cost design utilizes hexagonal driving and driven members in a telescoping package (see view *f*, Fig. 9-129), similar to the gear drive design. Wear life is usually reduced because the greater clearances required by straight hexagonal splines reduce the ability of the coupling to absorb shock loads. If the hexagonal surfaces are curved longitudinally to minimize clearances, the cost of manufacturing is almost as much as for gear couplings.

Holder Types

Full-floating holders are available in collet, bushing, and adjustable-flange types. Quick-change floating holders consist of a holder and a cartridge which holds the reamer. With the spindle stopped, the cartridge is lifted to remove the tool.

WORKHOLDING FOR REAMING

Jig design and the use of bushings for reaming are essentially the same as for drilling, discussed previously in this chapter. Major functions of the jigs and bushings are accurate locating, supporting, and securing of the workpieces, and precise guiding

of the tools. A difference for reaming is that closer tolerances are generally required on both the jigs and bushings.

Alignment

Close-tolerance reaming requires accurate alignment of the machine spindle with the guide bushing. Using the reamer as a locating device to force the workpiece into alignment is poor practice and can result in excessive wear, chipping, or breakage of the reamer.

When misalignment between the reamer and the hole to be machined amounts to about 30% or more of the stock to be removed, there is often wear on the reamer in the form of a thread. The lead of the thread is equal to the feed rate, and depth is equal to the misalignment. These threads appear to be more readily produced when tough and abrasive materials are reamed. When other materials are reamed, only excessive marginal wear may be noticed.

For precise hole locations, reaming jigs are essential. An ideal arrangement, especially for long holes is to guide the reamer on both sides of the workpiece as illustrated in Fig. 9-130, view *a*. Special piloted reamers are required for this purpose, with the pilots having grooves on their OD's to permit cutting fluid to lubricate the pilots and to assist in chip removal. The jig should be designed so that the pilot enters the bushing before the

WORKHOLDING FOR REAMING

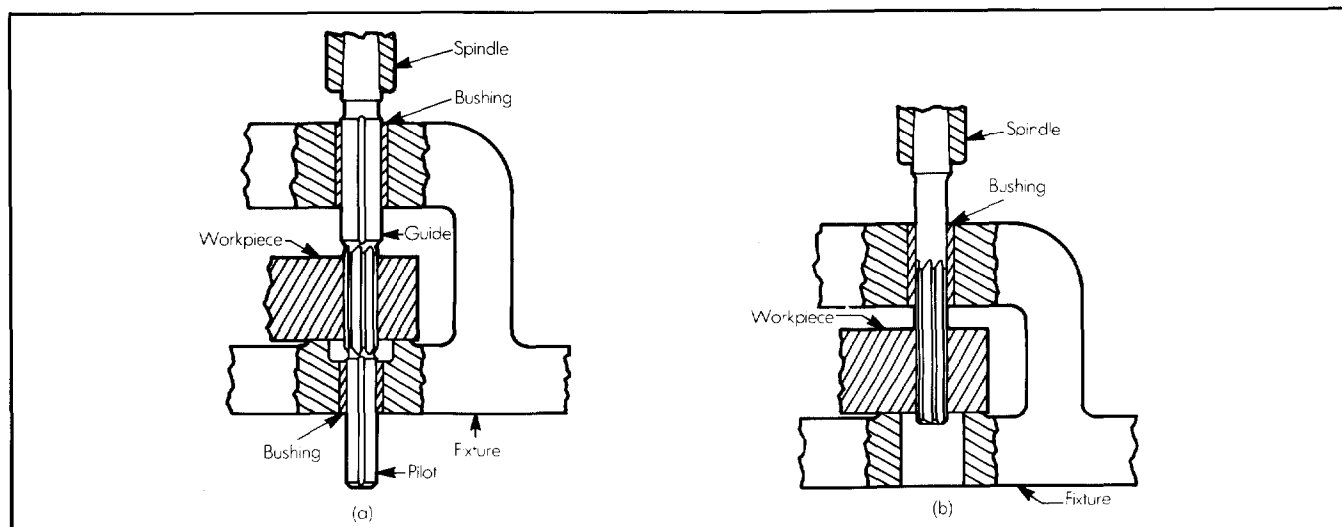


Fig. 9-130 Fixtures for guiding reamers through (a) long holes and (b) short holes.

TABLE 9-43
Tolerance Requirements for Reaming

Tolerance requirement for hole to be reamed, in. (mm)	Tolerance		Remarks
	Reamer OD, in. (mm)	Bushing ID,* in. (mm)	
0.0005 (0.013)	0.0002-0.0003 (0.005-0.008) under high limit of hole	0.0002-0.0003 (0.005-0.008) under high limit of hole	Carbide reamers and bushings necessary
0.001 (0.03)	0.0005-0.0006 (0.013-0.015) under high limit of hole	0.0003-0.0005 (0.005-0.013) under high limit of hole	Carbide reamers and bushings generally necessary
0.002 (0.05)	To mean tolerance of hole to be reamed	0.0005 (0.013) under high limit of hole	HSS reamers and bushings generally satisfactory

(Zagar, Inc.)

* Light lapping of bushing bore to a maximum of 0.0001" (0.003 mm) and selective matching with reamers to be used may be required.

reamer enters the workpiece, and the reamer should remain piloted until the tool finishes its cut. For short holes, the reamer can be guided only at the entry side of the hole (view *b*), with the bushing made to fit the OD of the reamer flutes.

Bushings for Reaming

Bushings for reaming are generally longer than for drilling, usually having a length equal to 3-4 times the reamer diameter. Chip clearance—the distance from the bottom of the bushing to the top of the workpiece—must be sufficient to permit chip ejection and minimize bushing wear, but it is generally less for reaming than for drilling. Many reaming experts recommend that the clearance distance be limited to one-fourth to one-half the tool diameter. Others suggest even less clearance—a maximum of 1/8 to 1/4" (3.2 to 6.3 mm), regardless of the reamer diameter.

Bushing bores must be held to close tolerances for precise reaming. Too small a bushing can result in tool seizure and possible breakage; too large an ID will result in out-of-round or bellmouthed holes. Tolerances that should be maintained on the bushing bores depend upon the tolerances required for the holes to be reamed. The data in Table 9-43 can be used as a guide. The larger the bushing diameter, the greater the tolerance on the bushing bore.

Carbide bushings offer the advantages of minimum wear and maintaining close tolerances. They are widely used for long production runs and for operations in which abrasive materials are being reamed; they eliminate galling in many applications. Roller or ball bearing, rotary-type pilot bushings also offer the advantages of minimum wear and maintaining close tolerances, plus they offer the capability of withstanding high loads.

OPERATING PARAMETERS FOR REAMING

OPERATING PARAMETERS FOR REAMING

Factors that must be established for efficient and economical reaming include the proper cutting speed, feed rate, and cutting fluid to be used. Other important considerations are resharpening the reamers and troubleshooting the operation.

Speeds and Feeds for Machine Reaming

Rearmers are operated at slower speeds and higher feed rates than twist drills of the same diameter. In general, the speeds of reamers may be 65-75% of those for drilling the same material, and the feeds are often two to three times those for drilling. High feed rates are required so that each flute has an adequate chip load.

Reamer feeds depend upon the type of material being reamed and the size or strength of the reamer. When soft materials such as aluminum or brass are being reamed, the feeds may vary from 0.001" (0.03 mm) per flute per revolution for small reamers to 0.005" (0.13 mm) per flute per revolution for large reamers. For hard materials such as hard cast irons and steels with a hardness of 330 Bhn or more, the feed per flute per revolution ranges from 0.0004-0.0015" (0.010-0.038 mm).

Table 9-44 presents some conservative recommendations with respect to speeds and feeds that can be used as starting points for reaming various materials with HSS reamers. Solid and carbide-tipped reamers can generally be operated at higher speeds and feeds (see Table 9-45).

Speeds and feeds for machine reaming vary, sometimes slightly and occasionally a great deal, although the same material is being reamed. These variations may be due to the finish and/or accuracy required, differences in the machine setup, or the design of the workpieces.

Speeds that are too low adversely affect the productivity obtainable with reamers without offering any substantial increase in tool life. Speeds that are too high may cause the workpiece material to cling to the edges and lands of the reamer, resulting in premature dulling of the cutting edges and the production of rough holes. In reaming hard materials, a reamer can be quickly ruined if operated at too high a speed. In all cases, the cutting speed should be adjusted to eliminate any chatter. One common method of selecting a reaming speed for a given job on a specific machine is to increase the speed until chatter occurs and then reduce the speed slightly. Lack of rigidity in the setup may necessitate lower speeds. When close tolerances and a fine finish are required, it is usually necessary to finish ream at considerably lower speeds.

Reamer feeds depend primarily on the type of reamer used, the amount of material to be removed, and the finish required. The more stock to be removed, the lower the feeds should be. Too high a feed tends to produce spiral marks or a wavy finish and may reduce the accuracy of the hole. Too fine a feed allows the reamer to idle in the cut and cause excessive wear and glazing of the hole. At all times the feed must be high enough to permit the reamer to cut rather than rub or burnish.

Block-type reamers. Cutting speeds for block reamers should be relatively low. When attempting to establish the proper operating parameters for a specific application, it is safer to increase the feed than the speed.

For reaming with HSS block tools, a safe starting speed of 25-50 sfm (7.6-15 m/min) should be used, keeping the speed low for the harder materials. A safe starting feed is 0.008-0.012 ipr (0.20-0.30 mm/rev). If satisfactory size and finish are obtained with no apparent overloading of the tools or machine, the feed

should be increased until it is the maximum obtainable without sacrificing any desirable results.

The conditions required for the best performance of the cast alloy and carbide block reamers vary too much to permit recommendations similar to those given for HSS and carbide reamers. Each individual job and the conditions it presents require individual study and planning.

Gun reamers. Speeds and feeds for gun reamers (gunbores) are basically the same as for gundrills (see Table 9-11). Gun reamers of the chips-back type require the same coolant pressure as gundrills, while gun reamers of the chips-ahead type require pressures of only 100-200 psi (690-1380 kPa). Whip limitations previously discussed (Table 9-11) do not apply to gun reamers of the chips-ahead type because of their round shanks.

Coolant-fed reamers. These tools can often be operated at higher speeds and feeds because of reduced friction and temperatures at the reamer/workpiece interface. Improved accuracies and smoother finishes are obtained in some applications.

Cutting Fluids for Reaming

For reaming operations, emphasis in the selection of a cutting fluid is usually based on the finish produced rather than the cooling properties of the fluid. Lubricating properties of the fluid, however, are also important to reduce frictional heat, improve the finish obtained, and reduce power requirements. In general, cutting fluids used for drilling are usually satisfactory for reaming. The properties of various cutting fluids and recommendations for their application, filtration, and maintenance are discussed in Chapter 4 of this volume, "Cutting Fluids and Industrial Lubricants." Filtering of the cutting fluids is important, especially when reaming cast irons and performing gunreaming.

Practically all materials require a cutting fluid for reaming, but cast irons are being reamed successfully dry. A vacuum or air blast is often used to remove the dry chips, but the use of air blasts requires guarding to protect the operator. In some applications, however, the cutting edges and margins pick up the cast iron and produce poor quality holes. In cases such as these, good results are being obtained by using a coolant-air mist.

Regrinding Reamers

Great care must be taken when reamers are being sharpened because of the accuracy required. Most finishing reamers must be sharpened to an accuracy of 0.0002" (0.005 mm) on the diameter, since they are used for sizing holes. In measuring a reamer, every set of opposing blades should be checked. Blades may be unevenly spaced to reduce chatter and to produce a better surface finish. Adjustable-style reamers must be set several thousandths of an inch oversize before grinding.

Rearmers should be resharpened before they become too dull. This minimizes the amount of stock that has to be removed during regrinding and prolongs the useful life of the tool. In adhering to this policy, sharpening should be limited to the entering taper or chamfer on the reamer. The chamfer must be ground concentric with the OD of the reamer. When sharpening the chamfer only, it is important that enough stock be ground to remove all traces of OD wear. Clearance on the chamfers of reamers varies with the job, with an angle of 7-12° being common. A secondary chamfer relief angle may be necessary if the flutes are relatively wide, as is common on expansion and carbide-tipped reamers, but is not normally required on solid-type HSS reamers. If it is necessary to regrind the periphery of

CHAPTER 9

OPERATING PARAMETERS FOR REAMING

TABLE 9-44
Recommended Speeds and Feeds for High-Speed Steel Reamers

Reamer Diam, in.	Speed, sfm (mm/min)	Feed Rate, ipr (mm/rev)	Speed, sfm (mm/min)	Feed Rate, ipr (mm/rev)	Speed, sfm (mm/min)	Feed Rate, ipr (mm/rev)	Speed, sfm (mm/min)	Feed Rate, ipr (mm/rev)
	Low-Carbon Steels, 120-200 Bhn		Low-Alloy Steels, 200-300 Bhn		High-Alloy Steels, 300-400 Bhn		Soft Cast Irons, 130 Bhn	
1/4	75 (23)	0.005 (0.13)	60 (18)	0.004 (0.10)	25 (7.6)	0.004 (0.10)	100 (30.5)	0.006 (0.15)
3/8	75 (23)	0.007 (0.18)	60 (18)	0.007 (0.18)	25 (7.6)	0.006 (0.15)	100 (30.5)	0.008 (0.20)
1/2	90 (27)	0.008 (0.20)	70 (21)	0.008 (0.20)	30 (9)	0.007 (0.18)	120 (36.6)	0.010 (0.25)
5/8	90 (27)	0.010 (0.25)	70 (21)	0.010 (0.25)	30 (9)	0.008 (0.20)	120 (36.6)	0.012 (0.30)
3/4	90 (27)	0.012 (0.30)	70 (21)	0.012 (0.30)	30 (9)	0.010 (0.25)	120 (36.6)	0.015 (0.38)
7/8	90 (27)	0.013 (0.33)	70 (21)	0.013 (0.33)	30 (9)	0.010 (0.25)	120 (36.6)	0.017 (0.43)
1 and over	90 (27)	0.020 (0.51)	70 (21)	0.015 (0.38)	30 (9)	0.012 (0.30)	120 (36.6)	0.020 (0.51)
	Medium Cast Irons, 175 Bhn		Hard Cast Irons, 230 Bhn		Malleable Cast Irons		Cast Brasses and Bronzes	
1/4	85 (26)	0.006 (0.15)	70 (21)	0.006 (0.15)	55 (17)	0.004 (0.10)	150 (46)	0.005 (0.13)
3/8	85 (26)	0.008 (0.20)	70 (21)	0.008 (0.20)	65 (20)	0.007 (0.18)	150 (46)	0.007 (0.18)
1/2	95 (29)	0.010 (0.25)	80 (24)	0.010 (0.25)	65 (20)	0.008 (0.20)	170 (52)	0.009 (0.23)
5/8	95 (29)	0.012 (0.30)	80 (24)	0.012 (0.30)	65 (20)	0.010 (0.25)	170 (52)	0.012 (0.30)
3/4	95 (29)	0.015 (0.38)	80 (24)	0.014 (0.36)	65 (20)	0.012 (0.30)	170 (52)	0.016 (0.41)
7/8	95 (29)	0.017 (0.43)	80 (24)	0.016 (0.41)	65 (20)	0.013 (0.33)	170 (52)	0.017 (0.43)
1 and over	95 (29)	0.020 (0.51)	80 (24)	0.016 (0.41)	65 (20)	0.015 (0.38)	170 (52)	0.020 (0.51)
	Cast Aluminum and Zinc Alloys		Plastics		Stainless Steels		Titanium Alloys	
1/4	140 (43)	0.006 (0.15)	70 (21)	0.005 (0.13)	25 (7.6)	0.002 (0.05)	30 (9)	0.006 (0.15)
3/8	140 (43)	0.008 (0.20)	70 (21)	0.007 (0.18)	25 (7.6)	0.004 (0.10)	30 (9)	0.008 (0.20)
1/2	160 (49)	0.010 (0.25)	80 (24)	0.008 (0.20)	30 (9)	0.006 (0.15)	40 (12)	0.010 (0.25)
5/8	160 (49)	0.012 (0.30)	80 (24)	0.010 (0.25)	30 (9)	0.008 (0.20)	40 (12)	0.010 (0.25)
3/4	160 (49)	0.015 (0.38)	80 (24)	0.012 (0.30)	35 (11)	0.010 (0.25)	40 (12)	0.012 (0.30)
7/8	160 (49)	0.017 (0.43)	80 (24)	0.013 (0.33)	40 (12)	0.012 (0.30)	60 (18)	0.015 (0.38)
1 and over	160 (49)	0.020 (0.51)	80 (24)	0.020 (0.51)	45 (14)	0.014 (0.36)	60 (18)	0.020 (0.51)

(TRW Tools Div.)

OPERATING PARAMETERS FOR REAMING

TABLE 9-45
Suggested Speeds and Feeds for Carbide Reamers

Material to be Reamed	Cutting Speed,* sfm (m/min)	Feed Rate, ipr, (mm/rev)
Steels, all types:		
250 Bhn or less	65-400 (19.8-122)	0.005-0.010 (0.13-0.25)
250-400 Bhn	45-150 (13.7-45.7)	0.004-0.006 (0.10-0.15)
400 Bhn or more	35-85 (10.7-25.9)	0.002-0.004 (0.05-0.10)
Cast irons:		
Malleable and cast	60-190 (18.3-57.9)	0.006-0.012 (0.15-0.30)
Nonferrous metals and nonmetallic materials	100-300 (30.5-91.4)	0.005-0.015 (0.13-0.38)

(TRW Tools Div.)

* When using reamers 1/4" or less diam, it is advisable to reduce speeds shown by about 25%. With reamers 3/4" or more diam, speeds can be increased about 25%.

the reamer, the correct circular land width must be maintained, as well as the correct back-taper.

Reamers can be mounted on centers or supported in close-fitting, concentric bushings on conventional tool and cutter grinders. Care is necessary to ensure that the chamfered front cutting edges are sharpened to equal height and with the same amount of relief. Grinding wheels used should be of fine grit and free cutting to produce keen, smooth cutting edges. Flaring-cup type grinding wheels are generally used.

On left-hand helix tools, it is important that the reamer be ground to leave a positive effective rake angle. This can best be done by reducing the 45° chamfer to about 30° or less. If the reamer has a very high left-hand helix angle (20° or more), a grind similar to the one shown in Fig. 9-111 is generally used.

It is occasionally desirable to eliminate the chamfer and to grind the reamer to cut square across the end. Reamers sharpened in this manner (see Fig. 9-116) have a cutting action much like that of end mills, discussed in Chapter 10, "Milling." Reamers ground with a square end-cut are sometimes required to correct a bent or misaligned hole, to bottom a hole, or to produce a square shoulder.

Grinds which cut only to the bottom of the flutes are relatively simple to produce. Grinding of reamers to end cut to smaller diameters or to the centers of the tools, however, is more complicated and requires notching of the reamer ends.

Only the rake surfaces of blades for adjustable floating reamers must be ground. The blades are sharpened in sets, using a fixture that holds the worn blades in position for resharpening. The fixture has a built-in positive rake (3 1/2°) and may be mounted on the table of a grinding machine parallel to the wheel centerline.

Sharpening carbide-tipped reamers. Carbide-tipped reamers may be reground on a standard cutter grinder with a 200-300 grit resinoid-bonded diamond cup wheel. The grinding wheel should be rotated in a direction to cut from the face of the tip toward its back to minimize the possibility of chipping or flaking the carbide (see Fig. 9-131). The use of a cutting fluid

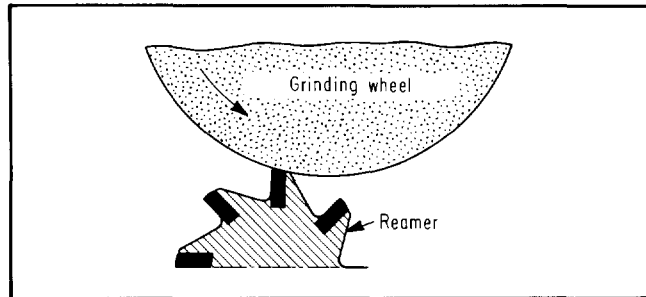


Fig. 9-131 Proper rotation of grinding wheel in relation to carbide tips on a reamer.

during grinding lengthens grinding-wheel life and lessens the danger of cracking a surface or crazing the carbide from overheating. Carbide tools should never be quenched in water after grinding.

In general, cylindrical regrinding of carbide reamers should be done only when absolutely necessary. Usually only the chamfer will need regrinding. Occasionally, the face of the tips will need a light polishing to remove any rounded edges or built-up portions.

Clearance on the chamfers of carbide reamers is usually less than that used on HSS reamers. A clearance angle of 5-8° is normal practice.

For very small diameter reductions (up to about 0.0005", 0.013 mm), when it is not practical to remove such a small amount with a tool and cutter grinder, it is necessary to lap the reamer. The lapping procedure is as follows:

1. Drill and ream a piece of cast iron or mild steel slightly deeper than the length of the carbide insert. The reamer to be lapped should be used to ream the hole.
2. Mix 180 or 240-grit diamond dust with a fine lightweight oil to form a paste solution. Apply this to the cylindrical land of the carbide inserts only. Insert the reamer in the hole and rotate it backward for 10-12 complete revolutions.
3. Remove the reamer, wipe two opposite flutes, and check for size. If necessary, repeat the lapping operation, adding diamond paste only when necessary until size is achieved.

Sharpening inserted-blade reamers. Three distinct grinding operations are involved when an inserted-blade reamer is sharpened: (1) circular grinding, (2) backing off, and (3) grinding the chamfer. The reamer should be mounted on centers so that the grinding wheel will revolve against the cutting edge (not with it) to avoid burning and featheredging. A cup or disc wheel should be employed to avoid hollow grinding.

To obtain accurately reamed holes within close limits, the reamer must be resharpened following any change in blade position (adjustment). This is because inserted-blade reamers are not considered adjustable within the meaning usually applied to hand tools. Leads may be reground without changing blade position at more frequent intervals than radial grinding.

For average work conditions and also for rough reaming of steel and tough bronze, the reamer should be ground as follows (Fig. 9-132):

1. Back-taper diameter at D should be 0.001 in./in. (or m/m) less than the cutting diameter at the end of the blade length.

OPERATING PARAMETERS FOR REAMING

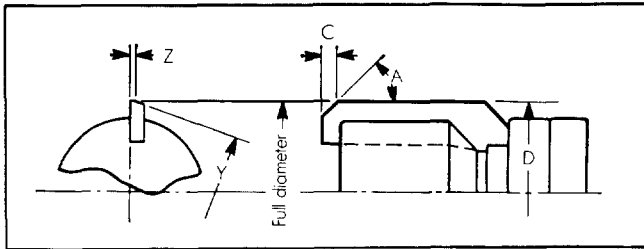


Fig. 9-132 General-purpose grind applied to an inserted-blade reamer.

2. Diameter across the heel of blade, Y , should be 0.003-0.005" (0.08-0.13 mm) less than the minimum cutting diameter.
3. Length of lead, C , should be about 1/16" (1.6 mm), depending on reamer size.
4. Angle of lead, A , should be 45° and backed off with clearance.
5. The peripheral cutting edge should be backed off close to the edge at Z , leaving a 0.005-0.015" (0.13-0.38 mm) cylindrical land.

When a reamer with two lead angles is ground, the main lead angle, A , as shown in Fig. 9-133 remains constant at 45°, as in the general-purpose grind, but it is necessary to vary the second lead angle, B , to obtain the best cutting conditions in various metals. For cast iron, bronze, aluminum, and most nonferrous alloys, angle B , should be 3° and its length, C , should be from 3/16 to 5/16" (4.8 to 7.9 mm). For steel, semisteel, and malleable iron, angle B should be 5° and its length C from 1/32 to 3/32" (0.8-2.4 mm).

For free cutting, both lead angles should be backed off. Back-taper, diameter across heel, and width of land should be nearly the same as used in the general-purpose grind, with the elements varying as conditions indicate. A maximum of back-taper should accompany a relatively wide land to assure sufficient hole clearance. When chips adhere to the land and cause a rough hole to be produced, it is an indication that the land is too wide.

A positive rake assists in reaming soft, stringy materials such as steel tubing and pipe, copper, soft aluminum, and the more stringy of the nonferrous alloys. One side of the rake angle, X , in Fig. 9-134 should blend into a radius at the body of the reamer to force the chip to curl. The rake angle should increase directly as the softness of the material to be reamed increases. For steel tubing and pipe, angle X should be about 5°, varying to as much as 10° for copper and aluminum.

When the cutting edge of the reamer blade picks up material, or when the hole produced is torn and rough, it may be advisable to increase the rake angle. Careful experimentation is sometimes necessary to develop the rake angle best suited to a particular application.

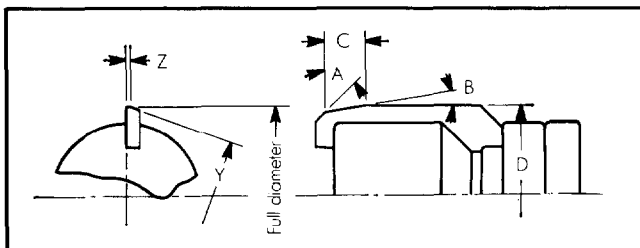


Fig. 9-133 Application of two lead angles to a reamer blade.

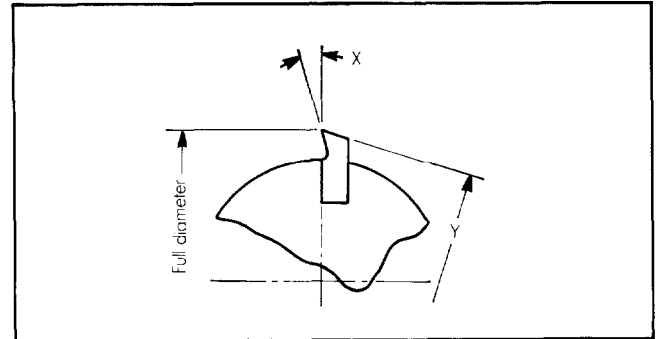


Fig. 9-134 Rake angle and radius applied to a reamer blade.

Sharpening block reamers. Although some manufacturers provide centers in blocks for regrinding and checking, it is customary to regrind most blocks in fixtures specifically designed for the purpose. As a general rule, the fixture corresponds to a short section of the bar in which the block is to be used; has hardened, ground, and lapped centers; and provides means for holding the blocks by the same surfaces as when the blocks are in use (Fig. 9-135). After circular grinding, the blades may be backed off in a manner similar to that for multiple-tooth cutter or reamer. Fixtures are available with additional eccentric centers to enable grinding of a curved or eccentric relief instead of a straight clearance.

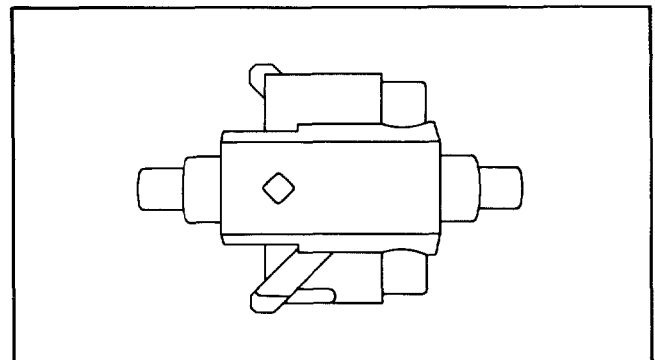


Fig. 9-135 Typical grinding fixture with block reamer in place.

The following general-purpose grinds have been found to be practical in establishing a starting point from which specific grinds may be developed to suit definite applications (Fig. 9-136):

1. Circular grind to full diameter with 0.0005-0.001" (0.013-0.03 mm) back-taper.
2. Back off in general to the same specifications as those used for regular reamers. The heel of the blade, A , should be 0.003-0.006" (0.08-0.15 mm) less than the full diameter, keeping the land 0.005-0.015" (0.13-0.38 mm) wide for cast iron and bronze, and from 0.020-0.030" (0.51-0.76 mm) for steel and copper. Increase the back-taper slightly when wide lands are used.
3. Grind a 1/32 to 1/16" (0.8 to 1.6 mm) wide x 45° lead at the cutting point.
4. If the block is used for roughing and the depth of cut is greater than the lead (ground in step 3), grind a 3-5° face clearance angle at E .

OPERATING PARAMETERS FOR REAMING

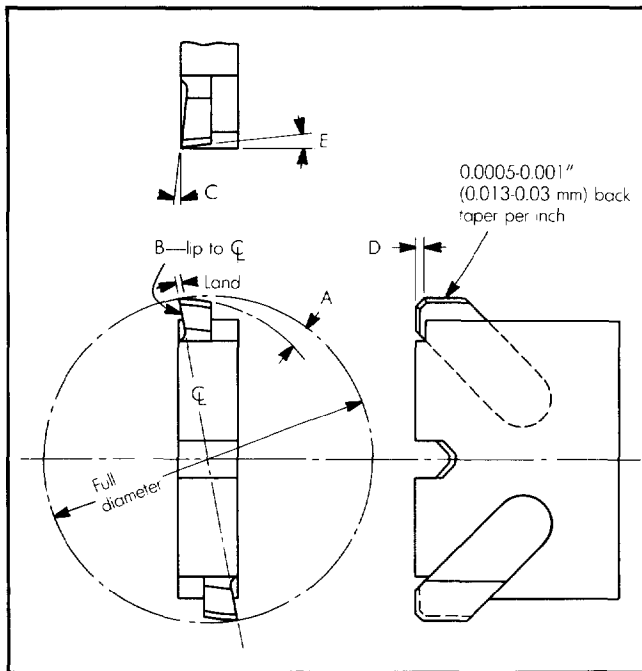


Fig. 9-136 General-purpose method for grinding block reamers.

Applying Reamers

Of major importance for successful and economical reaming is the selecting of the proper tools, machines, fixtures, and operating parameters. Selection depends primarily upon the following:

1. Material to be reamed, including its composition, hardness, and surface condition.
2. Holes to be reamed, including diameter, depth, accuracy, surface finish, and production requirements.
3. Machine and setup used, including type, condition, and rigidity.

Minimizing chatter. As is the case with other cutting tools, reamers do not work well when they chatter and can cause poor finishes and uneconomical operation. Carbide-tipped reamers in particular cannot withstand even momentary chatter at the start of the hole as this can result in chipping of the cutting corners. Major causes of chatter are lack of rigidity in the machine, the fixture, or the workpiece or improper design of the reamer.

Chatter may be reduced in several ways, including the following:

1. Improving machine rigidity by removing any looseness, eliminating backlash, replacing bearings, etc.
2. Increasing jig or fixture rigidity. The use of close fitting bushings, 0.0002-0.0003" (0.005-0.008 mm) over maximum reamer diameter, is helpful.
3. Increasing rigidity of the toolholder by eliminating looseness and reducing or removing float.
4. Reducing the cutting speed and/or increasing the feed rate.
5. Using power feed instead of hand feed.
6. Providing a small chamfer at the start of the hole before reaming.
7. Changing or modifying the design of the reamer.

When the holes are short, fairly large in diameter, and readily accessible, short reamers of comparatively large cross section can be used, thereby minimizing the chances of chatter. When holes are long, small in diameter, or relatively inaccessible, the avoidance of chatter is more difficult. The usual remedy is to use a reamer having helical flutes or pilots for supporting one or both ends of the flutes, or to change the speed/feed combination. Sometimes a combination of these remedies may be necessary.

Helical flutes are sometimes effective in reducing chatter. If possible, sufficient helix should be provided so that two or more flutes overlap in the length of the hole being reamed. When the helix angle is opposite the hand of cut, however, it should not be more than necessary, because the steeper the angle the more end pressure required to feed the reamer through the workpiece. An exception to this is in reaming tapered holes, because best results are obtained with a steep helix angle. The direction of helix should be opposite that of the cut to prevent the reamer from pulling itself through the hole ahead of the feed.

Pilots on reamers, supported by bushings in the fixtures, can improve alignment and minimize chatter. Diameters of the pilots should be as large as possible to provide maximum rigidity to the reamers and prevent torsional deflection and chatter.

Other methods of minimizing chatter include reducing the overhang of the reamer or spindle and using smaller lead-angle clearances on the reamer. For carbide-tipped reamers, chatter can often be reduced or eliminated by grinding a secondary lead on the carbide tips, as shown in Fig. 9-137. Reamer chatter can be eliminated in some applications by changing from a straight-fluted to a helically-fluted tool.

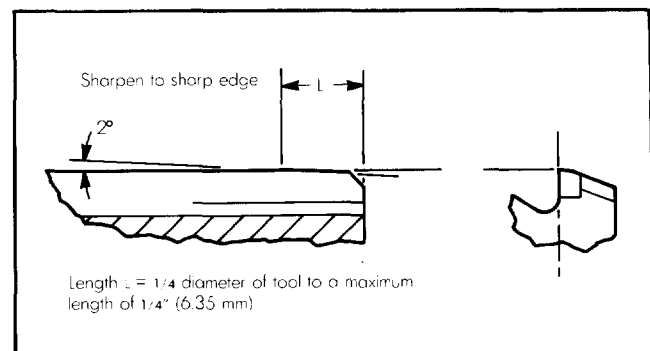


Fig. 9-137 Grinding a secondary lead on carbide tips of reamers may reduce or eliminate chatter. (TRW Tools Div.)

Effect of material reamed. The physical characteristics of the workpiece must be considered when a reamer is chosen for a particular application. Reamers for nonferrous metals, such as brass and aluminum, are cleared to a sharp edge or provided with a wider cylindrical margin than for steel; reamers for steel are cleared to within 0.008-0.015" (0.20-0.38 mm) of the cutting edge, leaving a cylindrical surface that aids materially in prolonging the life of the reamer.

Reamers for magnesium alloys, die-cast metals, and aluminum alloys. For these metals, reamers with left-hand helical flutes with little or no cylindrical margins and radial relief of 5-6° promotes shear cutting, which is desirable for aluminum alloys. Cylindrical margins are necessary for accurate reaming, but they should be reduced to 50% or less of normal width.

Reamers for Monel metal, nickel-chromium alloys, and stainless steels. For these materials, reamers with right-hand

OPERATING PARAMETERS FOR REAMING

helical flutes, radial cutting face, and narrow margins have proved most successful.

For all applications it is essential that reamers be kept sharp.

Use of hand reamers. When the workpiece is reasonably rigid and stock removal small, hand reaming can be performed by rotating the reamer by means of a double-end tap wrench applied to the driving square of the reamer. This type of wrench permits a balanced drive and is preferred to a single-end wrench, which can cause misalignment between reamer and hole axes.

Hand reamers should be rotated slowly and evenly, allowing alignment with the holes to be reamed. Wrenches should be large enough to permit a steady torque which will help to control vibration and chatter. The feed should be steady and large compared to that used in machine reaming. Feeds to one-fourth of the reamer diameter per revolution are not unusual.

When workpieces are small enough to be handled with ease, it is often advisable to place the reamer vertically in a vise and rotate the work down over the reamer by hand. If the workpiece is very light, a workholding device may be required to provide enough mass to dampen vibrations. The holding device should have two opposed handles, large enough in diameter to permit a steady, controlled torque.

Hand reamers should never be rotated backwards to remove them from the holes because this causes premature dulling of the tools. If possible, the reamers should be passed through the holes and removed from the opposite sides of the workpieces without stopping rotation. If this is not possible, the reamers should be withdrawn without stopping forward rotation.

Machine reaming with stationary tools. When reamers are used in lathes, screw machines, and other machine tools in which they are stationary and the workpieces revolve, misalignment between the reamer and workpiece/machine spindle axes is a common problem. Misalignment can result from many causes, including improper machine leveling, incorrect location or adjustment of tool slides, errors in the indexing mechanism, worn ways on the machine, and worn or dirty holes or sleeves in the toolholder. The misalignment can be minimized by correcting these causes or by using floating holders, discussed previously.

Reducing bellmouthing. A common problem in reaming is bellmouthing—holes being oversize at their entry ends. This condition always reflects misalignment and can be minimized by improved guiding by means of accurate bushings and pilot surfaces or by using floating holders. Bellmouthing can also be reduced by decreasing the bearing length of the cutting portion of the reamer.

Two ways of doing this are illustrated in Fig. 9-138. The method shown in view *a* involves reducing the diameter of the reamer 0.010-0.030" (0.25-0.76 mm), depending upon size and length, behind a short full-diameter section following the chamfer. The length of the full diameter section may range from 1/8 to 1/2" (3.2 to 12.7 mm) according to size and length. The second method, view *b*, involves grinding a back-taper of 0.008-0.015" per in. (or mm/mm) behind the short full-diameter section.

Improving accuracy. Holes that are critical with respect to both location and size are often reamed with two passes. The first pass is an end-cutting operation for accurate hole location, and the second, a finishing pass for size and finish. Step reamers are often used for these operations, with the large diameters of the reamers ground without any or a minimum of back-taper to

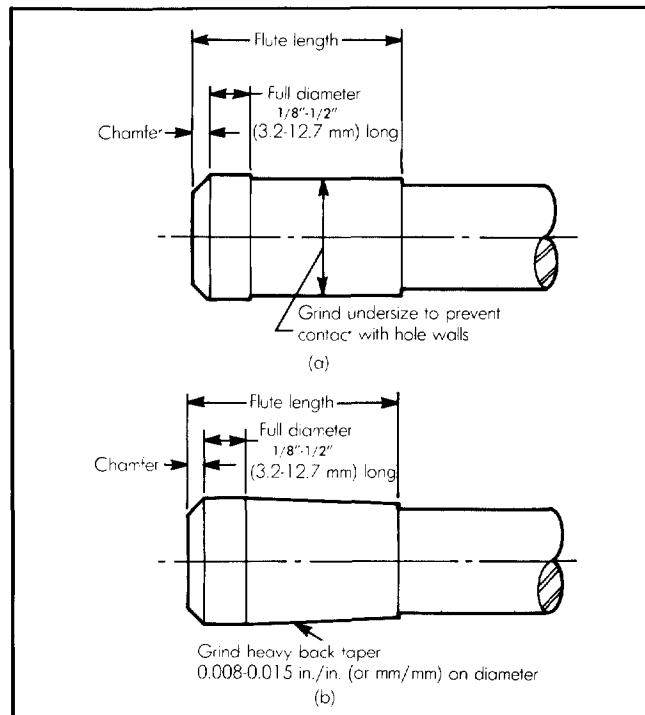


Fig. 9-138 Two methods of decreasing the bearing lengths of the cutting portions on reamers to reduce bellmouthing. (National Twist Drill)

reduce clearances in the bushings in which they are piloted. If a further reduction in clearance is required, rotary bushings can be employed. The end-cutting reaming operation is usually setup to cut 0.030-0.060" (0.76-1.52 mm) deeper than the finishing cut. This prevents the finishing reamer from bottoming out in the end-cut hole, which could cause the reamer to wander or chatter.

Improving finishes produced. To obtain smooth hole finishes reamers must operate smoothly without chatter or instability. With smooth operation, the finish attainable depends primarily upon the reamer design and operating conditions. It is difficult to produce smooth finishes that are less than about 120 μ in. (3.05 μ m) in soft mild steels because these materials tend to yield machined surfaces with a rough torn appearance.

One method of producing smoother finishes is to reduce the reamer feed per revolution. This places the helical paths of the several reamer flutes closer together. Feed rates as low as 0.0002-0.0005 ipr (0.005-0.013 mm/rev) per flute are used. Reamer life, however, will be longer if the maximum feasible feeds are used. Carbide reamers with honed edges are often used when fine finishes are required.

Removing the minimum practical amount of reaming stock allowance (see Table 9-35) often improves the finish by reducing the volume of chips and the resulting heat generation. Too little reaming stock, however, can cause problems because the reamer may not cut freely and can deflect the workpiece material rather than cut it. This can result in excessive heat, poor finish, and rapid reamer wear.

Troubleshooting reaming operations. Probable causes of reaming problems and possible solutions are tabulated in the troubleshooting chart presented in Table 9-46. Only one of the solutions listed should be tried at any one time to help determine the cause of the specific problem.

OPERATING PARAMETERS FOR REAMING

TABLE 9-46
Problems, Causes, and Solutions When using Reamers

Problems	Probable Causes	Possible Solutions
Oversize holes	Misalignment and/or vibration/chatter	Repair or change machine to increase rigidity. Realign and secure fixture and toolholder. Use floating or adjustable holder. Use close-fitting bushings. Reduce cutting speed first, and then feed rate if required. Use piloted and/or staggered-flute reamer.
	Excessive clearance in bushing or loose bushing	Replace bushing or tighten.
	Runout of reamer chamfer	Reground or replace reamer.
	Poor hole preparation	Improve hole prior to reaming.
	Excessive stock removal	Increase drill size, use core drill, or preream.
	Reamer hitting bottom in blind hole	Increase depth of drilled blind hole. Decrease depth of reamed hole.
	Galling	Use carbide reamer and bushing. Increase flow of cutting fluid.
	Dull reamer	Resharpen or replace reamer.
	Incorrect or damaged reamer	Change reamer.
	Excessive heat	Use cutting fluid, making sure it reaches cutting zone. Change cutting fluid being used. Increase flow of cutting fluid. Reduce feed rate and/or speed.
Undersize holes	Improper reamer size	Change reamer.
	Improperly sharpened reamer	Reground or replace reamer.
Bellmouthed holes	Misalignment and/or vibration/chatter	(See solutions under oversize holes.)
	Inadequate feed	Increase feed rate.
	Bent reamer shank	Replace reamer.
	Excessive tool overhang	Use shorter reamer if possible. Use piloted reamer or bushing.
	Excessive flute engagement	Shorten flute length. Grind relief on backs of flutes. Grind back taper on flutes.
Poor finish in holes	Dull or damaged reamer	Reground or replace reamer.
	Incorrect reamer	Change reamer type.
	Misalignment and/or vibration/chatter	(See solutions under oversize holes.)
	Excessive stock removal	Increase drill size, use core drill, or preream.
	Poor machinability of material being reamed	Change reamer material—try cobalt-containing HSS or carbide. Change cutting fluid being used.
	Reamer hitting bottom in blind hole	Increase depth of drilled blind hole. Decrease depth of reamed hole.
	Excessive speed	Reduce cutting speed. Change reamer material.
	Excessive heat	(See solutions under undersize holes.)
	Tool marks	Stop reamer rotation before withdrawing.
	Galling	Use carbide reamers and bushings.

CHAPTER 9

COUNTERBORES

TABLE 9-46—Continued

Problems	Probable Causes	Possible Solutions
Excessive reamer wear or breakage	Misalignment and/or vibration/chatter	(See solutions under oversize holes.)
	Incorrect operating parameters	Reduce cutting speed first, and then feed rate if required.
	Dirty or burred toolholder	Clean holder and remove burrs.
	Flute packing	Change setup and/or reamer to provide more flute space for chips.
	Excessive heat	(See solutions under undersize holes.)
	Dull or improperly sharpened reamer	Regrind or change grinding procedure.
	Reamer hitting bottom in blind hole	Increase depth of drilled blind hole. Decrease depth of reamed hole.
	Improper stock allowance	Increase or decrease as required.
	Poor hole preparation	Change hole production method. Avoid rough, bellmouthed, or tapered holes.

COUNTERBORING, SPOTFACING AND COUNTERSINKING

Counterboring, spotfacing, and countersinking are secondary operations performed with end-cutting tools. Enlarging a hole for a limited depth is called counterboring. If the cut is shallow so that it leaves only a finished face around the original hole, it is called spotfacing. The cutting of an angular opening into the end of a hole is countersinking, sometimes referred to as chamfering. Tools of this type often present difficulty because they are among the most inefficient of cutting tools due to the distance of the cutting edge from the support.

The counterbore is defined as a rotary, pilot-guided, end-cutting tool, having one or more cutting lips and usually having straight or helical flutes for the passage of chips and the admission of cutting fluid. A spotfacer is defined as a rotary, pilot-guided, end-cutting tool, having teeth on one or both ends. Spotfacers are mounted on either straight or taper-shank pilots (drive bars) and are used to produce flat surfaces normal to their axes of rotation. Spotfacers are designed as a variation of the counterbore to reach inaccessible areas. The drive bar acts as the pilot. A countersink is a rotary, end-cutting tool which may or may not be piloted for the purpose of beveling or tapering the work material around the periphery of a hole. The surface cut by the conical tool is concentric with and at an angle of less than 90° to the centerline of the hole.

COUNTERBORES

Six types of counterbores are in general use. They are:

1. *Solid counterbores.* These are made of one piece of tool material, including the pilot (Fig. 9-139, view *a*). One tool manufacturer brazes a pilot sleeve on the solid body (view *b*). The two-lip construction provides good chip space

while retaining the advantage of an unbroken surface on the pilot.

2. *Counterbores with interchangeable pilots.* These are counterbores having removable, mechanically held pilots.
3. *Interchangeable counterbores.* These are counterbores having end-cutting portions as well as pilots that are removable and interchangeable so that a series of cutters fit the same holder and a series of pilots fit the same cutter. These types of counterbores are generally available in standard sizes from 1/4 to 5" (6.3 to 127 mm) diam (Fig. 9-140).
4. *Inserted-blade counterbores.* These are counterbores which have replaceable, mechanically held blades. The blades may be either solid or tipped (Fig. 9-141).
5. *Indexable-insert counterbores.* These are counterbores with indexable-insert tips. They may be used in the same manner as interchangeable counterbores; however, they are more limited in application than other types (Fig. 9-142).
6. *Disposable-insert counterbores.* These counterbores or spotfacers have disposable inserts and removable pilots or pilot drills. The inserts may also be designed to chamfer the holes; depth setting remains constant when the inserts are changed.

Counterbores of the interchangeable type may be solid high-speed steel, or the body may be tipped with another type of tool material such as tungsten carbide (see Chapter 3, "Cutting Tool Materials").

Counterbores are frequently used in conjunction with a pilot that guides the tool in a previously prepared hole. Two types of

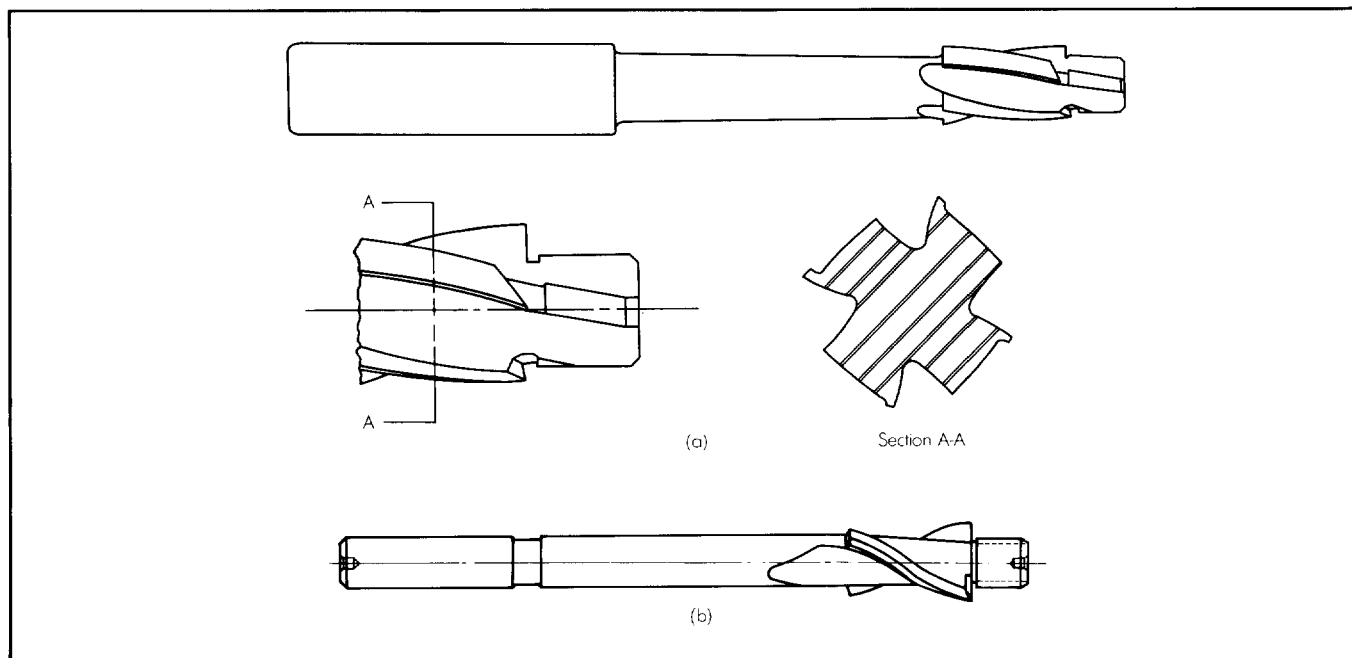


Fig. 9-139 Solid counterbore (a) is of one-piece construction. Counter bore (b) has pilot sleeve brazed to body. (Weldon Tool Co.)

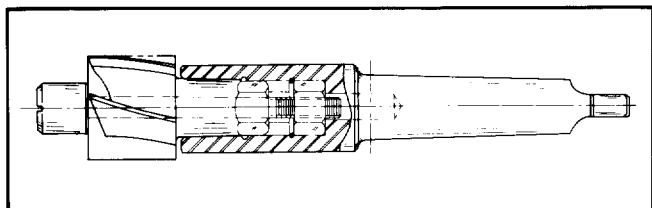


Fig. 9-140 Interchangeable counterbore with removable pilot and end-cutting portion.

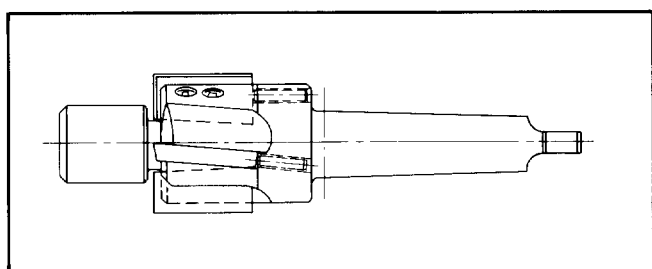


Fig. 9-141 Inserted-blade counterbore.

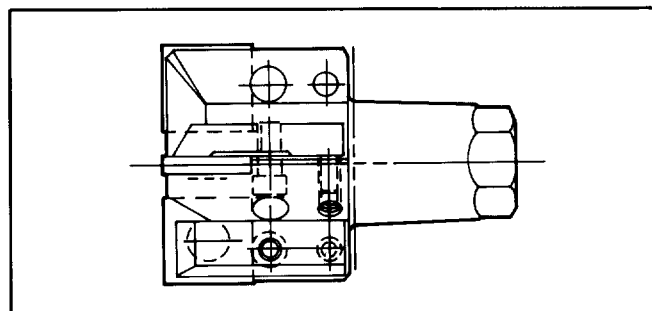


Fig. 9-142 Counterbore equipped with indexable inserts.

interchangeable pilots are generally used—the solid pilot and the roller pilot. Solid pilots are from 0.002-0.007" (0.05-0.18 mm) diam under nominal size of hole, dependent upon size—less undersize for smaller counterbores.

The hole must be reasonably straight and have a finish of 125 μ in. (3.2 μ m) or better. Roller pilots (Fig. 9-143) will improve counterbore performance if the hole is properly prepared. The pilot has 0.0005-0.001" (0.013-0.03 mm) clearance between stem and roller and 0.0005-0.001" clearance between pilot and hole. The hole must be straight and requires a finish of 64 μ in. (1.6 μ m) or better.

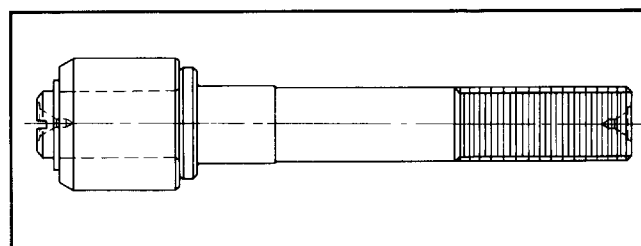


Fig. 9-143 Roller pilot for use with a counterbore.

Deep-hole counterboring is defined as a depth of counterboring greater than one-fourth the diameter of the cutter. This may be greater on some materials and also depends upon the pilot size in relation to the cutter diameter. The pilot size must be equal to or greater than the cutter-root diameter because chips can clog in the flutes if the area of the cut is too great for the chips to flow properly up the flutes. Chips can also clog if the cutting depth is too great. Too long a pilot tends to bind in the hole and may hit the bottom of the hole, the fixturing, or the work table. Roller pilots should be used if the hole finish is maintained. Holes should be rough reamed prior to counterboring.

Counterbores come with from one to seven flutes. Flutes may be odd or even. An odd number of flutes can in some cases

COUNTERBORES

reduce the tendency to chatter. The right-hand helical flutes of the counterbore give a positive axial rake to the cutter. Generally the cutting edge is slightly ahead of center, giving a negative radial rake. This may reduce chatter and directs the chips away from the center of the cutter toward the periphery and flute. Chatter can occur on a counterbore largely because of the distance from the tool support to the cutting edge and the high torque acting on the cutter. Usually counterboring is done with a very light chip load, which again can cause chatter problems.

Carbide-tipped counterbores work successfully on cast iron and, with a rigid setup, on steel or similar workpiece materials. Carbide-tipped counterbores require sufficient feed to obtain chip loads of not less than 0.004" (0.10 mm) per tooth. The pilot must not touch the carbide tips; therefore, a setscrew should be used, entering the side of the bore and holding the shank of the pilot.

Speeds and feeds for countersinking are generally slower than for drills of the same diameter. The data given in Table 9-47 can be used as a guide. Counterbores require more torque, but are generally more economical to operate than twist drills, especially when carbide counterbores with multiple cutting edges are operated at high cutting speeds.⁴

Among the most common applications for counterbores is producing the recesses for various types of cap screws and machine screws. Table 9-48 lists sizes of counterbores and pilots for standard socket-head cap screws. Table 9-49 lists sizes for machine screws and hexagon-head and fillister-head cap screws.

Several types of holding mechanisms are available for interchangeable-type counterbores (Fig. 9-144). The holder may have positive stops to control counterboring depth accurately. Figure 9-144, view *b*, illustrates a threaded-stop-nut holder. The ball-bearing stop nut (view *c*) indicates depth of counterboring when the stop nut begins to rotate.

There are many variations of counterbore-type tools used for special applications. Figure 9-145 illustrates a multiple-diameter counterbore, a multiple-diameter subland counterbore, and a port-contour cutter used specifically for hydraulic applications.

Counterbores, as well as spotfaces, can usually be machined with three or four-flute core drills sharpened to an included point angle of 180°. One or two flutes are usually sharpened to center when spotfacing is done without the benefit of a previously drilled hole. The other flutes provide additional support in the bushing, as well as balanced cutting action.

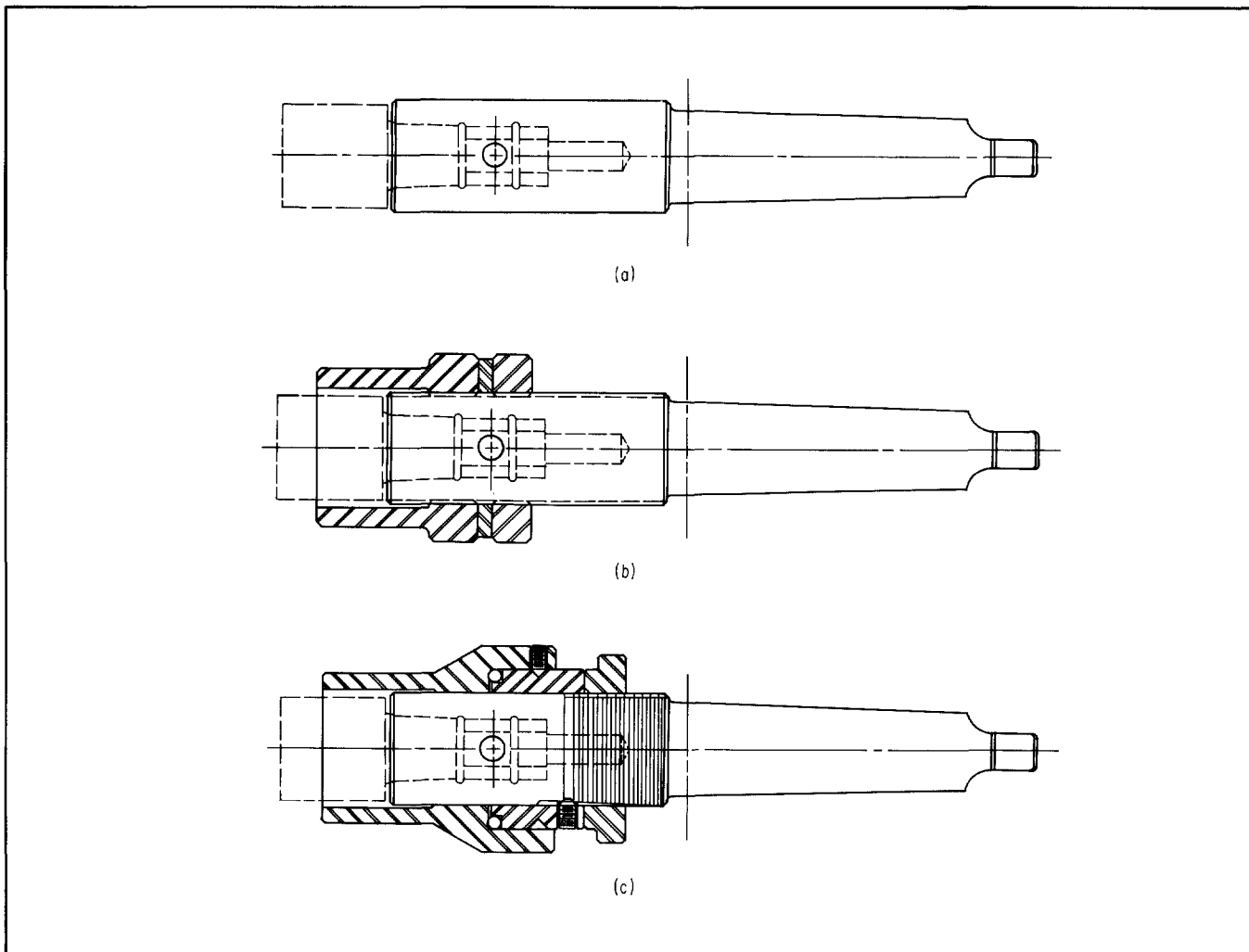


Fig. 9-144 Several types of holders for interchangeable-type counterbores.

TABLE 9-47
Speeds and Feeds for HSS and Carbide Counterbores

Material Machined	HSS Counterbores		Carbide-tipped Counterbores	
	Speed, sfm (m/min)	Feed ipr (mm/rev)	Speed, sfm (m/min)	Feed ipr (mm/rev)
Steels, soft	75-85 (23-26)	0.003-0.005 (0.08-0.13)	250-300 (76-91)	0.004-0.008 (0.10-0.20)
Alloy and tool steels	40-80 (12-24)	0.003-0.005 (0.08-0.13)	180-200 (55-61)	0.004-0.008 (0.10-0.20)
Cast irons, soft	120-140 (37-43)	0.005-0.007 (0.13-0.18)	300-350 (91-107)	0.006-0.012 (0.15-0.30)
Brass	150-300 (46-91)	0.005-0.010 (0.13-0.25)	400-700 (122-213)	0.006-0.012 (0.15-0.30)
Aluminum	400-800 (122-244)	0.005-0.015 (0.13-0.38)	500-800 (152-244)	0.006-0.012 (0.15-0.30)

TABLE 9-48
Counterbore and Pilot Sizes for Standard Socket-Head Cap Screws*

Nominal Size	Max Body Diam	Max Head Diam	Counterbore Diam	Pilot Diam Tap Drill		Pilot Diam Clearance Drill
				UNC	UNF	
6	0.1380	0.226	0.266	0.1094	0.113	9/64
8	0.1640	0.270	0.313	0.136	0.136	11/64
10	0.1900	5/16	0.359	0.1495	0.159	13/64
1 1/4	0.2500	3/8	0.406	0.201	0.213	9/32
5/16	0.3125	15/32	0.500	0.261	0.272	11/32
3/8	0.3750	9/16	0.594	0.316	0.332	13/32
7/16	0.4375	21/32	0.688	0.368	25/64	15/32
1/2	0.5000	3/4	0.781	27/64	29/64	17/32
5/8	0.6250	15/16	1.000	17/32	37/64	21/32
3/4	0.7500	1 1/8	1.188	21/32	11/16	25/32
7/8	0.8750	1 5/16	1.375	49/64	13/16	29/32
1	1.0000	1 1/2	1.563	7/8	59/64	1 1/32
1 1/8	1.1250	1 11/16	1.718	63/64	1 3/64	1 5/32
1 1/4	1.2500	1 7/8	2.000	1 7/64	1 11/64	1 9/32
1 3/8	1.3750	2 1/16	2.375	1 7/32	1 19/64	1 13/32
1 1/2	1.5000	2 1/4	2.281	1 11/32	1 27/64	1 17/32
1 3/4	1.7500	2 5/8	2.656	1 9/16		1 25/32
2	2.0000	3	3.032	1 25/32		2 1/32
2 1/4	2.2500	3 3/8	3.407	2 1/32		2 9/32
2 1/2	2.5000	3 3/4	3.782	2 1/4		2 17/32
2 3/4	2.7500	4 1/8	4.157			2 25/32
3	3.0000	4 1/2	4.532			3 1/32
3 1/4	3.2500	4 7/8	4.902			3 9/32
3 1/2	3.5000	5 1/4	5.282			3 17/32
3 3/4	3.7500	5 5/8	5.675			3 25/32
4	4.0000	6	6.032			4 1/32

* All dimensions in inches.

CHAPTER 9

COUNTERBORES

TABLE 9-49
Some Commercially Available Sizes of Counterbores for Cap Screws and Machine Screws*

Size thread NF and NC	Fillister-Head Cap Screw			Round or Hexagon-Head Cap Screw		Fillister-Head Machine Screw		Round or Hexagon-Head Machine Screw	
	Cutter for Head	Pilot for Body	Pilot for Taphole	Cutter for Head	Pilot for Body	Cutter for Head	Pilot for Body	Cutter for Head	Pilot for Body
.1/4-28	3/8	1/4	0.213	1/2	1/4	7/16	1/4	1/2	1/4
1/4-20	3/8	1/4	0.201	1/2	1/4	7/16	1/4	1/2	1/4
5/16-24	7/16	5/16	0.272	5/8	5/16	17/32	5/16	5/8	5/16
5/16-18	7/16	5/16	0.257	5/8	5/16	17/32	5/16	5/8	5/16
3/8-24	9/16	3/8	0.332	11/16	3/8	5/8	3/8	3/4	3/8
3/8-16	9/16	3/8	5/16	11/16	3/8	5/8	3/8	3/4	3/8
7/16-20	5/8	7/16	25/64	13/16	7/16	23/32	7/16	7/8	7/16
7/16-14	5/8	7/16	0.368	13/16	7/16	23/32	7/16	7/8	7/16
1/2-20	3/4	1/2	29/64	7/8	1/2	27/32	1/2	1	1/2
1/2-13	3/4	1/2	27/64	7/8	1/2	27/32	1/2	1	1/2
9/16-18	13/16	9/16	3 3/64	1	9/16				
9/16-12	13/16	9/16	31/64	1	9/16				
5/8-18	7/8	5/8	37/64	1 1/8	5/8				
5/8-11	7/8	5/8	17/32	1 1/8	5/8				
3/4-16	1	3/4	11/16	1 3/8	3/4				
3/4-10	1	3/4	21/32	1 3/8	3/4				
7/8-14	1 1/8	7/8	13/16	1 5/8	7/8				
7/8-9	1 1/8	7/8	49/64	1 5/8	7/8				
1-14	1 5/16	1	15/16	1 3/4	1				
1-8	1 5/16	1	7/8	1 3/4	1				

* All dimensions in inches.

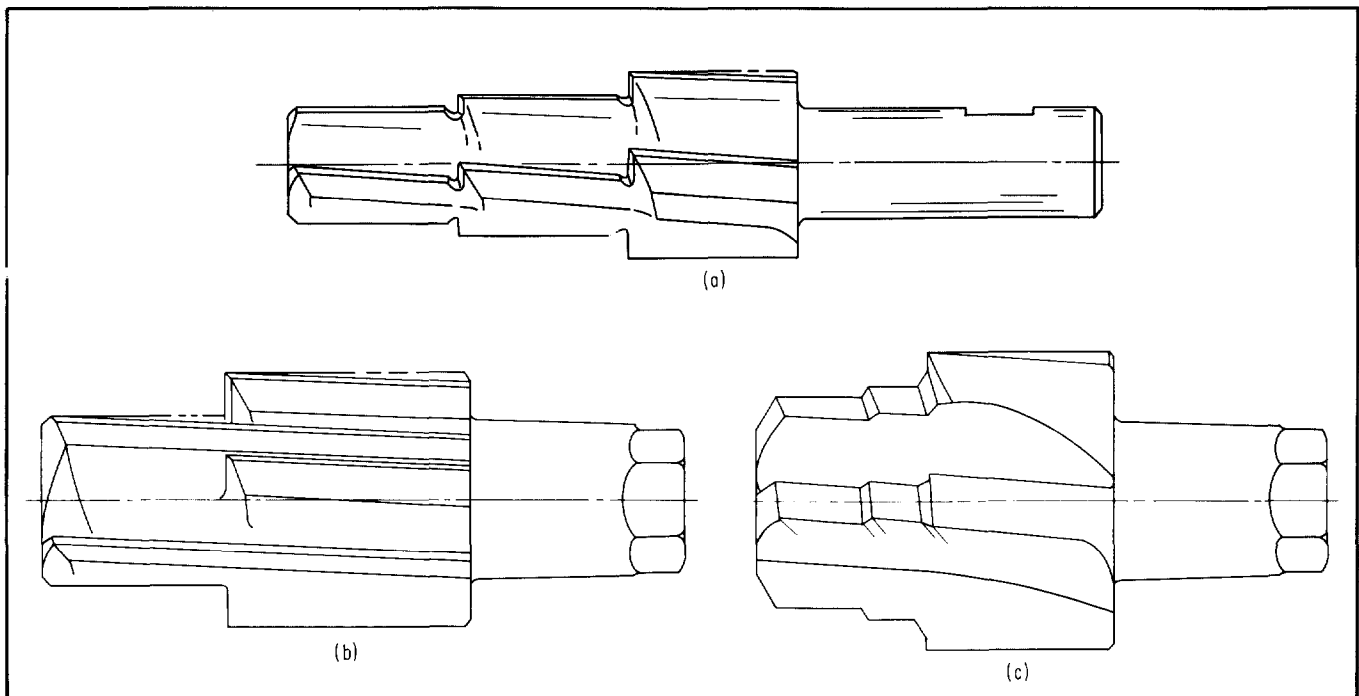


Fig. 9-145 Counterbore-type tools for special applications: (a) multidiameter counterbore, (b) multidiameter subland counterbore, and (c) port-contour cutter.

SPOTFACERS

A spotfacer, as its name implies, is designed for shallow machining of a surface for applications such as washers and bolt and nut heads. Figure 9-146 shows a standard back spotfacer with drive bar. These tools are designed to reach inaccessible spots in castings or similar workpieces. The body of the drive bar acts as a pilot, and the tool is generally run in reverse with the spindle moving toward the machine to produce the desired surface. Because of this, the standard single right-hand cutter actually appears to be a left-hand cutter when viewed in a normal manner. If the cutter is designed to spotface with a spindle feed away from the machine face, a left-hand cutter should be used. Cutters also come in double right-hand or double right-hand and left-hand types.

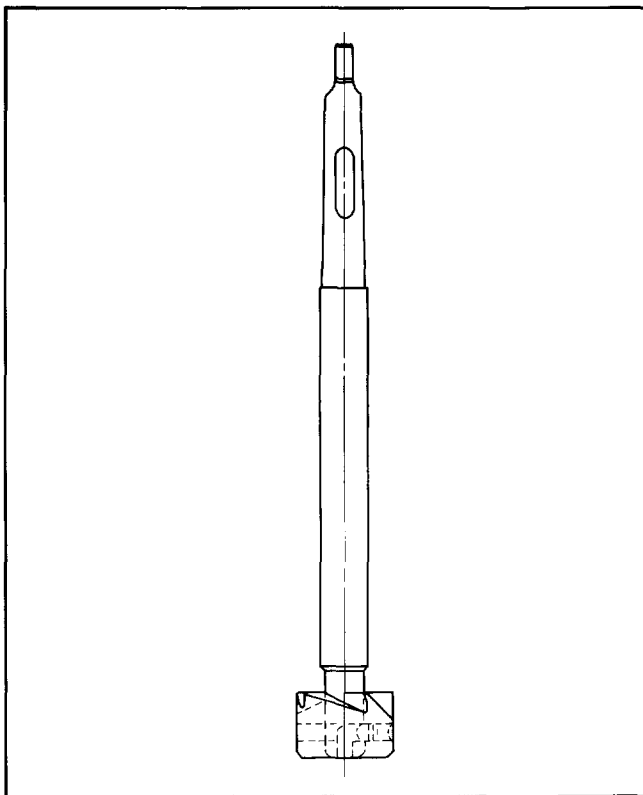


Fig. 9-146 Back spotface cutter and drive-bar assembly.

Automatic back spotfacing, counterboring, or back chamfering tools are also available. These tools consist of a spindle with a recess into which the cutter, called a wing, folds when the spindle enters the hole in the workpiece (see Fig. 9-147). When the cutter (wing) leaves the hole, it automatically extends due to centrifugal force. Then when the direction of spindle rotation is reversed and feed applied, the required operation is performed. With the exception of the smallest size tools, which use HSS cutters, the wings are provided with clamped carbide inserts.

The same rules apply for back spotfacing as for counterboring; however, most back spotface cutters are designed without deep flutes on the periphery and therefore are intended only for very shallow work. Designs are also available for deeper work and heavy-duty drives.

As with counterbores and countersinks, speeds and feeds for spotfacers are generally lower than those used for drilling (see

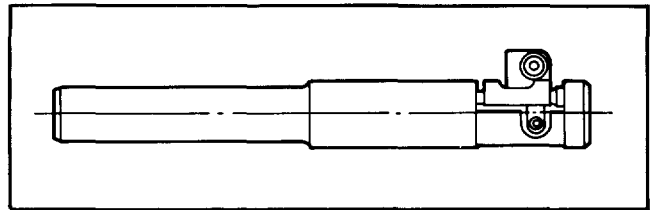


Fig. 9-147 Automatic back spotfacer. Cutter folds into recess when cutter enters hole. (Davis Tool Co.)

Table 9-47). As a rule of thumb the speed should be approximately three-fourths that used in drilling with feed the same penetration rate as for drilling; however, the load per tooth is less than in drilling because of the larger number of teeth used in a counterbore or similar type of tool. Because of the variations in the many jobs using this type of tooling, no hard-and-fast rules can be given for exact speeds and feeds to be used. If a hand feed is used, it is important to start the cut very carefully so that hogging does not occur with resultant breakage of the tool. It is also important to minimize backlash in the feeding mechanism of the machine.

COUNTERSINKS

Countersinks are generally classified in two general categories: (1) shank-type countersinks and (2) combined drills and countersinks. Commercially available combined drills and countersinks were discussed previously in this chapter under the subject of types of twist drills.

The types of countersinks shown in Fig. 9-148 may be broadly classified as machine countersinks. These are used mostly to produce countersunk holes for screw and rivet heads, as well as to chamfer and deburr holes. They are available with included point angles of 60°, 72°, 82°, and 90°.

View *a* of Fig. 9-149 illustrates a 100° aircraft countersink with disposable carbide insert. It has a one-piece body, pilot, and threaded shank, and is available in 90° and 82° designs and with a straight shank. View *b* shows a front and side view of a three-flute countersink large hole. This tool has removable HSS inserts and shanks.

Ship-plate countersinks are designed primarily for use in countersinking holes for rivet heads. They are made of high-speed steel only, with various point angles. Tables 9-50 and 9-51 give dimensions for commercially available ship-plate countersinks. Various countersinks are made to fit the same holders as used for holder-type counterbores. These are generally in two types: pilot-type countersinks and self-centering countersinks (Fig. 9-150).

Various designs of countersinks are available to minimize

TABLE 9-50
Some Commercially Available Sizes of Three-Flute Ship-Plate Countersinks

		Overall Length, in.		
Flute		Taper Shank		Point
Diam, in.	Length, in.	No. 3	No. 4	Diam, in.
1 1/2	5 3/8	10	11 1/8	1/2
1 3/4	5 3/8	10	11 1/8	9/16
2	5 3/8	10	11 1/8	5/8
2 1/4	5 3/8	---	11 1/8	11/16

COUNTERSINKS

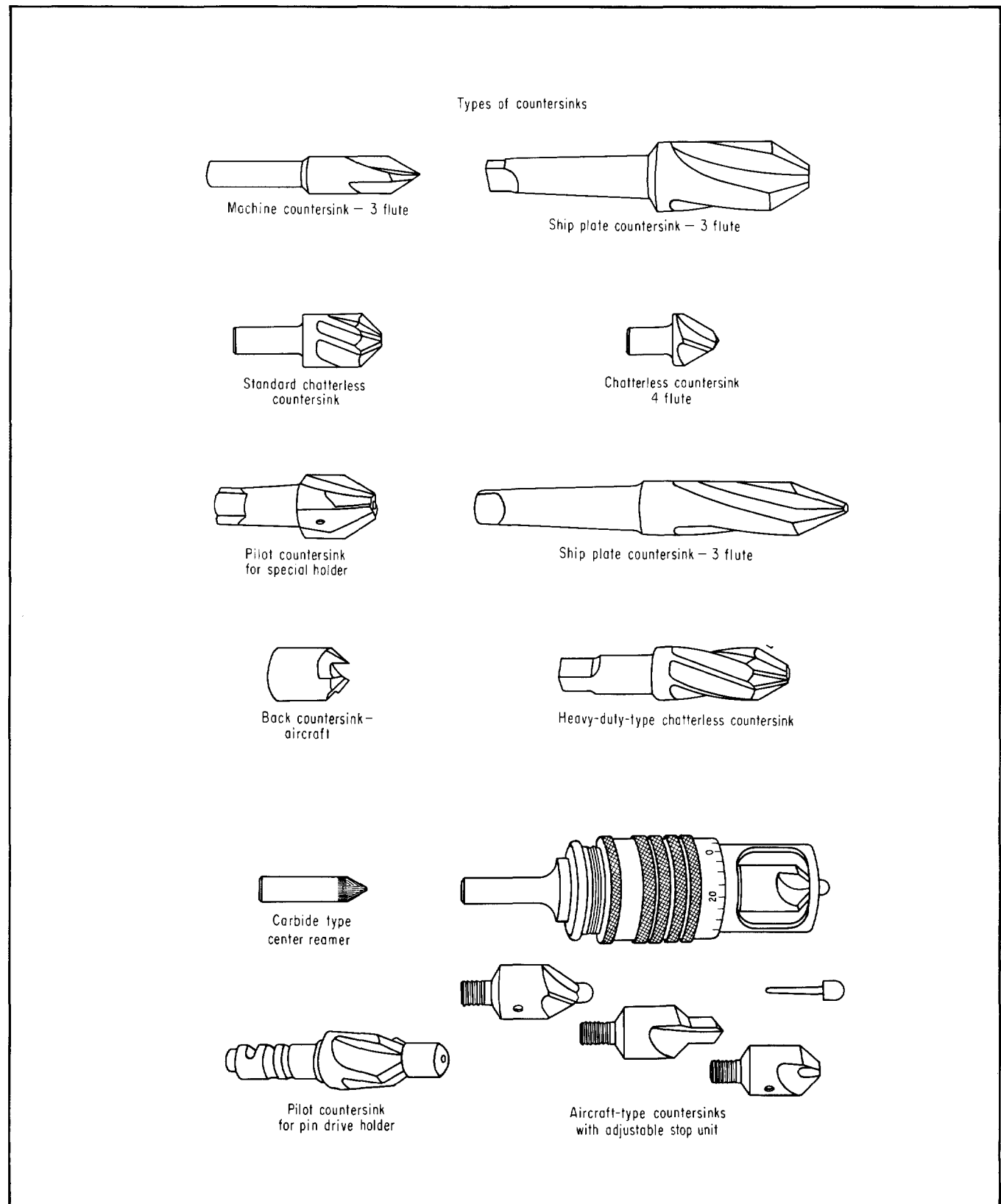


Fig. 9-148 Various types of countersinks broadly classified as machine countersinks.

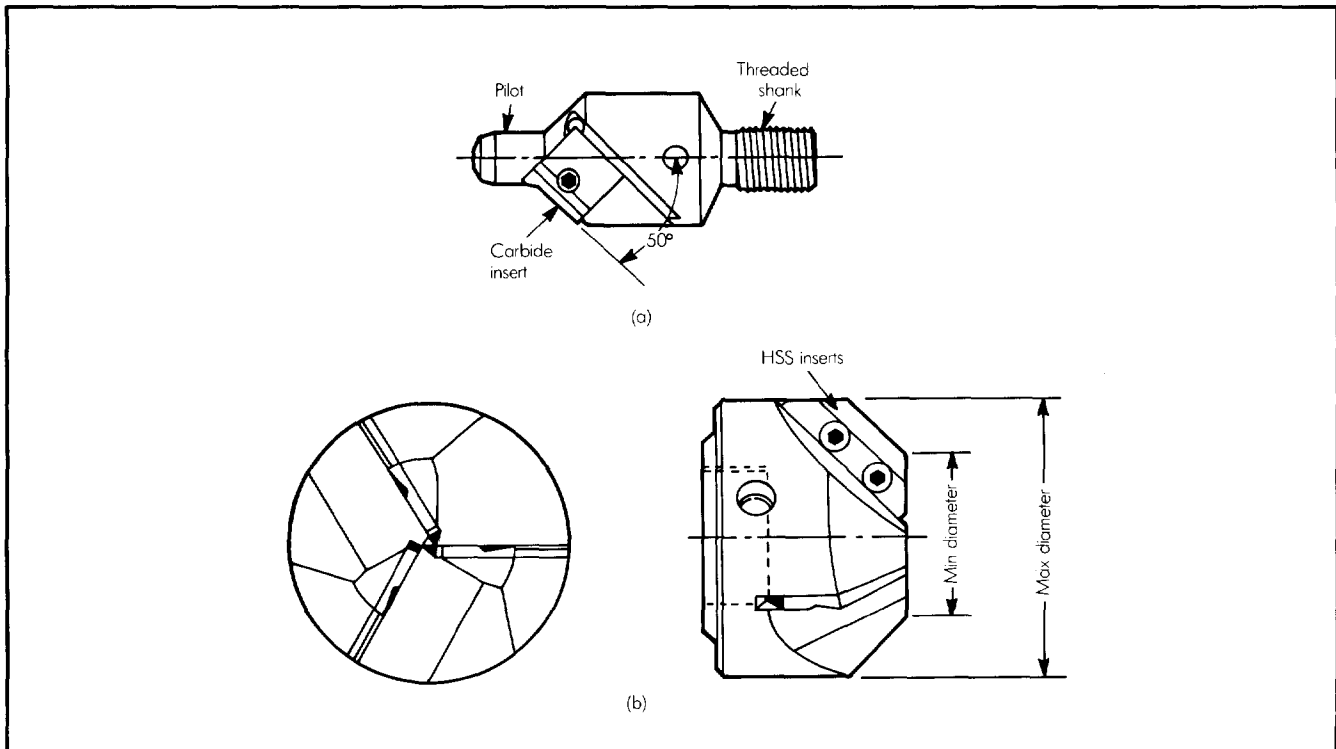


Fig. 9-149 Countersinks: (a) 100° type with disposable carbide insert for aircraft fasteners, (b) three-flute tool with HSS inserts for large holes. (Everede Tool Co.)

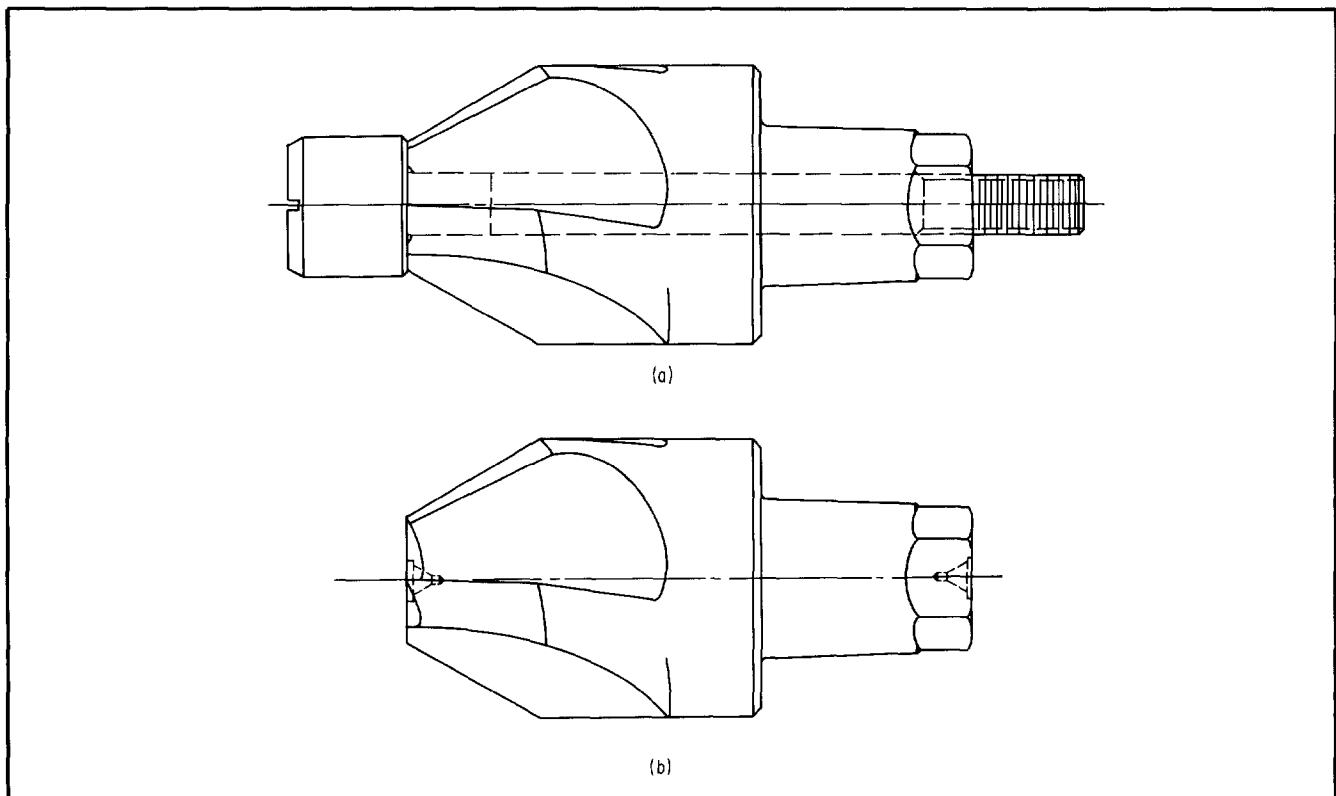


Fig. 9-150 (a) Pilot-type countersink and (b) self-centering type countersink.

COUNTERSINKS

chatter. The self-centering countersink in Fig. 9-150, view *b*, uses an odd number of flutes to help reduce chatter. Some countersinks are produced with radial flutes, which also reduce chatter. One design of chatterless countersink incorporates three main flutes with three intermediate flutes (Fig. 9-151). The same general rules for counterboring also apply when countersinking (see Table 9-47).

Another type of countersink features a single cutting edge for free-cutting action and minimum chatter. Chips produced by the edge of the hole in the tool pass through the hole and are ejected. These countersinks are available in pilotless, piloted (see Fig. 9-152, view *a*), and double-end styles with integral shanks, as well as in removable shank style (view *b*) with straight or taper shank.

Countersinks are available with tool steel bodies with disposable HSS or carbide inserts. Aircraft-type countersinks normally have an included point angle of 100° and frequently include a cutting radius at the junction of the conical and cylindrical surfaces. These cutters are made to rigid dimensional and quality standards because they must produce fastener holes in aircraft assemblies to close tolerances. They are frequently used in portable tools equipped with precise depth control devices.

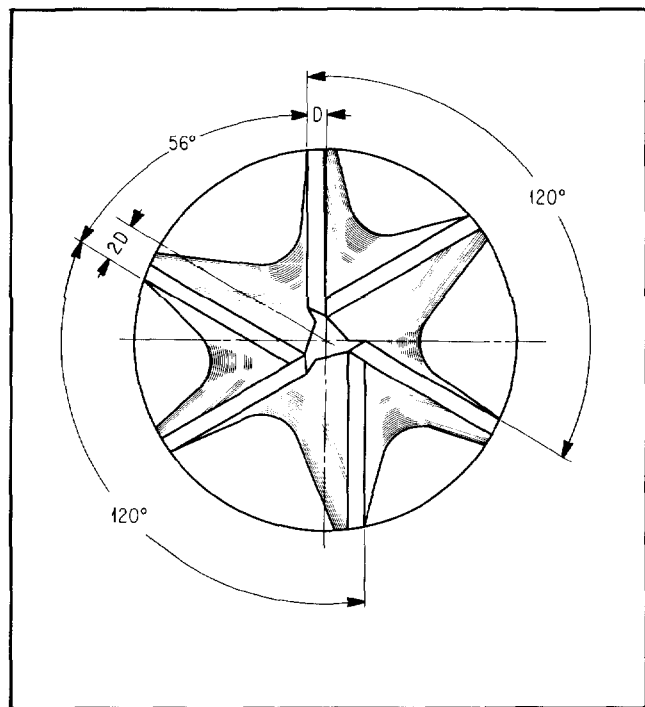


Fig. 9-151 Three main flutes and three intermediate flutes are used on this countersink to minimize chatter.

TABLE 9-51
Countersink Sizes Required for Various Rivet Sizes

Rivet Size, in.	Hole Size, in.	Countersink Face Diam, in.	Countersink Required, in.
5/8	11/16	1	1 1/2 or 1 3/4
3/4	13/16	1 3/16	1 1/2 or 1 3/4
7/8	15/16	1 3/8	1 3/4
1	1 1/16	1 9/16	1 3/4
1 1/8	1 3/16	1 3/4	2 or 2 1/4
1 1/4	1 5/16	1 15/16	2 or 2 1/4

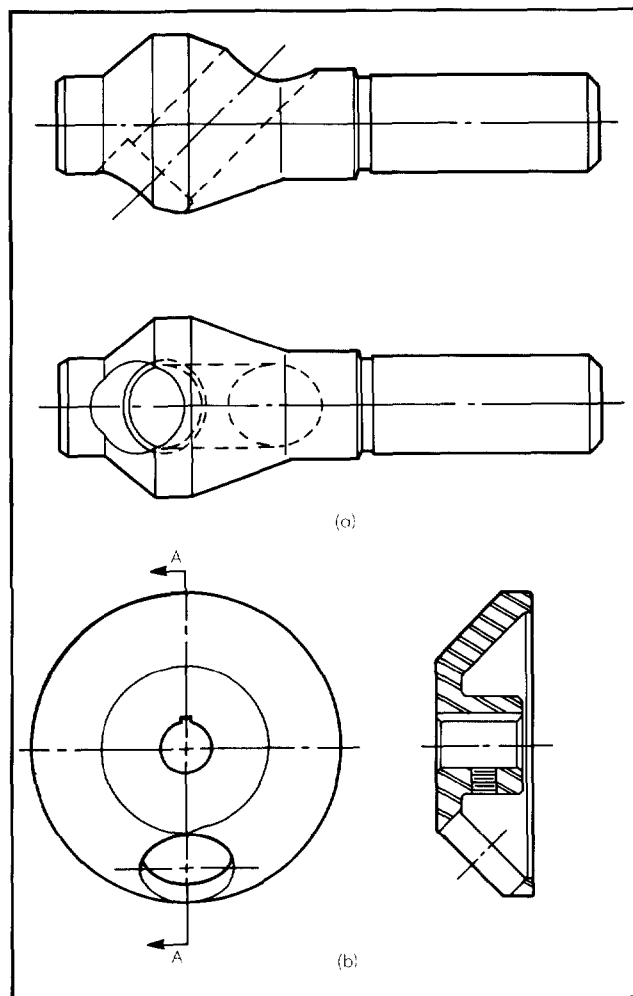


Fig. 9-152 Countersinks with single cutting edges: (a) piloted style with integral shank and (b) removable shank style. (Weldon Tool Co.)

References

1. R. L. Hatschek. "Fundamentals of Drilling." *American Machinist* (February 1979) pp. 108-130.
2. E. G. Hoffman. *Jig and Fixture Design* (Albany, NY: Delmar Publishers, 1980).
3. *Metal Cutting Tool Handbook*, (Cleveland: Metal Cutting Tool Institute, 1954).
4. C. H. Kahng and K. P. Rajurkar. *Fundamental Study on Counterboring*. SME Technical Paper MR77-349, 1977.

Bibliography

- Block, Frederick S. *Gundrilling*. The Eldorado Tool & Manufacturing Corp.
- Burant, Robert O. and McGinty, Michael J. "Cutting Tools/Drills—High-Speed Steel." *Manufacturing Engineering* (April 1979), pp. 65 & 66.
- . "Cutting Tools/Drills—Carbide." *Manufacturing Engineering* (May 1979), p. 65.
- Butrick, Frank M. *Engineering and Trouble Shooting of Spade Drill Applications*. SME Technical Paper MR76-353, 1976.
- Cannon, Michael R. *The Quick Change Chuck System*. SME Technical Paper MR81-976, 1981.
- Gillespie, LaRoux K. *An Overview of Drilling Approaches for Holes Smaller Than One Millimeter*. SME Technical Report MRR79-06, 1979.
- . *Drill Geometries for Miniature Holes*. SME Technical Report MRR81-08, 1981.
- Levin, Robert S. *Equipment and Techniques for Drilling Miniature Holes*. SME Technical Paper MR79-498, 1979.
- Metal Cuttings*. National Twist Drill, A Div. of Lear Siegler, Inc.
- Use and Care of Reamers*. The Cleveland Twist Drill Co.
- Use and Care of Twist Drills*. The Cleveland Twist Drill Co.
- Trost, Charles J. *Deep Hole Drilling*. SME Technical Paper MR78-133, 1978.
- Whalley, Howard M. *Advancements In Coolant Feeding Techniques*. SME Technical Paper MR76-352, 1976.
- Wick, Charles. "Automatic Adaptive Control of Machine Tools." *Manufacturing Engineering* (September 1977), pp. 38-45.
- . "The Advancing Technology of Small Hole Drilling." *Manufacturing Engineering* (October 1980), pp. 81-85.
- . "Advances in Coolant-fed Tooling." *Manufacturing Engineering* (November 1978), pp. 44-51.