13.4 MACHINING PROCESSES AND MACHINE TOOLS

by Serope Kalpakjian

REFERENCES: Arnone, "High Performance Machining," Hanser, 1998. "ASM Handbook," Vol. 16: "Machining," ASM International, 1989. Brown, "Advanced Machining Technology Handbook," McGraw-Hill, 1998. El-Hofy, "Advanced Machining Processes," McGraw-Hill, 2005. Erdel, "High-Speed Machining," Society of Manufacturing Engineers, 2003. Kalpakjian and Schmid, "Manufacturing Engineering and Technology," 5th ed., Prentice-Hall, 2006. Krar, "Grinding Technology," 2d ed., Delmar, 1995. "Machining Data Handbook," 3d ed., 2 vols., Machinability Data Center, 1980. "Metal Cutting Handbook," 7th ed., Industrial Press, 1989. Meyers and Slattery, "Basic Machining Reference Handbook," Industrial Press, 1988. Salmon, "Modern Grinding Process Technology," McGraw-Hill, 1992. Shaw, "Metal Cutting Principles," 2d ed., Oxford, 2005. Shaw, "Principles of Abrasive Processing," Oxford, 1996. Sluhan (ed.), "Cutting and Grinding Fluids: Selection and Application," Society of Manufacturing Engineers, 1992. Stephenson and Agapiou, "Metal Cutting: Theory and Practice," Dekker 1996. Trent and Wright, "Metal Cutting," 4th ed., Butterworth Heinemann, 2000. Walsh, "Machining and Metalworking Handbook," McGraw-Hill, 1994. Webster et al., "Abrasive Processes," Dekker, 1999. Weck, "Handbook of Machine Tools," 4 vols., Wiley, 1984.

INTRODUCTION

Machining processes, which include cutting, grinding, and various nonmechanical chipless processes, are desirable or even necessary for the following basic reasons: (1) Closer dimensional tolerances, surface roughness, or surface-finish characteristics may be required than are available by casting, forming, powder metallurgy, and other shaping processes; and (2) part geometries may be too complex or too expensive to be manufactured by other processes. However, machining processes inevitably waste material in the form of chips, production rates may be low, and unless carried out properly, the processes can have detrimental effects on the surface properties and performance of parts.

Traditional machining processes consist of turning, boring, drilling, reaming, threading, milling, shaping, planing, and broaching, as well as abrasive processes such as grinding, ultrasonic machining, lapping, and honing. Advanced processes include electrical and chemical means of material removal, as well as the use of abrasive jets, water jets, laser beams, and electron beams. This section describes the principles of these operations, the processing parameters involved, and the characteristics of the machine tools employed.

BASIC MECHANICS OF METAL CUTTING

The basic mechanics of chip-type machining processes (Fig. 13.4.1) are shown, in simplest two-dimensional form, in Fig. 13.4.2. A tool with a certain **rake angle** α (positive as shown) and **relief angle** moves along the surface of the workpiece at a depth t_1. The material ahead of the tool is sheared continuously along the **shear plane,** which makes an angle of ϕ with the surface of the workpiece. This angle is called the **shear angle** and, together with the rake angle, determines the chip thickness t_2. The ratio of t_1 to t_2 is called the **cutting ratio** r. The relationship between the shear angle, the rake angle, and the cutting ratio is given by the equation $\tan \phi = r \cos \alpha/(1 - r \sin \alpha)$. It can readily be seen that the shear angle is important in that it controls the thickness of the chip. This, in turn, has great influence on cutting performance. The **shear strain** that the material undergoes is given by the equation $\gamma = \cot \phi + \tan (\phi - \alpha)$. Shear strains in metal cutting are usually less than 5.

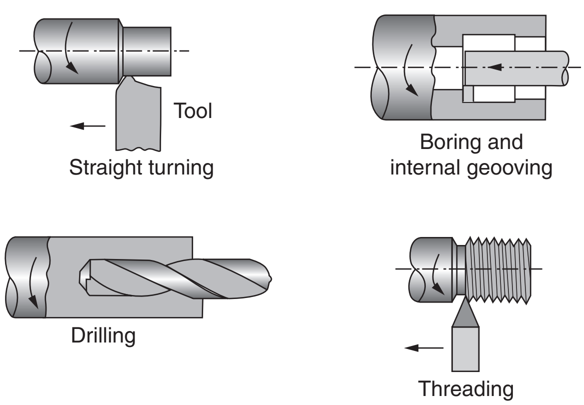


Fig. 13.4.1 Examples of chip-type machining operations.

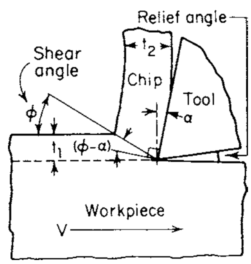


Fig. 13.4.2 Basic mechanics of metal cutting process.

Investigations have shown that the shear plane may be neither a plane nor a narrow zone, as assumed in simple analysis. Various formulas have been developed which define the shear angle in terms of such factors as the rake angle and the friction angle β. (See Fig. 13.4.3.)

Because of the large shear strains that the chip undergoes, it becomes hard and brittle. In most cases, the chip curls away from the tool. Among possible factors contributing to chip curl are nonuniform normal stress distribution on the shear plane, strain hardening, and thermal effects.

Regardless of the type of machining operation, some basic types of chips or combinations of these are found in practice (Fig. 13.4.4).

Continuous chips are formed by continuous deformation of the workpiece material ahead of the tool, followed by smooth flow of the chip along the tool face. These chips ordinarily are obtained in cutting ductile materials at high speeds.

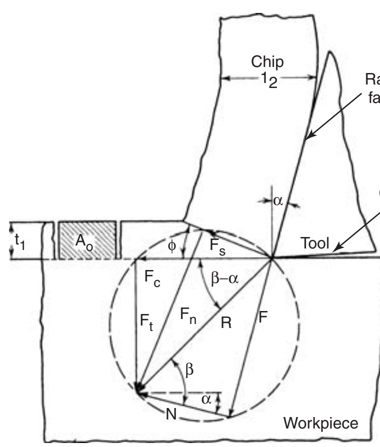


Fig. 13.4.3 Force system in metal cutting process.

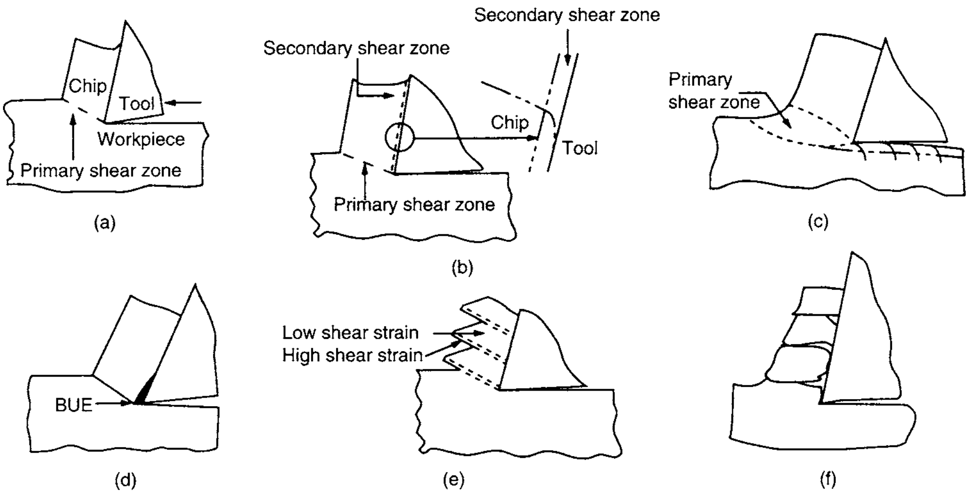


Fig. 13.4.4 Basic types of chips produced in metal cutting: (*a*) continuous chip with narrow, straight primary shear zone; (*b*) secondary shear zone at the tool-chip interface; (*c*) continuous chip with large primary shear zone; (*d*) continuous chip with built-up edge; (*e*) segmented or nonhomogeneous chip, (*f*) discontinuous chip. *(Source: After M. C. Shaw.)*

Discontinuous chips consist of segments which are produced by fracture of the metal ahead of the tool. The segments may be either loosely connected to each other or unconnected. Such chips are most often found in the machining of brittle materials or in cutting ductile materials at very low speeds or low or negative rake angles.

Inhomogeneous (serrated) chips consist of regions of large and small strain. Such chips are characteristic of metals with low thermal conductivity or metals whose yield strength decreases sharply with temperature. Chips from titanium alloys frequently are of this type.

Built-up edge chips consist of a mass of metal which adheres to the tool tip while the chip itself flows continuously along the rake face. This type of chip is often encountered in machining operations at low speeds and is associated with high adhesion between chip and tool and causes poor surface finish.

The **forces** acting on the cutting tool are shown in Fig. 13.4.3. The resultant force R has two components, F_c and F_t. The cutting force F_c in the direction of tool travel determines the amount of work done in cutting. The thrust force F_t does no work but, together with F_c, produces deflections of the tool. The resultant force also has two components on the shear plane: F_s is the force required to shear the metal along the shear plane, and F_n is the normal force on this plane. Two other force components also exist on the face of the tool: the friction force F and the normal force N.

Whereas the cutting force F_c is always in the direction shown in Fig. 13.4.3, the thrust force F_t may be in the opposite direction to that shown in the figure. This occurs when both the rake angle and the depth of cut are large, and friction is low.

From the geometry of Fig. 13.4.3, the following relationships can be derived: The **coefficient of friction** at the tool-chip interface is given by $\mu = (F_t + F_c \tan \alpha)/(F_c - F_t \tan \alpha)$. The **friction force** along the tool is $F = F_t \cos \alpha + F_c \sin \alpha$. The **shear stress** in the shear plane is $\tau = (F_c \sin \phi \cos \phi - F_t \sin^2 \phi)/A_0$, where A_0 is the cross-sectional area that is being cut from the workpiece.

The coefficient of friction on the tool face is a complex but important factor in cutting performance; it can be reduced by such means as the use of an effective cutting fluid, higher cutting speed, improved tool material and condition, or chemical additives in the workpiece material.

The net **power** consumed at the tool is $P = F_c V$. Since F_c is a function of tool geometry, workpiece material, and process variables, it is difficult reliably to calculate its value in a particular machining operation. Depending on workpiece material and the condition of the tool, **unit power** requirements in machining range between 0.2 hp·min/in^3 (0.55 W·s/mm^3) of metal removal for aluminum and magnesium alloys, to 3.5 for high-strength alloys. The power consumed is the product of unit power and rate of metal removal: P = (unit power)(vol/min).

The power consumed in cutting is transformed mostly to **heat.** Most of the heat is carried away by the chip, and the remainder is divided between the tool and the workpiece. An increase in cutting speed or feed will increase the proportion of the heat transferred to the chip. It has been observed that, in turning, the average interface **temperature** between the tool and the chip increases with cutting speed and feed, while the influence of the depth of cut on temperature has been found to be limited. Interface temperatures to the range of 1,500 to 2,000°F (800 to 1,100°C) have been measured in metal cutting. Generally the use of a cutting fluid removes heat and thus avoids temperature buildup on the cutting edge.

In cutting metal at high speeds, the chips may become very hot and cause safety hazards because of long spirals which whirl around and become entangled with the tooling. In such cases, **chip breakers** are introduced on the tool geometry, which curl the chips and cause them to break into short sections. Chip breakers can be produced on the face of the cutting tool or insert, or are separate pieces clamped on top of the tool or insert.

A phenomenon of great significance in metal cutting is **tool wear.** Many factors determine the type and rate at which wear occurs on the tool. The major critical variables that affect wear are tool temperature, type and hardness of tool material, grade and condition of workpiece, abrasiveness of the microconstituents in the workpiece material, tool geometry, feed, speed, and cutting fluid. The type of wear pattern that develops depends on the relative role of these variables.

Tool wear can be classified as (1) flank wear (Fig. 13.4.5); (2) crater wear on the tool face; (3) localized wear, such as the rounding of the cutting edge; (4) chipping or thermal softening and plastic flow of the cutting edge; (5) concentrated wear resulting in a deep groove at the edge of a turning tool, known as **wear notch.**

In general, the wear on the flank or relief side of the tool is the most dependable guide for **tool life.** A wear land of 0.060 in (1.5 mm) on high-speed steel tools and 0.015 in (0.4 mm) for carbide tools is usually used as the endpoint. The cutting speed is the variable which has the greatest influence on tool life. The relationship between tool life and cutting speed is given by the Taylor equation $VT^n = C$, where V is the cutting speed; T is the actual cutting time to develop a certain wear land, min; C is a constant whose value depends on workpiece material and process variables, numerically equal to the cutting speed that gives a tool life of 1 min; and n is the exponent whose value depends on workpiece material and other process variables.

Fig. 13.4.5 Types of tool wear in cutting.

The recommended cutting speed for a high-speed steel tool is generally the one which produces a 60- to 120-min tool life. With carbide tools, a 30- to 60-min tool life may be satisfactory. Values of n typically range from 0.08 to 0.2 for high-speed steels, 0.1 to 0.15 for cast alloys, 0.2 to 0.5 for uncoated carbides, 0.4 to 0.6 for coated carbides, and 0.5 to 0.7 for ceramics.

When tool-life equations are used, caution should be exercised in extrapolation of the curves beyond the operating region for which they are derived. In a log-log plot, tool life curves may be linear over a short cutting-speed range but are rarely linear over a wide range of cutting speeds. In spite of the considerable data obtained to date, no simple formulas can be given for quantitative relationships between tool life and various process variables for a wide range of materials and conditions.

An important aspect of machining on computer-controlled equipment is **tool-condition monitoring** while the machine is in operation with little or no supervision by an operator. Most state-of-the-art machine controls are now equipped with tool-condition monitoring systems. Two common techniques involve the use of (1) transducers that are installed on the tool holder and continually monitor torque and forces and (2) acoustic emission through a piezoelectric transducer. In both methods the signals are analyzed and interpreted automatically for tool wear or chipping, and corrective actions are taken before any significant damage is done to the workpiece.

A term commonly used in machining and comprising most of the items discussed above is **machinability.** This is best defined in terms of (1) tool life, (2) power requirement, and (3) surface integrity. Thus, a good machinability rating would indicate a combination of long tool life, low power requirement, and a good surface. However, it is difficult to develop quantitative relationships between these variables. Tool life is considered as the important factor and, in production, is usually expressed as the number of pieces machined between tool changes. Various tables are available in the literature that show the machinability rating for different materials; however, these ratings are relative. To determine the proper machining conditions for a given material, refer to the machining recommendations given later in this section.

The major factors influencing **surface finish** in machining are (1) the profile of the cutting tool in contact with the workpiece, (2) fragments of built-up edge left on the workpiece during cutting, and (3) vibration and chatter. Improvement in surface finish may be obtained to various degrees by increasing the cutting speed and decreasing the feed and depth of cut. Changes in cutting fluid, tool geometry, and tool material are also important; the microstructure and chemical composition of the material have great influence on surface finish.

As a result of mechanical working and thermal effects, **residual stresses** are generally developed on the surfaces of metals that have been machined or ground. These stresses may cause warping of the workpiece as well as affect the resistance to fatigue and stress corrosion. To minimize residual stresses, sharp tools, medium feeds, and medium depths of cut are recommended.

Because of plastic deformation, thermal effects, and chemical reactions during machining processes, alterations of machined surfaces may take place which can seriously affect the **surface integrity** of a part. Typical detrimental effects may be lowering of the fatigue strength of the part, distortion, changes in stress-corrosion properties, burns, cracks, and residual stresses. Improvements in surface integrity may be obtained by post-processing techniques such as polishing, sanding, peening, finish machining, and fine grinding.

Vibration in machine tools, a very complex behavior, is often the cause of premature tool failure or short tool life, poor surface finish, damage to the workpiece, and even damage to the machine itself. Vibration may be **forced** or **self-excited.** The term **chatter** is commonly used to designate self-excited vibrations in machine tools. The excited amplitudes are usually very high and may cause damage to the machine. Although there is no complete solution to all types of vibration problems, certain measures may be taken. If the vibration is being forced, it may be possible to remove or isolate the forcing element from the machine. In cases where the forcing frequency is near a natural frequency, either the forcing frequency or the natural frequency may be raised or lowered. Damping will also greatly reduce the amplitude. Self-excited vibrations are generally controlled by increasing the stiffness and damping of the machine tool. (See also Secs. 3 and 5.)

Good machining practice requires a rigid setup. The machine tool must be capable of providing the **stiffness** required for the machining conditions used. If a rigid setup is not available, the depth of cut must be reduced. Excessive tool overhang should be avoided, and in milling, cutters should be mounted as close to the spindle as possible. The length of end mills and drills should be kept to a minimum. Tools with large nose radius or with a long, straight cutting edge increase the possibility of chatter.

CUTTING-TOOL MATERIALS

A wide variety of **cutting-tool materials** are available. The selection of a proper material depends on such factors as the cutting operation involved, the machine to be used, the workpiece material, production requirements, cost, and surface finish and accuracy desired. The major qualities required in a cutting tool are (1) hot hardness, (2) resistance to mechanical impact and thermal shock, (3) wear resistance, and (4) chemical stability and inertness to the workpiece material being machined. (See Table 13.4.1 and Figs. 13.4.6 and 13.4.7.)

Materials for cutting tools include high-speed steels, cast alloys, carbides, ceramics or oxides, cubic boron nitride, and diamond. Understanding the different types of **tool steels** (see Sec. 6.2) requires knowledge of the role of different alloying elements. These elements are added to (1) obtain greater hardness and wear resistance, (2) obtain greater impact toughness, (3) impart hot hardness to the steel such that its hardness is maintained at high cutting temperatures, and (4) decrease distortion and warpage during heat treating.

Table 13.4.1 Characteristics of Cutting-Tool Materials

	High-speed steels	Cast cobalt alloys	Carbides	Coated carbides	Ceramics	Polycrystalline cubic boron nitride	Diamond
Hot hardness		increasing →					
Toughness		← increasing					
Impact strength		← increasing					
Wear resistance		increasing →					
Chipping resistance		← increasing					
Cutting speed		increasing →					
Thermal shock resistance		← increasing					
Tool material cost		increasing →					

NOTE: These tool materials have a wide range of compositions and properties; thus overlapping characteristics exist in many categories of tool materials.
SOURCE: After R Komanduri.

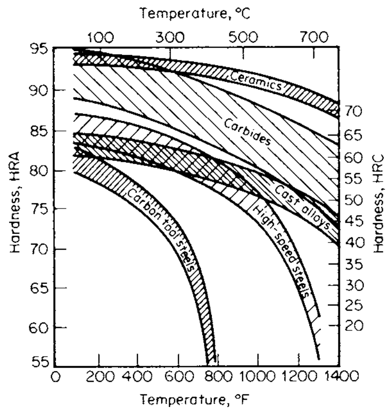


Fig. 13.4.6 Hardness of tool materials as a function of temperature.

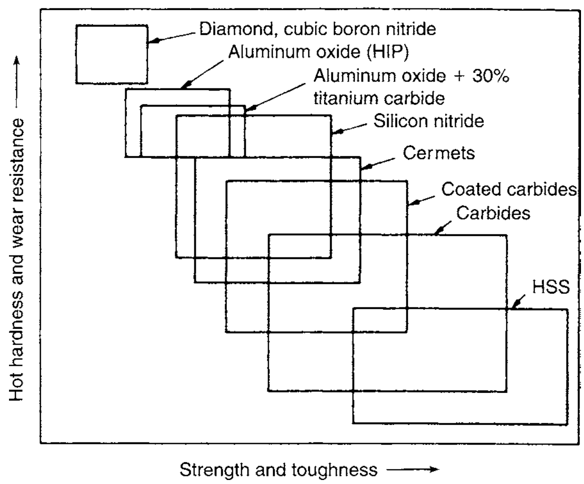


Fig. 13.4.7 Ranges of properties of various groups of tool materials.

Carbon forms a carbide with iron, making it respond to hardening and thus increasing the hardness, strength, and wear resistance. The carbon content of tool steels ranges from 0.6 to 1.4 percent. **Chromium** is added to increase wear resistance and toughness; the content ranges from 0.25 to 4.5 percent. **Cobalt** is commonly used in high-speed steels to increase hot hardness so that tools may be used at higher cutting speeds and still maintain hardness and sharp cutting edges; the content ranges from 5 to 12 percent. **Molybdenum** is a strong carbide-forming element and increases strength, wear resistance, and hot hardness. It is always used in conjunction with other alloying elements, and its content ranges to 10 percent. **Tungsten** promotes hot hardness and strength; content ranges from 1.25 to 20 percent. **Vanadium** increases hot hardness and abrasion resistance; in high-speed steels, it ranges from 1 to 5 percent.

High-speed steels are the most highly alloyed group among tool steels and maintain their hardness, strength, and cutting edge. With suitable procedures and equipment, they can be fully hardened with little danger of distortion or cracking. High-speed steel tools are widely used in operations using form tools, drilling, reaming, end-milling, broaching, tapping, and tooling for screw machines.

Cast alloys maintain high hardness at high temperatures and have good wear resistance. Cast-alloy tools, which are cast and ground into any desired shape, are composed of cobalt (38 to 53 percent), chromium (30 to 33 percent), and tungsten (10 to 20 percent). These alloys are recommended for deep roughing operations at relatively high speeds and feeds. Cutting fluids are not necessary and are usually used only to obtain a special surface finish.

Carbides have metal carbides as key ingredients and are manufactured by powder-metallurgy techniques. They have the following properties which make them very effective cutting-tool materials: (1) high hardness over a wide range of temperatures; (2) high elastic modulus, 2 to 3 times that of steel; (3) no plastic flow even at very high stresses; (4) low thermal expansion; and (5) high thermal conductivity. Carbides are used in the form of inserts or tips which are clamped or brazed to a steel shank. Because of the difference in coefficients of expansion, brazing should be done carefully. The mechanically fastened tool tips are called **inserts** (Fig. 13.4.8); they are available in different shapes, such as square, triangular, circular, and various special shapes.

There are three general groups of carbides in use: (1) tungsten carbide with cobalt as a binder, used in machining cast irons and nonferrous abrasive metals; (2) tungsten carbide with cobalt as a binder, plus a solid solution of WC-TiC-TaC-NbC, for use in machining steels;

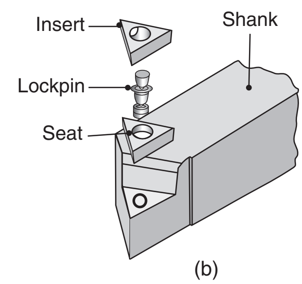


Fig. 13.4.8 (*a*) Insert clamped to shank of a toolholder; (*b*) insert clamped with wing lockpins.

and (3) titanium carbide with nickel and molybdenum as a binder, for use where cutting temperatures are high because of high cutting speeds or the high strength of the workpiece material. Carbides are classified by ISO and ANSI, as shown in Table 13.4.2 which includes recommendations for a variety of workpiece materials and cutting conditions. (See also Sec. 6.4.)

Coated carbides consist of conventional carbide inserts that are coated with a thin layer of titanium nitride, titanium carbide, titanium carbonitride, ceramic, polycrystalline diamond, or diamondlike carbon. The coating provides additional wear resistance while maintaining the strength and toughness of the carbide tool. Coatings are also applied to high-speed steel tools, particularly drills and taps. The desirable properties of individual coatings can be combined and optimized by using **multiphase coatings.** Carbide tools are now available with, e.g., a layer of titanium carbide over the carbide substrate, followed by aluminum oxide and then titanium nitride. Various alternating layers of coatings are also used, each layer being on the order of 80 to 400 μin (2 to 10 μm) thick.

Stiffness is of great importance when using carbide tools. Light feeds, low speeds, and chatter are deleterious. No cutting fluid is needed, but if one is used for cooling, it should be applied in large quantities and continuously to prevent heating and quenching.

Ceramic, or **oxide,** inserts consist primarily of fine aluminum oxide grains which have been bonded together. Minor additions of other elements help to obtain optimum properties.

Other ceramics include silicon nitride, with various additives such as aluminum oxide, yttrium oxide, and titanium carbide. Silicon-nitride-based ceramics include **sialon** (from silicon, aluminum, oxygen, and nitrogen) which has toughness, hot hardness, and good thermal-shock resistance. More recent developments include **whisker-reinforced** cutting tools, with enhanced toughness, cutting-edge strength, and thermal-shock resistance. A common whisker material is silicon carbide. Ceramic tools have very high abrasion resistance, are harder than carbides, and have less tendency to weld to metals during cutting. However, they generally lack impact toughness, and premature tool failure can result by chipping or general breakage. Ceramic tools have been found to be

Table 13.4.2 Classification of Tungsten Carbides According to Machining Applications

ISO standard	ANSI classification no. (grade)	Materials to be machined	Machining operation	Type of carbide	Characteristics of Cut	Characteristics of Carbide
K30–K40	C1	Cast iron, nonferrous metals, and nonmetallic materials requiring abrasion resistance	Roughing	Wear-resistant grades; generally straight WC-Co with varying grain sizes	Increasing cutting speed	Increasing hardness and wear resistance
K20	C2		General purpose		↓ ↑	↓ ↑
K10	C3		Light finishing			Increasing strength and binder content
K01	C4		Precision finishing		Increasing feed rate	
P30–P50	C5	Steels and steel alloys requiring crater and deformation resistance	Roughing	Crater-resistant grades; various WC-Co compositions with TiC and/or TaC alloys	Increasing cutting speed	Increasing hardness and wear resistance
P20	C6		General purpose		↓ ↑	↓ ↑
P10	C7		Light finishing			Increasing strength and binder content
P01	C8		Precision finishing		Increasing feed rate	

NOTE: The ISO and ANSI comparisons are approximate.

effective for high-speed, uninterrupted turning operations. Tool and setup geometry is important. Tool failures can be reduced by the use of rigid tool mountings and rigid machine tools. Included in oxide cutting-tool materials are **cermets** (such as 70 percent aluminum oxide and 30 percent titanium carbide), combining the advantages of ceramics and metals.

Polycrystalline diamond is used where good surface finish and dimensional accuracy are desired, particularly on soft nonferrous materials that are difficult to machine. The general properties of diamonds are extreme hardness, low thermal expansion, high heat conductivity, and a very low coefficient of friction. The polycrystalline diamond is bonded to a carbide substrate. **Single-crystal diamond** is also used as a cutting tool to produce extremely fine surface finish on nonferrous alloys, such as copper-base mirrors.

Next to diamond, **cubic boron nitride (cBN)** is the hardest material presently available. Polycrystalline cBN is bonded to a carbide substrate and used as a cutting tool. The cBN layer provides very high wear resistance and edge strength. It is chemically inert to iron and nickel at elevated temperatures; thus it is particularly suitable for machining high-temperature alloys and various ferrous alloys. Both diamond and cBN are also used as abrasives in grinding operations.

CUTTING FLUIDS

Cutting fluids, frequently referred to as lubricants or coolants, comprise those liquids and gases which are applied to the cutting zone in order to facilitate the cutting operation. A cutting fluid is used (1) to keep the tool cool and prevent it from being heated to a temperature at which the hardness and resistance to abrasion are reduced; (2) to keep the workpiece cool, thus preventing it from being machined in a warped shape to inaccurate final dimensions; (3) through lubrication to reduce friction and power consumption, wear on the tool, and generation of heat; (4) to provide a good finish on the workpiece; (5) to aid in providing a satisfactory chip formation; (6) to wash away the chips (this is particularly desirable in deep-hole drilling, hacksawing, milling, and grinding); and (7) to prevent corrosion of the workpiece and machine tool.

Classification Cutting fluids may be classified as follows: (1) emulsions, (2) oils, and (3) solutions (semisynthetics and synthetics). Cutting fluids are also classified as light-, medium-, and heavy-duty; light-duty fluids are for general-purpose machining. Induced **air blast** may be used with internal and surface grinding and polishing operations. Its main purpose is to remove the small chips or dust, although some cooling is also obtained, especially in machining of plastics.

Emulsions consist of a soluble oil emulsified with water in the ratio of 1 part oil to 10 to 100 parts water, depending upon the type of product and the operation. Emulsions have surface-active or **extreme-pressure** additives to reduce friction and provide an effective lubricant film under high pressure at the tool-chip interface during machining. Emulsions are low-cost cutting fluids and are used for practically all types of cutting and grinding when machining all types of metals. The more concentrated mixtures of oil and water, such as 1 : 10, are used for broaching, threading, and gear cutting. For most operations, a solution of 1 part soluble oil to 20 parts water is satisfactory.

A variety of **oils** are used in machining. They are used where lubrication rather than cooling is essential or on high-grade finishing cuts, although sometimes superior finishes are obtained with emulsions.

Oils generally used in machining are mineral oils with the following compositions: (1) straight mineral oil, (2) with fat, (3) with fat and sulfur, (4) with fat and chlorine, and (5) with fat, sulfur, and chlorine. The more severe the machining operation, the higher the composition of the oil. Broaching and tapping of refractory alloys and high-temperature alloys, for instance, require highly compounded oils. In order to avoid staining of the metal, aluminum and copper, for example, inhibited sulfur and chlorine are used.

Solutions are a family of cutting fluids that blend water and various chemical agents such as amines, nitrites, nitrates, phosphates, chlorine, and sulfur compounds. These agents are added for purposes of rust prevention, water softening, lubrication, and reduction of surface tension. Most of these chemical fluids are coolants but some are lubricants.

The **selection** of a cutting fluid for a particular operation requires consideration of several factors: cost, the workpiece material, the difficulty of the machining operation, the compatibility of the fluid with the workpiece material and the machine tool components, surface preparation, method of application and removal of the fluid, contamination of the cutting fluid with machine lubricants, and the treatment of the fluid after use. Also important are the **biological** and **ecological** aspects of the cutting fluid used. There may be potential health hazards to operating personnel from contact with or inhalation of mist or fumes from some fluids. Recycling and waste disposal are also important problems to be considered.

Methods of Application The most common method is **flood cooling** in quantities such as 3 to 5 gal/min (about 10 to 20 L/min) for single-point tools and up to 60 gal/min (230 L/min) per cutter for multiple-tooth cutters. Whenever possible, multiple nozzles should be used. In **mist cooling** a small jet equipment is used to disperse water-base fluids as very fine droplets in a carrier that is generally air at pressures 10 to

80 lb/in^2 (70 to 550 kPa). Mist cooling has a number of advantages, such as providing high-velocity fluids to the working areas, better visibility, and improving tool life in certain instances. The disadvantages are that venting is required and also the cooling capability is rather limited.

High-pressure refrigerated coolant systems are very effective in removing heat at high rates, particularly in computer-controlled machine tools. The fluid is directed generally at the relief angle of the cutting tools and at pressures as high as 5,000 lb/in^2 (35,000 kPa). Continuous filtering of the fluid is essential to eliminate any damage to workpiece surfaces due to the impact of any contaminants that may be present in the coolant system. More recent methods of application include delivering the coolant to the cutting zone through the tool and the machine spindle.

For economic as well as environmental reasons, an important trend is **near-dry** and **dry machining.** In near-dry machining, the cutting fluid typically consists of a fine mist of air containing a very small amount of cutting fluid (including vegetable oil) and is delivered through the machine spindle. Dry machining is carried out without any fluids but using appropriate cutting tools and processing parameters. Unlike other methods, however, dry machining cannot flush away the chips being produced; an effective means to do so is to use pressurized air.

MACHINE TOOLS

The general types of **machine tools** are lathes; turret lathes; screw, boring, drilling, reaming, threading, milling, and gear-cutting machines; planers and shapers; broaching, cutting-off, grinding, and polishing machines. Each of these is subdivided into many types and sizes. General items common to all machine tools are discussed first, and individual machining processes and equipment are treated later in this section.

Automation is the application of special equipment to control and perform manufacturing processes with little or no manual effort. It is applied to the manufacturing of all types of goods and processes, from the raw material to the finished product. Automation involves many activities, such as handling, processing, assembly, inspecting, and packaging. Its primary objective is to lower manufacturing cost through controlled production and quality, lower labor cost, reduced damage to work by handling, higher degree of safety for personnel, and economy of floor space. Automation may be partial, such as gaging in cylindrical grinding, or it may be total.

The conditions which play a role in decisions concerning automation are rising production costs, high percentage of rejects, lagging output, scarcity of skilled labor, hazardous working conditions, and work requiring repetitive operation. Factors which must be carefully studied before deciding on automation are high initial cost of equipment, maintenance problems, and type of product. (See also Sec. 16.)

Mass production with modern machine tools has been achieved through the development of self-contained **power-head** production units and the development of **transfer** mechanisms. Power-head units, consisting of a frame, electric driving motor, gearbox, tool spindles, etc., are available for many types of machining operations. Transfer mechanisms move the workpieces from station to station by various methods. Transfer-type machines can be arranged in several configurations, such as a straight line or a U pattern. Various types of machine tools for mass production can be built from components; this is known as the **building-block** principle. Such a system combines flexibility and adaptability with high productivity. (See **machining centers.**)

Numerical control (NC), which is a method of controlling the motions of machine components by numbers, was first applied to machine tools in the 1950s. Numerically controlled machine tools are classified according to the type of cutting operation. For instance, in drilling and boring machines, the positioning and the cutting take place sequentially (point to point), whereas in die-sinking machines, positioning and cutting take place simultaneously. The latter are often described as **continuous-path** machines, and since they require more exacting specifications, they give rise to more complex problems. Machines now perform over a very wide range of cutting conditions without requiring adjustment to eliminate chatter, and to improve accuracy. Complex contours can be machined which would be almost impossible by any other method. A large variety of programming systems has been developed.

The control system in NC machines has been converted to computer control with various software. In **computer numerical control** (CNC), a microcomputer is a part of the control panel of the machine tool. The advantages of computer numerical control are ease of operation, simpler programming, greater accuracy, versatility, and lower maintenance costs.

Further developments in machine tools are **machining centers.** This is a machine equipped with as many as 200 tools and with an automatic tool changer (Fig. 13.4.9). It is designed to perform various operations on different surfaces of the workpiece, which is placed on a pallet capable of as much as five-axis movement (three linear and two rotational). Machining centers, which may be vertical or horizontal spindle, have flexibility and versatility that other machine tools do not have, and thus they have become the first choice in machine selection in modern manufacturing plants and shops. They have the capability of tool and part checking, tool-condition monitoring, in-process and postprocess gaging, and inspection of machined surfaces. **Universal machining centers** are the latest development, and they have both vertical and horizontal spindles. **Turning centers** are a further development of computer-controlled lathes and have great flexibility. Many centers are now constructed on a **modular** basis, so that various accessories and peripheral equipment can be installed and modified depending on the type of product to be machined.

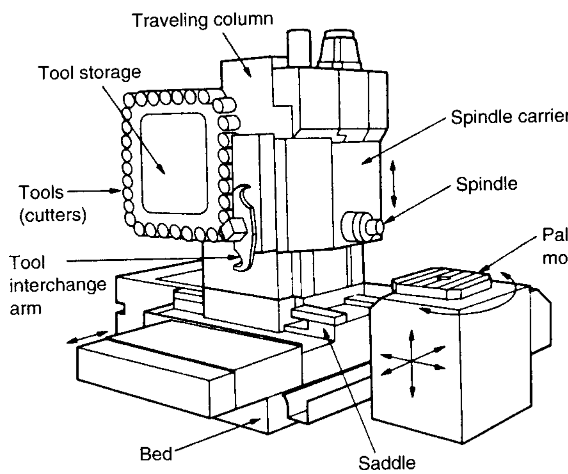


Fig. 13.4.9 Schematic of a horizontal spindle machining center, equipped with an automatic tool changer. Tool magazines can store 200 different cutting tools.

An approach to optimize machining operations is **adaptive control.** While the material is being machined, the system senses operating conditions such as forces, tool-tip temperature, rate of tool wear, and surface finish, and converts these data into feed and speed control that enables the machine to cut under optimum conditions for maximum productivity. Combined with numerical controls and computers, adaptive controls are expected to result in increased efficiency of metalworking operations.

With the advent of sophisticated computers and various software, modern manufacturing has evolved into **computer-integrated manufacturing (CIM).** This system involves the coordinated participation of computers in all phases of manufacturing. **Computer-aided design** combined with **computer-aided manufacturing (CAD/CAM)** results in a much higher productivity, better accuracy and efficiency, and reduction in design effort and prototype development. CIM also involves the management of the factory, inventory, and labor, and it integrates all these activities, eventually leading to untended factories.

The highest level of sophistication is reached with a **flexible manufacturing system (FMS).** Such a system is made of **manufacturing cells** and an automatic materials-handling system interfaced with a central computer. The manufacturing cell is a system in which CNC machines are used to make a specific part or parts with similar shape. The workstations, i.e., several machine tools, are placed around an **industrial robot** which automatically loads, unloads, and transfers the parts. FMS has the capability to optimize each step of the total manufacturing operation, resulting in the highest possible level of efficiency and productivity.

The proper design of **machine-tool structures** requires analysis of such factors as form and materials of structures, stresses, weight, and manufacturing and performance considerations. The best approach to obtain the ultimate in machine-tool accuracy is to employ both improvements in structural stiffness and compensation of deflections by use of special controls. The C-frame structure has been used extensively in the past because it provides ready accessibility to the working area of the machine. With the advent of computer control, the box-type frame with its considerably improved static stiffness becomes practical since the need for manual access to the working area is greatly reduced. The use of a box-type structure with thin walls can provide low weight for a given stiffness. The light-weight-design principle offers high dynamic stiffness by providing a high natural frequency of the structure through combining high static stiffness with low weight rather than through the use of large mass. (Dynamic stiffness is the stiffness exhibited by the system when subjected to dynamic excitation where the elastic, the damping, and the inertia properties of the structure are involved; it is a frequency-dependent quantity.)

TURNING

Turning is a machining operation for all types of metallic and nonmetallic materials and is capable of producing circular parts with straight or various profiles. The cutting tools may be single-point or form tools. The most common machine tool used is a **lathe;** modern lathes are computer-controlled and can achieve high production rates with little labor. The basic operation is shown in Fig. 13.4.10, where the workpiece is held in a chuck and rotates at *N* r/min; a cutting tool moves along the length of the piece at a feed *f* (in/r or mm/r) and removes material at a radial depth *d,* reducing the diameter from D_0 to D_f.

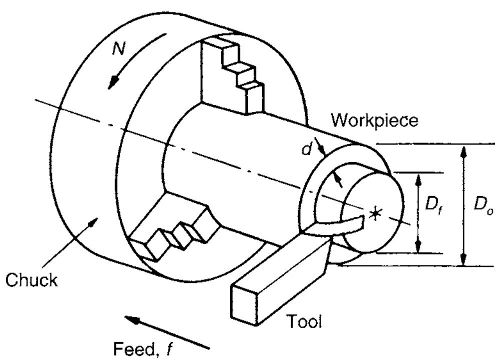


Fig. 13.4.10 A turning operation on a round workpiece held in a three-jaw chuck.

Lathes generally are considered to be the oldest member of machine tools, having been first developed in the late eighteenth century. The most common lathe is called an engine lathe because it was one of the first machines driven by Watt's steam engine. The basic lathe has the following main parts: bed, headstock, tailstock, and carriage. The types of lathes available for a variety of applications may be listed as follows: engine lathes, bench lathes, horizontal turret lathes, vertical lathes, and automatics. A great variety of lathes and attachments are available within each category, also depending on the production rate required.

It is common practice to specify the size of an engine lathe by giving the **swing** (diameter) and the **distance between centers** when the tailstock is flush with the end of the bed. The maximum swing over the ways is usually greater than the nominal swing. The **length of the bed** is given frequently to specify the overall length of the bed. A lathe size is indicated thus: 14 in (356 mm) (swing) by 30 in (762 mm) (between centers) by 6 ft (1,830 mm) (length of bed). Lathes are made for light-, medium-, or heavy-duty work.

All geared-head lathes, which are single-pulley (belt-driven or arranged for direct-motor drive through short, flat, or V belts, gears, or silent chain), increase the power of the drive and provide a means for obtaining 8, 12, 16, or 24 spindle speeds. The teeth may be of the spur, helical, or herringbone type and may be ground or lapped after hardening.

Variable speeds are obtained by driving with adjustable-speed dc shunt-wound motors with stepped field-resistance control or by electronics or motor-generator system to give speed variation in infinite steps. AC motors driving through infinitely variable speed transmissions of the mechanical or hydraulic type are also in general use.

Modern lathes, most of which are now computer-controlled **(turning centers),** are built with the speed capacity, stiffness, and strength capable of taking full advantage of new and stronger tool materials. The main drive-motor capacity of lathes ranges from fractional to more than 200 hp (150 kW). Speed preselectors, which give speed as a function of work diameter, are introduced, and variable-speed drives using dc motors with panel control are standard on many lathes. Lathes with contour facing, turning, and boring attachments are also available.

Tool Shapes for Turning

The standard **nomenclature** for single-point tools, such as those used on lathes, planers, and shapers, is shown in Fig. 13.4.11. Each tool consists of a shank and point. The point of a single-point tool may be formed by grinding on the end of the shank; it may be forged on the end of the shank and subsequently ground; a tip or insert may be clamped or brazed to the end of the shank (see Fig. 13.4.8). The **best tool shape** for each material and each operation depends on many factors. For specific information and recommendations, the various sources listed in the References should be consulted. See also Table 13.4.3.

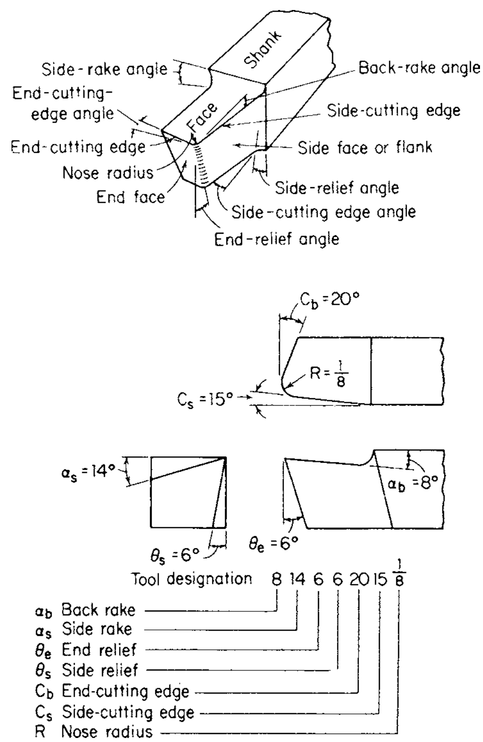


Fig. 13.4.11 Standard nomenclature for single-point cutting tools.

Table 13.4.3 Recommend Tool Geometry for Turning, deg

Material	High-speed steel and cast-alloy tools					Carbide tools (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 alloys	20	15	12	10	5	0	5	5	5	15
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, ferritic	5	8	5	5	15	0	5	5	5	15
Stainless steels, austenitic	0	10	5	5	15	0	5	5	5	15
Stainless steels, martensitic	0	10	5	5	15	−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
Thermosetting plastics	0	0	20−30	15−20	10	0	15	5	5	15

SOURCE: "Matchining Data Handbook," published by the Machinability Data Center, Metcut Research Associates, Inc.

Positive **rake angles** improve the cutting operation with regard to forces and deflection; however, a high positive rake angle may result in early failure of the cutting edge. Positive rake angles are generally used in lower-strength materials. For higher-strength materials, negative rake angles may be used. **Back rake** usually controls the direction of chip flow and is of less importance than the **side rake.** The purpose of **relief angles** is to avoid interference and rubbing between the workpiece and tool flank surfaces. In general, they should be small for high-strength materials and larger for softer materials. Excessive relief angles may weaken the tool. The **side cutting-edge angle** influences the length of chip contact and the true feed. This angle is often limited by the workpiece geometry, e.g., the shoulder contour. Large angles are apt to cause tool chatter. Small **end cutting-edge angles** may create excessive force normal to the workpiece, and large angles may weaken the tool point. The purpose of the **nose radius** is to give a smooth surface finish and to obtain longer tool life by increasing the strength of the cutting edge. The nose radius should be tangent to the cutting-edge angles. A large nose radius gives a stronger tool and may be used for roughing cuts; however, large radii may lead to tool chatter. A small nose radius reduces forces and is therefore preferred on thin or slender workpieces.

Turning Recommendations Recommendations for tool materials, depth of cut, feed, and cutting speed for turning a variety of materials are given in Table 13.4.4. The cutting speeds for high-speed steels for turning, which are generally M2 and M3, are about one-half those for uncoated carbides. A general **troubleshooting guide** for turning operations is given in Table 13.4.5. The range of applicable cutting speeds and feeds for a variety of tool materials is shown in Fig. 13.4.12.

High-Speed Machining To increase productivity and reduce machining costs, there is a continuing trend to increase cutting speeds, especially in turning, milling, boring, and drilling. High-speed machining is a general term used to describe this trend, where speeds typically range as follows: High speed: up 6,000 ft/min (1,800 m/min): very high speed: up to 60,000 ft/min (18,000 m/min); and ultrahigh-speed, higher than this range. Because of the high speeds involved, important considerations in these operations include inertia effects, spindle design, bearings, and power; stiffness and accuracy of the machine tools; selection of appropriate cutting tools; and chip removal systems.

Hard Turning and Machining As workpiece hardness increases, its machinability decreases and there may be difficulties with traditional machining operations regarding surface finish, surface integrity, and tool life. With advances in cutting tools and the availability of rigid and powerful machine tools and work-holding devices, however, it is now possible to machine hard materials, including heat-treated steels, with high dimensional accuracy. Hard machining can compete well with grinding processes and has been shown to be economical for parts such as shafts, gears, pinions, and various automotive components.

Ultraprecision Machining To respond to increasing demands for special parts with surface finish and dimensional accuracies on the order of a nanometre (10^{-9} m), several important developments have been taking place in advanced machining. A common example of ultraprecision machining is **diamond turning,** typically using a single-crystal diamond cutting tool and rigid machine tools. Applications for such parts and components are in the computer, electronic, nuclear, and defense industries.

Turret Lathes

Turret lathes are used for the production of parts in moderate quantities and produce interchangeable parts at low production cost. Turret lathes may be chucking, screw machine, or universal. The universal machine may be set up to machine bar stock as a screw machine or have the work held in a chuck. These machines may be semiautomatic, i.e., so arranged that after a piece is chucked and the machine started, it will complete the machining cycle automatically and come to a stop. They may be horizontal or vertical and single- or multiple-spindle; many of these lathes are now computer-controlled and have a variety of features.

The basic principle of the turret lathe is that, with standard tools, setups can be made quickly so that combined, multiple, and successive cuts can be made on a part. By **combined** cuts, tools on the cross slide operate simultaneously with those on the turret, e.g., facing from the cross slide and boring from the turret. **Multiple** cuts permit two or more tools to operate from either or both the cross slide or turret. By **successive** cuts, one tool may follow another to rough or finish a surface; e.g., a hole may be drilled, bored, and reamed at one chucking. In the tool-slide machine only roughing cuts, such as turn and face, can be made in one machine.

Ram-type turret lathes have the turret mounted on a ram which slides in a separate base. The base is clamped at a position along the bed to suit a long or short workpiece. A cross slide can be used so that combined cuts can be taken from the turret and the cross slide at the same time. Turret and cross slide can be equipped with manual or power feed. The short stroke of the turret slide limits this machine to comparatively short light work, in both small and quantity-lot production.

Saddle-type turret lathes have the turret mounted on a saddle which slides directly on the bed. Hence, the length of stroke is limited only by the length of bed. A separate square-turret carriage with longitudinal and transverse movement can be mounted between the head and the hex-turret saddle so that combined cuts from both stations at one time are possible. The saddle type of turret lathe generally has a large hollow vertically faced turret for accurate alignment of the tools.

Screw Machines

When turret lathes are set up for bar stock, they are often called **screw machines.** Turret lathes that are adaptable only to bar-stock work are

Table 13.4.4 General Recommendations for Turning Operations

		General-purpose starting conditions			Range for roughing and finishing		
Workpiece material	Cutting tool	Depth of cut, mm (in)	Feed, mm/r (in/r)	Cutting speed, m/min (ft/min)	Depth of cut, mm (in)	Feed, mm/r (in/r)	Cutting speed, m/min (ft/min)
Low-C and free-machining steels	Uncoated carbide	1.5–6.3 (0.06–0.25)	0.35 (0.014)	90 (300)	0.5–7.6 (0.02–0.30)	0.15–1.1 (0.006–0.045)	60–135 (200–450)
	Ceramic-coated carbide	1.5–6.3 (0.06–0.25)	0.35 (0.014)	245–275 (800–900)	0.5–7.6 (0.02–0.30)	0.15–1.1 (0.006–0.045)	120–425 (400–1,400)
	Triple-coated carbide	1.5–6.3 (0.06–0.25)	0.35 (0.014)	185–200 (600–650)	0.5–7.6 (0.02–0.30)	0.15–1.1 (006–045)	90–245 (300–800)
	TiN-coated carbide	1.5–6.3 (0.06–0.25)	0.35 (0.014)	105–150 (350–500)	0.5–7.6 (0.02–0.30)	0.15–1.1 (0.006–0.045)	60–230 (200–750)
	$A1_2O_3$ ceramic	1.5–6.3 (0.06–0.25)	0.25 (0.010)	395–440 (1,300–1,450)	0.5–7.6 (0.02–0.30)	0.15–1.1 (0.006–0.045)	365–550 (1,200–1,800)
	Cermet	1.5–6.3 (0.06–0.25)	0.30 (0.012)	215–290 (700–950)	0.5–7.6 (0.02–0.30)	0.15–1.1 (0.006–0.045)	105–455 (350–1,500)
Medium- and high-C steels	Uncoated carbide	1.2–4.0 (0.05–0.20)	0.30 (0.012)	75 (250)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	45–120 (150–400)
	Ceramic-coated carbide	1.2–4.0 (0.05–0.20)	0.30 (0.012)	185–230 (600–750)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	120–410 (400–1,350)
	Triple-coated carbide	1.2–4.0 (0.050–0.20)	0.30 (0.012)	120–150 (400–500)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	75–215 (250–700)
	TiN-coated carbide	1.2–4.0 (0.05–0.20)	0.30 (0.012)	90–200 (300–650)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	45–215 (150–700)
	$A1_2O_3$ ceramic	1.2–4.0 (0.05–0.20)	0.25 (0.010)	335 (1,100)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	245–455 (800–1,500)
	Cermet	1.2–4.0 (0.05–0.20)	0.25 (0.010)	170–245 (550–800)	2.5–7.6 (0.10–0.30)	0.15–0.75 (0.006–0.03)	105–305 (350–1,000)
Cast iron, gray	Uncoated carbide	1.25–6.3 (0.05–0.25)	0.32 (0.013)	90 (300)	0.4–12.7 (0.015–0.5)	0.1–0.75 (0.004–0.03)	75–185 (250–600)
	Ceramic-coated carbide	1.25–6.3 (0.05–0.25)	0.32 (0.013)	200 (650)	0.4–12.7 (0.015–0.5)	0.1–0.75 (0.004–0.03)	120–365 (400–1,200)
	TiN-coated carbide	1.25–6.3 (0.05–0.25)	0.32 (0.013)	90–135 (300–450)	0.4–12.7 (0.015–0.5)	0.1–0.75 (0.004–0.03)	60–215 (200–700)
	$A1_2O_3$ ceramic	1.25–6.3 (0.05–0.25)	0.25 (0.010)	455–490 (1,500–1,600)	0.4–12.7 (0.015–0.5)	0.1–0.75 (0.004–0.03)	365–855 (1,200–2,800)
	SiN ceramic	1.25–6.3 (0.05–0.25)	0.32 (0.013)	730 (2,400)	0.4–12.7 (0.015–0.5)	0.1–0.75 (0.004–0.03)	200–990 (650–3,250)
Stainless steel, austenitic	Triple-coated carbide	1.5–4.4 (0.06–0.175)	0.35 (0.014)	150 (500)	0.5–12.7 (0.02–0.5)	0.08–0.75 (0.003–0.03)	75–230 (250–750)
	TiN-coated carbide	1.5–4.4 (0.06–0.175)	0.35 (0.014)	85–160 (275–525)	0.5–12.7 (0.02–0.5)	0.08–0.75 (0.003–0.03)	55–200 (175–650)
	Cermet	1.5–4.4 (0.06–0.175)	0.30 (0.012)	185–215 (600–700)	0.5–12.7 (0.02–0.5)	0.08–0.75 (0.003–0.03)	105–290 (350–950)
High-temperature alloys, nickel base	Uncoated carbide	2.5 (0.10)	0.15 (0.006)	25–45 (75–150)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	15–30 (50–100)
	Ceramic-coated carbide	2.5 (0.10)	0.15 (0.006)	45 (150)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	20–60 (65–200)
	TiN-coated carbide	2.5 (0.10)	0.15 (0.006)	30–55 (95–175)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	20–85 (60–275)
	Al_2O_3 ceramic	2.5 (0.10)	0.15 (0.006)	260 (850)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	185–395 (600–1,300)
	SiN ceramic	2.5 (0.10)	0.15 (0.006)	215 (700)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	90–215 (300–700)
	Polycrystalline cBN	2.5 (0.10)	0.15 (0.006)	150 (500)	0.25–6.3 (0.01–0.25)	0.1–0.3 (0.004–0.012)	120–185 (400–600)
Titanium alloys	Uncoated carbide	1.0–3.8 (0.04–0.15)	0.15 (0.006)	35–60 (120–200)	0.25–6.3 (0.01–0.25)	0.1–0.4 (0.004–0.015)	10–75 (30–250)
	TiN-coated carbide	1.0–3.8 (0.04–0.15)	0.15 (0.006)	30–60 (100–200)	0.25–6.3 (0.01–0.25)	0.1–0.4 (0.004–0.015)	(10–100) (30–325)
Aluminum alloys free-machining	Uncoated carbide	1.5–5.0 (0.06–0.20)	0.45 (0.018)	490 (1,600)	0.25–8.8 (0.01–0.35)	0.08–0.62 (0.003–0.025)	200–670 (650–2,000)
	TiN-coated carbide	1.5–5.0 (0.06–0.20)	0.45 (0.018)	550 (1,800)	0.25–8.8 (0.01–0.35)	0.08–0.62 (0.003–0.025)	60–915 (200–3,000)
	Cermet	1.5–5.0 (0.06–0.20)	0.45 (0.018)	490 (1,600)	0.25–8.8 (0.01–0.35)	0.08–0.62 (0.003–0.025)	215–795 (700–2,600)
	Polycrystalline diamond	1.5–5.0 (0.06–0.20)	0.45 (0.018)	760 (2,500)	0.25–8.8 (0.01–0.35)	0.08–0.62 (0.003–0.025)	305–3,050 (1,000–10,000)
High-silicon copper alloys	Polycrystalline diamond	1.5–5.0 (0.06–0.20)	0.45 (0.018)	530 (1,700)	0.25–8.8 (0.01–0.35)	0.08–0.62 (0.003–0.025)	365–915 (1,200–3,000)
	Uncoated carbide	1.5–5.0 (0.06–0.20)	0.25 (0.010)	260 (850)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	105–535 (350–1,750)

Table 13.4.4 General Recommendations for Turning Operations (*Continued*)

Workpiece material	Cutting tool	General-purpose starting conditions: Depth of cut, mm (in)	Feed, mm/r (in/r)	Cutting speed, m/min (ft/min)	Range for roughing and finishing: Depth of cut, mm (in)	Feed, mm/r (in/r)	Cutting speed, m/min (ft/min)
High-silicon copper alloys (*cont.*)	Ceramic-coated carbide	1.5–5.0 (0.06–0.20)	0.25 (0.010)	365 (1,200)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	215–670 (700–2,200)
	Triple-coated carbide	1.5–5.0 (0.06–0.20)	0.25 (0.010)	215 (700)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	90–305 (300–1,000)
	TiN-coated carbide	1.5–5.0 (0.06–0.20)	0.25 (0.010)	90–275 (300–900)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	45–455 (150–1,500)
	Cermet	1.5–5.0 (0.06–0.20)	0.25 (0.010)	245–425 (800–1,400)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	200–610 (650–2,000)
	Polycrystalline diamond	1.5–5.0 (0.06–0.20)	0.25 (0.010)	520 (1,700)	0.4–7.5 (0.015–0.3)	0.15–0.75 (0.006–0.03)	275–915 (900–3,000)
Tungsten alloys	Uncoated carbide	2.5 (0.10)	0.2 (0.008)	75 (250)	0.25–5.0 (0.01–0.2)	0.12–0.45 (0.005–0.018)	55–120 (175–400)
	TiN-coated carbide	2.5 (0.10)	0.2 (0.008)	85 (275)	0.25–5.0 (0.01–0.2)	0.12–0.45 (0.005–0.018)	60–150 (200–500)
Thermoplastics and thermosets	TiN-coated carbide	1.2 (0.05)	0.12 (0.005)	170 (550)	0.12–5.0 (0.005–0.20)	0.08–0.35 (0.003–0.015)	90–230 (300–750)
	Polycrystalline diamond	1.2 (0.05)	0.12 (0.005)	395 (1,300)	0.12–5.0 (0.005–0.20)	0.08–1.35 (0.003–0.015)	150–730 (500–2,400)
Composites, graphite-reinforced	TiN-coated carbide	1.9 (0.075)	0.2 (0.008)	200 (650)	0.12–6.3 (0.005–0.25)	0.12–1.5 (0.005–0.06)	105–290 (350–950)
	Polycrystalline diamond	1.9 (0.075)	0.2 (0.008)	760 (2,500)	0.12–6.3 (0.005–0.25)	0.12–1.5 (0.005–0.06)	550–1,310 (1,800–4,300)

NOTE: Cutting speeds for high-speed-steel tools are about one-half those for uncoated carbides.
SOURCE: Based on data from Kennametal Inc.

Table 13.4.5 General Troubleshooting Guide for Turning Operations

Problem	Probable causes
Tool breakage	Tool material lacks toughness; improper tool angles; machine tool lacks stiffness; worn bearings and machine components; cutting parameters too high
Excessive tool wear	Cutting parameters too high; improper tool material; ineffective cutting fluid; improper tool angles
Rough surface finish	Built-up edge on tool; feed too high; tool too sharp, chipped, or worn; vibration and chatter
Dimensional variability	Lack of stiffness of machine tool and work-holding devices; excessive temperature rise; tool wear
Tool chatter	Lack of stiffness of machine tool and work-holding devices; excessive tool overhang; machining parameters not set properly

constructed for light work. As with turret lathes, they have spring collets for holding the bars during machining and friction fingers or rolls to feed the bar stock forward. Some bar-feeding devices are operated by hand and others semiautomatically.

Automatic screw machines may be classified as single-spindle or multiple-spindle. Single-spindle machines rotate the bar stock from which the part is to be made. The tools are carried on a turret and on cross slides or on a circular drum and on cross slides. Multiple-spindle machines have four, five, six, or eight spindles, each carrying a bar of the material from which the piece is to be made. Capacities range from ⅛ to 6 in (3 to 150 mm) diam of bar stock.

Feeds of forming tools vary with the width of the cut. The wider the forming tool and the smaller the diameter of stock, the smaller the feed. On multiple-spindle machines, where many tools are working simultaneously, the feeds should be such as to reduce the actual cutting time to a minimum. Often only one or two tools in a set are working up to capacity, as far as actual speed and feed are concerned.

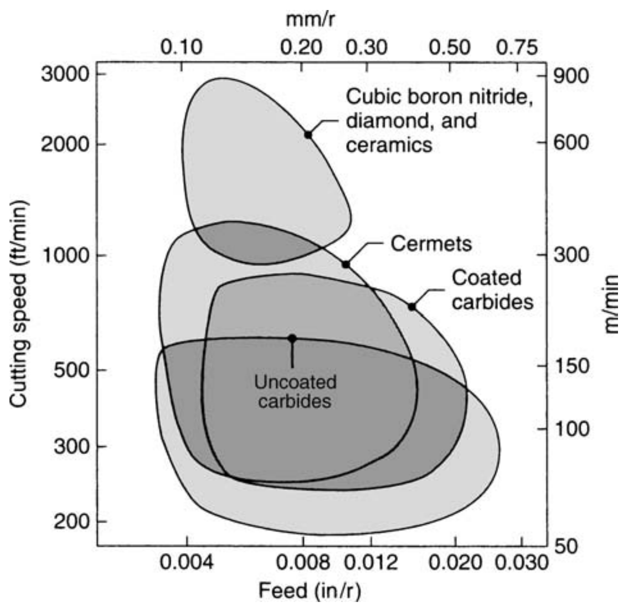


Fig. 13.4.12 Range of applicable cutting speeds and feeds for various tool materials.

BORING

Boring is a machining process for producing internal straight cylindrical surfaces or profiles, with process characteristics and tooling similar to those for turning operations.

Boring machines are of two general types, horizontal and vertical, and are frequently referred to as horizontal boring machines and vertical boring and turning mills. A classification of boring machines comprises horizontal boring, drilling, and milling machines; vertical boring and turning mills; vertical multispindle cylinder boring mills; vertical cylinder boring mills; vertical turret boring mills (vertical turret lathes); carwheel boring mills; diamond or precision boring machines (vertical and horizontal); and jig borers.

The **horizontal type** is made for both precision work and general manufacturing. It is particularly adapted for work not conveniently revolved, for milling, slotting, drilling, tapping, boring, and reaming long holes, and for making interchangeable parts that must be produced without jigs and fixtures. The machine is universal and has a wide range of speeds and feeds, for a face-mill operation may be followed by one with a small-diameter drill or end mill.

Vertical boring mills are adapted to a wide range of faceplate work that can be revolved. The advantage lies in the ease of fastening a workpiece to the horizontal table, which resembles a four-jaw independent chuck with extra radial T slots, and in the lessened effect of centrifugal forces arising from unsymmetrically balanced workpieces.

A jig-boring machine has a single-spindle sliding head mounted over a table adjustable longitudinally and transversely by lead screws which roughly locate the work under the spindle. Precision setting of the table may be obtained with end measuring rods, or it may depend only on the accuracy of the lead screw. These machines, made in various sizes, are used for accurately finishing holes and surfaces in definite relation to one another. They may use drills, rose or fluted reamers, or single-point boring tools. The latter are held in an adjustable **boring head** by which the tool can be moved eccentrically to change the diameter of the hole.

Precision-boring machines may have one or more spindles operating at high speeds for the purpose of boring to accurate dimensions such surfaces as wrist-pin holes in pistons and connecting-rod bushings.

Boring Recommendations Boring recommendations for tool materials, depth of cut, feed, and cutting speed are generally the same as those for turning operations (see Table 13.4.4). However, tool deflections, chatter, and dimensional accuracy can be significant problems because the boring bar has to reach the full length to be machined and space within the workpiece may be limited. Boring bars have been designed to dampen vibrations and reduce chatter during machining.

DRILLING

Drilling is a commonly employed hole-making process that uses a **drill** as a cutting tool for producing round holes of various sizes and depths. Drilled holes may be subjected to additional operations for better surface finish and dimensional accuracy, such as reaming and honing, described later in this section.

Drilling machines are intended for drilling holes, tapping, counterboring, reaming, and general boring operations. They may be classified into a large variety of types.

Vertical drilling machines are usually designated by a dimension which roughly indicates the diameter of the largest circle that can be drilled at its center under the machine. This dimensioning, however, does not hold for all makes of machines. The sizes begin with about 6 and continue to 50 in. Heavy-duty drill presses of the vertical type, with all-geared speed and feed drive, are constructed with a box-type column instead of the older cylindrical column.

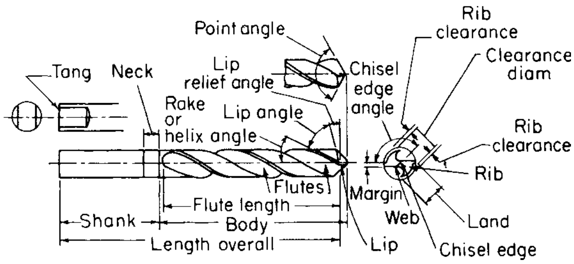


Fig. 13.4.13 Straight shank twist drill.

The size of a **radial drill** is designated by the length of the arm. This represents the radius of a piece which can be drilled in the center.

Twist drills (Fig. 13.4.13) are the most common tools used in drilling and are made in many sizes and lengths. For years they have been grouped according to numbered sizes, 1 to 80, inclusive, corresponding approximately to Stub's steel wire gage; some by lettered sizes A to Z, inclusive; some by fractional inches from 1⁄64 up, and the group of millimetre sizes.

Straight-shank twist drills of fractional size and various lengths range from 1⁄64 in diam to 1¼ in by 1⁄64 in increments; to 1½ in by 1⁄32 in; and to 2 in by 1⁄16 in. **Taper-shank drills** range from ⅛ in diam to 1¾ in by 1⁄64 increments; to 2¼ in by 1⁄32 in; and to 3½ in by 1⁄16 in. Larger drills are made by various drill manufacturers. Drills are also available in metric dimensions.

Tolerances have been set on the various features of all drills so that the products of different manufacturers will be interchangeable in the user's plants.

Twist drills are decreased in diameter from point to shank (back taper) to prevent binding. If the web is increased gradually in thickness from point to shank to increase the strength, it is customary to reduce the helix angle as it approaches the shank. The shape of the groove is important, the one that gives a straight cutting edge and allows a full curl to the chip being the best. The **helix angles** of the flutes vary from 10 to 45°. The standard **point angle** is 118°. There are a number of **drill grinders** on the market designed to give the proper angles. The point may be ground either in the **standard** or the **crankshaft** geometry. The drill geometry for high-speed steel twist drills for a variety of workpiece materials is given in Table 13.4.6.

Among the common **types of drills** (Fig. 13.4.14) are the combined drill and countersink or **center drill,** a short drill used to center shafts before squaring and turning: the **step drill,** with two or more diameters; the **spade drill** which has a removable tip or bit clamped in a holder on the drill shank, used for large and deep holes; the **trepanning tool** used to cut a core from a piece of metal instead of reducing all the metal

Table 13.4.6 Recommended Drill Geometry for High-Speed Steel Twist Drillls

Material	Point angle, deg	Lip relief angle, deg	Chisel edge angle, deg	Helix angle, deg	Point grind
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
Steels	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-temperature 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

SOURCE: "Machining Data Handbook," published by the Machinability Data Center, Metcut Research Associates Inc.

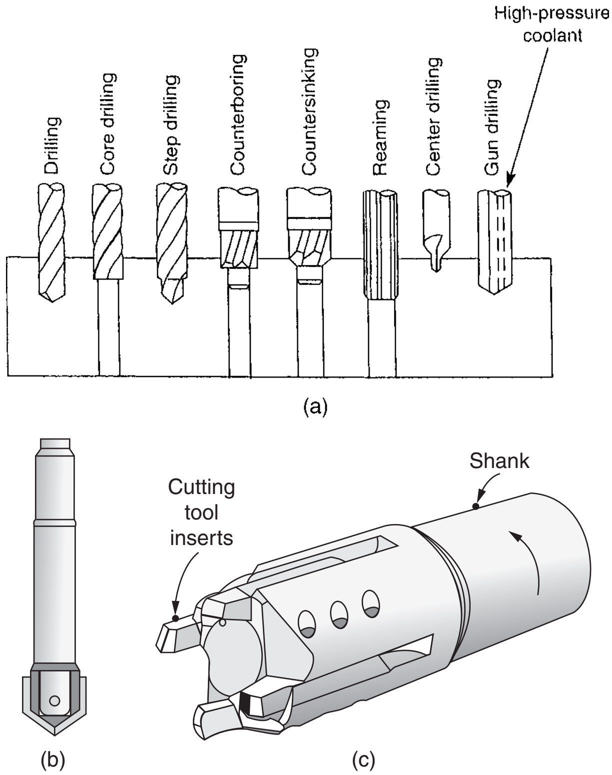


Fig. 13.4.14 (*a*) Various types of drills and drilling and reaming operations; (*b*) spade drill; (*c*) trepanning tool with four cutting-tool inserts.

removed to chips; the **gun drill,** run at a high speed under a light feed, and used to drill small long holes; the **core drill** used to bore out cored holes; the **oil-hole drill,** having holes or tubes in its body through which oil is forced to the cutting lips; **three-** and **four-fluted drills,** used to enlarge holes after a leader hole has been cored, punched, or drilled with a two-fluted drill; **twist drills** made from flat high-speed steel or drop-forged to desired shape and then twisted. Drills are also made of solid carbide or of high-speed steel with an insert of carbide to form the chisel edge and both cutting edges. They are used primarily for drilling abrasive or very hard materials.

Drilling Recommendations The most common tool material for drills is high-speed steel M1, M7, and M10. General recommendations for speeds and feeds in drilling a variety of materials are given in Table 13.4.7. Hole depth is also a factor in selecting drilling parameters. A general **troubleshooting guide** for drilling is given in Table 13.4.8.

Table 13.4.8 General Troubleshooting Guide for Drilling Operations

Problem	Probable causes
Drill breakage	Dull drill; drill seizing in hole because of chips clogging flutes; feed too high; lip relief angle too small
Excessive drill wear	Cutting speed too high; ineffective cutting fluid; rake angle too high; drill burned and strength lost when sharpened
Tapered hole	Drill misaligned or bent; lips not equal; web not central
Oversize hole	Same as above; machine spindle loose; chisel edge not central; side pressure on workpiece
Poor hole surface finish	Dull drill; ineffective cutting fluid; welding of workpiece material on drill margin; improperly ground drill; improper alignment

REAMING

A **reamer** is a multiple-cutting edge tool used to enlarge or finish holes, and to provide accurate dimensions as well as good finish. Reamers are of two types: (1) rose and (2) fluted.

The **rose reamer** is a heavy-bodied tool with end cutting edges. It is used to remove considerable metal and to true up a hole preparatory to flute reaming. It is similar to the three- and four-fluted drills. Wide cylindrical lands are provided back of the flute edges.

Fluted reamers cut principally on the periphery and remove only 0.004 to 0.008 in (0.1 to 0.2 mm) on the bore. Very narrow cylindrical margins are provided back of the flute edges, 0.012 to 0.015 in (0.3 to 0.4 mm) wide for machine-finish reaming and 0.004 to 0.006 in (0.1 to 0.15 mm) for hand reaming, to provide free cutting of the edges due to the slight body taper and also to pilot the reamer in the hole. The hole to be flute- or finish-reamed should be true. A rake of 5° is recommended for most reaming operations. A reamer may be straight or helically fluted. The latter provides much smoother cutting and gives a better finish.

Expansion reamers permit a slight expansion by a wedge so that the reamer may be resharpened to its normal size or for job shop use; they provide slight variations in size. **Adjustable reamers** have means of adjusting inserted blades so that a definite size can be maintained through numerous grindings and fully worn blades can be replaced with new ones. **Shell reamers** constitute the cutting portion of the tool which fits interchangeably on arbors to make many sizes available or to make replacement of worn-out shells less costly. Reamers float in their holding fixtures to ensure alignment, or they should be piloted in guide bushings above and below the work. They may also be held rigidly, such as in the tailstock of a lathe.

The **speed** of high-speed steel reamers should be two-thirds to three-quarters and **feeds** usually are two or three times that of the corresponding drill size. The most common tool materials for reamers are M1, M2, and M7 high-speed steels and C2 carbide.

Table 13.4.7 General Recommendations for Drilling

Workpiece material	Surface speed		Feed, mm/r (in/r), Drill diameter		r/min	
	m/min	ft/min	1.5 mm (0.060 in)	12.5 mm (0.5 in)	1.5 mm	12.5 mm
Aluminum alloys	30–120	100–400	0.025 (0.001)	0.30 (0.012)	6,400–25,000	800–3,000
Magnesium alloys	45–120	150–400	0.025 (0.001)	0.30 (0.012)	9,600–25,000	1,100–3,000
Copper alloys	15–60	50–200	0.025 (0.001)	0.25 (0.010)	3,200–12,000	400–1,500
Steels	20–30	60–100	0.025 (0.001)	0.30 (0.012)	4,300–6,400	500–800
Stainless steels	10–20	40–60	0.025 (0.001)	0.18 (0.007)	2,100–4,300	250–500
Titanium alloys	6–20	20–60	0.010 (0.0004)	0.15 (0.006)	1,300–4,300	150–500
Cast irons	20–60	60–200	0.025 (0.001)	0.30 (0.012)	4,300–12,000	500–1,500
Thermoplastics	30–60	100–200	0.025 (0.001)	0.13 (0.005)	6,400–12,000	800–1,500
Thermosets	20–60	60–200	0.025 (0.001)	0.10 (0.004)	4,300–12,000	500–1,500

NOTE: As hole depth increases, speeds and feeds should be reduced. Selection of speeds and feeds also depends on the specific surface finish required.

THREADING

Threads may be formed on the outside or inside of a cylinder or cone (1) with single-point threading tools (see Fig. 13.4.1), (2) with threading chasers, (3) with taps, (4) with dies, (5) by thread milling, (6) by thread rolling, and (7) by grinding. There are numerous types of taps, such as hand, machine screw, pipe, and combined pipe tap and drill. Small taps usually have no radial relief. They may be made in two, three, or four flutes. Large taps may have still more flutes.

The **feed** of a tap depends upon the lead of the screw thread. The **cutting speed** depends upon numerous factors: Hard tough materials, great length of hole, taper taps, and full-depth thread reduce the speed; long chamfer, fine pitches, and a cutting fluid applied in quantity increase the speed. Taps are cut or formed by grinding. The ground-thread taps may operate at much higher speeds than the cut taps. Speeds may range from 3 ft/min (1 m/min) for high-strength steels to 150 ft/min (45 m/min) for aluminum and magnesium alloys. Common high-speed steel tool materials for taps are M1, M7, and M10.

Threading dies, used to produce external threads, may be solid, adjustable, spring-adjustable, or self-opening die heads. Replacement chasers are used in die heads and may be of the fixed or self-opening type. These chasers may be of the radial type, hobbed or milled; of the tangenital type; or of the circular type. Emulsions and oils are satisfactory for most threading operations.

For thread rolling, see Sec. 13.2.

MILLING

Milling is one of the most versatile machining processes and is capable of producing a variety of shapes involving flat surfaces, slots, and contours (Fig. 13.4.15). **Milling machines** use cutters with multiple teeth in contrast with the single-point tools of the lathe and planer.

Milling-machine classification is based on design, operation, or purpose. **Knee-and-column** type milling machines have the table and saddle supported on the vertically adjustable knee gibbed to the face of the column. The table is fed longitudinally on the saddle, and the latter transversely on the knee to give three feeding motions.

Knee-type machines are made with horizontal or vertical spindles. The **horizontal** universal machines have a swiveling table for cutting helices. The plain machines are used for jobbing or production work, the universal for toolroom work. **Vertical** milling machines with fixed or sliding heads are otherwise similar to the horizontal type. They are used for face or end milling and are frequently provided with a rotary table for making cylindrical surfaces.

The **fixed-bed** machines have a spindle mounted in a head dovetailed to and sliding on the face of the column. The table rests directly on the bed. They are simple and rigidly built and are used primarily for high-production work. These machines are usually provided with work-holding fixtures and may be constructed as plain or multiple-spindle machines, simple or duplex.

Planer-type millers are used only on the heaviest work. They are used to machine a number of surfaces on a particular part or group of parts arranged in series in fixtures on the table.

Milling Cutters

Milling cutters are made in a wide variety of shapes and sizes. The nomenclature of tooth parts and angles is standardized as in Fig. 13.4.16. Milling cutters may be classified in various ways, such as purpose or use of the cutters (Woodruff keyseat cutters, T-slot cutters, gear cutters, etc.); construction characteristics (solid cutters, carbide-tipped cutters, etc.); method of mounting (arbor type, shank type, etc.); and relief of teeth. The latter has two categories: profile cutters which produce flat, curved, or irregular surfaces, with the cutter teeth sharpened on the land; and formed cutters which are sharpened on the face to retain true cross-sectional form of the cutter.

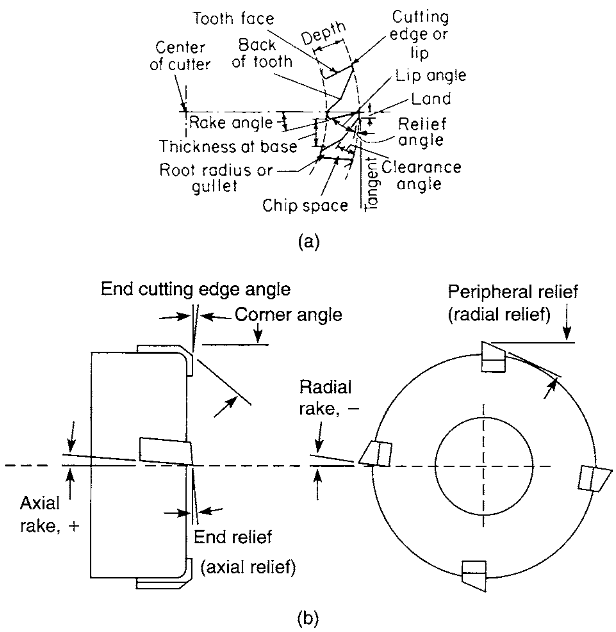


Fig. 13.4.16 (*a*) Plain milling cutter teeth; (*b*) face milling cutter.

Two kinds of milling are generally considered to represent all forms of milling processes: **peripheral (slab)** and **face** milling. In the peripheral-milling process the axis of the cutter is parallel to the surface milled, whereas in face milling, the cutter axis is generally at a right angle to the surface. The peripheral-milling process is also divided into two types: **conventional (up) milling** and **climb (down) milling.** Each has its advantages, and the choice depends on a number of factors such as the type and condition of the equipment, tool life, surface finish, and machining parameters.

Milling Recommendations Recommendations for tool materials, feed per tooth, and cutting speed for milling a variety of materials are

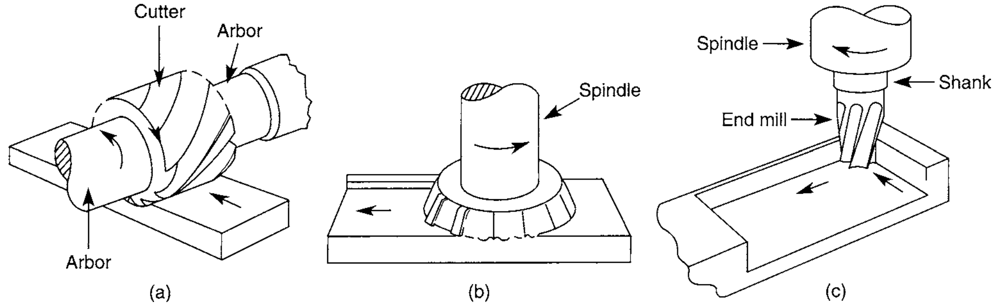


Fig. 13.4.15 Basic types of milling cutters and operations. (*a*) Peripheral milling; (*b*) face milling; (*c*) end milling.

Table 13.4.9 General Recommendations for Milling Operations

Workpiece material	Cutting tool	General-purpose starting conditions: Feed, mm/tooth (in/tooth)	General-purpose starting conditions: Speed, m/min (ft/min)	Range of conditions: Feed, mm/tooth (in/tooth)	Range of conditions: Speed, m/min (ft/min)
Low-C and free-machining steels	Uncoated carbide, coated carbide, cermets	0.13–0.20 (0.005–0.008)	120–180 (400–600)	0.085–0.38 (0.003–0.015)	90–425 (300–1,400)
Alloy steels					
Soft	Uncoated, coated, cermets	0.10–0.18 (0.004–0.007)	90–170 (300–550)	0.08–0.30 (0.003–0.012)	60–370 (200–1,200)
Hard	Cermets, PcBN	0.10–0.15 (0.004–0.006)	180–210 (600–700)	0.08–0.25 (0.003–0.010)	75–460 (250–1,500)
Cast iron, gray					
Soft	Uncoated, coated, cermets, SiN	0.10–0.20 (0.004–0.008)	120–760 (400–2,500)	0.08–0.38 (0.003–0.015)	90–1,370 (300–4,500)
Hard	Cermets, SiN, PcBN	0.10–0.20 (0.004–0.008)	120–210 (400–700)	0.08–0.38 (0.003–0.015)	90–460 (300–1,500)
Stainless steel, austenitic	Uncoated, coated, cermets	0.13–0.18 (0.005–0.007)	120–370 (400–1,200)	0.08–0.38 (0.003–0.015)	90–500 (300–1,800)
High-temperature alloys, nickel base	Uncoated, coated, cermets, SiN, PcBN	0.10–0.18 (0.004–0.007)	30–370 (100–1,200)	0.08–0.38 (0.003–0.015)	30–550 (90–1,800)
Titanium alloys	Uncoated, coated, cermets	0.13–0.15 (0.005–0.006)	50–60 (175–200)	0.08–0.38 (0.003–0.015)	40–140 (125–450)
Aluminum alloys					
Free-machining	Uncoated, coated, PCD	0.13–0.23 (0.005–0.009)	610–900 (2,000–3,000)	0.08–0.46 (0.003–0.018)	300–3,000 (1,000–10,000)
High-silicon	PCD	0.13 (0.005)	610 (2,000)	0.08–0.38 (0.003–0.015)	370–910 (1,200–3,000)
Copper alloys	Uncoated, coated, PCD	0.13–0.23 (0.005–0.009)	300–760 (1,000–2,500)	0.08–0.46 (0.003–0.018)	90–1,070 (300–3,500)
Thermoplastics and thermosets	Uncoated, coated, PCD	0.13–0.23 (0.005–0.009)	270–460 (900–1,500)	0.08–0.46 (0.003–0.018)	90–1,370 (300–4,500)

NOTE: Depths of cut, *d*, usually are in the range of 1–8 mm (0.04–0.3 in). PcBN: polycrystalline cubic boron nitride; PCD: polycrystalline diamond.
SOURCE: Based on data from Kennametal Inc.

Table 13.4.10 General Troubleshooting Guide for Milling Operations

Problem	Probable causes
Tool breakage	Tool material lacks toughness; improper tool angles; cutting parameters too high
Tool wear excessive	Cutting parameters too high; improper tool material; improper tool angles; improper cutting fluid
Rough surface finish	Feed too high; spindle speed too low; too few teeth on cutter; tool chipped or worn; built-up edge; vibration and chatter
Tolerances too broad	Lack of spindle stiffness; excessive temperature rise; dull tool; chips clogging cutter
Workpiece surface burnished	Dull tool; depth of cut too low; radial relief angle too small
Back striking	Dull cutting tools; cutter spindle tilt; negative tool angles
Chatter marks	Insufficient stiffness of system; external vibrations; feed, depth, and width of cut too large
Burr formation	Dull cutting edges or too much honing; incorrect angle of entry or exit; feed and depth of cut too high; incorrect insert geometry
Breakout	Lead angle too low; incorrect cutting edge geometry; incorrect angle of entry or exit; feed and depth of cut too high

given in Table 13.4.9. A general **troubleshooting guide** for milling operations is given in Table 13.4.10.

GEAR MANUFACTURING

(See also Sec. 8.)

Gear Cutting Most gear-cutting processes can be classified as either **forming or generating.** In a forming process, the shape of the tool is reproduced on the workpiece; in a generating process, the shape produced on the workpiece depends on both the shape of the tool and the relative motion between the tool and the workpiece during the cutting operation. In general, a generating process is more accurate than a forming process.

In the **form cutting** of gears, the tool has the shape of the space between the teeth. For this reason, form cutting will produce precise tooth profiles only when the cutter is accurately made and the tooth space is of constant width, such as on spur and helical gears. A form cutter may cut or finish one of or all the spaces in one pass. Single-space cutters may be disk-type or end-mill-type milling cutters. In all single-space operations, the gear blank must be retracted and indexed, i.e., rotated one tooth space, between each pass.

Single-space form milling with disk-type cutters is particularly suitable for gears with large teeth, because, as far as metal removal is concerned, the cutting action of a milling cutter is more efficient than that of the tools used for generating. Form milling of spur gears is done on machines that retract and index the gear blank automatically.

For the same tooth size (pitch), the shape (profile) of the teeth on an involute gear depends on the number of teeth on the gear. Most gears have active profiles that are wholly, partially, or approximately involute,

Table 13.4.11

No. of cutter	1	2	3	4	5	6	7	8
No. of teeth	135–∞	55–134	35–54	27–34	21–26	17–20	14–16	12 and 13
For more accurate gears, 15 cutters are available								
No. of cutter	1	1½	2	2½	3	3½	4	4½
No. of teeth	135–∞	80–134	55–79	42–54	35–41	30–34	26–29	23–25
No. of cutter	5	5½	6	6½	7	7½	8	
No. of teeth	21 and 22	19 and 20	17 and 18	15 and 16	14	13	12	

and, consequently, accurate form cutting would require a different cutter for each number of teeth. In most cases, satisfactory results can be obtained by using the eight cutters for each pitch that are commercially available. Each cutter is designed to cut a range of tooth numbers; the no. 1 cutter, for example, cuts from 135 teeth to a rack, and the no. 8 cuts 12 and 13 teeth. (See Table 13.4.11.)

In a **gear generating machine,** the generating tool can be considered as one of the gears in a conjugate pair and the gear blank as the other gear. The correct relative motion between the tool arbor and the blank arbor is obtained by means of a train of indexing gears within the machine.

One of the most valuable properties of the involute as a gear-tooth profile is that if a cutter is made in the form of an involute gear of a given pitch and any number of teeth, it can generate all gears of all tooth numbers of the same pitch and they will all be conjugate to one another. The generating tool may be a pinion-shaped cutter, a rack-shaped (straight) cutter, or a hob, which is essentially a series of racks wrapped around a cylinder in a helical, screwlike form.

On a **gear shaper,** the generating tool is a pinion-shaped cutter that rotates slowly at the proper speed as if in mesh with the blank; the cutting action is produced by a reciprocation of the cutter parallel to the work axis. These machines can cut spur and helical gears, both internal and external; they can also cut continuous-tooth helical (herringbone) gears and are particularly suitable for cluster gears, or gears that are close to a shoulder.

On a **rack shaper** the generating tool is a segment of a rack that moves perpendicular to the axis of the blank while the blank rotates about a fixed axis at the speed corresponding to conjugate action between the rack and the blank; the cutting action is produced by a reciprocation of the cutter parallel to the axis of the blank. Since it is impracticable to have more than 6 to 12 teeth on a rack cutter, the cutter must be disengaged from the blank at suitable intervals and returned to the starting point, the blank meanwhile remaining fixed. These machines can cut both spur and helical external gears.

A gear-cutting **hob** (Fig. 13.4.17) is basically a worm, or screw, made into a generating tool by cutting a series of longitudinal slots or "gashes" to form teeth; to form cutting edges, the teeth are "backed off," or relieved, in a lathe equipped with a backing-off attachment. A hob may have one, two, or three threads; on involute hobs with a single thread, the generating portion of the hob-tooth profile usually has straight sides (like an involute rack tooth) in a section taken at right angles to the thread.

In addition to the conjugate rotary motions of the hob and workpiece, the hob must be fed parallel to the workpiece axis for a distance greater than the face width of the gear. The feed, per revolution of the workpiece, is produced by the feed gears, and its magnitude depends on the material, pitch, and finish desired; the feed gears are independent of the indexing gears. The hobbing process is continuous until all the teeth are cut.

The same machines and the same hobs that are used for cutting spur gears can be used for helical gears; it is only necessary to tip the hob axis so that the hob and gear pitch helices are tangent to one another and to correlate the indexing and feed gears so that the blank and the hob are advanced or retarded with respect to each other by the amount required to produce the helical teeth. Some hobbing machines have a differential gear mechanism that permits the indexing gears to be selected as for spur gears and the feed gearing to be chosen independently.

The threads of worms are usually cut with a disk-type milling cutter on a thread-milling machine and finished, after hardening, by grinding.

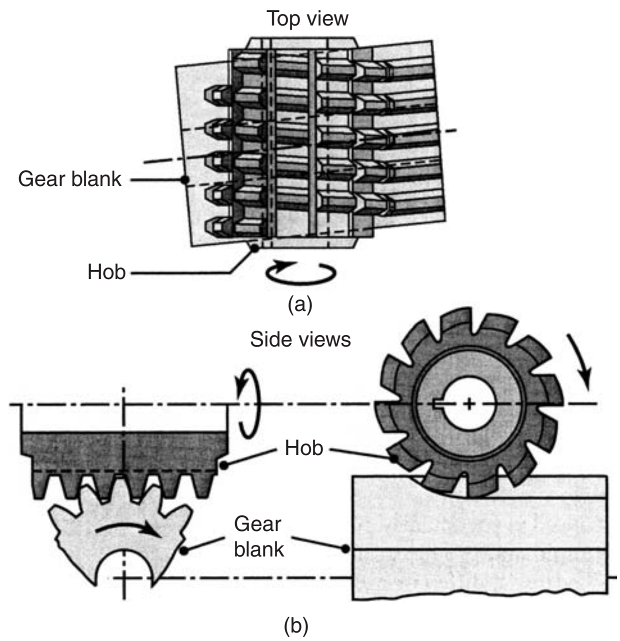


Fig. 13.4.17 A gear-cutting hob.

Worm gears are usually cut with a hob on the machines used for hobbing spur and helical gears. Except for the gashes, the relief on the teeth, and an allowance for grinding, the hob is a counterpart of the worm. The hob and workpiece axes are inclined to one another at the shaft angle of the worm and gear set, usually 90°. The hob may be fed in to full depth in a radial (to the blank) direction or parallel to the hob axis.

Although it is possible to approximate the true shape of the teeth on a straight bevel gear by taking two or three cuts with a form cutter on a milling machine, this method, because of the taper of the teeth, is obviously unsuited for the rapid production of accurate teeth. Most straight bevel gears are roughed out in one cut with a form cutter on machines that index automatically and then finished to the proper shape on a generator.

The generating method used for straight bevel gears is analogous to the rack-generating method used for spur gears. Instead of using a rack with several complete teeth, however, the cutter has only one straight cutting edge that moves, during generation, in the plane of the tooth of a basic crown gear conjugate to the gear being generated. A crown gear is the rack among bevel gears; its pitch surface is a plane, and its teeth have straight sides.

The generating cutter moves back and forth across the face of the bevel gear like the tool on a shaper; the "generating roll" is obtained by rotating the gear slowly relative to the tool. In practice two tools are used, one for each side of a tooth; after each tooth has been generated, the gear must be retracted and indexed to the next tooth.

The machines used for cutting spiral bevel gears operate on essentially the same principle as those used for straight bevel gears; only the cutter is different. The spiral cutter is basically a disk that has a number of straight-sided cutting blades protruding from its periphery on one side to form the rim of a cup. The machines have means for indexing, retracting, and producing a generating roll; by disconnecting the roll gears, spiral bevel gears can be form cut.

Gear Shaving For improving the surface finish and profile accuracy of cut spur and helical gears (internal and external), gear shaving, a free-cutting gear finishing operation that removes small amounts of metal from the working surfaces of the teeth, is employed. The teeth on the shaving cutter, which may be in the form of a pinion (spur or helical) or a rack, have a series of sharp-edged rectangular grooves running from tip to root. The intersection of the grooves with the tooth profiles creates cutting edges; when the cutter and the workpiece, in tight mesh, are caused to move relative to one another along the teeth, the cutting edges remove metal from the teeth of the work gear. Usually the cutter drives the workpiece, which is free to rotate and is traversed past the cutter parallel to the workpiece axis. Shaving requires less time than grinding, but ordinarily it cannot be used on gears harder than approximately 400 HB (42 HRC).

Gear Grinding Machines for the grinding of spur and helical gears utilize either a forming or a generating process. For form grinding, a disk-type grinding wheel is dressed to the proper shape by a diamond held on a special dressing attachment; for each number of teeth a special index plate, with V-type notches on its periphery, is required. When grinding helical gears, means for producing a helical motion of the blank must be provided.

For grinding-generating, the grinding wheel may be a disk-type, double-conical wheel with an axial section equivalent to the basic rack of the gear system. A master gear, similar to the gear being ground, is attached to the workpiece arbor and meshes with a master rack; the generating roll is created by rolling the master gear in the stationary rack.

Spiral bevel and hypoid gears can be ground on the machines on which they are generated. The grinding wheel has the shape of a flaring cup with a double-conical rim having a cross section equivalent to the surface that is the envelope of the rotary cutter blades.

Gear Rolling The cold-rolling process is used for the finishing of spur and helical gears for automatic transmissions and power tools; in some cases it has replaced gear shaving. It differs from cutting in that the metal is not removed in the form of chips but is displaced under heavy pressure. (See Sec. 13.2.)

There are two main types of cold-rolling machines, namely, those employing dies in the form of racks or gears that operate in a parallel-axis relationship with the blank and those employing worm-type dies that operate on axes at approximately 90° to the workpiece axis. The dies, under pressure, create the tooth profiles by the plastic deformation of the blank.

When racks are used, the process resembles thread rolling; with gear-type dies the blank can turn freely on a shaft between two dies, one mounted on a fixed head and the other on a movable head. The dies have the same number of teeth and are connected by gears to run in the same direction at the same speed. In operation, the movable die presses the blank into contact with the fixed die, and a conjugate profile is generated on the blank. On some of these machines the blank can be fed axially, and gears can be rolled in bar form to any convenient length.

On machines employing worm-type dies, the two dies are diametrically opposed on the blank and rotate in opposite directions. The speeds of the blank and the dies are synchronized by change gears, like the blank and the hob on a hobbing machine; the blank is fed axially between the dies.

PLANING AND SHAPING

Planers are used to rough and finish large flat surfaces, although arcs and special forms can be made with proper tools and attachments. Surfaces to be finished by scraping, such as ways and long dovetails and, particularly, parts of machine tools, are, with few exceptions, planed. With fixtures to arrange parts in parallel and series, quantities of small parts can be produced economically on planers. **Shapers** are used for miscellaneous planing, surfacing, notching, key seating, and production of flat surfaces on small parts. The tool is held in a holder supported on a clapper on the end of a ram which is reciprocated hydraulically or by crank and rocker arm, in a straight line.

BROACHING

Broaching is a production process whereby a cutter, called a **broach,** is used to finish internal or external surfaces such as holes of circular, square, or irregular section, keyways, the teeth of internal gears, multiple spline holes, and flat surfaces. Broaching round holes gives greater accuracy and better finish than reaming, but since the broach may be guided only by the workpiece it is cutting, the hole may not be accurate with respect to previously machined surfaces. Where such accuracy is required, it is better practice to broach first and then turn other surfaces with the workpiece mounted on a mandrel. The broach is usually long and is provided with many teeth so graded in size that each takes a small chip when the tool is pulled or pushed through the previously prepared leader hole or past the surface.

The main features of the broach are the pitch, degree of taper or increase in height of each successive tooth, relief, tooth depth, and rake.

The **pitch** of the teeth, i.e., the distance from one tooth to the next, depends upon tooth strength, length of cut, shape and size of chips, etc. The pitch should be as coarse as possible to provide ample chip clearance, but at least two teeth should be in contact with the workpiece at all times. The formula $p = 0.35\sqrt{l}$ may be used, where p is pitch of the roughing teeth and l the length of hole or surface, in. An average pitch for small broaches is ⅛ to ¼ in (3.175 to 6.35 mm) and for large ones ½ to 1 in (12.7 to 25.4 mm). Where the hole or other surface to be broached is short, the teeth are often cut on an angle or helix, so as to give more continuous cutting action by having at least two teeth cutting simultaneously.

The degree of **taper,** or increase in size per tooth, depends largely on the hardness or toughness of the material to be broached and the finish desired. The degree of taper or feed for broaching cast iron is approximately double that for steel. Usually the first few teeth coming in contact with the workpiece are undersize but of uniform taper to take the greatest feeds per tooth, but as the finished size is approached, the teeth take smaller and smaller feeds with several teeth at the finishing end of nearly zero taper. In some cases, for soft metals and even cast iron, the large end is left plain or with rounded lands a trifle larger than the last cutting tooth so as to burnish the surface. For medium-sized broaches, the taper per tooth is 0.001 to 0.003 in (0.025 to 0.076 mm). Large broaches remove 0.005 to 0.010 in (0.127 to 0.254 mm) per tooth or even more. The teeth are given a **front rake** angle of 5 to 15° to give a curl to the chip, provide a cleaner cut surface, and reduce the power consumption. The **land** back of the cutting edge, which may be 1⁄64 to 1⁄16 in (0.4 to 1.6 mm) wide, usually is provided with a land relief varying from 1 to 3° with a clearance of 30 to 45°.

The heavier the feed per tooth or the longer the surface being broached, the greater must be the **chip clearance** or space between successive teeth for the chips to accumulate. The root should be a smooth curve.

Broaches are generally made of M2 or M7 high-speed steel; carbide is also used for the teeth of large broaches. Broaches of complicated shape are likely to warp during the heat-treating process. For this reason, in hardening, they are often heated in a vertical cylindrical furnace and quenched by being hung in an air blast furnished from small holes along the side of pipes placed vertically about the broach.

Push broaches are usually shorter than pull broaches, being 6 to 14 in (150 to 350 mm) long, depending on their diameter and the amount of metal to be removed. In many cases, for accuracy, four to six broaches of the push type constitute a set used in sequence to finish the surface being broached. Push broaches usually have a large cross-sectional area so as to be sufficiently rigid. With **pull broaches,** pulling tends to straighten the hole, whereas pushing permits the broaches to follow any irregularity of the leader hole. Pull broaches are attached to the crosshead of the broaching machine by means of a key slot and key, by a threaded connection, or by a head that fits into an automatic broach puller. The threaded connection is used where the broach is not removed from the drawing head while the workpiece is placed over the cutter, as in cutting a keyway. In enlarging holes, however, the small end of the pull broach must first be extended through the reamed, drilled, or cored hole and then fixed in the drawing head before being pulled through the workpiece.

Broaching Machines Push broaching is done on machines of the press type with a sort of fixture for holding the workpiece and broach or on presses operated by power. They are usually vertical and may be driven hydraulically or by screw, rack, or crank. The pull type of broach may be either vertical or horizontal. The ram may be driven hydraulically or by screw, rack, or crank. Both are made in the duplex- and multiple-head type.

Processing Parameters for Broaching Cutting speeds for broaching may range from 5 ft/min (1.5 m/min) for high-strength materials to as high as 30 to 50 ft/min (9 to 15 m/min) for aluminum and magnesium alloys. The most common tool materials are M2 and M7 high-speed steels and carbides. An emulsion is often used for broaching for general work, but oils may also be used.

CUTTING OFF

Cutting off involves parting or slotting bars, tubes, plate, or sheet by various means. The machines come in various types such as a lathe (using a single-point cutting tool), hacksaws, band saws, circular saws, friction saws, and thin abrasive wheels. Cutting off may also be carried out by shearing and cropping, as well as using flames and laser beams.

In **power hacksaws,** the frame in which the blade is strained is reciprocated above the workpiece which is held in a vise on the bed. The cutting feed is effected by weighting the frame, with 12 to 50 lb (55 to 225 N) of force from small to large machines; adding weights or spring tension giving up to 180 lb (800 N); providing a positive screw feed or a friction screw feed; and by a hydraulic feed mechanism giving forces up to 300 lb (1.34 kN) between the blade and workpiece. With highspeed steel blades, cutting speeds range from about 30 strokes per minute for high-strength materials to 180 strokes per minute for carbon steels.

Hacksaw **blades** for hand frames are made 8,10, and 12 in long, $\frac{7}{16}$ to $\frac{9}{16}$ in wide, and 0.025 in thick. Number of teeth per inch for cutting soft steel or cast iron, 14; tool steel and angle iron, 18; brass, copper, and heavy tubing, 24; sheet metal and thin tube, 32. Blades for power hacksaws are made of alloy steels and of high-speed steels. Each length is made in two or more widths. The coarsest teeth should be used on large workpieces and with heavy feeds.

Band saws, vertical, horizontal, and universal, are used for cutting off. The kerf or width of cut is small with a consequently small loss in expensive material. The teeth of band saws, like those of hacksaws, are set with the regular alternate type, one bent to the right and the next to the left; or with the alternate and center set, in which one tooth is bent to the right, the second to the left, and the third straight in the center. With high-speed steel saw blades, band speeds range from about 30 ft/min (10 m/min) for high-temperature alloys to about 400 (120) for carbon steels. For aluminum and magnesium alloys the speed ranges up to about 1,300 ft/min (400 m/min) with high-carbon blades. The band speed should be decreased as the workpiece thickness increases.

Circular saws are made in a wide variety of styles and sizes. Circular saws may have teeth of several shapes, as radial face teeth for small fine-tooth saws; radial face teeth with a land for fine-tooth saws; alternate bevel-edged teeth to break up the chips, with every other tooth beveled 45° on each side with the next tooth plain; alternate side-beveled teeth; and one tooth beveled on the right, the next on the left, and the third beveled on both sides, each leaving slightly overlapping flats on the periphery of the teeth. The most common high-speed steels for circular saws are M2 and M7; saws also may have welded carbide teeth for better performance.

Abrasive cutoff wheels are made of thin resinoid or rubber bonded abrasives. The wheels operate at surface speeds of 12,000 to 16,000 ft/min (3,600 to 4,800 m/min). These wheels are used for cutting off tubes, shapes, and hardened high-speed steel.

Friction sawing machines are used largely for cutting off structural shapes. Peripheral speeds of about 20,000 ft/min (6,000 m/min) are used. The wheels may be plain on the periphery, V-notched, or with milled square notches.

ABRASIVE PROCESSES

(See also Sec. 6.)

Abrasive processes consist of a variety of operations in which the tool is made of an abrasive material, the most common examples of which are grinding (using wheels, known as **bonded abrasives**), honing, and lapping. An abrasive is a small, nonmetallic hard particle having sharp cutting edges and an irregular shape. Abrasive processes, which can be performed on a wide variety of metallic and nonmetallic materials, remove material in the form of tiny chips and produce surface finishes and dimensional accuracies that are generally not obtainable through other machining or manufacturing processes.

Grinding wheels have characteristics influenced by (1) type of abrasive; (2) grain size; (3) grade; (4) structure; and (5) type of bond. (See Figs. 13.4.18 and 13.4.19.)

Selection of Abrasive Although a number of natural abrasives are available, such as emery, corundum, quartz, garnet, and diamond, the most commonly used abrasives in grinding wheels are **aluminum oxide** and **silicon carbide,** the former being more commonly used than the latter. Aluminum oxide is softer than silicon carbide, and because of its friability and low attritious wear it is suitable for most applications. Silicon carbide is used for grinding aluminum, magnesium, titanium, copper, tungsten, and rubber. It is also used for grinding very hard and brittle materials such as carbides, ceramics, and stones. **Diamond** and **cubic boron nitride** grains are used to grind very hard materials and are known as **superabrasives.**

Selection of **grain size** depends on the rate of material removal desired and the surface finish. Coarse grains are used for fast removal of stock; fine grain for low removal rates and for fine finish. Coarse grains are also used for ductile materials and a finer grain for hard and brittle materials.

The **grade** of a grinding wheel is a measure of the strength of its bond. The force that acts on the grain in grinding depends on process variables (such as speeds, depth of cut, etc.) and the strength of the work material. Thus a greater force on the grain will increase the possibility of dislodging the grain; if the bond is too strong, the grain will tend to get dull, and if it is too weak then wheel wear will be high. If glazing occurs, the wheel is **acting hard;** reducing the wheel speed or increasing the work speed or the depth of cut causes the wheel to **act softer.** If the wheel breaks down too rapidly, reversing this procedure will make the wheel act harder. Harder wheels are generally recommended for soft work materials, and vice versa.

A variety of **bond** types are used in grinding wheels; these are generally categorized as **organic** and **inorganic.** Organic bonds are materials such as resin, rubber, shellac, and other similar bonding agents. Inorganic materials are glass, clay, porcelain, sodium silicate, magnesium oxychloride, and metal. The most common bond type is the vitrified bond which is composed of clay, glass, porcelain, or related ceramic materials. This type of bond is brittle and produces wheels that are rigid, porous, and resistant to oil and water. The most flexible bond is rubber which is used in making very thin, flexible wheels. Wheels subjected to bending strains should be made with organic bonds. In selecting a bonding agent, attention should be paid to its sensitivity to temperature, stresses, and grinding fluids, particularly over a period of time. The term **reinforced** as applied to grinding wheels indicates a class of organic wheels which contain one or more layers of strengthening fabric or filament, such as fiberglass. This term does not cover wheels with reinforcing elements such as steel rings, steel cup backs, or wire or tape winding. Fiberglass and filament reinforcing increases the ability of wheels to withstand operational forces when cracked.

The **structure** of a wheel is important in two aspects: It supplies a clearance for the chip, and it determines the number of cutting points on the wheel.

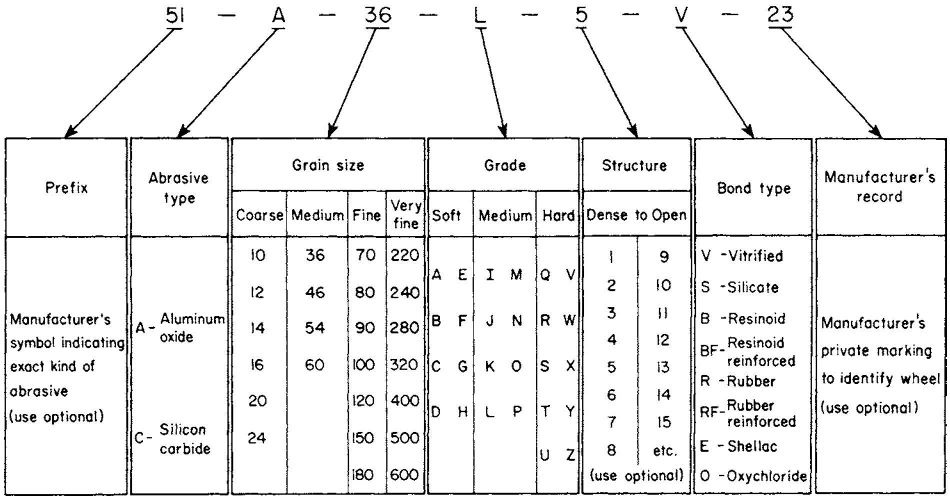


Fig. 13.4.18 Standard marking system chart for aluminum oxide and silicon carbide bonded abrasives.

In addition to wheel characteristics, grinding wheels come in a very large variety of shapes and dimensions. They are classified as **types,** such as type 1: straight wheels, type 4: taper side wheels, type 12: dish wheels, etc.

The **grinding ratio** is defined as the ratio of the volume of material removed to the volume of wheel wear. The grinding ratio depends on parameters such as the type of wheel, workpiece speed, wheel speed, cross-feed, down-feed, and the grinding fluid used. Values ranging from a low of 2 to over 200 have been observed in practice. A high grinding ratio, however, may not necessarily result in the best surface integrity of the part.

Wheel Speeds Depending on the type of wheel and the type and strength of bond, wheel speeds for standard applications range between 4,500 and 16,000 surface ft/min (1,400 and 4,800 m/min). The lowest speeds are for low-strength, inorganic bonds whereas the highest speeds are for high-strength organic bonds. The majority of surface grinding operations are carried out at speeds from 5,500 to 6,500 ft/min (1,750 to 2,000 m/min). The trend is toward high-efficiency grinding where wheel speeds from 12,000 to 18,000 ft/min (3,600 to 5,500 m/min) are employed.

It has been found that, by increasing the wheel speed, the rate of material removal can be increased, thus making the process more economical. This, of course, requires special grinding wheels to withstand the high stresses. Design changes or improvements involve items such as a composite wheel with a vitrified bond on the outside and a resinoid bond toward the center of the wheel; elimination of the central hole of the wheel by providing small bolt holes; and clamping of wheel segments instead of using a one-piece wheel. Grinding machines for such high-speed applications have requirements such as rigidity, high work and wheel speeds, high power, and special provisions for safety.

Workpiece speeds depend on the size and type of workpiece material and on whether it is rigid enough to hold its shape. In surface grinding, table speeds generally range from 50 to 100 ft/min (15 to 30 m/ min); for cylindrical grinding, work speeds from 70 to 100 ft/min (20 to 30 m/min), and for internal grinding they generally range from 75 to 200 ft/min (20 to 60 m/min).

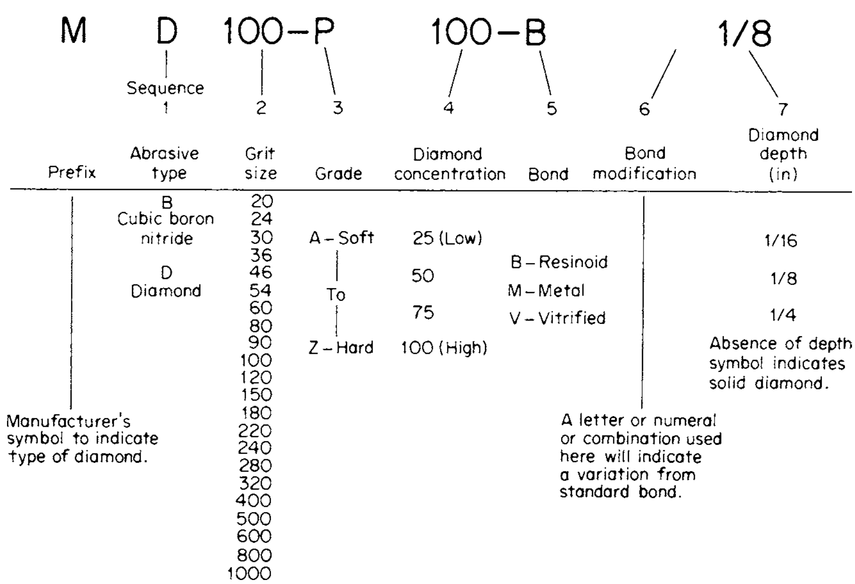


Fig. 13.4.19 Standard marking system chart for diamond tool and cubic boron nitride (cBN) bonded abrasives.

Cross-feed depends on the width of the wheel. In roughing, the workpiece should travel past the wheel ¾ to ⅞ of the width of the wheel for each revolution of the work. As the workpiece travels past the wheel with a helical motion, the preceding rule allows a slight overlap. In finishing, a finer feed is used, generally 1⁄10 to ¼ of the width of the wheel for each revolution of the workpiece.

Depth of Cut In the roughing operation, the depth of cut should be all the wheel will stand. This varies with the hardness of the material and the diameter of the workpiece. In the finishing operation, the depth of cut is always small: 0.0005 to 0.001 in (0.013 to 0.025 mm). Good results as regards finish are obtained by letting the wheel run over the workpiece several times without cross-feeding.

Grinding Allowances From 0.005 to 0.040 in (0.13 to 1 mm) is generally removed from the diameter in rough grinding in a cylindrical machine. For finishing, 0.002 to 0.010 in (0.05 to 0.25 mm) is common. Workpieces can be finished by grinding to a tolerance of 0.0002 in (0.005 mm) and a surface roughness of 50+ μin R_q (1.2 μm).

In situations where grinding leaves unfavorable surface **residual stresses,** the technique of **gentle** or **low-stress** grinding may be employed. This generally consists of removing a layer of about 0.010 in (0.25 mm) at depths of cut of 0.0002 to 0.0005 in (0.005 to 0.013 mm) with wheel speeds that are lower than the conventional 5,500 to 6,500 ft/min.

Truing and Dressing In **truing,** a diamond supported in the end of a soft steel rod held rigidly in the machine is passed over the face of the wheel two or three times to remove just enough material to give the wheel its true geometric shape. **Dressing** is a more severe operation of removing the dull or loaded surface of the wheel. Abrasive sticks or wheels or steel star wheels are pressed against and moved over the wheel face.

Safety If not stored, handled, and used properly, a grinding wheel can be a very dangerous tool. Because of its mass and high rotational speed, a grinding wheel has considerable energy and, if it fractures, it can cause serious injury and even death to the operator or to personnel nearby. A safety code B7.1 entitled "The Use, Care, and Protection of Abrasive Wheels" is available from ANSI; other safety literature is available from the Grinding Wheel Institute and from the National Safety Council.

The salient features of safety in the use of grinding wheels may be listed as follows: Wheels should be stored and handled carefully; a wheel that has been dropped should not be used. Before it is mounted on the machine, a wheel should be visually inspected for possible cracks; a simple "ring" test may be employed whereby the wheel is tapped gently with a light nonmetallic implement and if it sounds cracked, it should not be used. The wheel should be mounted properly with the required blotters and flanges, and the mounting nut tightened not excessively. The label on the wheel should be read carefully for maximum operating speed and other instructions. An appropriate guard should always be used with the machine, whether portable or stationary.

Newly mounted wheels should be allowed to run idle at the operating speed for at least 1 min before grinding. The operator should always wear safety glasses and should not stand directly in front of a grinding wheel when a grinder is started. If a grinding fluid is used, it should be turned off first before stopping the wheel to avoid creating an out-of-balance condition. Because for each type of operation and workpiece material there usually is a specific type of wheel recommended, the operator must make sure that the appropriate wheel has been selected.

Grinders

Grinding machines may be classified as to purpose and type as follows: for **rough removal of stock,** the swinging-frame, portable, flexible shaft, two-wheel stand, and disk; **cutting off** or parting, the cutting-off machine; **surface finishing,** band polisher, two-wheel combination, two-wheel polishing machine, two-wheel buffing machine, and semiautomatic polishing and buffing machine; **precision grinding,** tool post, cylindrical (plain and universal), crankshaft, centerless, internal, and surface (reciprocating table with horizontal or vertical wheel spindle, and rotary table with horizontal or vertical wheel spindle); **special form** grinders, gear or worm, ball-bearing balls, cams, and threads; and **tool and cutter** grinders for single-point tools, drills, and milling cutters, reamers, taps, dies, knives, etc.

Grinding equipment of all types (many with computer controls) has been improved during the past few years so as to be more rigid, provide more power to the grinding wheel, and provide automatic cycling, loading, clamping, wheel dressing, and automatic feedback.

Centerless grinders are used to good advantage where large numbers of relatively small pieces must be ground and where the ground surface has no exact relation to any other surface except as a whole; the work is carried on a support between two abrasive wheels, one a normal grinding wheel, the second a rubber-bonded wheel, rotating at about 1⁄20th the grinding speed, and is tilted 3 to 8° to cause the work to rotate and feed past the grinding wheel (see also Sec. 6).

The **cylindrical grinder** is a companion machine to the engine lathe; shafts, cylinders, rods, studs, and a wide variety of other cylindrical parts are first roughed out on the lathe, then finished accurately to size by the cylindrical grinder. The work is carried on centers, rotated slowly, and traversed past the face of a grinding wheel.

Universal grinders are cylindrical machines arranged with a swiveling table so that both straight and taper internal and external work can be ground. **Drill grinders** are provided with rests so mounted that by a simple swinging motion, correct cutting angles are produced automatically on the lips of drills; a cupped wheel is usually employed. **Internal grinders** are used for finishing the holes in bushings, rolls, sleeves, cutters, and the like; spindle speeds from 15,000 to 30,000 r/min are common.

Horizontal surface grinders range from small capacity, used mainly in tool making or small production work, to large sizes used for production work.

Vertical surface grinders are used for producing flat surfaces on production work. **Vertical** and **horizontal disk grinders** are used for surfacing. Grinding machines are used for **cutting off** steel, especially tubes, structural shapes, and hard metals. A thin resinoid or rubber-bonded wheel is used, with aluminum oxide abrasive for all types of steel, aluminum, brass, bronze, nickel, Monel, and Stellite; silicon carbide for cast iron, copper, carbon, glass, stone, plastics, and other nonmetallic materials; and diamond for cemented carbides and ceramics.

Belt grinders use a coated abrasive belt running between pulleys. Belt grinding is generally considered to be a roughing process, but finer finishes may be obtained by using finer grain size. Belt speeds generally range from 2,000 to 10,000 ft/min (600 to 3,000 m/min) with grain sizes ranging between 24 and 320, depending on the workpiece material and the surface finish desired. The process has the advantage of high-speed material removal and is applied to flat as well as irregular surfaces.

Although grinding is generally regarded as a finishing operation, it is possible to increase the rate of stock removal whereby the process becomes, in certain instances, competitive with milling. This type of grinding operation is usually called **creep-feed grinding.** It uses equipment such as reciprocating table or vertical-spindle rotary table surface grinders with capacities up to 300 hp (220 kW). The normal stock removal may range up to ¼ in (6.4 mm) with wheel speeds between 3,400 and 5,000 surface ft/min (1,000 and 1,500 m/min).

Finishing Operations

Polishing is an operation by which scratches or tool marks or, in some instances, rough surfaces left after forging, rolling, or similar operations are removed. It is not a precision operation. The nature of the polishing process has been debated for a long time. Two mechanisms appear to play a role: One is fine-scale abrasion, and the other is softening of surface layers. In addition to removal of material by the abrasive particles, the high temperatures generated because of friction soften the asperities of the surface of the workpiece, resulting in a smeared surface layer. Furthermore, chemical reactions may also take place in polishing whereby surface irregularities are removed by chemical attack.

Polishing is usually done in stages. The first stage is rough polishing, using abrasive grain sizes of about 36 to 80, followed by a second stage, using an abrasive size range of 80 to 120, a third stage of size 150 and finer, etc., with a final stage of buffing. For the first two steps the polishing wheels are used dry. For finishing, the wheels are first worn down a little and then coated with tallow, oil, beeswax, or similar substances. This step is partly polishing and partly buffing, as additional abrasive is often added in cake form with the grease. The cutting action is freer, and the life of the wheel is prolonged by making the wheel surface flexible. Buffing wheels are also used for the finishing step when tallow, etc., containing coarse or fine abrasive grains is periodically rubbed against the wheel.

Polishing wheels consisting of wooden disks faced with leather, turned to fit the form of the piece to be polished, are used for flat surfaces or on work where it is necessary to maintain square edges. A large variety of other types of wheels are in common use. Compress wheels are used extensively and are strong, durable, and easily kept in balance. They consist of a steel center the rim of which holds a laminated surface of leather, canvas, linen, felt, rubber, etc., of various degrees of pliability. Wheels of solid leather disks of walrus hide, buffalo hide, sheepskin, or bull's neck hide, or of soft materials such as felt, canvas, and muslin, built up of disks either loose, stitched, or glued, depending on the resiliency or pliability required, are used extensively for polishing as well as buffing. **Belts** of cloth or leather are often charged with abrasive for polishing flat or other workpieces. Wire brushes may be used with no abrasive for a final operation to give a satin finish to nonferrous metals.

For most polishing operations speeds range from 5,000 to 7,500 surface ft/min (1,500 to 2,250 m/min). The higher range is for high-strength steels and stainless steels. Excessively high speeds may cause burning of the workpiece and glazing.

Mirrorlike finishes may be obtained by **electropolishing,** a process that is the reverse of electroplating; it is particularly useful for polishing irregularly shaped workpieces which otherwise would be difficult to polish uniformly. A more recent specialized process is **magnetic-field polishing,** in which fine abrasive polishing particles are suspended in a magnetic fluid. This process is effective for polishing ceramic balls, such as ball bearings. The **chemical-mechanical polishing** process combines the actions of abrasive particles, suspended in a water-based solution, with a chemistry selected to cause controlled corrosion. It produces exceptionally flat surfaces with very fine surface finish, particularly important in polishing silicon wafers for the semiconductor industry.

Buffing is a form of finish polishing in which the surface finish is improved; very little material is removed. The powdered abrasives are applied to the surface of the wheel by pressing a mixture of abrasive and tallow or wax against the face for a few seconds. The abrasive is replenished periodically. The wheels are made of a soft pliable material, such as soft leather, felt, linen, or muslin, and rotated at high speed.

A variety of buffing compounds are available: aluminum oxide, chromium oxide, soft silica, rouge (iron oxide), pumice, lime compounds, emery, and crocus. In cutting down nonferrous metals, Tripoli is used; and for steels and stainless steels, aluminum oxide is the common abrasive. For coloring, soft silica, rouge, and chromium oxide are the more common compounds used. Buffing speeds range from 6,000 to 10,000 surface ft/min (1,800 to 3,000 m/min); the higher speeds are for steels, although the speed may be as high at 12,000 surface ft/min (3,600 m/min) for coloring brass and copper.

Lapping is a process of producing extremely smooth and accurate surfaces by rubbing the surface which is to be lapped against a mating form which is called a **lap.** The lap may either be charged with a fine abrasive and moistened with oil or grease, or the fine abrasive may be introduced with the oil. If a part is to be lapped to a final accurate dimension, a mating form of a softer material such as soft close-grained cast iron, copper, brass, or lead is made up. Aluminum oxide, silicon carbide, and diamond grits are used for lapping. Lapping requires considerable time. No more than 0.0002 to 0.0005 in (0.005 to 0.013 mm) should be left for removal by this method. Surface plates, rings, and plugs are common forms of laps. For most applications grit sizes range between 100 and 800, depending on the finish desired. For most efficient lapping, speeds generally range from 300 to 800 surface ft/min (150 to 240 m/min) with pressures of 1 to 3 lb/in^2 (7 to 21 kPa) for soft materials and up to 10 lb/in^2 (70 kPa) for harder materials.

Honing is an operation similar to lapping. Instead of a metal lap charged with fine abrasive grains, a honing stone made of fine abrasives is used. Small stones of various cross-sectional shapes and lengths are manufactured for honing the edges of cutting tools. Automobile cylinders are honed for fine finish and accurate dimensions. This honing usually follows a light-finish reaming operation or a precision-boring operation using diamonds or carbide tools. The tool consists of several honing stones adjustable at a given radius or forced outward by springs or a wedge forced mechanically or hydraulically and is given a reciprocating (25 to 40 per min) and a rotating motion (about 300 r/min) in the cylinder which is flooded with kerosine.

Hones operate at speeds of 50 to 200 surface ft/min (15 to 60 m/min) and use universal joints to allow the tool to center itself in the workpiece. The automatic pressure-cycle control of hone expansion, in which the pressure is reduced in steps as the final finish is reached, removes metal 10 times as fast as with the spring-expanded hone. Rotational and reciprocating movements are provided to give an uneven ratio and thus prevent an abrasive grain from ever traversing its own path twice.

Superfinishing is a honing process. Formed honing stones bear against the workpiece previously finished to 0.0005 in (0.013 mm) or at the most to 0.001 in (0.025 mm) by a very light pressure which gradually increases to several pounds per square inch (1 lb/in^2 = 0.0069 MPa) of stone area in proportion to the development of the increased area of contact between the workpiece and stone. The workpiece or tool rotates and where possible is reciprocated slowly over the surface which may be finished in a matter of 20 s to a surface quality of 1 to 3 μin (0.025 to 0.075 μm). Superfinishing is applied to many types of workpieces such as crankshaft pins and bearings, cylinder bores, pistons, valve stems, cams, and other metallic moving parts.

Deburring involves removing burrs (thin ridges, generally triangular, resulting from operations such as punching and blanking of sheet metals, and from machining and drilling) along the edges of a workpiece. Several deburring operations are available, the most common ones being manual filing, wire brushing, using abrasives (emery paper, belts, abrasive blasting), and vibratory and barrel finishing. Deburring operations can also be carried out using programmable robots.

MACHINING AND GRINDING OF PLASTICS

The low strength of thermoplastics permits high cutting speeds and feeds, but the low heat conductivity and greater resilience require increased reliefs and less rake in order to avoid undersize cutting. Hard and sharp tools should be used. Plastics are usually abrasive and cause the tools to wear or become dull rapidly. Dull tools generate heat and cause the tools to cut to shallow depths. The depth of cut should be small. When high production justifies the cost, diamond turning and boring tools are used. Diamond tools maintain sharp cutting edges and produce an excellent machined surface. They are particularly advantageous when a more abrasive plastic such as reinforced plastic is machined.

A cutting fluid, such as a small blast of air or a stream of water, improves the **turning** and cutting of plastics as it prevents the heating of the tool and causes the chips to remain brittle and to break rather than become sticky and gummy. A zero or slightly negative back rake and a relief angle of 8 to 12° should be used. For thermoplastics cutting speeds generally range from 250 to 400 ft/min (75 to 120 m/min) and for thermosetting plastics from 400 to 1,000 ft/min (120 to 300 m/min). Recommended tool materials are M2 and T5 high-speed steels and C2 carbide.

In **milling** plastics, speeds of 400 to 1,000 ft/min (120 to 300 m/min) should be used, with angles similar to those on a single-point tool. From 0 to 10° negative rake may be used. Good results have been obtained by hobbing plastic gears with carbide-tipped hobs. Recommended tool materials are M2 and M7 high-speed steels and C2 carbide.

In **drilling,** speeds range from 150 to 400 ft/min (45 to 120 m/min), and the recommended drill geometry is given in Table 13.4.6. Tool materials are M1, M7, and M10 high-speed steel. Usually the drill cuts undersize; drills 0.002 to 0.003 in (0.05 to 0.075 mm) oversize should be used.

In **sawing** plastics, either precision or buttress tooth forms may be used, with a pitch ranging from 3 to 14 teeth/in (1.2 to 5.5 teeth/cm), the thicker the material the lower the number of teeth per unit length of saw. Cutting speeds for thermoplastics range from 1,000 to 4,000 ft/min (300 to 1,200 m/min) and for thermosetting plastics from about 3,000 to 5,500 ft/min (900 to 1,700 m/min), with the higher speeds for thinner stock. High-carbon-steel blades are recommended. An air blast is helpful in preventing the chips from sticking to the saw. Abrasive saws operating at 3,500 to 6,000 ft/min (1,000 to 1,800 m/min) are also used for cutting off bars and forms.

Plastics are tapped and threaded with standard tools. Ground M1, M7, or M10 high-speed steel **taps** with large polished flutes are recommended. **Tapping speeds** are usually from 25 to 50 ft/min (8 to 15 m/min); water serves as a good cutting fluid as it keeps the material brittle and prevents sticking in the flutes. **Thread cutting** is generally accomplished with tools similar to those used on brass.

Reaming is best accomplished in production by using tools of the expansion or adjustable type with relatively low speeds but high feeds. Less material should be removed in reaming plastics than in reaming other materials.

Polishing and **buffing** are done on many types of plastics. Polishing is done with special compounds containing wax or a fine abrasive. Buffing wheels for plastics should have loose stitching. Vinyl plastics can be buffed and polished with fabric wheels of standard types, using light pressures.

Thermoplastics and thermosets can be **ground** with relative ease, usually by using silicon carbide wheels. As in machining, temperature rise should be minimized.

MACHINING AND GRINDING OF CERAMICS

The technology of machining and grinding of ceramics, as well as composite materials, has advanced rapidly, resulting in good surface characteristics and product integrity. Ceramics can be machined with carbide, high-speed steel, or diamond tools, although care should be exercised because of the brittle nature of ceramics and the resulting possible surface damage. **Machinable ceramics** have been developed which minimize machining problems. Grinding of ceramics is usually done with diamond wheels.

ADVANCED MACHINING PROCESSES

In addition to the mechanical methods of material removal described above, there are a number of other important processes which may be preferred over conventional methods. Among the important factors to be considered are the hardness of the workpiece material, the shape of the part, its sensitivity to thermal damage, residual stresses, tolerances, and economics. Some of these processes produce a heat-affected layer on the surface; improvements in surface integrity may be obtained by postprocessing techniques such as polishing or peening. Almost all machines are now computer-controlled.

Electric-discharge machining (EDM) is based on the principle of erosion of metals by spark discharges. Figure 13.4.20 gives a schematic diagram of this process. The spark is a transient electric discharge through the space between two charged electrodes, which are the tool and the workpiece. The discharge occurs when the potential difference between the tool and the workpiece is large enough to cause a breakdown in the medium (which is called the **dielectric fluid** and is usually a hydrocarbon) and to procure an electrically conductive spark channel. The breakdown potential is usually established by connecting the two electrodes to the terminals of a capacitor charged from a power source. The spacing between the tool and workpiece is critical; therefore, the feed is controlled by servomechanisms. The dielectric fluid has the additional functions of providing a cooling medium and carrying away particles produced by the electric discharge. The discharge is repeated rapidly, and each time a minute amount of workpiece material is removed.

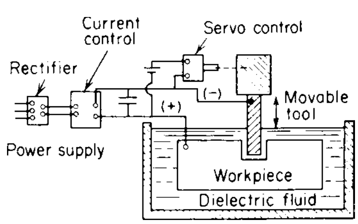


Fig. 13.4.20 Schematic diagram of the electric-discharge machining process.

The rate of metal removal depends mostly on the average current in the discharge circuit; it is also a function of the electrode characteristics, the electrical parameters, and the nature of the dielectric fluid. In practice, this rate is normally varied by changing the number of discharges per second or the energy per discharge. Rates of metal removal may range from 0.01 to 25 in^3/h (0.17 to 410 cm^3/h), depending on surface finish and tolerance requirements. In general, higher rates produce rougher surfaces. Surface finishes may range from 1,000 μin R_q (25 μm) in roughing cuts to less than 25 μin (0.6 μm) in finishing cuts.

The response of materials to this process depends mostly on their thermal properties. Thermal capacity and conductivity, latent heats of melting and vaporization are important. Hardness and strength do not necessarily have significant effect on metal removal rates. The process is applicable to all materials which are sufficiently good conductors of electricity. The tool has great influence on permissible removal rates. It is usually made of graphite, copper-tungsten, or copper alloys. Tools have been made by casting, extruding, machining, powder metallurgy, and other techniques and are made in any desired shape. Tool wear is an important consideration, and in order to control tolerances and minimize cost, the ratio of tool material removed to workpiece material removed should be low. This ratio varies with different tool and workpiece material combinations and with operating conditions. Therefore, a particular tool material may not be best for all workpieces. Tolerances as low as 0.0001 to 0.0005 in (0.0025 to 0.0127 mm) can be held with slow metal removal rates. In machining some steels, tool wear can be minimized by reversing the polarity and using copper tools. This is known as "no-wear EDM."

The electric-discharge machining process has numerous applications, such as machining cavities and dies, cutting small-diameter holes, blanking parts from sheets, cutting off rods of materials with poor machinability, and flat or form grinding. It is also applied to sharpening tools, cutters, and broaches. The process can be used to generate almost any geometry if a suitable tool can be fabricated and brought into close proximity to the workpiece.

Thick plates may be cut with **wire EDM** (Fig. 13.4.21). A slowly moving wire travels a prescribed path along the workpiece and cuts the metal with the sparks acting like saw teeth. The wire, usually about 0.01 in (0.25 mm) in diameter, is made of brass, copper, or tungsten and is generally used only once. The process is also used in making tools and dies from hard materials, provided that they are electrically conducting.

Electric-discharge grinding (EDG) is similar to the electric-discharge machining process with the exception that the electrode is in the form of

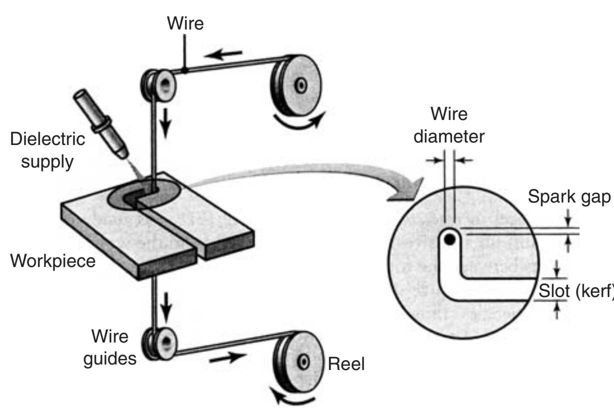


Fig. 13.4.21 Schematic diagram of the wire EDM process.

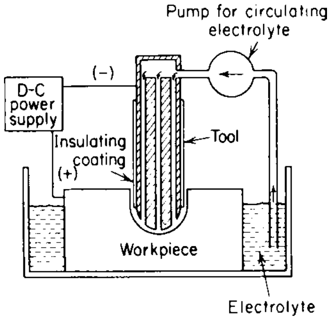


Fig. 13.4.22 Schematic diagram of the electrochemical machining process.

a grinding wheel. Removal rates are up to 1.5 in^3/h (25 cm^3/h) with practical tolerances on the order of 0.001 in (0.025 mm). A graphite or brass electrode wheel is operated around 100 to 600 surface ft/min (30 to 180 m/min) to minimize splashing of the dielectric fluid. Typical applications of this process are in grinding of carbide tools and dies, thin slots in hard materials, and production grinding of intricate forms.

The **electrochemical machining (ECM)** process (Fig. 13.4.22) uses electrolytes which dissolve the reaction products formed on the workpiece by electrochemical action; it is similar to a reverse electroplating process. The electrolyte is pumped at high velocities through the tool. A gap of 0.005 to 0.020 in (0.13 to 0.5 mm) is maintained. A dc power supply maintains very high current densities between the tool and the workpiece. In most applications, a current density of 1,000 to 5,000 A is required per in^2 of active cutting area. The rate of metal removal is proportional to the amount of current passing between the tool and the workpiece. Removal rates up to 1 in^3/min (16 cm^3/min) can be obtained with a 10,000-A power supply. The penetration rate is proportional to the current density for a given workpiece material.

The process leaves a burr-free surface. It is also a cold machining process and does no thermal damage to the surface of the workpiece. Electrodes are normally made of brass or copper; stainless steel, titanium, sintered copper-tungsten, aluminum, and graphite have also been used. The electrolyte is usually a sodium chloride solution up to 2.5 lb/gal (300 g/L); other solutions and proprietary mixtures are also available. The amount of overcut, defined as the difference between hole diameter and tool diameter, depends upon cutting conditions. For production applications, the average overcut is around 0.015 in (0.4 mm). The rate of penetration is up to 0.750 in/min (20 mm/min).

Very good surface finishes may be obtained with this process. However, sharp square corners or sharp corners and flat bottoms cannot be machined to high accuracies. The process is applied mainly to round or odd-shaped holes with straight parallel sides. It is also applied to cases where conventional methods produce burrs which are costly to remove. The process is particularly economical for materials with a hardness above 400 HB.

The **electrochemical grinding (ECG)** process (Fig. 13.4.23) is a combination of electrochemical machining and abrasive cutting where most of the metal removal results from the electrolytic action. The process consists of a rotating cathode, a neutral electrolyte, and abrasive particles in contact with the workpiece. The equipment is similar to a

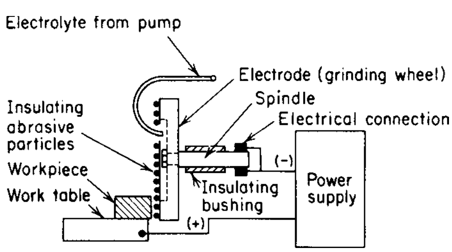


Fig. 13.4.23 Schematic diagram of the electrochemical grinding process.

conventional grinding machine except for the electrical accessories. The cathode usually consists of a metal-bonded diamond or aluminum oxide wheel. An important function of the abrasive grains is to maintain a space for the electrolyte between the wheel and workpiece.

Surface finish, precision, and metal-removal rate are influenced by the composition of the electrolyte. Aqueous solutions of sodium silicate, borax, sodium nitrate, and sodium nitrite are commonly used as electrolytes. The process is primarily used for tool and cutter sharpening and for machining of high-strength materials.

A combination of the electric-discharge and electrochemical methods of material removal is known as **electrochemical discharge grinding (ECDG).** The electrode is a pure graphite rotating wheel which electrochemically grinds the workpiece. The intermittent spark discharges remove oxide films that form as a result of electrolytic action. The equipment is similar to that for electrochemical grinding. Typical applications include machining of fragile parts and resharpening or form grinding of carbides and tools such as milling cutters.

In **chemical machining (CM)** material is removed by chemical or electrochemical dissolution of preferentially exposed surfaces of the workpiece. Selective attack on different areas is controlled by masking or by partial immersion. There are two processes involved: chemical **milling** and chemical **blanking.** Milling applications produce shallow cavities for overall weight reduction, and are also used to make tapered sheets, plates, or extrusions. Masking with paint or tapes is common. Masking materials may be elastomers (such as butyl rubber, neoprene, and styrene-butadiene) or plastics (such as polyvinyl chloride, polystyrene, and polyethylene). Typical blanking applications are decorative panels, printed-circuit etching, and thin stampings. Etchants are solutions of sodium hydroxide for aluminum, and solutions of hydrochloric and nitric acids for steel.

Ultrasonic machining (USM) is a process in which a tool is given a high-frequency, low-amplitude oscillation, which, in turn, transmits a high velocity to fine abrasive particles that are present between the tool and the workpiece. Minute particles of the workpiece are chipped away on each stroke. Aluminum oxide, boron carbide, or silicone carbide grains are used in a water slurry (usually 50 percent by volume), which also carries away the debris. Grain size ranges from 200 to 1,000 (see Sec. 6 and Figs. 13.4.18 and 13.4.19).

The equipment consists of an electronic oscillator, a transducer, a connecting cone or toolholder, and the tool. The oscillatory motion is obtained most conveniently by magnetostriction, at approximately 20,000 Hz and a stroke of 0.002 to 0.005 in (0.05 to 0.13 mm). The tool material is normally cold-rolled steel or stainless steel and is brazed, soldered, or fastened mechanically to the transducer through a toolholder. The tool is ordinarily 0.003 to 0.004 in (0.075 to 0.1 mm) smaller than the cavity it produces. Tolerances of 0.0005 in (0.013 mm) or better can be obtained with fine abrasives. For best results, roughing cuts should be followed with one or more finishing operations with finer grits. The ultrasonic machining process is used in drilling holes, engraving, cavity sinking, slicing, broaching, etc. It is best suited to materials which are hard and brittle, such as ceramics, carbides, borides, ferrites, glass, precious stones, and hardened steels.

In **water jet machining (WJM),** water is ejected from a nozzle at pressures as high as 200,000 lb/in^2 (1,400 MPa) and acts as a saw. The process is suitable for cutting and deburring of a variety of materials such as polymers, paper, and brick in thicknesses ranging from 0.03 to 1 in (0.8 to 25 mm) or more. The cut can be started at any location, wetting is minimal, and no deformation of the rest of the piece takes place. Abrasives can be added to the water stream to increase material removal rate, and this is known as **abrasive water jet machining (AWJM).**

In **abrasive-jet machining** (AJM), material is removed by fine abrasive particles (aluminum oxide or silicon carbide) carried in a high-velocity stream of air, nitrogen, or carbon dioxide. The gas pressure ranges up to 120 lb/in^2 (800 kPa), providing a nozzle velocity of up to 1,000 ft/s (300 m/s). Nozzles are made of tungsten carbide or sapphire. Typical applications are in drilling, sawing, slotting, and deburring of hard, brittle materials such as glass.

In **laser-beam machining (LBM),** material is removed by converting electric energy into a narrow beam of light and focusing it on the workpiece. The high energy density of the beam is capable of melting and vaporizing all materials, and consequently, there is a thin heat-affected zone. The most commonly used laser types are CO_2 (pulsed or continuous-wave) and Nd:YAG. Typical applications include cutting a variety of metallic and nonmetallic materials, drilling (as small as 0.0002 in or 0.005 mm in diameter), and marking. The efficiency of cutting increases with decreasing thermal conductivity and reflectivity of the material. Because of the inherent flexibility of the process, programmable and computer-controlled laser cutting is now becoming important, particularly in cutting profiles and multiple holes of various shapes and sizes on large sheets. Cutting speeds may range up to 25 ft/ min (7.5 m/min).

The **electron-beam machining (EBM)** process removes material by focusing high-velocity electrons on the workpiece. Unlike lasers, this process is carried out in a vacuum chamber and is used for drilling small holes, scribing, and cutting slots in all materials, including ceramics.

13.5 SURFACE TEXTURE DESIGNATION, PRODUCTION, AND QUALITY CONTROL

by Ali M. Sadegh

REFERENCES: American National Standards Institute, "Surface Texture," ANSI/ASME B 46.1-1985, and "Surface Texture Symbols," ANSI Y 14.36-1978. Broadston, "Control of Surface Quality," Surface Checking Gage Co., Hollywood, CA. ASME, "Metals Engineering Design Handbook," McGraw-Hill SME, "Tool and Manufacturing Engineers Handbook," McGraw-Hill.

Rapid changes in the complexity and precision requirements of mechanical products since 1945 have created a need for improved methods of determining, designating, producing, and controlling the surface texture of manufactured parts. Although standards are aimed at standardizing methods for measuring by using stylus probes and electronic transducers for surface quality control, other descriptive specifications are sometimes required, i.e., interferometric light bands, peak-to-valley by optical sectioning, light reflectance by commercial glossmeters, etc. Other parameters are used by highly industrialized foreign countries to solve their surface specification problems. These include the high-spot counter and bearing area meter of England (Talysurf); the total peak-to-valley, or R_1, of Germany (Perthen); and the *R* or average amplitude of surface deviations of France. In the United States, peak counting is used in the sheet-steel industry, instrumentation is available (Bendix), and a standard for specification, SAE J-911, exists.

Surface texture control should be considered for many reasons, among them being the following:

1. Advancements in the technology of metal-cutting tools and machinery have made the production of higher-quality surfaces possible.
2. Products are now being designed that depend upon proper quality control of critical surfaces for their successful operation as well as for long, troublefree performance in service.
3. Remote manufacture and the necessity for controlling costs have made it preferable that finish requirements for all the critical surfaces of a part be specified on the drawing.
4. The design engineer, who best understands the overall function of a part and all its surfaces, should be able to determine the requirement for surface texture control where applicable and to use a satisfactory standardized method for providing this information on the drawing for use by manufacturing departments.
5. Manufacturing personnel should know what processes are able to produce surfaces within specifications and should be able to verify that the production techniques in use are under control.
6. Quality control personnel should be able to check conformance to surface texture specifications if product quality is to be maintained and product performance and reputation ensured.

DESIGN CRITERIA

Surfaces produced by various processes exhibit distinct differences in texture. These differences make it possible for honed, lapped, polished, turned, milled, or ground surfaces to be easily identified. As a result of its unique character, the surface texture produced by any given process can be readily compared with other surfaces produced by the same process through the simple means of comparing the average size of its irregularities, using applicable standards and modern measurement methods. It is then possible to predict and control its performance with considerable certainty by limiting the range of the average size of its characteristic surface irregularities. Surface texture standards make this control possible.

Variations in the texture of a critical surface of a part influence its ability to resist wear and fatigue; to assist or destroy effective lubrication; to increase or decrease its friction and/or abrasive action on other parts, and to resist corrosion, as well as affect many other properties that may be critical under certain conditions.

Clay has shown that the load-carrying capacity of nitrided shafts of varying degrees of roughness, all running at 1,500 r/min in diamond-turned lead-bronze bushings finished to 20 μin (0.50 μm), varies as shown in Fig. 13.5.1. The effects of roughness values on the friction between a flat slider on a well-lubricated rotating disk are shown in Fig. 13.5.2.

Surface texture control should be a normal design consideration under the following conditions:

1. For those parts whose roughness must be held within closely controlled limits for optimum performance. In such cases, even the process may have to be specified. Automobile engine cylinder walls, which should be finished to about 13 μin (0.32 μm) and have a circumferential (ground) or an angular (honed) lay, are an example. If too rough, excessive wear occurs, if too smooth, piston rings will not seat properly, lubrication is poor, and surfaces will seize or gall.
2. Some parts, such as antifriction bearings, cannot be made too smooth for their function. In these cases, the designer must optimize the tradeoff between the added costs of production and various benefits derived from added performance, such as higher reliability and market value.

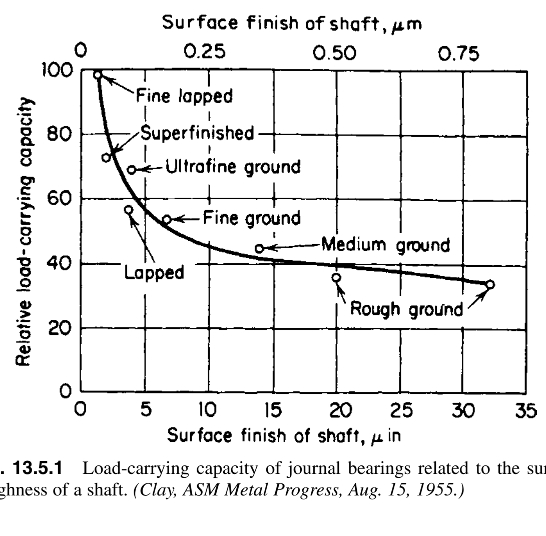


Fig. 13.5.1 Load-carrying capacity of journal bearings related to the surface roughness of a shaft. *(Clay, ASM Metal Progress, Aug. 15, 1955.)*

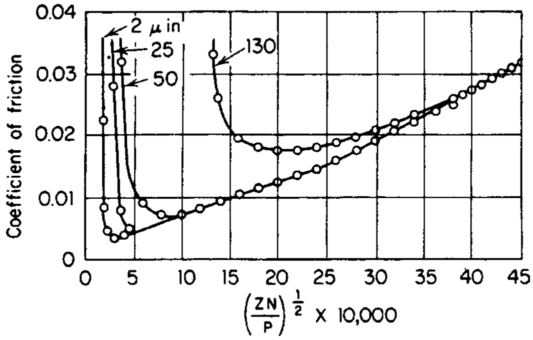


Fig. 13.5.2 Effect of surface texture on friction with hydrodynamic lubrication using a flat slider on a rotating disk. Z = oil viscosity, cP; N = rubbing speed, ft/min; P = load, lb/in^2.

3. There are some parts where surfaces must be made as smooth as possible for optimum performance regardless of cost, such as gages, gage blocks, lenses, and carbon pressure seals.

4. In some cases, the nature of the most satisfactory finishing process may dictate the surface texture requirements to attain production efficiency, uniformity, and control even though the individual performance of the part itself may not be dependent on the quality of the controlled surface. Hardened steel bushings, e.g., which must be ground to close tolerance for press fit into housings, could have outside surfaces well beyond the roughness range specified and still perform their function satisfactorily.

5. For parts which the shop, with unjustified pride, has traditionally finished to greater perfection than is necessary, the use of proper surface texture designations will encourage rougher surfaces on exterior and other surfaces that do not need to be finely finished. Significant cost reductions will accrue thereby.

It is the designer's responsibility to decide which surfaces of a given part are critical to its design function and which are not. This decision should be based upon a full knowledge of the part's function as well as of the performance of various surface textures that might be specified. From both a design and an economic standpoint, it may be just as unsound to specify too smooth a surface as to make it too rough—or to control it at all if not necessary. Wherever normal shop practice will produce acceptable surfaces, as in drilling, tapping, and threading, or in keyways, slots, and other purely functional surfaces, unnecessary surface texture control will add costs which should be avoided.

Whereas each specialized field of endeavor has its own traditional criteria for determining which surface finishes are optimum for adequate performance, Table 13.5.1 provides some common examples for design review, and Table 13.5.6 provides data on the surface texture ranges that can be obtained from normal production processes.

DESIGNATION STANDARDS, SYMBOLS, AND CONVENTIONS

The precise definition and measurement of surface texture irregularities of machined surfaces are almost impossible because the irregularities are very complex in shape and character and, being so small, do not lend themselves to direct measurement. Although both their shape and length may affect their properties, control of their average height and direction usually provides sufficient control of their performance. The standards do not specify the surface texture suitable for any particular application, nor the means by which it may be produced or measured. Neither are the standards concerned with other surface qualities such as appearance, luster, color, hardness, microstructure, or corrosion and wear resistance, any of which may be a governing design consideration.

The standards provide definitions of the terms used in delineating critical surface-texture qualities and a series of symbols and conventions suitable for their designation and control. In the discussion which follows, the reference standards used are "Surface Texture" (ANSI/ ASME B46.1-1985) and "Surface Texture Symbols" (ANSI Y 14.36-1978).

The basic ANSI symbol for designating surface texture is the checkmark with horizontal extension shown in Fig. 13.5.3. The symbol with the triangle at the base indicates a requirement for a machining allowance, in preference to the old *f* symbol. Another, with the small circle in the base, prohibits machining; hence surfaces must be produced without the removal of material by processes such as cast, forged, hot- or cold-finished, die-cast, sintered- or injection-molded, to name a few. The surface-texture requirement may be shown at A; the machining allowance at B; the process may be indicated above the line at C;

Table 13.5.1 Typical Surface Texture Design Requirements

(250 μin)	6.3 √	Clearance surfaces Rough machine parts
(125 μin)	3.2 √	Mating surfaces (static) Chased and cut threads Clutch-disk faces Surfaces for soft gaskets
(63 μin)	1.60 √	Piston-pin bores Brake drums Cylinder block, top Gear locating faces Gear shafts and bores Ratchet and pawl teeth Milled threads Rolling surfaces Gearbox faces Piston crowns Turbine-blade dovetails
(32 μin)	0.80 √	Broached holes Bronze journal bearings Gear teeth Slideways and gibs Press-fit parts Piston-rod bushings Antifriction bearing seats Sealing surfaces for hydraulic tube fittings
(16 μin)	0.40 √	Motor shafts Gear teeth (heavy loads) Spline shafts O-ring grooves (static) Antifriction bearing bores and faces Camshaft lobes Compressor-blade airfoils Journals for elastomer lip seals
(13 μin)	0.32 √	Engine cylinder bores Piston outside diameters Crankshaft bearings
(8 μin)	0.20 √	Jet-engine stator blades Valve-tappet cam faces Hydraulic-cylinder bores Lapped antifriction bearings
(4 μin)	0.10 √	Ball-bearing races Piston pins Hydraulic piston rods Carbon-seal mating surfaces
(2 μin)	0.050 √	Shop-gage faces Comparator anvils
(1 μin)	0.025 √	Bearing balls Gages and mirrors Micrometre anvils

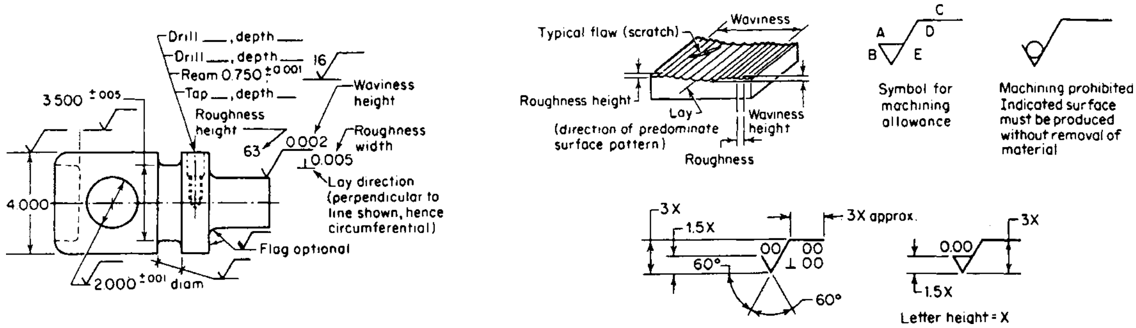


Fig. 13.5.3 Application and use of surface texture symbols.

the roughness width cutoff (sampling length) at D, and the lay at E. The ANSI symbol provides places for the insertion of numbers to specify a wide variety of texture characteristics, as shown in Table 13.5.2.

Control of **roughness,** the finely spaced surface texture irregularities resulting from the manufacturing process or the cutting action of tools or abrasive grains, is the most important function accomplished through the use of these standards, because roughness, in general, has a greater effect on performance than any other surface quality. The roughness-height index value is a number which equals the arithmetic average deviation of the minute surface irregularities from a hypothetical perfect surface, expressed in either millionths of an inch (microinches, μin, 0.000001 in) or in micrometres, μm, if drawing dimensions are in metric, SI units. For control purposes, roughness-height values are taken from Table 13.5.3, with those in boldface type given preference.

The term *roughness cutoff,* a characteristic of tracer-point measuring instruments, is used to limit the length of trace within which the asperities of the surface must lie for consideration as roughness. Asperity spacings greater than roughness cutoff are then considered as waviness.

Waviness refers to the secondary irregularities upon which roughness is superimposed, which are of significantly longer wavelength and are usually caused by machine or work deflections, tool or workpiece vibration, heat treatment, or warping. Waviness can be measured by a dial indicator or a profile recording instrument from which roughness has been filtered out. It is rated as maximum peak-to-valley distance and is indicated by the preferred values of Table 13.5.4. For fine waviness control, techniques involving contact-area determination in percent (90, 75, 50 percent preferred) may be required. Waviness control by interferometric methods is also common, where notes, such as "Flat within XX helium light bands," may be used. Dimensions may be determined from the precision length table (see Sec. 1).

Lay refers to the direction of the predominant visible surface roughness pattern. It can be controlled by use of the approved symbols given in Table 13.5.5, which indicate desired lay direction with respect to the boundary line of the surface upon which the symbol is placed.

Flaws are imperfections in a surface that occur only at infrequent intervals. They are usually caused by nonuniformity of the material, or they result from damage to the surface subsequent to processing, such as scratches, dents, pits, and cracks. Flaws should not be considered in surface texture measurements, as the standards do not consider or classify them. Acceptance or rejection of parts having flaws is strictly a matter of judgment based upon whether the flaw will compromise the intended function of the part.

To call attention to the fact that surface texture values are specified on any given drawing, a note and typical symbol may be used as follows:

√ Surface texture per ANSI B46.1

Values for nondesignated surfaces can be limited by the note

$\overset{xx}{\surd}$ All machined surfaces except as noted

MEASUREMENT

Two general methods exist to measure surface texture: **profile methods** and **area methods.** Profile methods measure the contour of the surface in a plane usually perpendicular to the surface. Area methods measure an area of a surface and produce results that depend on area-averaged properties.

Table 13.5.2 Application of Surface Texture Values to Surface Symbols

(63) 1.6√	Roughness average rating is placed at the left of the long leg. The specification of only one rating shall indicate the maximum value and any lesser value shall be acceptable. Specify in micrometres (microinches).	(63) 1.6 / 3.5 √ (machining allowance)	Machining is required to produce the surface. The basic amount of stock provided for machining is specified at the left of the short leg of the symbol. Specify in millimetres (inches).
(63) 1.6 / (32) 0.8√	The specification of maximum value and minimum value roughness average ratings indicates permissible range of value rating. Specify in micrometres (microinches).	(63) 1.6 √ (machining prohibited)	Removal of material by machining is prohibited.
		(32) 0.8√ ⊥	Lay designation is indicated by the lay symbol placed at the right of the long leg.
(32) 0.8√ 0.05	Maximum waviness height rating is placed above the horizontal extension. Any lesser rating shall be acceptable. Specify in millimetres (inches).	(32) 0.8√ 2.5 (0.100)	Roughness sampling length or cutoff rating is placed below the horizontal extension. When no value is shown, 0.80 mm is assumed. Specify in millimetres (inches).
(32) 0.8√ 0.05 – 100	Maximum waviness spacing rating is placed above the horizontal extension and to the right of the waviness height rating. Any lesser rating shall be acceptable. Specify in millimetres (inches).	(32) 0.8√ ⊥0.5	Where required, maximum roughness spacing shall be placed at the right of the lay symbol. Any lesser rating shall be acceptable. Specify in millimetres (inches).

Table 13.5.3 Preferred Series Roughness Average Values R_a, μm and μin

μm	μin	μm	μin	μm	μin	μm	μin	μm	μin
0.012	0.5	0.125	5	0.50	20	2.00	80	8.0	320
0.025	1	0.15	6	0.63	25	2.50	100	10.0	400
0.050	**2**	**0.20**	**8**	**0.80**	**32**	**3.20**	**125**	**12.5**	**500**
0.075	3	0.25	10	1.00	40	4.0	160	15.0	600
0.10	**4**	0.32	13	1.25	50	5.0	200	20.0	800
		0.40	**16**	**1.60**	**63**	**6.3**	**250**	**25.0**	**1,000**

Table 13.5.4 Preferred Series Maximum Waviness Height Values

mm	in	mm	in	mm	in
0.0005	0.00002	0.008	0.0003	0.12	0.005
0.0008	0.00003	0.012	0.0005	0.20	0.008
0.0012	0.00005	0.020	0.0008	0.25	0.010
0.0020	0.00008	0.025	0.001	0.38	0.015
0.0025	0.0001	0.05	0.002	0.50	0.020
0.005	0.0002	0.08	0.003	0.80	0.030

Another categorization is by **contact methods** and **noncontact methods.** Contact methods include stylus methods (tracer-point analysis) and capacitance methods. Noncontact methods include light section microscopy, optical reflection measurements, and interferometry.

Replicas of typical standard machined surfaces provide less accurate but often adequate reference and control of rougher surfaces with R_a over 16 μin.

The United States and 25 other countries have adopted the **roughness average R_a** as the standard measure of surface roughness. (See ANSI/ASME B46.1-1985.)

PRODUCTION

Various production processes can produce surfaces within the ranges shown in Table 13.5.6. For production efficiency, it is best that critical areas requiring surface texture control be clearly designated on drawings so that proper machining and adequate protection from damage during processing will be ensured.

SURFACE QUALITY VERSUS TOLERANCES

It should be remembered that surface quality and tolerances are distinctly different attributes that are controlled for completely separate purposes. **Tolerances** are established to limit the range of the size of a part at the time of manufacture, as measured with gages, micrometres,

Table 13.5.5 Lay Symbols

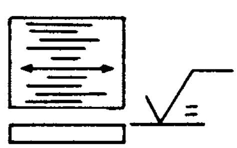
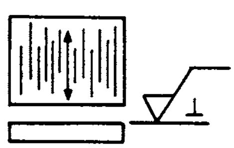
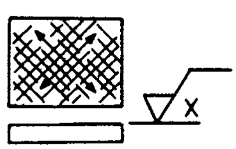
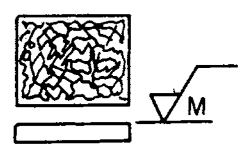
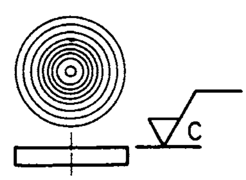
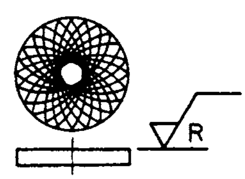
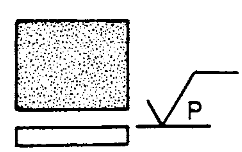
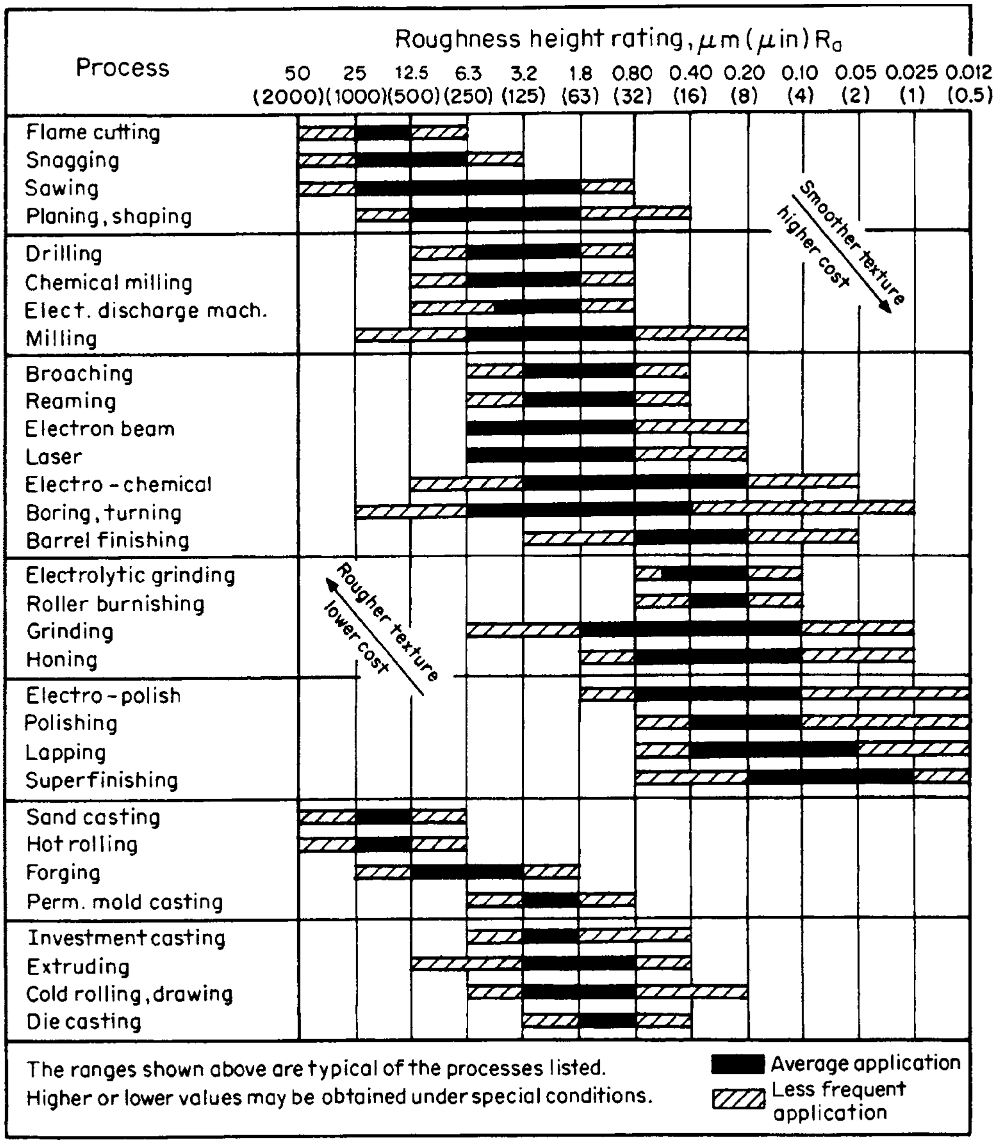
Lay symbol	Interpretation	Example showing direction of tool marks
=	Lay approximately parallel to the line representing the surface to which the symbol is applied	
⊥	Lay approximately perpendicular to the line representing the surface to which the symbol is applied	
X	Lay angular in both directions to line representing the surface to which symbol is applied	
M	Lay multidirectional	
C	Lay approximately circular relative to the center of the surface to which the symbol is applied	
R	Lay approximately radial relative to the center of the surface to which the symbol is applied	
P	Lay particulate, nondirectional, or protuberant	

Table 13.5.6 Surface-Roughness Ranges of Production Processes



or other traditional measuring devices having anvils that make contact with the part. **Surface quality** controls, on the other hand, serve to limit the minute surface irregularities or asperities that are formed by the manufacturing process. These lie under the gage anvils during measurement and **do not use up tolerances.**

QUALITY CONTROL (SIX SIGMA)

Quality control is a system that outlines the policies and procedures necessary to improve and control the various processes in manufacturing that will ultimately lead to improved business performance.

Six Sigma is a quality management program to achieve “six sigma” levels of quality. It was pioneered by Motorola in the mid-1980s and has spread to many other manufacturing companies. In statistics, sigma refers to the standard deviation of a set of data. Therefore, **six sigma** refers to six standard deviations. Likewise, **three sigma** refers to three standard deviations. In probability and statistics, the standard deviation is the most commonly used measure of statistical dispersion; i.e., it measures the degree to which values in a data set are spread. The standard deviation is defined as the square root of the variance, i.e., the root mean square (rms) deviation from the average. It is defined in this way to give us a measure of dispersion.

Assuming that defects occur according to a standard normal distribution, this corresponds to approximately 2 quality failures per million parts manufactured. In practical application of the six sigma methodology, however, the rate is taken to be 3.4 per million.

Initially, many believed that such high process reliability was impossible, and **three sigma** (67,000 defects per million opportunities, or DPMO) was considered acceptable. However, market leaders have measurably reached six sigma in numerous processes.