МИНИСТЕРСТВО ОБРАЗОВАНИЯ И НАУКИ УКРАИНЫ

Национальный аэрокосмический университет им. Н.Е. Жуковского
«Харьковский авиационный институт»

M. K. Knyazyev Sergeyev S. V.

М.К. Князев, С.В. Сергеев

CUTTING PROCESSES FOR PRODUCING ROUND SHAPES

Textbook

МЕТОДЫ ОБРАБОТКИ РЕЗАНИЕМ ЦИЛИНДРИЧЕСКИХ ПОВЕРХНОСТЕЙ

Учебное пособие

Харьков «ХАИ» 2002

УДК 621.7

Cutting processes for producing round shapes / M. K. Knyazyev, S. V. Sergeyev. –Textbook. – Kharkov: National aerospace university "KhAI", 2002. – 21 p.

Методы обработки резанием цилиндрических поверхностей / М.К. Князев, С.В. Сергеев. – Учеб. пособие. – Харьков: Нац. аэрокосмический ун-т «Харьк. авиац. ин-т», 2002. – 21 с.

The condensed description of machining processes for producing round shapes with cutting tools is submitted. Recommendations on selection of material and cutting edge geometrical parameters of tools, machining conditions and cutting fluids usage are cited. The textbook was developed on the basis of original sources.

Представлено краткое описание методов обработки цилиндрических поверхностей лезвийным инструментом. Приведены рекомендации по выбору материала и геометрических параметров режущих кромок инструмента, технологических режимов обработки и использованию смазывающе-охлаждающих жидкостей. Пособие разработано на основе оригинальных источников.

Для студентов механических специальностей.

Fig. 18. Tables 7. Bibliogr.: 3 titles.

Рис. 18. Табл. 7. Библиогр.: 3 назв.

Рецензенты: канд. техн. наук, доц. Н. Ф. Савченко,
канд. филол. наук, доц. Н. Г. Долинина

Introduction

This issue is devoted to the processes that produce parts that are basically round in shape. Typical products made include parts as small as miniature screws for eyeglass-frame hinges and as large as shafts, pistons, cylinders, gun barrels, and turbines for hydroelectric power plants. These processes are usually performed by turning the workpiece on a lathe. *Turning* means that the part is rotating while it is being machined. The starting material is usually a workpiece that has been made by other processes, such as casting, shaping, forging, extrusion, and drawing. Turning processes are versatile and capable of producing a wide variety of shapes (Fig. 1):

- **Turning** straight, conical, curved, or grooved workpieces (Figs. la-d), such as shafts, spindles, pins, handles, and various machine components.
- **Facing**, to produce a flat surface at the end of the part (Fig. 1e), such as parts that are attached to other components, or to produce grooves for O-ring seats (Fig. 1f).
- Producing various shapes by **form tools** (Fig.1g), such as for functional purposes or for appearance.
- **Boring,** to enlarge a hole made by a previous process or in a tubular workpiece or to produce internal grooves (Fig.1h).
- **Drilling,** to produce a hole (Fig.1i), which may be followed by boring to improve its accuracy and surface finish.
- **Parting**, also called *cutting off*, to cut a piece from the end of a part, as in making slugs or blanks for additional processing into discrete products (Fig.1j).
- **Threading**, to produce external and internal threads in workpieces (Fig.1k).

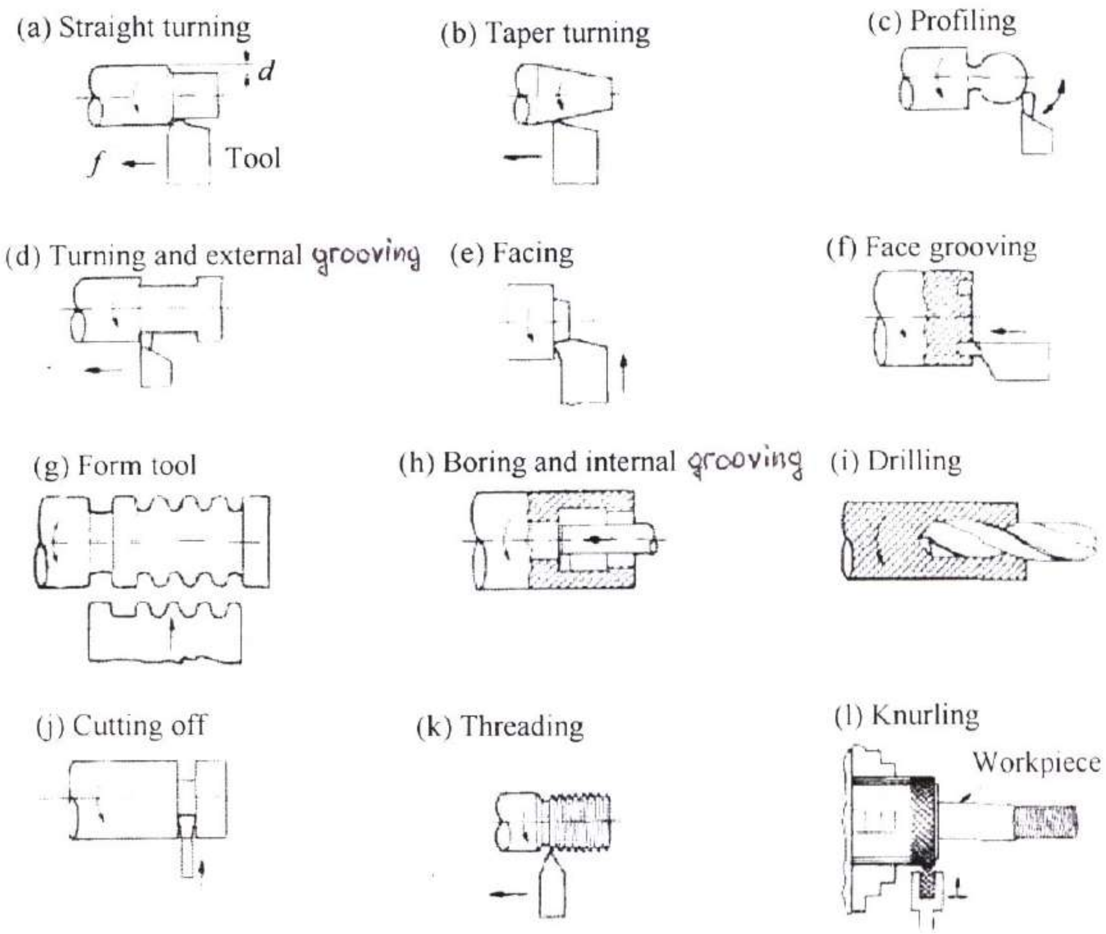

FIGURE 1. Various cutting operations that can be performed on a lathe.

- **Knurling**, to produce a regularly shaped roughness on cylindrical surfaces, as in making knobs (Fig. 1*l*).

These operations may be performed at various rotational speeds of the workpiece, depths of cut, *d*, and feed, *f*, depending on the workpiece and tool materials, the surface finish and dimensional accuracy required, and the capacity of the machine tool. Cutting speeds are usually in the range of 0.15–4 m/s (30–800 ft/min). *Roughing cuts*, which are performed for large-scale material removal, usually involve depths of cut greater than 0.5 mm (0.02 in.) and feeds on the order of 0.2–2 mm/rev (0.008–0.08 in./rev). *Finishing cuts* usually involve lower depths of cut and feed.

Lathes

Lathes are generally considered to be the oldest machine tools. Although woodworking lathes were first developed during the period 1000–1 B.C., metalworking lathes with lead screws were not built until the late 1700s. The most common lathe, shown schematically in Fig. 2, was originally called an *engine lathe* because it was powered with overhead pulleys and belts from nearby engines. Although simple and versatile, an engine lathe requires a skilled machinist because all controls are manipulated by hand. Consequently, it is inefficient for repetitive operations and for large production runs. However, various types of automation can be added to improve efficiency.

Lathe components

Lathes are equipped with a variety of components and accessories. The basic components of a common lathe are described in the following paragraphs.

Bed. The bed supports all the other major components of the lathe. Beds have a large mass and are built rigidly, usually from gray or nodular cast iron. The top portion of the bed has two *ways*,

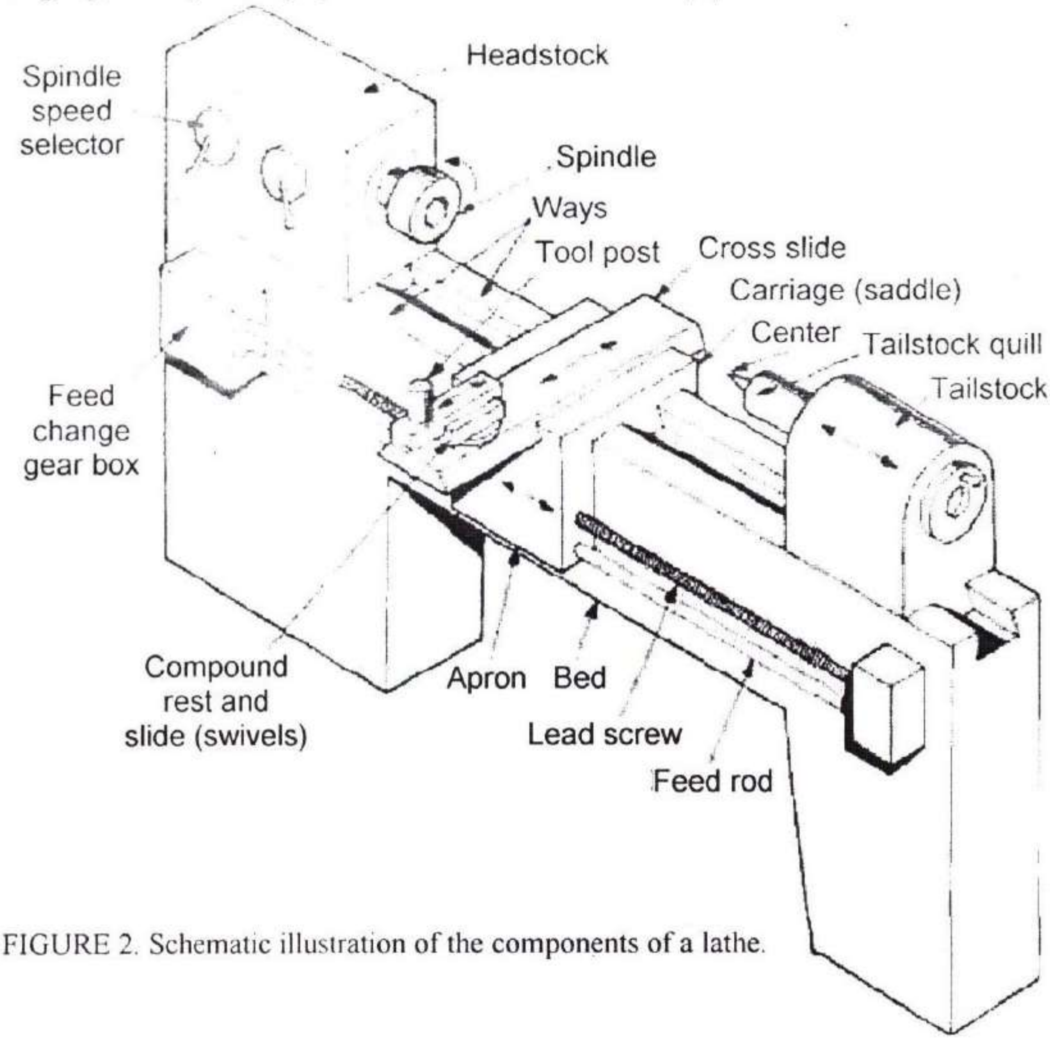

FIGURE 2. Schematic illustration of the components of a lathe.

with various cross-sections, that are hardened and machined accurately for wear resistance and dimensional accuracy during use.

Carriage. The carriage or carriage assembly slides along the ways and consists of an assembly of the cross-slide, tool post, and apron. The cutting tool is mounted on the *tool post,* usually with a *compound rest* that swivels for tool positioning and adjustment. The cross-slide moves radially in and out, thus controlling the radial position of the cutting tool, as in facing operations. The *apron* is equipped with mechanisms for both manual and mechanized movement of the carriage and the cross-slide, by means of the lead screw.

Headstock. The headstock is fixed to the bed and is equipped with motors, pulleys, and V-belts that supply power to the *spindle* at various rotational speeds. The speeds can be set through manually controlled selectors. Most headstocks are equipped with a set of gears, and some have various drives to provide a continuously variable speed range to the spindle. Headstocks have a hollow spindle to which workholding devices, such as *chucks* and *collets,* are attached, and long bars can be fed through for various turning operations.

Tailstock. The tailstock, which can slide along the ways and be clamped at any position, supports the other end of the workpiece. It is equipped with a center that may be fixed (*dead center),* or it may be free to rotate with the workpiece *(live center).* Drills and reamers can be mounted on the tailstock *quill* (a hollow cylindrical part with a tapered hole) to produce axial holes in the workpiece.

Feed rod and lead screw. The feed rod is powered by a set of gears from the headstock. It rotates during operation of the lathe and provides movement to the carriage and the cross-slide by means of gears, a friction clutch, and a keyway along the length of the rod. The lead screw is used for cutting threads accurately. Closing a split nut around the lead screw engages it with the carriage.

Lathe specifications

A lathe is usually specified by its *swing,* that is, the maximum diameter of the workpiece that can be machined (Table 1), by the maximum distance between the headstock and tailstock centers, and by the length of the bed. Thus, for example, a lathe may have the following size: 360 mm (14 in.) (swing) by 760 mm (30 in.) (between centers) by 1830 mm (6 ft) (length of bed). Lathes are available in a variety of styles, types of construction, stiffnesses, and power.

Bench lathes are placed on a workbench; they have fractional horsepower, are usually operated by hand feed, and are used for precision machining small workpieces. *Toolroom lathes* have high precision, thus enabling the machining of parts to close tolerances. *Engine lathes* are available in a wide range of sizes and are used for a variety of turning operations. In *gap lathes* a section of the bed in front of the headstock can be removed to accommodate larger diameter workpieces. *Special-purpose lathes* are used for applications such as railroad wheels, gun barrels, and rolling-mill rolls, with workpiece sizes as large as 1.7 m in diameter by 8 m long (66 in. × 25 ft), and capacities of 300 kW (400 hp). Spindle speeds are usually 2000 rpm, although they may range from 4000 rpm to 10,000 rpm for special applications, but may be only about 200 rpm for large lathes. Cost of engine lathes ranges from about $2000 for bench types to about $100,000 for larger units.

TABLE 2. TYPICAL CAPACITIES AND MAXIMUM WORKPIECE DIMENSIONS FOR MACHINE TOOLS

MACHINE TOOL	MAXIMUM DIMENSION (m)	POWER (kW)	MAXIMUM rpm
Lathes (swing/length)			
Bench	0.3/1	<1	3000
Engine	3/5	70	4000
Turret	0.5/1.5	60	3000
Automatic screw	0.1/0.3	20	10,000
Boring machines (work diameter/length)			
Vertical spindle	4/3	200	300
Horizontal spindle	1.5/2	70	1000
Drilling machines			
Bench and column (drill diameter)	0.1	10	12,000
Radial (column to spindle distance)	3	—	—
Numerical control (table travel)	4	—	—

Note: Larger capacities are available for special applications.

Workholding devices

Workholding devices are important in machine tools. In a lathe, one end of the workpiece is clamped to the spindle by a chuck, collet, face plate, or mandrel – or between centers.

A *chuck* is usually equipped with three or four *jaws*. Three-jaw chucks generally have a geared-scroll design that makes the jaws self-centering (Fig. 3) and hence are used for round workpieces, such as bar stock, pipes, and tubing. Workpieces can be centered within 0.025 mm (0.001 in.). Four-jaw chucks (independent chucks) have jaws that can be moved and adjusted independently of each other and thus can be used for square or rectangular, as well as odd-shaped, workpieces. They are more ruggedly constructed than three-jaw chucks and hence are used for heavy workpieces.

The jaws in both types of chuck can be reversed to permit clamping of the workpieces on either outside surfaces (as shown in Fig. 3) or on inside surfaces of hollow workpieces, such as pipes and tubing. Chucks are available in various designs and sizes. Their selection depends on the type and speed of operation, workpiece size, production and accuracy requirements, and the jaw forces required. The magnitude of jaw forces is important to ensure that the part does not slip in the chuck during machining. High spindle speeds can reduce jaw forces significantly because of centrifugal forces. Chucks are actuated manually with a special key or are power actuated. Because it takes longer to operate them, manually actuated chucks are usually used for toolroom and limited production runs.

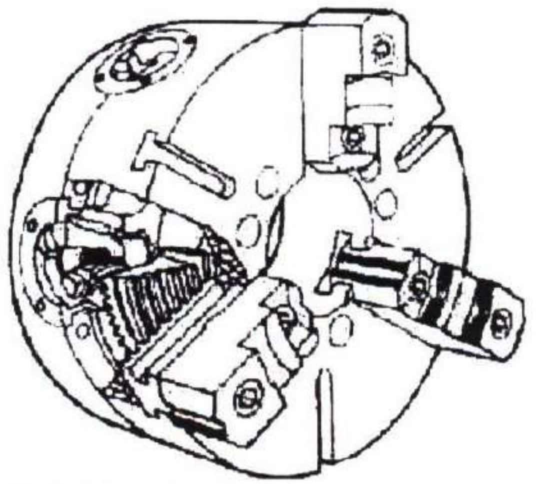

FIGURE 3. A typical three-jaw self-centering chuck of gear-scrolled design.

To meet the increasing demands for stiffness, precision, versatility, power, and high cutting speeds in modern machine tools, major advances have been made in the design of workholding devices. *Power chucks,* actuated pneumatically or hydraulically, are now used in automated equipment for high production rates. Also available are several types of power chucks with lever or wedge type mechanisms to actuate the jaws. These chucks have jaw movements that are limited to about 13 mm (0.5 in.).

A *collet* is basically a longitudinally split, tapered bushing. The workpiece is placed inside the collet, and the collet is pulled (draw-in collet; Fig. 4a) or pushed (push-out collet; Fig. 4b) into the spindle by mechanical means. The tapered surfaces shrink the segments of the collet radially, tightening the workpiece. Collets are used for round workpieces and are available in a range of internal diameters. Because the radial movement of the collet segments are small, workpieces should generally be within 0.125 mm (0.005 in.) of the nominal size of the collet.

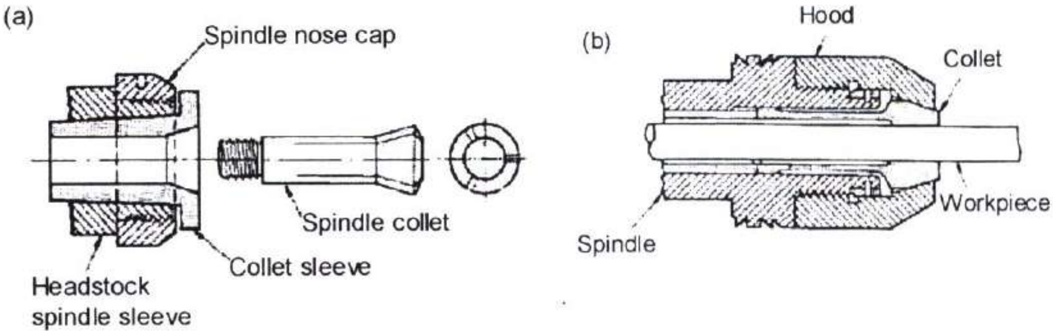

FIGURE 4. (a) Schematic illustrations of a draw-in type collet. The round workpiece is placed in the collet hole, and the conical surfaces of the collet are forced inward by pulling it with a draw bar into the sleeve. (b) A push-out type collet.

Face plates are used for clamping irregularly shaped workpieces. The plates are round and have several slots and holes through which the workpiece is bolted or clamped. *Mandrels* (Fig. 5) are placed inside hollow or tubular workpieces and are used to hold workpieces that require machining on both ends or their cylindrical surfaces.

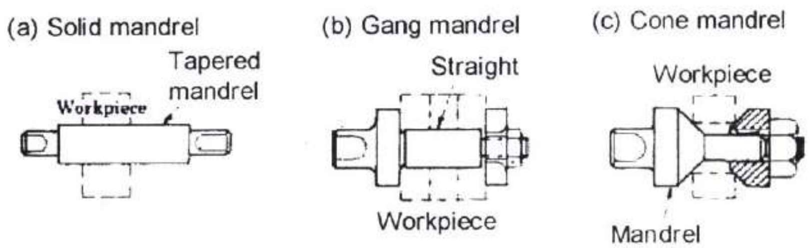

FIGURE 5. Various types of mandrels to hold workpieces for turning.

Accessories and attachments

Several devices are available as accessories and attachments for lathes. Among these devices are (1) carriage and cross-slide stops with various designs to stop the carriage at a predetermined distance along the bed; (2) devices for turning parts with various tapers or radii; (3) milling, sawing, gear-cutting, and grinding attachments; and (4) various attachments for boring, drilling, and thread cutting.

Lathe operations

In a typical turning operation, the workpiece is clamped by any one of the workholding devices that has been described. Long and slender parts are supported by a *steady rest* or *follow rest* placed on the bed; otherwise the part will deflect under the cutting forces. These rests are equipped with three adjustable fingers, which support the workpiece while allowing it to rotate freely. Steady rests are clamped to the ways, whereas follow rests travel with the carriage. The cutting tool, attached to the tool post and driven by the lead screw, removes material by traveling along the bed. A right-hand tool travels toward the headstock and a left-hand tool toward the tailstock. Workpiece facing is done by moving the tool radially, with the cross-slide, and clamping the carriage for better dimensional accuracy.

Form tools are used to produce various shapes on round workpieces by turning (see Fig. 1g). The tool moves radially inward to machine the part. Machining by form cutting is not suitable for deep and narrow grooves or sharp corners. To avoid vibration, the formed length should not be greater than about 2.5 times the minimum diameter of the part.

The boring operation on a lathe is similar to turning. Boring is performed on hollow workpieces or in a hole made previously by drilling or other means. The workpiece is held in a chuck or some other suitable workholding device.

Drilling can be performed on a lathe by mounting the drill in a drill chuck into the tailstock quill (a tubular shaft). The workpiece is placed in a workholder on the headstock, and the quill is advanced by rotating the hand wheel. Holes drilled in this manner may not be concentric because of the drill's radial drifting. The concentricity of the hole is improved by boring the drilled hole. Drilled holes may be reamed on lathes in a manner similar to drilling.

The tools for parting, grooving, thread cutting, and various other operations are specially shaped for the particular purpose or are available as inserts. Knurling is performed on a lathe with hardened rolls (see Fig. 1*l*). The surface of the rolls is a replica of the profile to be generated. The rolls are pressed radially against the rotating workpiece while the tool moves axially along the part.

Turning parameters

The majority of turning operations involve simple single-point cutting tools. The geometry of a typical right-hand cutting tool for turning is shown in Fig. 6. Such tools are described by a standardized nomenclature. Each group of tool and workpiece materials has an optimum set of tool angles, which was developed largely through experience (Table 2).

Rake angles are important in controlling the direction of chip flow and to the strength of the tool tip. Positive angles improve the cutting operation by reducing forces and temperatures. However, positive angles produce a small included angle of the tool tip, which, depending on the toughness of the tool material, may cause premature tool failure. Side rake angle is more important than back rake angle, although the latter usually controls the direction of chip flow.

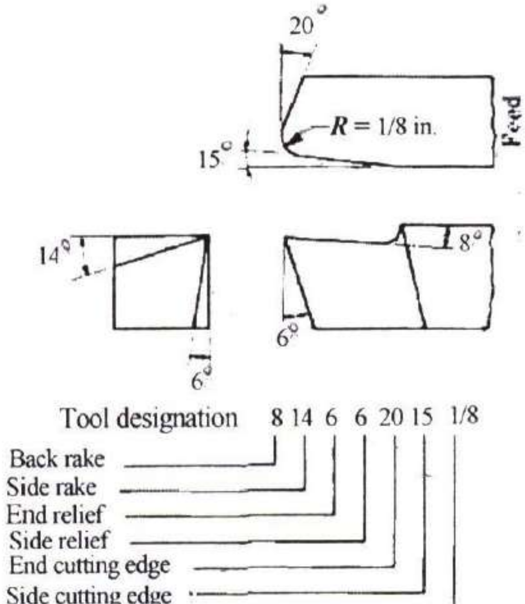

FIGURE 6. Designation and symbols for a right-hand cutting tool.

Relief angles control interference and rubbing at the tool-workpiece interface. If the relief angle is too large, the tool may chip off; if too small, flank wear may be excessive. *Cutting edge angles* affect chip formation, tool strength, and cutting forces to various degrees. *Nose radius* affects surface finish and tool-tip strength. The sharper the radius, the rougher will be the surface finish of the workpiece and the lower will be the strength of the tool. However, large nose radii can lead to tool chatter. Recommendations for cutting-tool selection are given in Table 3. The most commonly used tool materials are M2 and M3 tool steels and C-6 and C-7 carbides.

Conventional cutting-speed ranges for a variety of materials are presented in Table 4. Although many metals and nonmetallic materials can be machined without a cutting fluid, in many cases the application of fluids can improve the operation. Recommendations for cutting fluids are given in Table 5.

TABLE 2. GENERAL RECOMMENDATIONS FOR TURNING TOOLS

MATERIAL	HIGH-SPEED STEEL					CARBIDE (INSERTS)				
	BACK RAKE	SIDE RAKE	END RELIEF	SIDE RELIEF	SIDE AND END CUTTING EDGE	BACK RAKE	SIDE RAKE	END RELIEF	SIDE RELIEF	SIDE AND END CUTTING EDGE
Aluminum and magnesium alloys	*20*	15	12	10	5	*0*	5	5	5	15
Copper alloys	5	10	8	8	5	0	5	5	5	15
Steels	10	12	5	5	15	–5	–5	5	5	15
Stainless steels	5	8–10	5	5	15	–5–0	–5–5	5	5	15
High-temperature alloys	0	10	5	5	15	5	0	5	5	45
Refractory alloys	0	20	5	5	5	0	0	5	5	15
Titanium alloys	0	5	5	5	15	–5	–5	5	5	5
Cast irons	5	10	5	5	15	–5	–5	5	5	15
Thermoplastics	0	0	20–30	15–20	10	0	0	20–30	15–20	10
Thermosets	0	0	20–30	15–20	10	0	15	5	5	15

TABLE 3. COMMON TOOL MATERIALS FOR CUTTING OPERATIONS

WORKPIECE	TURNING	DRILLING, REAMING, TAPPING	MILLING	BROACHING, GEAR CUTTING
Aluminum and copper alloys	HSS, WC, CA, CC, D	HSS, WC	HSS, WC, CA	HSS
Carbon steels and cast irons	HSS, WC, CA, CC, C	HSS, WC	HSS, WC, CA	HSS, WC, CA
Alloys steels and alloy cast irons	WC, CA, CC, C, CBN	HSS, WC	WC, CA	HSS, WC
High-temperature alloys and titanium	HSS, WC, CA, CC, C	HSS, WC	WC, CA	HSS, WC
Plastics and composites	WC, CA, CBN, D	WC	WC	HSS

Note: HSS, high-speed steel; WC, tungsten carbide; CA, cast alloy; CC, coated carbide;
C, ceramic and cermet; CBN, cubic boron nitride; and D, diamond.

TABLE 4. RECOMMENDATIONS FOR CUTTING SPEEDS IN TURNING

WORKPIECE MATERIAL	CUTTING SPEED (m/s)	
	HSS	*WC*
Aluminum alloys	3-4	5-7
Magnesium alloys	4	10
Copper alloys	0.5-2	1-5
Steels	0.5-1	1-3
Stainless steels	0.15-0.5	1-2
High-temperature alloys	0.05-0.1	0.15-0.3
Titanium alloys	0.15-1	0.5-2
Cast irons	0.15-0.5	0.5-2
Thermoplastics	1.5-2	2-3
Thermosets	1-2	1-4

Note:

(a) Depth of cut is usually 4 mm for rough turning and 0.7 mm for finish turning

(b) Feeds for rough turning range from 0.2 mm/rev for materials with high hardness, to 2 mm/rev for lower hardness. Finishing cuts require lower feeds.

(c) Cutting speeds are for uncoated tools. Speeds for coated tools are from 25–75 percent higher.

(d) Cutting speeds for ceramic tools can be 2–3 times higher than the values indicated.

(e) Cutting speed for diamond tools is usually 4–15 m/s, depth of cut 0.05–0.2 mm, and feed 0.02-0.05 mm/rev.

(f) As hardness increases, cutting speed, feed, and depth of cut should be decreased.

(g) Speeds for free-machining metals are higher than those indicated.

(h) Speeds for other cutting processes are generally lower by as much as 75 percent.

TABLE 5. GENERAL RECOMMENDATIONS FOR CUTTING FLUIDS FOR MACHINING

MATERIAL	TYPE OF FLUID
Aluminum	D, MO, E, MO + FO, CSN
Beryllium	MO, E, CSN
Copper	D, E, CSN, MO + FO
Lead	D
Magnesium	D, MO, MO + FO
Nickel	MO, E, CSN
Refractory	MO, E, EP
Steels (carbon and low alloy)	D, MO, E, CSN, EP
Steels (stainless)	D, MO, E, CSN
Titanium	CSN, EP, MO
Zinc	D, MO, E, CSN
Zirconium	D, E, CSN

Note: CSN, chemicals and synthetics; D, dry; E, emulsion; EP, extreme pressure; FO, fatty oil; and MO, mineral oil.

Turning process capabilities

The surface finish and dimensional accuracy obtained in turning and related operations (Figs. 7 and 8) depends on factors such as the characteristics and condition of the machine tool, stiffness, vibration and chatter, process parameters, tool shape and wear, cutting fluids, machinability of the workpiece material, and operator skill. As a result, a wide range of surface finishes is obtained, as shown in Fig. 7.

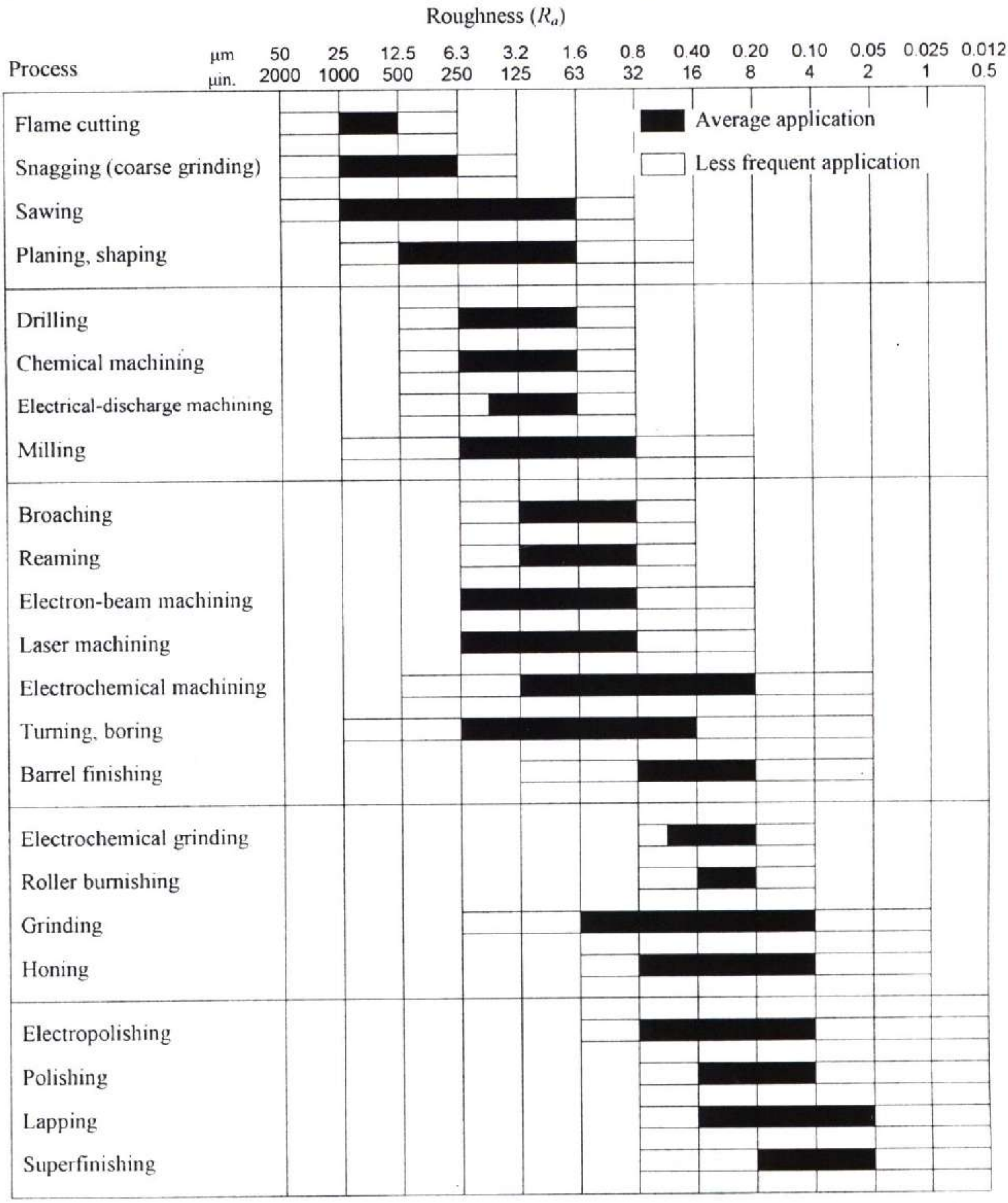

FIGURE 7. Range of surface roughnesses obtained in various machining processes. Note the wide range within each group.

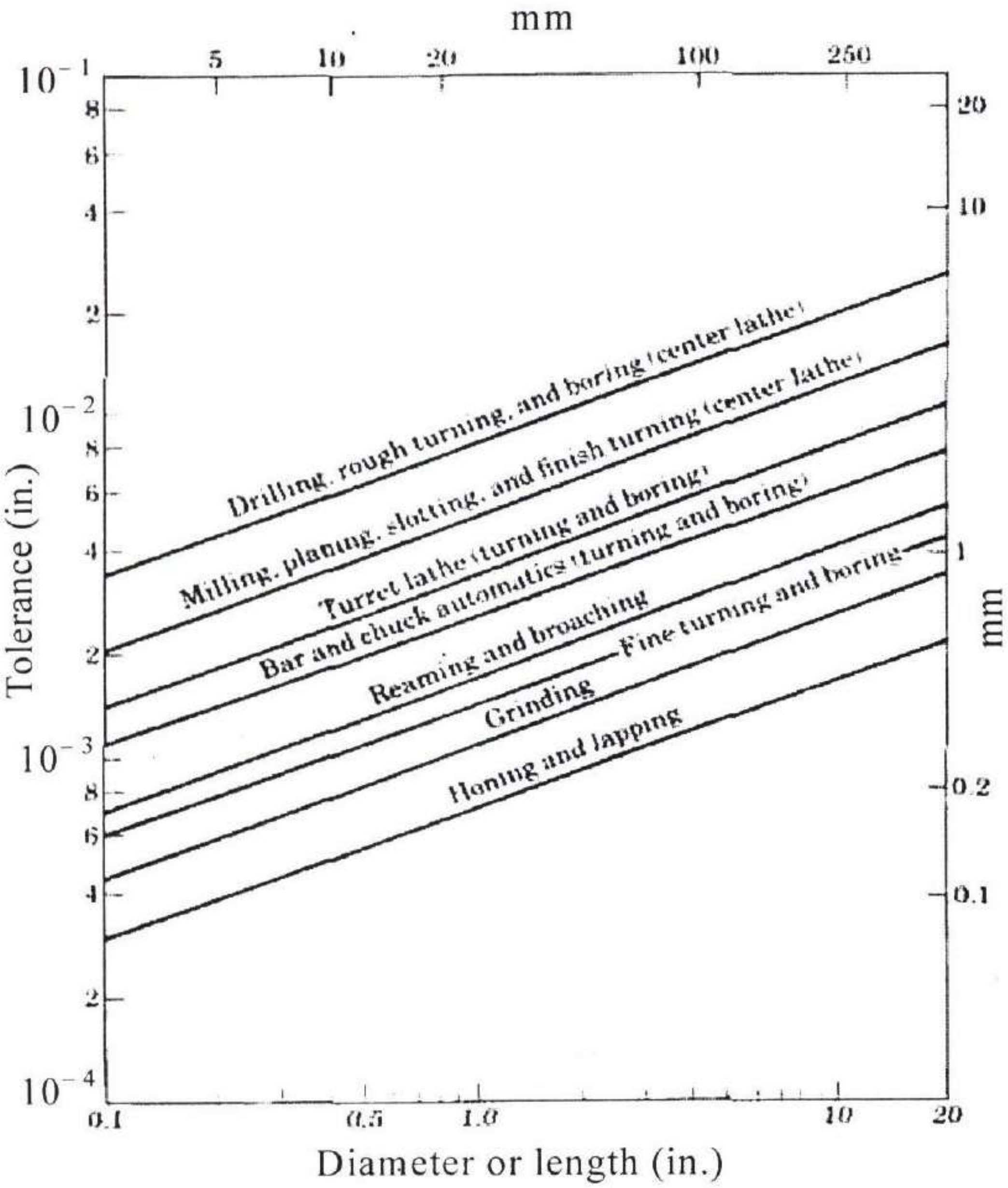

FIGURE 8. Range of tolerances obtained in various machining processes as a function of workpiece size. *Source:* Adapted from *Manufacturing Planning and Estimating Handbook*, McGraw-Hill, 1963.

Screw-thread cutting on a lathe

A typical thread-cutting operation on a lathe is shown in Fig. 9(a). The cutting tool, whose shape depends on the type of thread to be cut, is mounted on a holder that is moved along the length of the workpiece by the lead screw (see Fig. 2). The axial movement of the tool in relation to the rotation of the workpiece determines the lead of the screw thread. Thus for a fixed spindle rpm, the slower the tool movement, the finer the thread will be. In thread cutting, the cutting tool may be fed radially into the workpiece, thus cutting both sides of the thread at the same time as in form cutting. However, this method usually produces a poor surface finish. Generally, a number of passes in the sequence shown in Fig. 9(b) are required to produce good dimensional accuracy and surface finish. Although cutting threads on lathes is an old and versatile method, it requires considerable operator skill and is a slow, hence uneconomical, process. Consequently, except for small production runs, it has been largely replaced by other methods, such as thread rolling and automatic screw machining.

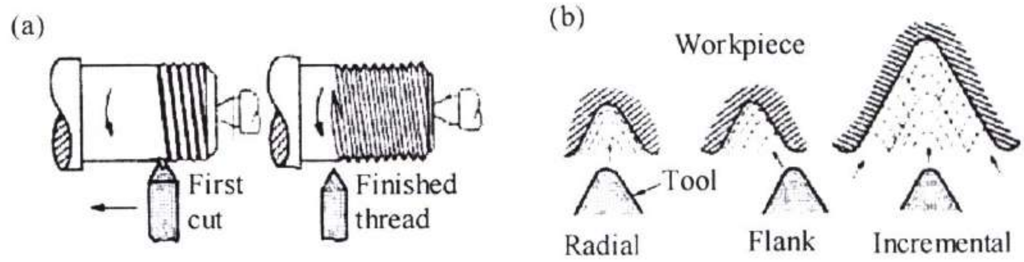

FIGURE 9 (a) Cutting screw threads on a lathe with a single-point cutting tool. (b) Cutting screw threads with a single-point tool in several passes. This process is normally utilized for large threads.

The production rate in cutting screw threads can be increased with tools called *die-head chasers* (Figs. 10a and b). These tools typically have four cutters with multiple teeth and can be adjusted radially. After threads are cut, the cutters open automatically (hence they are also called *self-opening die heads)* by rotating around their axes to allow the part to be removed. *Solid threading dies* (Fig. 10c) are also available for cutting straight or tapered screw threads. These dies are used mostly in threading ends of pipes and tubing and are not suitable for production work.

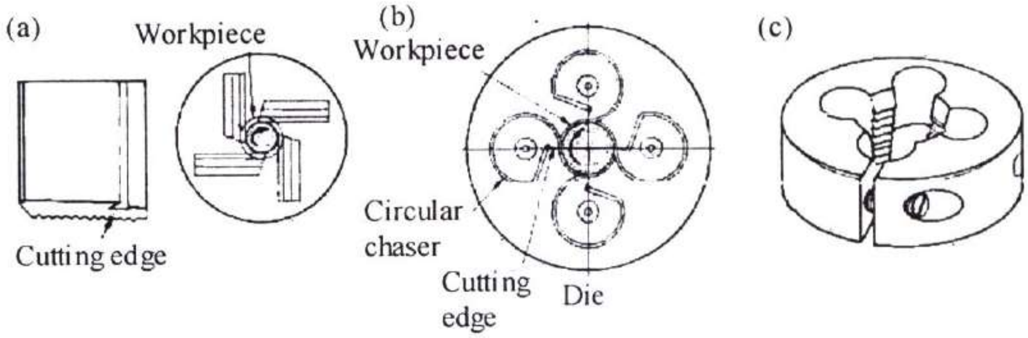

FIGURE 10. (a) Straight chasers for cutting threads on a lathe. (b) Circular chasers. (c) A solid threading die.

Drilling and Holemaking Operations

When inspecting various products around us, we realize that the vast majority have a variety of holes. Holes are used either for assembly with fasteners such as bolts, screws, and rivets or to provide access inside machinery or products. Note, for example, the number of rivets on an airplane's fuselage or the bolts in various components under the hood of an automobile, each rivet or bolt requiring a hole. Holemaking is thus among the most important operations in manufacturing.

Drills

One of the most common of all machining processes is drilling. Drills usually have a high length-to-diameter ratio (Fig. 11), hence are capable of producing deep holes. However, they are somewhat flexible, depending on their diameter, and should be used with care in order to drill holes accurately and to prevent the drill from breaking. Furthermore, the chips that are produced within the workpiece have to move in the direction opposite to the axial movement of the drill. Consequently, chip disposal and the effectiveness of cutting fluids can be significant problems in drilling.

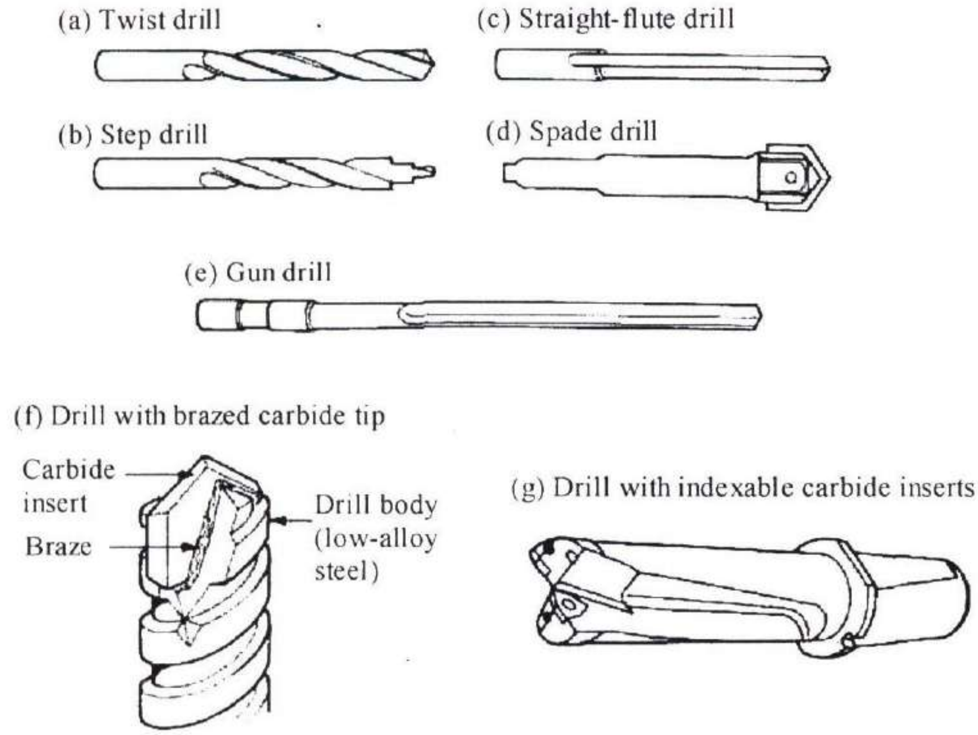

FIGURE 11. Various types of drills.

Twist drill. The most common drill is the standard-point twist drill (Fig. 12). The main features of the drill point are a *point angle, lip-relief angle, chisel-edge angle,* and *helix angle.* The geometry of the drill tip is such that the normal rake angle and velocity of the cutting edge vary with the distance from the center of the drill. In the outer regions of the hole the rake angle is positive, the cutting speed is high and two chips produced are continuous. In the center, however, the rake angle is highly negative and the speed is low, producing a highly deformed, coarse chip.

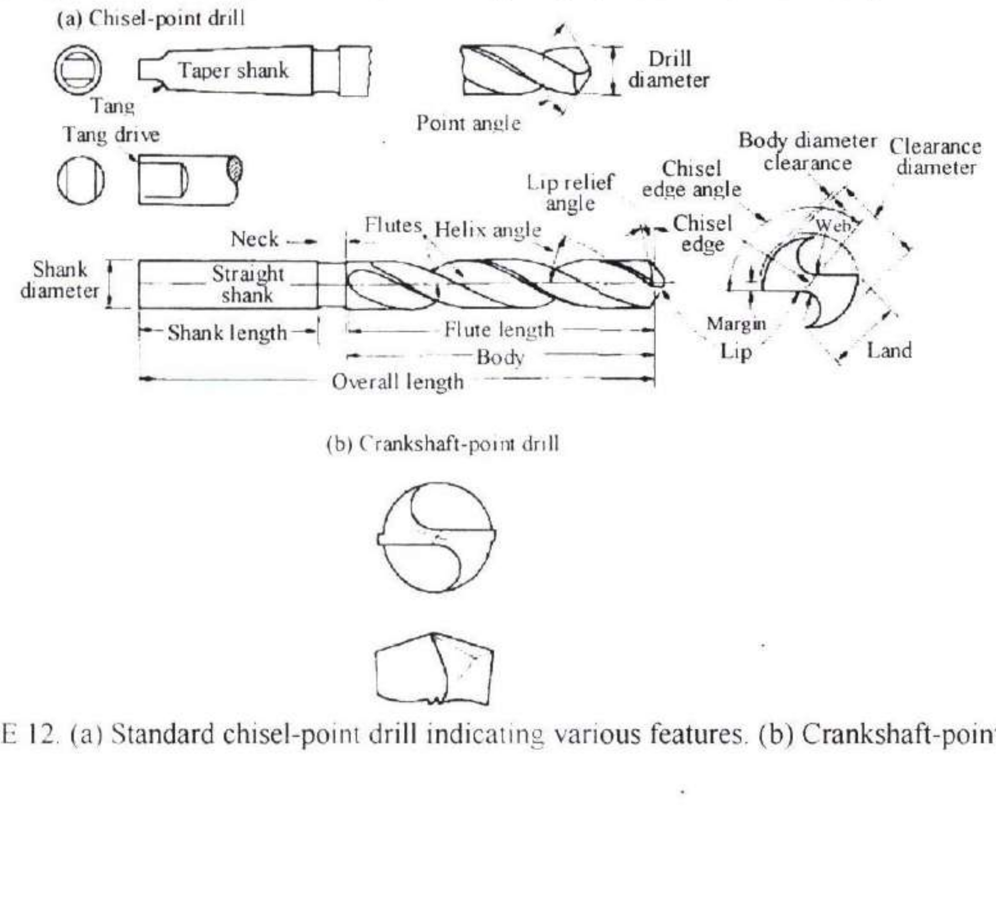

FIGURE 12. (a) Standard chisel-point drill indicating various features. (b) Crankshaft-point drill.

Generally, two spiral grooves *(flutes)* run the length of the drill, and the chips produced are guided upward through these grooves. The grooves also serve as passageways to enable the cutting fluid to reach the cutting edges. Drills are available with chip-breaker features ground along the cutting edges. This feature is important in drilling with automated machinery where disposal of long chips without operator interference is important. Drills are also provided with internal longitudinal holes through which cutting fluids are forced, thus improving lubrication and washing away the chips.

General recommendations for drill geometry for various workpiece materials are presented in Table 6. These angles are based on experience in drilling operations and are designed to produce accurate holes, minimize drilling forces and torque, and optimize drill life. Various other drill-point geometries have been developed to improve drill performance, but they require special drilling techniques and equipment.

TABLE 6. GENERAL RECOMMENDATIONS FOR DRILL GEOMETRY FOR HIGH-SPEED STEEL TWIST DRILLS

WORKPIECE MATERIAL	POINT ANGLE	LIP-RELIEF ANGLE	CHISEL-EDGE ANGLE	HELIX ANGLE	POINT
Aluminum alloys	90-118	12-15	125-135	24-48	Standard
Magnesium alloys	70-118	12-15	120-135	30-45	Standard
Copper alloys	118	12-15	125-135	10-30	Standard
Steel	118	10-15	125-135	24-32	Standard
High-strength steels	118-135	7-10	125-135	24-32	Crankshaft
Stainless steels, low strength	118	10-12	125-135	24-32	Standard
Stainless steels, high strength	118-135	7-10	120-130	24-32	Crankshaft
High-temp. alloys	118-135	9-12	125-135	15-30	Crankshaft
Refractory alloys	118	7-10	125-135	24-32	Standard
Titanium alloys	118-135	7-10	125-135	15-32	Crankshaft
Cast irons	118	8-12	125-135	24-32	Standard
Plastics	60-90	7	120-135	29	Standard

Other types of drills. Several types of drills are shown in Figs. 11 and 13. A *step drill* produces holes of two or more different diameters. A *core drill* has a square end and is used to make an existing hole larger. *Counterboring* and *countersinking drills* produce depressions on the surface to accommodate the heads of screws and bolts. A *center drill* is a short drill and is used to help start a hole and guide the drill for regular drilling. *Spade drills* have a removable tip or bit and are used to produce large and deep holes. They have the advantages of higher stiffness (because of the absence of flutes in the body of the drill), ease of grinding the cutting edge, and lower cost. *Crankshaft drills* (Fig. 12b) have good centering ability, and because chips tend to break up easily these drills are suitable for drilling deep holes.

Gun drilling. Developed originally for drilling gun barrels, hence the word gun, *gun drilling* requires a special drill (see Figs. 11e and 13) and is used for drilling deep holes. Hole depth-to-diameter ratios can be 300 or higher. The thrust force (the radial force that tends to push the drill sideways) is balanced by bearing pads on the drill that slide along the inside surface of hole (Fig. 14). Thus a gun drill is a self-centering – an important factor in drilling straight, deep holes. The cutting fluid is forced under high pressure through a longitudinal hole in the body of the drill. In addition to its lubricating and cooling functions, the fluid flushes out chips that otherwise would trapped in the hole and interfere with the drilling operation. Cutting speeds are usually high and feeds are low.

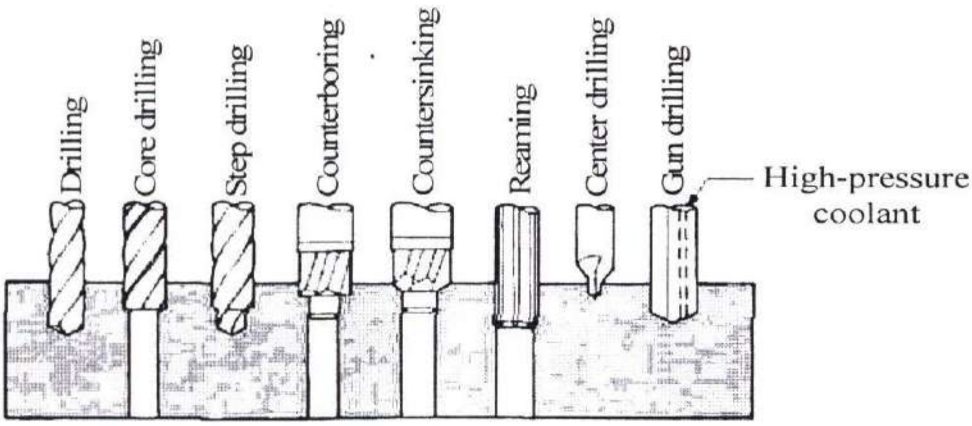

FIGURE 13. Various types of drills and drilling operations.

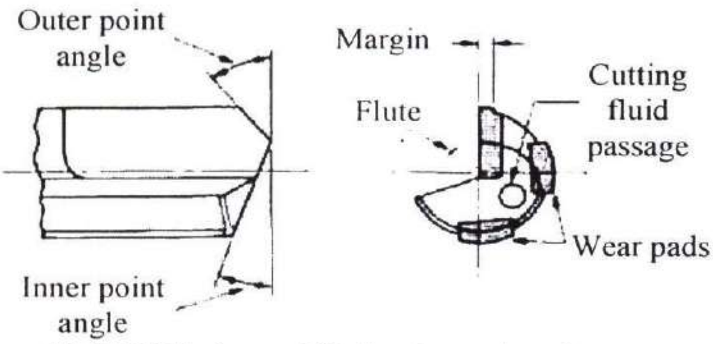

FIGURE 14. A gun drill showing various features.

Trepanning. In *trepanning* the cutting tool (Fig. 15a) produces a hole by removing a disk-shaped piece (core), usually from flat plates. Thus a hole is produced without reducing all the material to chips. The process can be used to make disks up to 150 mm (6 in.) in diameter from flat sheet or plate. Trepanning can also be used to make circular grooves in which O-rings are to be placed (see also Fig. 1f). Trepanning can be done on lathes, drill presses (Fig. 15b), or other machines, using single point or multipoint tools. A variation of trepanning is *gun-trepanning*, which uses a cutting tool similar to a gun drill except that the tool has a central hole in it.

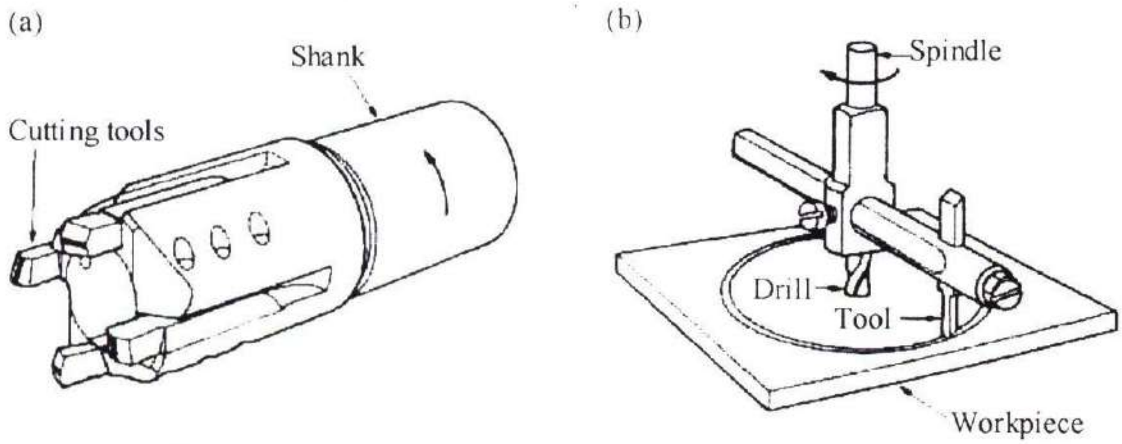

FIGURE 15. (a) Trepanning tool. (b) Trepanning with a drill-mounted single cutter.

Drilling practice

Drills and similar holemaking tools are usually held in *drill chucks* which may be tightened with or without keys. Special chucks and collets, with various quick-change features that do not require stopping the spindle, are available for use on automated machinery. Because a drill doesn't have a centering action, it tends to "walk" on the workpiece surface at the beginning of the operation. This problem is particularly severe with small-diameter drills. To start a hole properly, the drill should be guided to keep it from deflecting sideways. A small starting hole can be made with a center drill, fixtures (such as a bushing) can be used, or the drill point may be ground to an S shape *(spiral point)*. This shape's self-centering characteristic eliminates center drilling, produces accurate holes, and improves drill life. These factors are particularly important in automated production with computer numerical control machines.

Chip removal during drilling can be difficult, especially for deep holes in soft and ductile materials. The drill should be retracted periodically to remove chips that may have accumulated along the flutes; otherwise the drill may break because of excessive torque.

Because of its rotary motion, drilling produces holes with walls that have circumferential marks. In contrast, punched holes have longitudinal lines. This difference is significant in terms of the hole's fatigue properties. Drills generally leave a burr on the bottom surface upon breakthrough, thus necessitating deburring operations.

Rotational speeds in drilling can range up to 30,000 rpm for drills less than 1 mm in diameter. Recommended ranges of drilling speed and feeds are given in Table 7.

TABLE 7. GENERAL RECOMMENDATIONS FOR SPEEDS AND FEEDS IN DRILLING

		FEED (mm/rev) DRILL DIAMETER (mm)	
WORKPIECE MATERIAL	SPEED (m/s)	1.5	50
Aluminum alloys	0.5–2	0.025	0.75
Magnesium alloys	0.75–2	0.025	0.75
Copper alloys	0.25–1	0.025	0.65
Steels	0.3–0.5	0.025	0.75
Stainless steels	0.2–0.3	0.025	0.45
Titanium alloys	0.1–0.3	0.01	0.3
Cast irons	0.3–1	0.025	0.75
Thermoplastics	0.5–1	0.025	0.3
Thermosets	0.3–1	0.025	0.4

Note: As hole depth increases, speeds and feeds should be reduced.

Drilling Machines

Drilling machines are used for drilling holes, tapping, reaming, and other general-purpose, small-diameter boring operations. These machine tools are generally vertical. The most common vertical type is the drill press, the major components of which are shown in Fig. 16. The workpiece is placed on an adjustable table, either by clamping it directly into the slots and holes on the table or by using a vise, which in turn can be clamped to the table. The workpiece should be properly clamped, both for safety and accuracy, because the drilling torque can be high enough to rotate the workpiece. The drill is lowered manually by hand wheel or by power feed at preset rates.

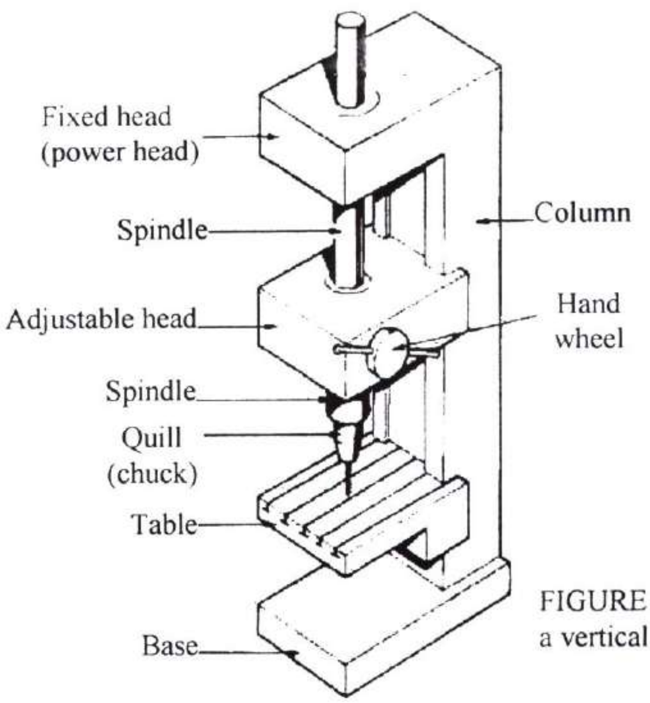

FIGURE 16. Schematic illustration of the components of a vertical drill press.

Manual feeding requires some skill in judging the appropriate feed rate. In order to maintain proper cutting speeds at the cutting edges of drills, the spindle speed on drilling machines has to be adjustable to accommodate different sizes of drills. Adjustments are made by means of pulleys, gear boxes, or variable-speed motors. Drill presses are usually designated by the largest workpiece diameter that can be accommodated on the table. Sizes typically range from 150 mm to 1250 mm (6 in. to 50 in.).

Types of drilling machines range from simple *bench-type* units, used to drill small-diameter holes, to large *radial drills,* which can accommodate large work-pieces. The distance between the column and the spindle center can be as large as 3 m (10 ft). The drill head of *universal* drilling machines can be swiveled to drill holes at an angle. Recent developments in drilling machines include computer numerical control, three-axis machines in which various drilling operations are performed automatically and in the desired sequence with the use of a turret. Note that the turret holds several different tools.

Reaming and Reamers

Reaming is an operation used to make an existing hole dimensionally more accurate than can be obtained by drilling alone and to improve its surface finish. The most accurate holes are produced by the following sequence of operations:

- Centering
- Drilling
- Boring
- Reaming

For even better accuracy and surface finish, holes may be internally ground and honed.

A reamer (Fig. 17a) is a multiple-cutting-edge tool with straight or helically fluted edges, which removes very little material. The shanks may be straight or tapered, as in drills. The basic types of reamers are hand and machine.

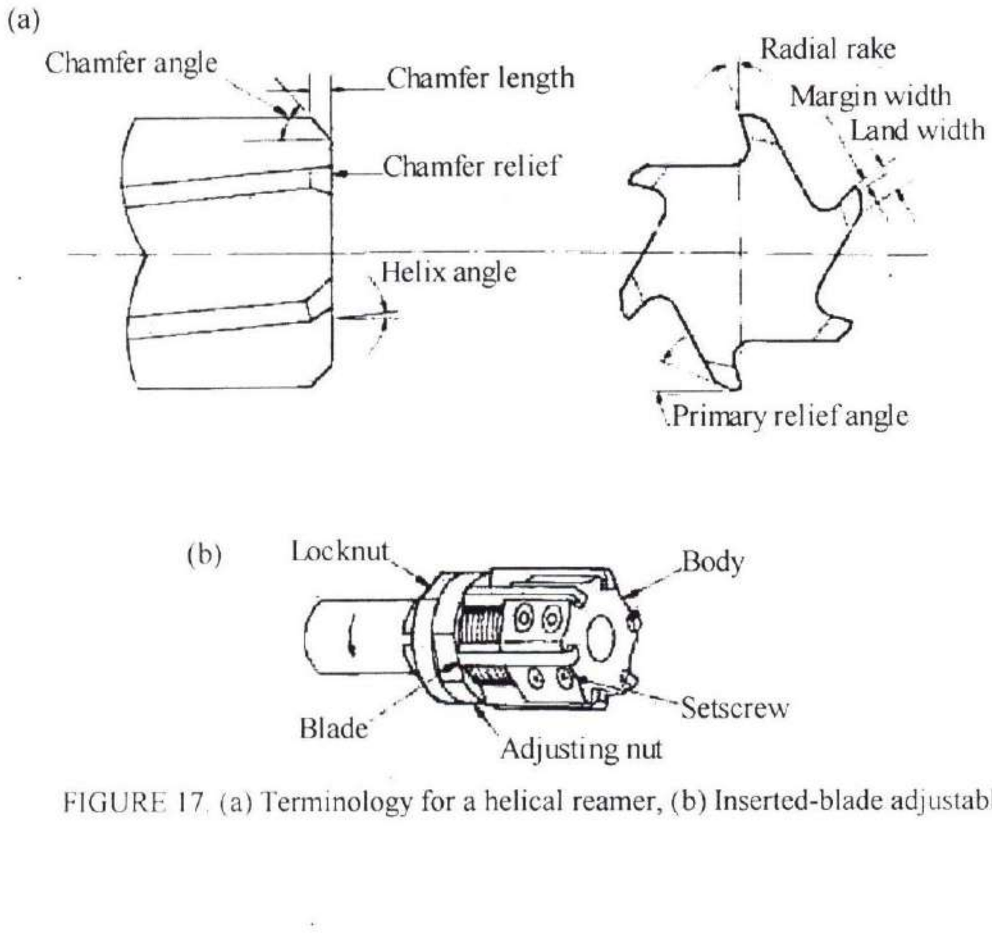

FIGURE 17. (a) Terminology for a helical reamer, (b) Inserted-blade adjustable reamer.

Hand reamers are straight or have a tapered end in the first third of their length. Various *machine reamers*, also called chucking reamers because they are mounted in a chuck and operated by a machine, are available. There are two types of chucking reamers. *Rose* reamers have cutting edges with wide margins and no relief (see Fig. 17a). They remove considerable material and true up a hole for flute reaming. *Fluted* reamers have small margins and relief, with a rake angle of about 5°. They are usually used for light cuts of about 0.1 mm (0.004 in.) on the hole diameter. *Shell* reamers, which are hollow and mounted on an arbor, are generally used for holes larger than 20 mm (0.75 in.). *Expansion* reamers are adjustable for small variations in hole size and also to compensate for wear of the reamer's cutting edges. *Adjustable* reamers (Fig. 17b) can be set for specific hole diameters and are thus versatile.

Reamers are usually made of high-speed steels (MI, M2, and M7) or solid carbides (C-2) – or have carbide cutting edges. Reaming speeds are generally lower than drilling for the same material, but feeds are higher. Proper reamer maintenance and reconditioning are important for hole accuracy and surface finish. Reamers may be held rigidly, as in a chuck, or they may float in their holding fixtures to ensure alignment or be piloted in guide bushings placed above and below the workpiece.

Tapping and Taps

Internal threads in workpieces can be produced by tapping. A tap is basically a threading tool with multiple cutting teeth (Fig. 18a). Taps are generally available with three or four flutes. Three-fluted taps are stronger because of the larger amount of material available in the flute. *Tapered taps* are designed to reduce the torque required for tapping through holes. *Bottoming taps* are for tapping blind holes to their full depth. *Collapsible taps* are for large diameter holes. After tapping has been completed, the tap is mechanically collapsed and, without having to rotate it, removed from the hole. Tap sizes range up to 100 mm (4 in.).

Chip removal can be a significant problem during tapping because of the small clearances involved. If chips aren't removed properly, the resulting excessive torque can break the tap. The use of a proper cutting fluid and periodic reversal and removal of the tap from the hole are effective means of chip removal and improving the quality of the tapped hole. The tapping is among the most severe cutting operations, requiring low cutting speeds and effective cutting fluids.

Tapping may be done by hand or in drilling machines, lathes, or automatic screw machines, using tapping heads to hold the taps. Special tapping machines are also available with features for multiple tapping operations. The system for automatic tapping of nuts shown in Fig. 18(b) can achieve a production rate of 500 pieces per hour. With proper lubrication, tap life may be as high as 10,000 holes tapped. Tap life can be determined by the same technique used to measure drill life. Taps are usually made of carbon steels for light-duty applications and high-speed steels (MI, M2, M7, and M10) for production work.

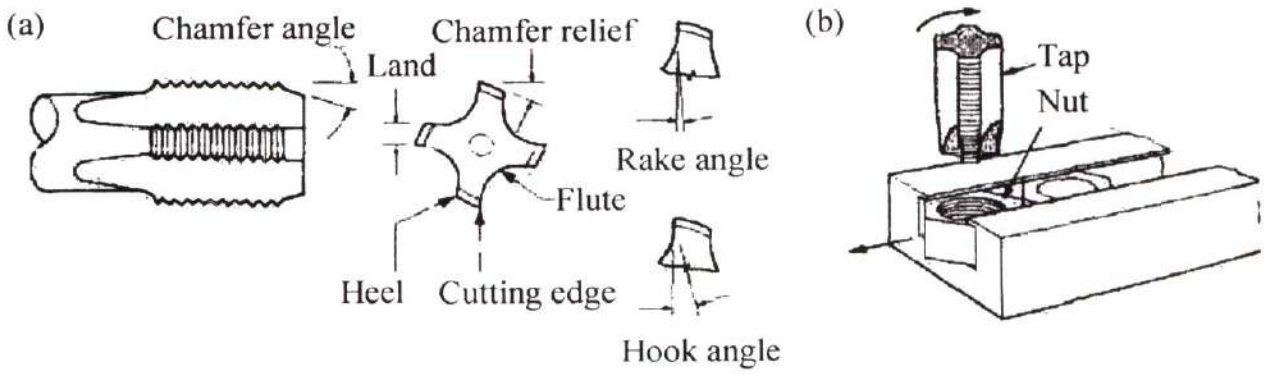

FIGURE 18. (a) Terminology for a tap. (b) Tapping of steel nuts in production.

SUMMARY

Cutting processes that produce external and internal circular profiles are turning, boring, and drilling. Reaming, tapping, and die threading are processes for finishing workpieces. Chip formation in all these processes is essentially the same. However, because of the three-dimensional nature of the cut, chip movement and its control are important considerations, since otherwise they interfere with the cutting operation. Chip removal can be a significant problem especially in drilling and tapping and can lead to tool breakage. Each process should be studied in order to understand the interrelationships of design parameters, such as dimensional accuracy, surface finish, and integrity, and process parameters, such as speed, feed, depth of cut, tool material and shape, and cutting fluids.

Design guidelines should be followed carefully to take full advantage of the capabilities of each process. Parts to be machined may have been produced by casting, forging, extrusion, powder metallurgy, etc. The closer the blank to be machined to the final shape desired, the fewer the number and extent of machining processes required. Such net-shape manufacturing is of major significance in minimizing costs.

References

1. Manufacturing Engineering and Technology. / Serope Kalpakjian. Illinois Institute of Technology: Addison-Wesley Publishing Company. 1990. – 1224 p.
2. Modern Manufacturing Process Engineering / Benjamin W. Neibel, Alan B. Draper, Richard A. Wysk. Copyright © 1989 by McGraw-Hill, Inc. - 896 p.
3. Metal Process Engineering / Under the editorship of prof. P. Polukhin. English translation, Mir Publishers. 1977. – 426 p.

Михаил Климович Князев

Сергей Валерьевич Сергеев

МЕТОДЫ ОБРАБОТКИ РЕЗАНИЕМ ЦИЛИНДРИЧЕСКИХ ПОВЕРХНОСТЕЙ

Редактор Т. А. Ястремская

Св. план, 2002
Подписано в печать 17.04.2002
Формат 60х84 1/16.Бум. офс. № 2 . Офс. печ.
Усл. печ.л. 1,2. Уч-изд.л 1,31. Т. 25 экз. Заказ 230. Цена свободная

Национальный аэрокосмический университет им. Н. Е. Жуковского
«Харьковский авиационный институт»
61070, Харьков-70, ул. Чкалова, 17
http://www.khai.edu
Издательский центр «ХАИ»
61070, Харьков-70, ул. Чкалова, 17
izdat@khai.edu