22 Cutting-Tool Technology

Chapter Contents

Machining operations are accomplished using cutting tools. The high forces and temperatures during machining create a very harsh environment for the tool. If cutting force becomes too high, the tool fractures. If cutting temperature becomes too high, the tool material softens and fails. If neither of these conditions causes the tool to fail, continual wear of the cutting edge ultimately leads to failure.

Cutting tool technology has two principal aspects: tool material and tool geometry. The first is concerned with developing materials that can withstand the forces, temperatures, and wearing action in the machining process. The second deals with optimizing the geometry of the cutting tool for the tool material and for a given operation. It is appropriate to begin by considering tool life, because this is a prerequisite for much of the subsequent discussion on tool materials. It also seems appropriate to include a section on cutting fluids at the end of this chapter; cutting fluids are often used in machining operations to prolong the life of a cutting tool.

22.1 Tool Life

As suggested by the opening paragraph, there are three possible modes by which a cutting tool can fail in machining:

1. ***Fracture failure***. This mode of failure occurs when the cutting force at the tool point becomes excessive, causing it to fail suddenly by brittle fracture.
2. ***Temperature failure***. This failure occurs when the cutting temperature is too high for the tool material, causing the material at the tool point to soften, which leads to plastic deformation and loss of the sharp edge.
3. ***Gradual wear***. Gradual wearing of the cutting edge causes loss of tool shape, reduction in cutting efficiency, an acceleration of wearing as the tool becomes heavily worn, and finally tool failure in a manner similar to a temperature failure.

Fracture and temperature failures result in premature loss of the cutting tool. These two modes of failure are therefore undesirable. Of the three possible tool failures, gradual wear is preferred because it leads to the longest possible use of the tool, with the associated economic advantage of that longer use.

Product quality must also be considered when attempting to control the mode of tool failure. When the tool point fails suddenly during a cut, it often causes damage to the work surface. This damage requires either rework of the surface or possible scrapping of the part. The damage can be avoided by selecting cutting conditions that favor gradual wearing of the tool rather than fracture or temperature failure, and by changing the tool before the final catastrophic loss of the cutting edge occurs.

22.1.1 TOOL WEAR

Gradual wear occurs at two principal locations on a cutting tool: the top rake face and the flank. Accordingly, two main types of tool wear can be distinguished: crater wear and flank wear, illustrated in Figures 22.1 and 22.2. A single-point tool is used to explain tool wear and the mechanisms that cause it. ***Crater wear***, Figure 22.2(a), consists of a cavity in the rake face of the tool that forms and grows from the action of the chip sliding against the surface. High stresses and temperatures characterize the tool–chip contact interface, contributing to the wearing action. The crater can be measured either by its depth or its area. ***Flank wear***, Figure 22.2(b), occurs on the flank, or relief face, of the tool. It results from rubbing between the newly generated work surface and the flank face adjacent to the cutting edge. Flank wear is measured by the width of the wear band, FW. This wear band is sometimes called the flank wear ***land***.

Certain features of flank wear can be identified. First, an extreme condition of flank wear often appears on the cutting edge at the location corresponding to the original surface of the work part. This is called ***notch wear***. It occurs because the original work surface is harder and/or more abrasive than the internal material, which could be caused by work hardening from cold drawing or previous machining, sand particles in the surface from casting, or other reasons. As a consequence of the harder surface, wear is accelerated at this location. A second region of flank wear that can be identified is ***nose radius wear***; this occurs on the nose radius leading into the end cutting edge.

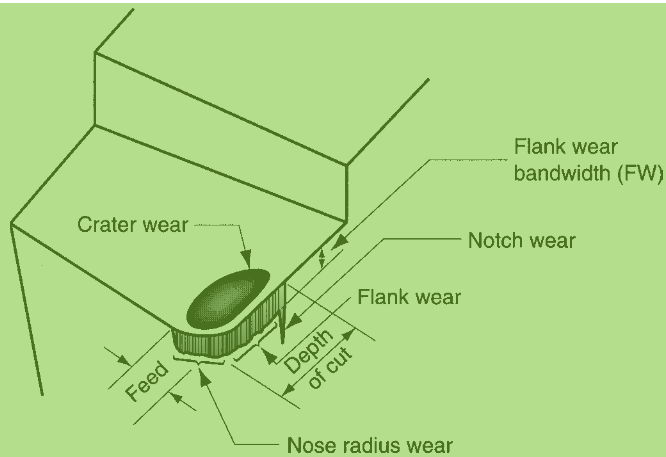
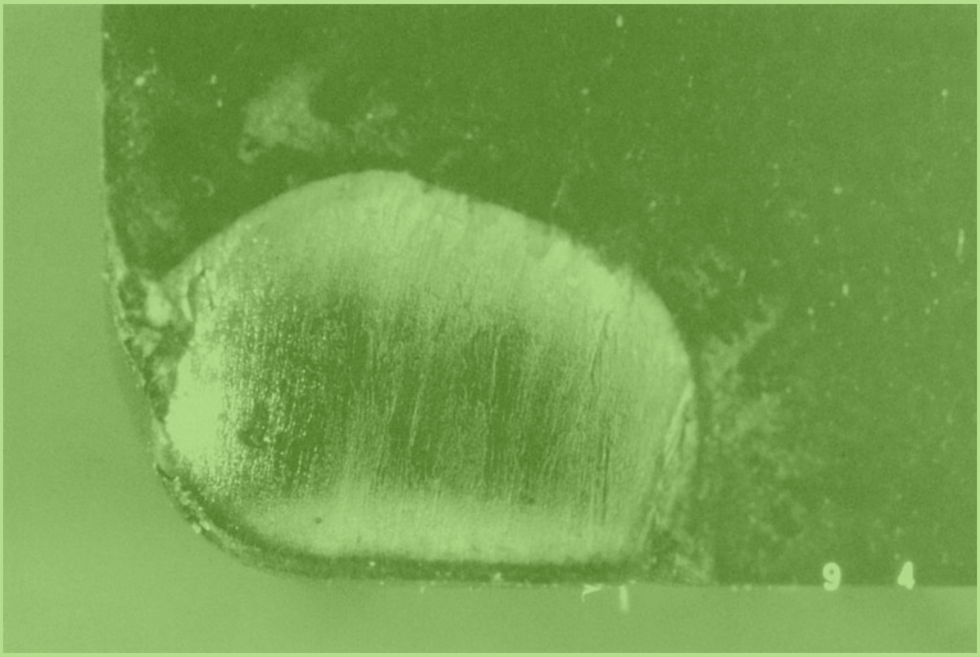
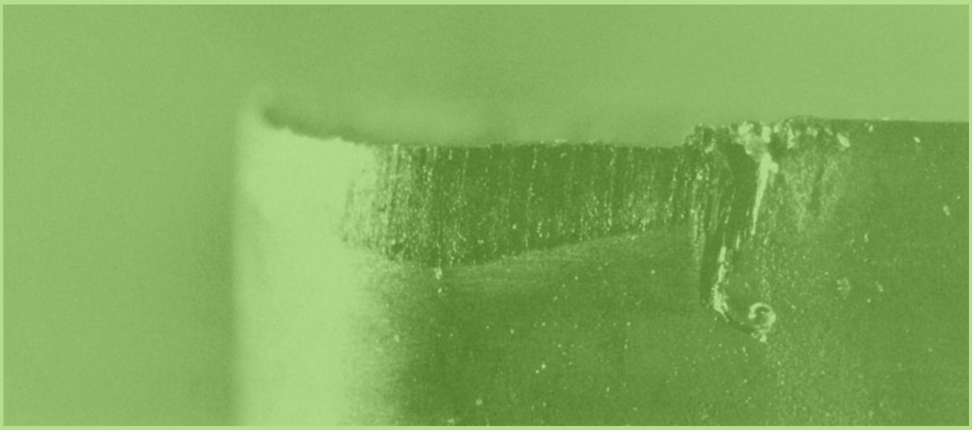


FIGURE 22.1 Diagram of worn cutting tool, showing the principal locations and types of wear that occur.



(a)



(b)

FIGURE 22.2 (a) Crater wear and (b) flank wear on a cemented carbide tool, as seen through a toolmaker's microscope. (Courtesy of George E. Kane Manufacturing Technology Laboratory, Lehigh University; photos by J. C. Keefe.)

The mechanisms that cause wear at the tool–chip and tool–work interfaces in machining can be summarized as follows:

- ***Abrasion***. This is a mechanical wearing action caused by hard particles in the work material gouging and removing small portions of the tool. This abrasive action occurs in both flank wear and crater wear; it is a significant cause of flank wear.
- ***Adhesion***. When two metals are forced into contact under high pressure and temperature, adhesion (welding) occurs between them. These conditions are present between the chip and the rake face of the tool. As the chip flows across

the tool, small particles of the tool adhere to the chip and are broken away from the surface, resulting in attrition of the surface.

- ***Diffusion***. This is a process in which an exchange of atoms takes place across a close contact boundary between two materials (Section 4.3). In the case of tool wear, diffusion occurs at the tool–chip boundary, causing the tool surface to become depleted of the atoms responsible for its hardness. As this process continues, the tool surface becomes more susceptible to abrasion and adhesion. Diffusion is believed to be a principal mechanism of crater wear.
- ***Chemical reactions***. The high temperatures and clean surfaces at the tool–chip interface in machining at high speeds can result in chemical reactions, in particular, oxidation, on the rake face of the tool. The oxidized layer, being softer than the parent tool material, is sheared away, exposing new material to sustain the reaction process.
- ***Plastic deformation***. Another mechanism that contributes to tool wear is plastic deformation of the cutting edge. The cutting forces acting on the cutting edge at high temperature cause the edge to deform plastically, making it more vulnerable to abrasion of the tool surface. Plastic deformation contributes mainly to flank wear.

Most of these tool-wear mechanisms are accelerated at higher cutting speeds and temperatures. Diffusion and chemical reaction are especially sensitive to elevated temperature.

22.1.2 TOOL LIFE AND THE TAYLOR TOOL LIFE EQUATION

As cutting proceeds, the various wear mechanisms result in increasing levels of wear on the cutting tool. The general relationship of tool wear versus cutting time is shown in Figure 22.3. Although the relationship shown is for flank wear, a similar relationship occurs for crater wear. Three regions can usually be identified in the typical wear growth curve. The first is the ***break-in period***, in which the sharp cutting edge wears rapidly at the beginning of its use. This first region occurs within the first few minutes of cutting. The break-in period is followed by wear that occurs at a fairly uniform rate. This is called the ***steady-state wear*** region. In the figure, this region is pictured as a linear function of time, although there are deviations from the straight line in actual machining. Finally, wear reaches a level at which the wear rate begins to accelerate. This marks the beginning of the ***failure region***, in which cutting

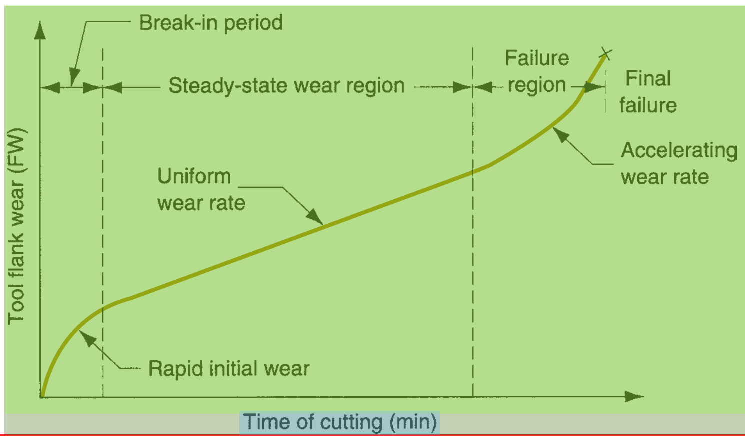


FIGURE 22.3 Tool wear as a function of cutting time. Flank wear (FW) is used here as the measure of tool wear. Crater wear follows a similar growth curve.

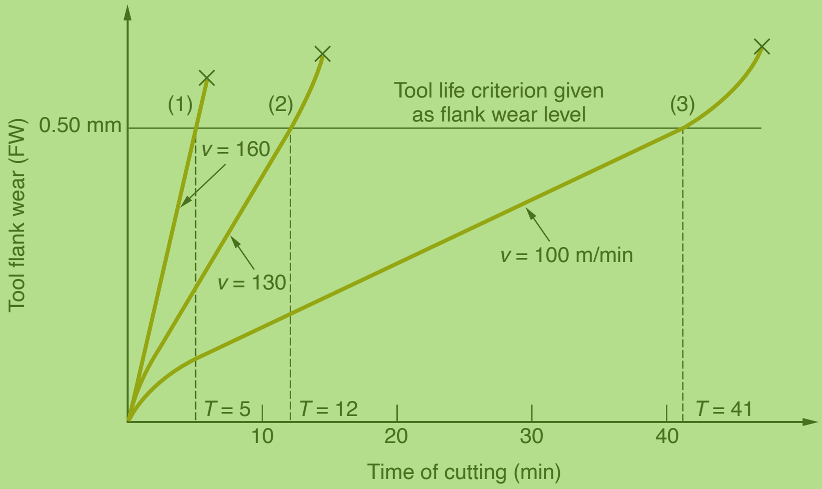


FIGURE 22.4 Effect of cutting speed on tool flank wear (FW) for three cutting speeds. Hypothetical values of speed and tool life are shown for a tool life criterion of 0.50-mm flank wear.

temperatures are higher, and the general efficiency of the machining process is reduced. If allowed to continue, the tool finally fails by temperature failure.

The slope of the tool wear curve in the steady-state region is affected by work material and cutting conditions. Harder work materials cause the wear rate (slope of the tool wear curve) to increase. Increased speed, feed, and depth of cut have a similar effect, with speed being the most important of the three. If the tool wear curves are plotted for several different cutting speeds, the results appear as in Figure 22.4. As cutting speed is increased, wear rate increases so the same level of wear is reached in less time.

Tool life is defined as the length of cutting time that the tool can be used. Operating the tool until final catastrophic failure is one way of defining tool life. This is indicated in Figure 22.4 by the end of each tool wear curve. However, in production, it is often a disadvantage to use the tool until this failure occurs because of difficulties in resharpening the tool and problems with work surface quality. As an alternative, a level of tool wear can be selected as a criterion of tool life, and the tool is replaced when wear reaches that level. A convenient tool life criterion is a certain flank wear value, such as 0.5 mm (0.020 in), illustrated as the horizontal line on the graph. When each of the three wear curves intersects that line, the life of the corresponding tool is defined as ended. If the intersection points are projected down to the time axis, the values of tool life can be identified.

Taylor Tool Life Equation If the tool life values for the three wear curves in Figure 22.4 are plotted on a natural log–log graph of cutting speed versus tool life, the resulting relationship is a straight line as shown in Figure 22.5.[1]

The discovery of this relationship around 1900 is credited to F. W. Taylor. It can be expressed in equation form and is called the Taylor tool life equation:

$$vT^n = C \tag{22.1}$$

[1]The reader may have noted in Figure 22.5 that the dependent variable (tool life) has been plotted on the horizontal axis and the independent variable (cutting speed) on the vertical axis. Although this is a reversal of the normal plotting convention, it nevertheless is the way the Taylor tool life relationship is usually presented.

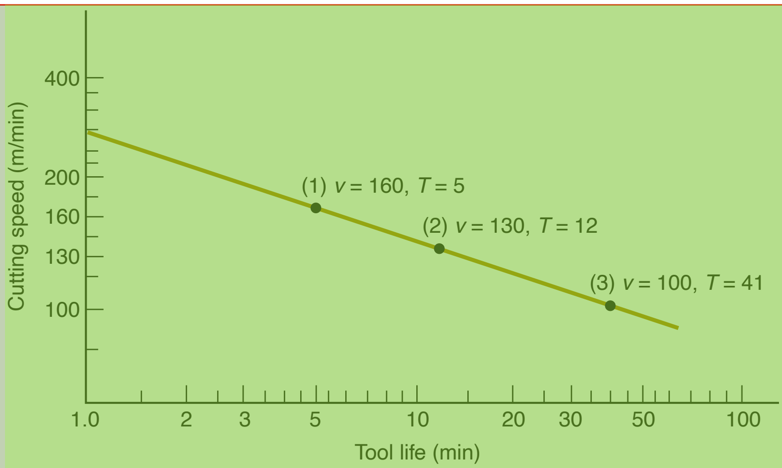


FIGURE 22.5 Natural log–log plot of cutting speed vs. tool life.

where v = cutting speed, m/min (ft/min); T = tool life, min; and n and C are parameters whose values depend on feed, depth of cut, work material, tooling (material in particular), and the tool life criterion used. The value of n is relative constant for a given tool material, whereas the value of C depends on tool material, work material, and cutting conditions.

Basically, Equation (22.1) states that higher cutting speeds result in shorter tool lives. Relating the parameters n and C to Figure 22.5, n is the slope of the plot (expressed in linear terms rather than in the scale of the axes), and C is the intercept on the speed axis. C represents the cutting speed that results in a 1-min tool life.

The problem with Equation (22.1) is that the units on the right-hand side of the equation are not consistent with the units on the left-hand side. To make the units consistent, the equation should be expressed in the form

$$vT^n = C\,(T_{ref}{}^n) \tag{22.2}$$

where T_{ref} = a reference value for C. T_{ref} is simply 1 min when m/min (ft/min) and minutes are used for v and T, respectively. The advantage of Equation (22.2) is seen when it is desired to use the Taylor equation with units other than m/min (ft/min) and minutes—for example, if cutting speed were expressed as m/sec and tool life as sec. In this case, T_{ref} would be 60 sec and C would therefore be the same speed value as in Equation (22.1), although converted to units of m/sec. The slope n would have the same numerical value as in Equation (22.1).

Example 22.1 Taylor tool life equation

Determine the values of C and n in the plot of Figure 22.5, using two of the three points on the curve and solving simultaneous equations of the form of Equation (22.1).

Solution: Choosing the two extreme points: v = 160 m/min, T = 5 min; and v = 100 m/ min, T = 41 min; the two equations are:

$$160(5)^n = C$$

$$100(41)^n = C$$

Setting the left-hand sides of each equation equal,

$$160(5)^n = 100\,(41)^n$$

Taking the natural logarithms of each term,

$$\ln(160) + n\ln(5) = \ln(100) + n\ln(41)$$

$$5.0752 + 1.6094\,n = 4.6052 + 3.7136\,n$$

$$0.4700 = 2.1042\,n$$

$$n = \frac{0.4700}{2.1042} = \mathbf{0.223}$$

Substituting this value of n into either starting equation, the value of C is obtained:

$$C = 160(5)^{0.223} = \mathbf{229}$$

or

$$C = 100(41)^{0.223} = 229$$

The Taylor tool life equation for the data of Figure 22.5 is therefore $vT^{0.223} = 229$

An expanded version of Equation (22.2) can be formulated to include the effects of feed, depth of cut, and even work material hardness:

$$vT^n f^m d^p H^q = K T_{ref}^{\,n} f_{ref}^{\,m} d_{ref}^{\,p} H_{ref}^{\,q} \tag{22.3}$$

where f = feed, mm (in); d = depth of cut, mm (in); H = hardness, expressed in an appropriate hardness scale; m, p, and q are exponents whose values are experimentally determined for the conditions of the operation; K = a constant analogous to C in Equation (22.2); and f_{ref}, d_{ref}, and H_{ref} are reference values for feed, depth of cut, and hardness. The values of m and p, the exponents for feed and depth, are less than 1.0. This indicates the greater effect of cutting speed on tool life because the exponent of v is 1.0. After speed, feed is next in importance, so m has a value greater than p. The exponent for work hardness, q, is also less than 1.0.

Perhaps the greatest difficulty in applying Equation (22.3) in a practical machining operation is the tremendous amount of machining data that would be required to determine the parameters of the equation. Variations in work materials and testing conditions also cause difficulties by introducing statistical variations in the data. Equation (22.3) is valid in indicating general trends among its variables, but not in its ability to accurately predict tool life performance. To reduce these problems and make the scope of the equation more manageable, some of the terms are usually eliminated. For example, omitting depth and hardness reduces Equation (22.3) to the following:

$$vT^n f^m = KT_{\text{ref}}{}^n f_{\text{ref}}{}^m \tag{22.4}$$

where the terms have the same meaning as before, except that the constant K will have a slightly different interpretation.

Tool Life Criteria in Production Although flank wear is the tool life criterion in the previous discussion of the Taylor equation, this criterion is not very practical in a factory environment because of the difficulties and time required to measure flank wear. Following are nine alternative tool life criteria that are more convenient to use in a production machining operation, some of which are admittedly subjective:

1. Complete failure of the cutting edge (fracture failure, temperature failure, or wearing until complete breakdown of the tool has occurred). This criterion has disadvantages, as discussed earlier.
2. Visual inspection of flank wear (or crater wear) by the machine operator (without a toolmaker's microscope). This criterion is limited by the operator's judgment and ability to observe tool wear with the naked eye.
3. Fingernail test across the cutting edge by the operator to test for irregularities.
4. Changes in the sound emitting from the operation, as judged by the operator.
5. Chips become ribbony, stringy, and difficult to dispose of.
6. Degradation of the surface finish on the work.
7. Increased power consumption in the operation, as measured by a wattmeter connected to the machine tool.
8. Workpiece count. The operator is instructed to change the tool after a certain specified number of parts have been machined.
9. Cumulative cutting time. This is similar to the previous workpiece count, except that the length of time the tool has been cutting is monitored. This is possible on machine tools controlled by computer; the computer is programmed to keep data on the total cutting time for each tool.

22.2 Tool Materials

The three modes of tool failure allow us to identify three important properties required in a tool material:

- ***Toughness***. To avoid fracture failure, the tool material must possess high toughness. Toughness is the capacity of a material to absorb energy without failing. It is usually characterized by a combination of strength and ductility in the material.
- ***Hot hardness***. Hot hardness is the ability of a material to retain its hardness at high temperatures. This is required because of the high-temperature environment in which the tool operates.

➢ ***Wear resistance***. Hardness is the single most important property needed to resist abrasive wear. All cutting-tool materials must be hard. However, wear resistance in metal cutting depends on more than just tool hardness, because of the other tool-wear mechanisms. Other characteristics affecting wear resistance include surface finish on the tool (a smoother surface means a lower coefficient of friction), chemistry of tool and work materials, and whether a cutting fluid is used.

Cutting-tool materials achieve this combination of properties in varying degrees. In this section, the following cutting-tool materials are discussed: (1) high-speed steel and its predecessors, plain carbon and low alloy steels, (2) cast cobalt alloys, (3) cemented carbides, cermets, and coated carbides, (4) ceramics, (5) synthetic diamond and cubic boron nitride. Before examining these individual materials, a brief overview and technical comparison will be helpful. The historical development of these materials is described in Historical Note 22.1. Commercially, the most important tool materials are high-speed steel and cemented carbides, cermets, and coated carbides. These two categories account for more than 90% of the cutting tools used in machining operations.

Table 22.1 and Figure 22.6 present data on properties of various tool materials. The properties are those related to the requirements of a cutting tool: hardness, toughness, and hot hardness. Table 22.1 lists room temperature hardness and transverse rupture strength for selected materials. Transverse rupture strength (Section 3.1.3) is a property used to indicate toughness for hard materials. Figure 22.6 shows hardness as a function of temperature for several of the tool materials discussed in this section.

TABLE • 22.1 Typical hardness values (at room temperature) and transverse rupture strengths for various tool materials.[a]

		Transverse Rupture Strength	
Material	**Hardness**	**MPa**	**lb/in²**
Plain carbon steel	60 HRC	5200	750,000
High-speed steel	65 HRC	4100	600,000
Cast cobalt alloy	65 HRC	2250	325,000
Cemented carbide (WC)			
Low Co content	93 HRA, 1800 HK	1400	200,000
High Co content	90 HRA, 1700 HK	2400	350,000
Cermet (TiC)	2400 HK	1700	250,000
Alumina (Al_2O_3)	2100 HK	400	60,000
Cubic boron nitride	5000 HK	700	100,000
Polycrystalline diamond	6000 HK	1000	150,000
Natural diamond	8000 HK	1500	215,000

Compiled from [4], [9], [17], and other sources.

[a]*Note:* The values of hardness and TRS are intended to be comparative and typical. Variations in properties result from differences in composition and processing.

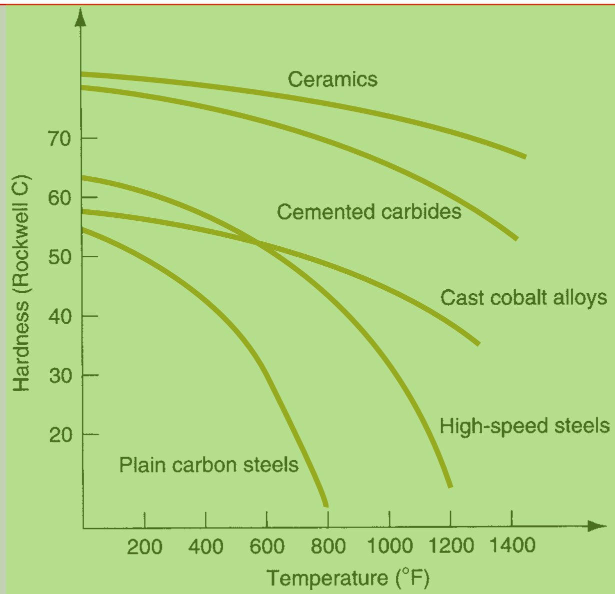


FIGURE 22.6 Typical hot hardness relationships for selected tool materials. Plain carbon steel shows a rapid loss of hardness as temperature increases. High-speed steel is substantially better, while cemented carbides and ceramics are significantly harder at elevated temperatures.

Historical Note 22.1 *Cutting-tool materials*

In 1800, England was leading the Industrial Revolution, and iron was the leading metal in the revolution. The best tools for cutting iron were made of cast steel by the crucible process, invented in 1742 by B. Huntsman. Cast steel, whose carbon content lies between wrought iron and cast iron, could be hardened by heat treatment to machine the other metals. In 1868, R. Mushet discovered that by alloying about 7% tungsten in crucible steel, a hardened tool steel was obtained by air quenching after heat treatment. Mushet's tool steel was far superior to its predecessor in machining.

Frederick W. Taylor stands as an important figure in the history of cutting tools. Starting around 1880 at Midvale Steel in Philadelphia and later at Bethlehem Steel in Bethlehem, Pennsylvania, he began a series of experiments that lasted a quarter century, yielding a much improved understanding of the metal-cutting process. Among the developments resulting from the work of Taylor and colleague Maunsel White at Bethlehem was ***high-speed steel*** (HSS), a class of highly alloyed tool steels that permitted substantially higher cutting speeds than previous cutting tools. The superiority of HSS resulted not only from greater alloying, but also from refinements in heat treatment. Tools of the new steel allowed cutting speeds more than twice those of Mushet's steel and almost four times those of plain carbon cast steels.

Tungsten carbide (WC) was first synthesized in the late 1890s. It took nearly three decades before a useful cutting tool material was developed by sintering the WC with a metallic binder to form ***cemented carbides.*** These were first used in metal cutting in the mid-1920s in Germany, and in the late 1920s in the United States (Historical Note 7.2). ***Cermet*** cutting tools based on titanium carbide were first introduced in the 1950s, but their commercial importance dates from the 1970s. The first ***coated carbides,*** consisting of one coating on a WC–Co substrate, were first used around 1970. Coating materials included TiC, TiN, and Al_2O_3. Modern coated carbides have three or more coatings of these and other hard materials.

Attempts to use ***alumina ceramics*** in machining date from the early 1900s in Europe. Their brittleness inhibited success in these early applications. Processing refinements over many decades have resulted in property improvements in these materials. U.S. commercial use of ceramic cutting tools dates from the mid-1950s.

The first industrial diamonds were produced by the General Electric Company in 1954. They were single crystal diamonds that were applied with some success in grinding operations starting around 1957. Greater acceptance of diamond cutting tools has resulted from the use of ***sintered polycrystalline diamond*** (SPD), dating from the early 1970s. A similar tool material, sintered ***cubic boron nitride,*** was first introduced in 1969 by GE under the trade name Borazon.

TABLE • 22.2 Representative values of n and C in the Taylor tool life equation, Equation (22.1), for selected tool materials.

		C			
		Nonsteel Cutting		Steel Cutting	
Tool Material	***n***	**m/min**	**(ft/min)**	**m/min**	**ft/min**
Plain carbon tool steel	0.1	70	(200)	20	60
High-speed steel	0.125	120	(350)	70	200
Cemented carbide	0.25	900	(2700)	500	1500
Cermet	0.25			600	2000
Coated carbide	0.25			700	2200
Ceramic	0.6			3000	10,000

Compiled from [4], [9], and other sources.
The parameter values are approximated for turning at feed = 0.25 mm/rev (0.010 in/rev) and depth = 2.5 mm (0.100 in). Nonsteel cutting refers to easy-to-machine metals such as aluminum, brass, and cast iron. Steel cutting refers to the machining of mild (unhardened) steel. It should be noted that significant variations in these values can be expected in practice.

In addition to these property comparisons, it is useful to compare the materials in terms of the parameters n and C in the Taylor tool life equation. In general, the development of new cutting-tool materials has resulted in increases in the values of these two parameters. Table 22.2 lists representative values of n and C for selected cutting-tool materials.

The chronological development of tool materials has generally followed a path in which new materials have permitted higher and higher cutting speeds to be achieved. Table 22.3 identifies the cutting-tool materials, together with their approximate year of introduction and typical maximum allowable cutting speeds at which they can be used.

TABLE • 22.3 Cutting-tool materials with their approximate dates of initial use and allowable cutting speeds.

		Allowable Cutting Speed[a]			
		Nonsteel Cutting		Steel Cutting	
Tool Material	**Year of Initial Use**	**m/min**	**ft/min**	**m/min**	**ft/min**
Plain carbon tool steel	1800s	Below 10	Below 30	Below 5	Below 15
High-speed steel	1900	25–65	75–200	17–33	50–100
Cast cobalt alloys	1915	50–200	150–600	33–100	100–300
Cemented carbides (WC)	1930	330–650	1000–2000	100–300	300–900
Cermets (TiC)	1950s			165–400	500–1200
Ceramics (Al_2O_3)	1955			330–650	1000–2000
Synthetic diamonds	1954, 1973	390–1300	1200–4000		
Cubic boron nitride	1969			500–800	1500–2500
Coated carbides	1970			165–400	500–1200

[a]Compiled from [9], [12], [16], [19], and other sources.

Dramatic increases in machining productivity have been made possible by advances in tool material technology, as indicated in the table. Machine tool practice has not always kept pace with cutting-tool technology. Limitations on horsepower, machine tool rigidity, spindle bearings, and the widespread use of older equipment in industry have acted to underutilize the possible upper speeds permitted by available cutting tools.

22.2.1 HIGH-SPEED STEEL AND ITS PREDECESSORS

Before the development of high-speed steel, plain carbon steel and Mushet's steel were the principal tool materials for metal cutting. Today, these steels are rarely used in industrial machining applications. The plain carbon steels used as cutting tools could be heat-treated to achieve relatively high hardness (~ Rockwell C 60), because of their fairly high carbon content. However, with low alloying levels, they possess poor hot hardness (Figure 22.6), which renders them unusable in metal cutting except at speeds too slow to be practical by today's standards. Mushet's steel contained alloying elements tungsten (4%–12%) and manganese (2%–4%) in addition to carbon. It was displaced by the introduction of high speed steel and other advances in tool steel metallurgy.

High-speed steel (HSS) is a highly alloyed tool steel capable of maintaining hardness at elevated temperatures better than high carbon and low alloy steels. Its good hot hardness permits tools made of HSS to be used at higher cutting speeds. Compared with the other tool materials at the time of its development, it was truly deserving of its name "high speed." A wide variety of high-speed steels is available, but they can be divided into two basic types: (1) tungsten-type, designated T-grades by the American Iron and Steel Institute (AISI); and (2) molybdenum-type, designated M-grades by AISI.

Tungsten-type HSS contains tungsten (W) as its principal alloying ingredient. Additional alloying elements are chromium (Cr), and vanadium (V). One of the original and best known HSS grades is T1, or 18-4-1 high-speed steel, containing 18% W, 4% Cr, and 1% V. ***Molybdenum HSS*** grades contain combinations of tungsten and molybdenum (Mo), plus the same additional alloying elements as in the T-grades. Cobalt (Co) is sometimes added to HSS to enhance hot hardness. Of course, high-speed steel contains carbon, the element common to all steels. Typical alloying contents and functions of each alloying element in HSS are listed in Table 22.4.

Commercially, high-speed steel is one of the most important cutting-tool materials in use today, despite the fact that it was introduced more than a century ago. HSS is especially suited to applications involving complicated tool geometries, such as drills, taps, milling cutters, and broaches. These complex shapes are generally easier and less expensive to produce from unhardened HSS than other tool materials. They can then be heat-treated so that cutting-edge hardness is very good (Rockwell C 65) whereas toughness of the internal portions of the tool is also good. HSS cutters possess better toughness than any of the harder nonsteel tool materials used for machining, such as cemented carbides and ceramics. Even for single-point tools, HSS is popular among machinists because of the ease with which a desired tool geometry can be ground into the tool point. Over the years, improvements have been made in the metallurgical formulation and processing of HSS so that this class of tool material remains competitive in many applications. Also, HSS tools, drills in particular, are often coated with a thin film of titanium nitride (TiN) to provide significant increases in cutting performance. Physical vapor deposition processes (Section 27.5.1) are commonly used to coat these HSS tools.

TABLE • 22.4 Typical contents and functions of alloying elements in high-speed steel.

Alloying Element	Typical Content in HSS, % by Weight	Functions in High-Speed Steel
Tungsten	T-type HSS: 12–20	Increases hot hardness
	M-type HSS: 1.5–6	Improves abrasion resistance through formation of hard carbides in HSS
Molybdenum	T-type HSS: none	Increases hot hardness
	M-type HSS: 5–10	Improves abrasion resistance through formation of hard carbides in HSS
Chromium	3.75–4.5	Depth hardenability during heat treatment
		Improves abrasion resistance through formation of hard carbides in HSS Corrosion resistance (minor effect)
Vanadium	1–5	Combines with carbon for wear resistance
		Retards grain growth for better toughness
Cobalt	0–12	Increases hot hardness
Carbon	0.75–1.5	Principal hardening element in steel
		Provides available carbon to form carbides with other alloying elements for wear resistance

22.2.2 CAST COBALT ALLOYS

Cast cobalt alloy cutting tools consist of cobalt, around 40% to 50%; chromium, about 25% to 35%; and tungsten, usually 15% to 20%; with trace amounts of other elements. These tools are made into the desired shape by casting in graphite molds and then grinding to final size and cutting-edge sharpness. High hardness is achieved as cast, an advantage over HSS, which requires heat treatment to achieve its hardness. Wear resistance of the cast cobalts is better than high-speed steel, but not as good as cemented carbide. Toughness of cast cobalt tools is better than carbides but not as good as HSS. Hot hardness also lies between these two materials.

As might be expected from their properties, applications of cast cobalt tools are generally between those of high-speed steel and cemented carbides. They are capable of heavy roughing cuts at speeds greater than HSS and feeds greater than carbides. Work materials include both steels and nonsteels, as well as nonmetallic materials such as plastics and graphite. Today, cast cobalt alloy tools are not nearly as important commercially as either high-speed steel or cemented carbides. They were introduced around 1915 as a tool material that would allow higher cutting speeds than HSS. The carbides were subsequently developed and proved to be superior to the cast Co alloys in most cutting situations.

22.2.3 CEMENTED CARBIDES, CERMETS, AND COATED CARBIDES

Cermets are defined as composites of ***cer***amic and ***met***allic materials (Section 9.2.1). Technically speaking, cemented carbides are included within this definition; however, cermets based on WC–Co, including WC–TiC–TaC–Co, are known as carbides (cemented carbides) in common usage. In cutting-tool terminology, the term cermet is applied to ceramic-metal composites containing TiC, TiN, and certain other ceramics not including WC. One of the advances in cutting-tool materials involves the application of a very thin coating to a WC–Co substrate. These tools are called coated carbides. Thus, there are three important and closely related tool materials to discuss: (1) cemented carbides, (2) cermets, and (3) coated carbides.

Cemented Carbides Cemented carbides (also called ***sintered carbides***) are a class of hard tool material formulated from tungsten carbide (WC, Section 7.3.2) using powder metallurgy techniques (Chapter 15) with cobalt (Co) as the binder (Sections 9.2.1 and 16.3.1). There may be other carbide compounds in the mixture, such as titanium carbide (TiC) and/or tantalum carbide (TaC), in addition to WC.

The first cemented carbide cutting tools were made of WC–Co (Historical Note 7.2) and could be used to machine cast irons and nonsteel materials at cutting speeds faster than those possible with high-speed steel and cast cobalt alloys. However, when the straight WC–Co tools were used to cut steel, crater wear occurred rapidly, leading to early failure of the tools. A strong chemical affinity exists between steel and the carbon in WC, resulting in accelerated wear by diffusion and chemical reaction at the tool–chip interface for this work-tool combination. Consequently, straight WC–Co tools cannot be used effectively to machine steel. It was subsequently discovered that additions of titanium carbide and tantalum carbide to the WC–Co mix significantly retarded the rate of crater wear when cutting steel. These new WC–TiC–TaC–Co tools could be used for steel machining. The result is that cemented carbides are divided into two basic types: (1) nonsteel-cutting grades, consisting of only WC–Co; and (2) steel-cutting grades, with combinations of TiC and TaC added to the WC–Co.

The general properties of the two types of cemented carbides are similar: (1) high compressive strength but low-to-moderate tensile strength; (2) high hardness (90 to 95 HRA); (3) good hot hardness; (4) good wear resistance; (5) high thermal conductivity; (6) high modulus of elasticity—E values up to around 600×10^3 MPa (90×10^6 lb/in^2); and (7) toughness lower than high-speed steel.

Nonsteel-cutting grades refer to those cemented carbides that are suitable for machining aluminum, brass, copper, magnesium, titanium, and other nonferrous metals; anomalously, gray cast iron is included in this group of work materials. In the nonsteel-cutting grades, grain size and cobalt content are the factors that influence properties of the cemented carbide material. The typical grain size found in conventional cemented carbides ranges between 0.5 and 5 μm (20 and 200 μ-in). As grain size is increased, hardness and hot hardness decrease, but transverse rupture strength increases.[2] The typical cobalt content in cemented carbides used for cutting tools is 3 to 12%. The effect of cobalt content on hardness and transverse rupture strength is shown in Figure 9.9. As cobalt content increases, TRS improves at the expense of hardness and wear resistance. Cemented carbides with low percentages of cobalt content (3%–6%) have high hardness and low TRS, whereas carbides with high Co (6%–12%) have high TRS but lower hardness (Table 22.1). Accordingly, cemented carbides with higher cobalt are used for roughing operations and interrupted cuts (such as milling), while carbides with lower cobalt (therefore, higher hardness and wear resistance) are used in finishing cuts.

[2] The effect of grain size (GS) on transverse rupture strength (TRS) is more complicated than reported here. Published data indicate that the effect of GS on TRS is influenced by cobalt content. At lower Co contents (< 10%), TRS does indeed increase as GS increases, but at higher Co contents (> 10%) TRS decreases as GS increases [2], [5].

TABLE • 22.5 The ANSI C-grade classification system for cemented carbides.

Machining Application	Nonsteel-cutting Grades	Steel-cutting Grades	Cobalt and Properties
Roughing	C1	C5	High Co for max. toughness
General purpose	C2	C6	Medium to high Co
Finishing	C3	C7	Medium to low Co
Precision finishing	C4	C8	Low Co for max. hardness
Work materials	Al, brass, Ti, cast iron	Carbon & alloy steels	
Typical ingredients	WC–Co	WC–TiC–TaC–Co	

Steel-cutting grades are used for low carbon, stainless, and other alloy steels. For these carbide grades, titanium carbide and/or tantalum carbide is substituted for some of the tungsten carbide. TiC is the more popular additive in most applications. Typically, from 10% to 25% of the WC might be replaced by combinations of TiC and TaC. This composition increases the crater wear resistance for steel cutting, but tends to adversely affect flank wear resistance for nonsteel-cutting applications. That is why two basic categories of cemented carbide are needed.

One of the important developments in cemented carbide technology in recent years is the use of very fine grain sizes (submicron sizes) of the various carbide ingredients (WC, TiC, and TaC). Although small grain size is usually associated with higher hardness but lower transverse rupture strength, the decrease in TRS is reduced or reversed at the submicron particle sizes. Therefore, these ultrafine grain carbides possess high hardness combined with good toughness.

Since the two basic types of cemented carbide were introduced in the 1920s and 1930s, the increasing number and variety of engineering materials have complicated the selection of the most appropriate cemented carbide for a given machining application. To address the problem of grade selection, two classification systems have been developed: (1) the ANSI[3] C-grade system, developed in the United States starting around 1942; and (2) the ISO R513-1975(E) system, introduced by the International Organization for Standardization (ISO) around 1964. In the C-grade system, summarized in Table 22.5, machining grades of cemented carbide are divided into two basic groups, corresponding to nonsteel-cutting and steel-cutting categories. Within each group there are four levels, corresponding to roughing, general purpose, finishing, and precision finishing.

The ISO R513-1975(E) system, titled "Application of Carbides for Machining by Chip Removal," classifies all machining grades of cemented carbides into three basic groups, each with its own letter and color code, as summarized in Table 22.6. Within each group, the grades are numbered on a scale that ranges from maximum hardness to maximum toughness. Harder grades are used for finishing operations (high speeds, low feeds and depths), whereas tougher grades are used for roughing operations. The ISO classification system can also be used to recommend applications for cermets and coated carbides.

The two systems map into each other as follows: The ANSI C1 through C4-grades map into the ISO K-grades, but in reverse numerical order, and the ANSI C5 through C8 grades translate into the ISO P-grades, but again in reverse numerical order.

[3]ANSI = American National Standards Institute.

TABLE • 22.6 ISO R513-1975(E) "Application of Carbides for Machining by Chip Removal."

Group	Carbide Type	Work Materials	Number Scheme (Cobalt and Properties)
P (blue)	Highly alloyed WC–TiC–TaC–Co	Steel, steel castings, ductile cast iron (ferrous metals with long chips)	P01 (low Co for maximum hardness) to P50 (high Co for maximum toughness)
M (yellow)	Alloyed WC–TiC–TaC–Co	Free-cutting steel, gray cast iron, austenitic stainless steel, superalloys	M10 (low Co for maximum hardness) to M40 (high Co for maximum toughness)
K (red)	Straight WC–Co	Nonferrous metals and alloys, gray cast iron (ferrous metals with short chips), nonmetallics	K01 (low Co for maximum hardness) to K40 (high Co for maximum toughness)

Cermets Although cemented carbides are technically classified as cermet composites, the term ***cermet*** in cutting-tool technology is generally reserved for combinations of TiC, TiN, and titanium carbonitride (TiCN), with nickel and/or molybdenum as binders. Some of the cermet chemistries are more complex (e.g., ceramics such as Ta_xNb_yC and binders such as Mo_2C). However, cermets exclude metallic composites that are primarily based on WC–Co. Applications of cermets include high-speed finishing and semifinishing of steels, stainless steels, and cast irons. Higher speeds are generally allowed with these tools compared with steel-cutting carbide grades. Lower feeds are typically used so that better surface finish is achieved, often eliminating the need for grinding.

Coated Carbides The development of coated carbides around 1970 represented a significant advance in cutting-tool technology. ***Coated carbides*** are a cemented carbide insert coated with one or more thin layers of wear resistant material, such as titanium carbide, titanium nitride, and/or aluminum oxide (Al_2O_3). The coating is applied to the substrate by chemical vapor deposition or physical vapor deposition (Section 27.5). The coating thickness is only 2.5 to 13 μm (0.0001–0.0005 in). It has been found that thicker coatings tend to be brittle, resulting in cracking, chipping, and separation from the substrate.

The first generation of coated carbides had only a single layer coating (TiC, TiN, or Al_2O_3). More recently, coated inserts have been developed that consist of multiple layers. The first layer applied to the WC–Co base is usually TiN or TiCN because of good adhesion and similar coefficient of thermal expansion. Additional layers of various combinations of TiN, TiCN, Al_2O_3, and TiAlN are subsequently applied (see Figure 27.8).

Coated carbides are used to machine cast irons and steels in turning and milling operations. They are best applied at high cutting speeds in situations in which dynamic force and thermal shock are minimal. If these conditions become too severe, as in some interrupted cut operations, chipping of the coating can occur, resulting in premature tool failure. In this situation, uncoated carbides formulated for toughness are preferred. When properly applied, coated carbide tools usually permit increases in allowable cutting speeds compared with uncoated cemented carbides.

Use of coated carbide tools is expanding to nonferrous metal and nonmetal applications for improved tool life and higher cutting speeds. Different coating materials are required, such as chromium carbide (CrC), zirconium nitride (ZrN), and diamond [11].

22.2.4 CERAMICS

Cutting tools made from ceramics were first used commercially in the United States in the mid-1950s, although their development and use in Europe dates back to the early 1900s. Today's ceramic cutting tools are composed primarily of fine-grained ***aluminum oxide*** (Al_2O_3), pressed and sintered at high pressures and temperatures with no binder into insert form. The aluminum oxide is usually very pure (99% is typical), although some manufacturers add other oxides (such as zirconium oxide) in small amounts. In producing ceramic tools, it is important to use a very fine grain size in the alumina powder, and to maximize density of the mix through high-pressure compaction to improve the material's low toughness.

Aluminum oxide cutting tools are most successful in high-speed turning of cast iron and steel. Applications also include finish turning of hardened steels using high cutting speeds, low feeds and depths, and a rigid work setup. Many premature fracture failures of ceramic tools occur because of nonrigid machine tool setups, which subject the tools to mechanical shock. When properly applied, ceramic cutting tools can be used to obtain very good surface finish. Ceramics are not recommended for heavy interrupted cut operations (e.g., rough milling) because of their low toughness. In addition to its use as inserts in conventional machining operations, Al_2O_3 is widely used as an abrasive in grinding and other abrasive processes (Chapter 24).

Other commercially available ceramic cutting-tool materials include silicon nitride (SiN), ***sialon*** (silicon nitride and aluminum oxide, SiN–Al_2O_3), aluminum oxide and titanium carbide (Al_2O_3–TiC), and aluminum oxide reinforced with single crystal-whiskers of silicon carbide. These tools are usually intended for special applications, a discussion of which is beyond the scope of this introductory treatment.

22.2.5 SYNTHETIC DIAMONDS AND CUBIC BORON NITRIDE

Diamond is the hardest material known (Section 7.6.1). By some measures of hardness, diamond is three to four times as hard as tungsten carbide or aluminum oxide. Since high hardness is one of the desirable properties of a cutting tool, it is natural to think of diamonds for machining and grinding applications. Synthetic diamond cutting tools are made of sintered polycrystalline diamond (SPD), which dates from the early 1970s. ***Sintered polycrystalline diamond*** is fabricated by sintering fine-grained diamond crystals under high temperatures and pressures into the desired shape. Little or no binder is used. The crystals have a random orientation and this adds considerable toughness to the SPD tools compared with single crystal diamonds. Tool inserts are typically made by depositing a layer of SPD about 0.5 mm (0.020 in) thick on the surface of a cemented carbide base. Very small inserts have also been made of 100% SPD.

Applications of diamond cutting tools include high-speed machining of nonferrous metals and abrasive nonmetals such as fiberglass, graphite, and wood. Machining of steel, other ferrous metals, and nickel-based alloys with SPD tools is not practical because of the chemical affinity that exists between these metals and carbon (a diamond, after all, is carbon).

Next to diamond, ***cubic boron nitride*** (Section 7.3.3) is the hardest material known, and its fabrication into cutting tool inserts is basically the same as SPD, that is, coatings on WC–Co inserts. Cubic boron nitride (symbolized cBN) does not react chemically with iron and nickel as SPD does; therefore, the applications of cBN-coated tools are for machining steel and nickel-based alloys. Both SPD and cBN tools are expensive, as one might expect, and the applications must justify the additional tooling cost.

22.3 Tool Geometry

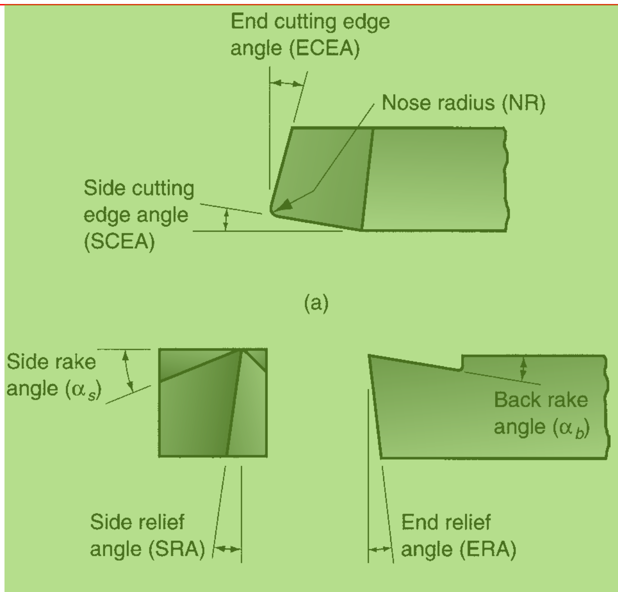
One important way to classify cutting tools is according to the machining process. Thus, there are turning tools, cutoff tools, milling cutters, drill bits, reamers, taps, and many other cutting tools that are named for the operation in which they are used, each with its own tool geometry—in some cases quite unique.

As indicated in Section 20.1, cutting tools can be divided into single-point tools and multiple-cutting-edge tools. Single-point tools are used in turning, boring, shaping, and planing. Multiple-cutting-edge tools are used in drilling, reaming, tapping, milling, broaching, and sawing. Many of the principles that apply to single-point tools also apply to the other cutting-tool types, simply because the mechanism of chip formation is basically the same for all machining operations.

Table 22.7 presents a troubleshooting guide that summarizes many of the actions that can be taken to reduce tooling problems that might result from nonoptimal application of cutting conditions, tool material, and/or tool geometry.

TABLE • 22.7 Troubleshooting Guide for Cutting Tool Problems.

Problem	Possible Solutions
Fracture failure	Increase rigidity of setup (e.g., larger toolholder)
	Reduce feed and/or depth of cut
	Increase cutting speed
	Use tool material with greater toughness (e.g., if ceramic change to carbide)
	Increase nose radius and/or side cutting edge angle
	Use smaller relief angle on cutting edge
Temperature failure	Use coolant type cutting fluid (Section 22.4.1)
	Reduce cutting speed
	Reduce feed and/or depth of cut
	Use cutting tool material with higher hot hardness (e.g., if high speed steel change to carbide, if carbide select a grade with lower binder content)
Wear too rapid	Use lubricant type cutting fluid (Section 22.4.1)
	Reduce cutting speed
	Use cutting tool material with higher wear resistance (e.g., if cemented carbide change to coated carbide)
	Increase relief angle, nose radius, and/or side cutting edge angle
	Use cutting tool with finer finish on rake face



(b) Tool signature: α_b, α_s, ERA, SRA, ECEA, SCEA, NR

FIGURE 22.7 (a) Seven elements of single-point tool geometry, and (b) the tool signature convention that defines the seven elements.

22.3.1 SINGLE-POINT TOOL GEOMETRY

The general shape of a single-point cutting tool is illustrated in Figure 20.4(a). Figure 22.7 shows a more detailed drawing.

The rake angle of a cutting tool has previously been treated as one parameter. In a single-point tool, the orientation of the rake face is defined by two angles, ***back rake angle*** (α_b) and ***side rake angle*** (α_s). Together, these angles are influential in determining the direction of chip flow across the rake face. The flank surface of the tool is defined by the ***end relief angle*** (ERA) and ***side relief angle*** (SRA). These angles determine the amount of clearance between the tool and the freshly cut work surface. The cutting edge of a single-point tool is divided into two sections, side cutting edge and end cutting edge. These two sections are separated by the tool point, which has a certain radius, called the nose radius. The ***side cutting edge angle*** (SCEA) determines the entry of the tool into the work and can be used to reduce the sudden force the tool experiences as it enters a work part. ***Nose radius*** (NR) determines to a large degree the texture of the surface generated in the operation. A very pointed tool (small nose radius) results in very pronounced feed marks on the surface (Section 23.2.2). ***End cutting edge angle*** (ECEA) provides a clearance between the trailing edge of the tool and the newly generated work surface, thus reducing rubbing and friction against the surface.

In all, there are seven elements of tool geometry for a single-point tool. When specified in the following order, they are collectively called the ***tool geometry signature***: back rake angle, side rake angle, end relief angle, side relief angle, end cutting edge angle, side cutting edge angle, and nose radius. For example, a single-point tool used in turning might have the following signature: 5, 5, 7, 7, 20, 15, 0.8 mm.

Chip Breakers Chip disposal is a problem that is often encountered in turning and other continuous operations. Long, stringy chips are often generated, especially when turning ductile materials at high speeds. These chips cause a hazard to the machine

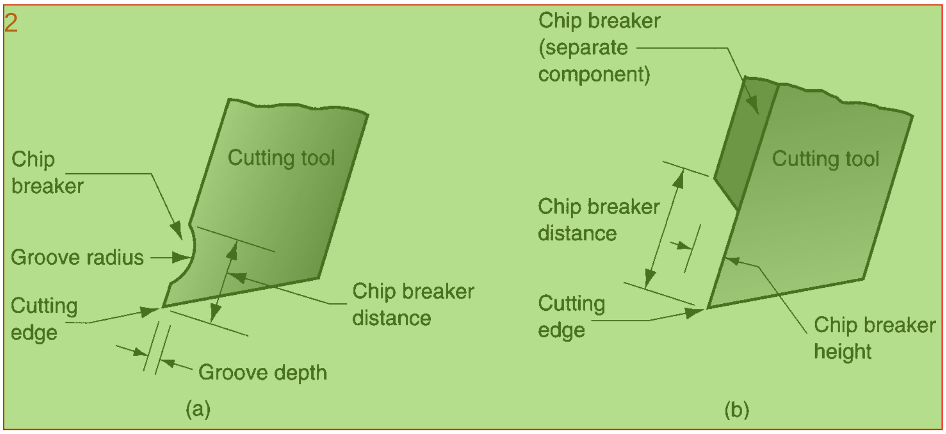


FIGURE 22.8 Two methods of chip breaking in single-point tools: (a) groove-type and (b) obstruction-type chip breakers.

operator and the work part finish, and they interfere with automatic operation of the turning process. ***Chip breakers*** are frequently used with single-point tools to force the chips to curl more tightly than they would naturally be inclined to do, thus causing them to fracture. There are two principal forms of chip breaker design commonly used on single-point turning tools, illustrated in Figure 22.8: (a) groove-type chip breaker designed into the cutting tool itself, and (b) obstruction-type chip breaker designed as an additional device on the rake face of the tool. The chip breaker distance can be adjusted in the obstruction-type device for different cutting conditions.

Effect of Tool Material on Tool Geometry It was noted in the discussion of the Merchant equation (Section 20.3.2) that a positive rake angle is generally desirable because it reduces cutting forces, temperature, and power consumption. High speed steel cutting tools are almost always ground with positive rake angles, typically ranging from $+5°$ to $+20°$. HSS has good strength and toughness, so that the thinner cross section of the tool created by high positive rake angles does not usually cause a problem with tool breakage. HSS tools are predominantly made of one piece. The heat treatment of high-speed steel can be controlled to provide a hard cutting edge while maintaining a tough inner core.

With the development of the very hard tool materials (e.g., cemented carbides and ceramics), changes in tool geometry were required. As a group, these materials have higher hardness and lower toughness than HSS. Also, their shear and tensile strengths are low relative to their compressive strengths, and their properties cannot be manipulated through heat treatment like those of HSS. Finally, cost per unit weight for these very hard materials is higher than the cost of HSS. These factors have affected cutting-tool design for the very hard tool materials in several ways.

First, the very hard materials must be designed with either negative rake or small positive angles. This change tends to load the tool more in compression and less in shear, thus favoring the high compressive strength of these harder materials. Cemented carbides, for example, are used with rake angles typically in the range from $-5°$ to $+10°$. Ceramics have rake angles between $-5°$ and $-15°$. Relief angles are made as small as possible ($5°$ is typical) to provide as much support for the cutting edge as possible.

Another difference is the way in which the cutting edge of the tool is held in position. The alternative ways of holding and presenting the cutting edge for

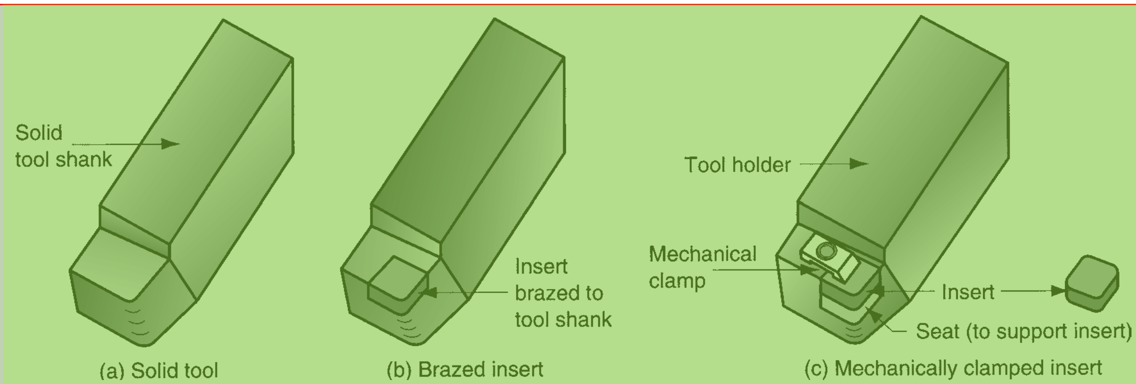


FIGURE 22.9 Three ways of holding and presenting the cutting edge for a single-point tool: (a) solid tool, typical of HSS; (b) brazed insert, one way of holding a cemented carbide insert; and (c) mechanically clamped insert, used for cemented carbides, ceramics, and other very hard tool materials.

a single-point tool are illustrated in Figure 22.9. The geometry of a HSS tool is ground from a solid shank, as shown in part (a) of the figure. The higher cost and differences in properties and processing of the harder tool materials have given rise to the use of inserts that are either brazed or mechanically clamped to a toolholder. Part (b) shows a brazed insert, in which a cemented carbide insert is brazed to a tool shank. The shank is made of tool steel for strength and toughness. Part (c) illustrates one possible design for mechanically clamping an insert in a toolholder. Mechanical clamping is used for cemented carbides, ceramics, and the other hard materials. The significant advantage of the mechanically clamped insert is that each insert contains multiple cutting edges. When an edge wears out, the insert is unclamped, indexed (rotated in the toolholder) to the next edge, and reclamped in the toolholder. When all of the cutting edges are worn, the insert is discarded and replaced.

Inserts Cutting-tool inserts are widely used in machining because they are economical and adaptable to many different types of machining operations: turning, boring, threading, milling, and even drilling. They are available in a variety of shapes and sizes for the variety of cutting situations encountered in practice. A square insert is shown in Figure 22.9(c). Other common shapes used in turning operations are displayed in Figure 22.10. In general, the largest point angle should be selected for strength and economy. Round inserts possess large point angles (and large nose radii) just because of their shape. Inserts with large point angles are inherently stronger and less likely to chip or break during cutting, but they require more power, and there is a greater likelihood of vibration. The economic advantage of round inserts is that they can be indexed multiple times for more cuts per insert. Square inserts present four cutting edges, triangular shapes have three edges, whereas rhombus shapes have only two. Fewer edges are a cost disadvantage. If both sides of the insert can be used (e.g., in most negative rake angle applications), then the number of cutting edges is doubled. Rhombus shapes are used (especially with acute point angles) because of their versatility and accessibility when a variety of operations are to be performed. These shapes can be more readily positioned in tight spaces and can be used not only for turning but also for facing, Figure 21.6(a), and contour turning, Figure 21.6(c).

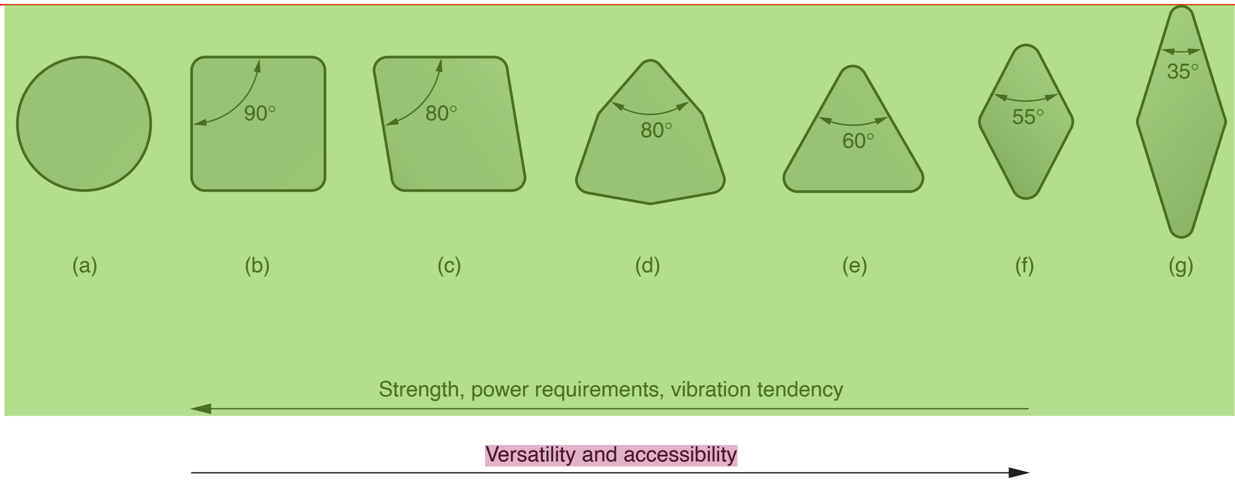


FIGURE 22.10 Common insert shapes: (a) round, (b) square, (c) rhombus with two 80° point angles, (d) hexagon with three 80° point angles, (e) triangle (equilateral), (f) rhombus with two 55° point angles, (g) rhombus with two 35° point angles. Also shown are typical features of the geometry. Strength, power requirements, and tendency for vibration increase with geometries on the left; whereas versatility and accessibility tend to be better with geometries on the right

Inserts are usually not made with perfectly sharp cutting edges, because a sharp edge is weaker and fractures more easily, especially for the very hard and brittle tool materials from which inserts are made (cemented carbides, coated carbides, cermets, ceramics, cBN, and diamond). Some kind of shape alteration is commonly performed on the cutting edge at an almost microscopic level. The effect of this ***edge preparation*** is to increase the strength of the cutting edge by providing a more gradual transition between the clearance edge and the rake face of the tool. Three common edge preparations are shown in Figure 22.11: (a) radius or edge rounding, also referred to as honed edge, (b) chamfer, and (c) land. For comparison, a perfectly sharp cutting edge is shown in (d). The radius in (a) is typically only about 0.025 mm (0.001 in), and the land in (c) is 15° or 20°. Combinations of these edge preparations are often applied to a single cutting edge to maximize the strengthening effect.

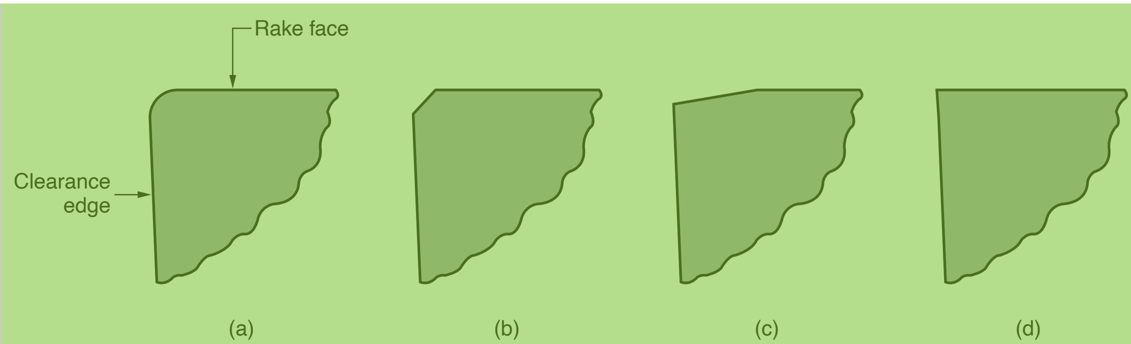


FIGURE 22.11 Three types of edge preparation that are applied to the cutting edge of an insert: (a) radius, (b) chamfer, (c) land, and (d) perfectly sharp edge (no edge preparation).

22.3.2 MULTIPLE-CUTTING-EDGE TOOLS

Most multiple-cutting-edge tools are used in machining operations in which the tool is rotated. Primary examples are drilling and milling. On the other hand, broaching and some sawing operations (hack sawing and band sawing) use multiple-cutting-edge tools that operate with a linear motion. Other sawing operations (circular sawing) use rotating saw blades.

Drills Various cutting tools are available for hole making, but the ***twist drill*** is by far the most common. It comes in diameters ranging from about 0.15 mm (0.006 in) to as large as 75 mm (3.0 in). Twist drills are widely used in industry to produce holes rapidly and economically.

The standard twist drill geometry is illustrated in Figure 22.12. The body of the drill has two spiral ***flutes*** (the spiral gives the twist drill its name). The angle of the spiral flutes is called the ***helix angle***, a typical value of which is around 30°. While drilling, the flutes act as passageways for extraction of chips from the hole. Although it is desirable for the flute openings to be large to provide maximum clearance for the chips, the body of the drill must be supported over its length. This support is provided by the ***web***, which is the thickness of the drill between the flutes.

The point of the twist drill has a conical shape. A typical value for the ***point angle*** is 118°. The point can be designed in various ways, but the most common design is a ***chisel edge***, as in Figure 22.12. Connected to the chisel edge are two cutting edges (sometimes called lips) that lead into the flutes. The portion of each flute adjacent to the cutting edge acts as the rake face of the tool.

The cutting action of the twist drill is complex. The rotation and feeding of the drill bit result in relative motion between the cutting edges and the workpiece to form the chips. The cutting speed along each cutting edge varies as a function of the distance from the axis of rotation. Accordingly, the efficiency of the cutting action varies, being most efficient at the outer diameter of the drill and least efficient at the center. In fact, the relative velocity at the drill point is zero, so no cutting takes place. Instead, the chisel edge of the drill point pushes aside the material at the center as it penetrates into the hole; a large thrust force is required to drive the twist drill forward into the hole. Also, at the beginning of the operation, the rotating chisel edge tends to wander on the surface of the work part, causing loss of positional accuracy. Various alternative drill point designs have been developed to address this problem.

Chip removal can be a problem in drilling. The cutting action takes place inside the hole, and the flutes must provide sufficient clearance throughout the length of

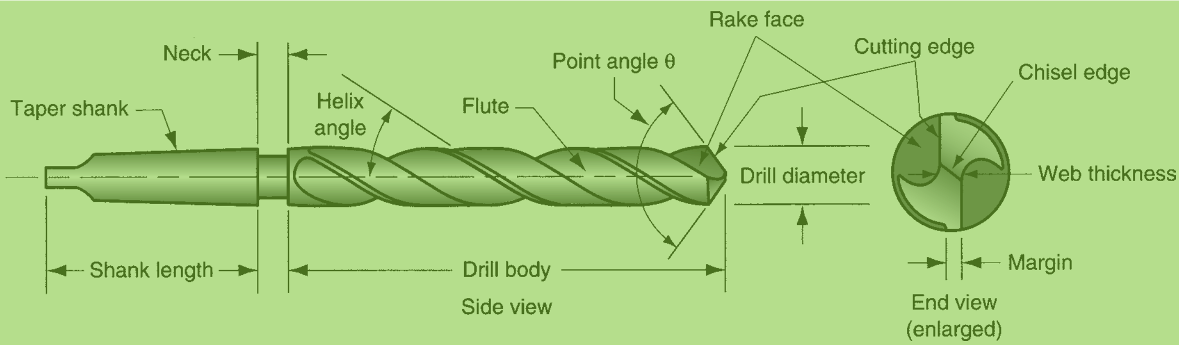


FIGURE 22.12 Standard geometry of a twist drill.

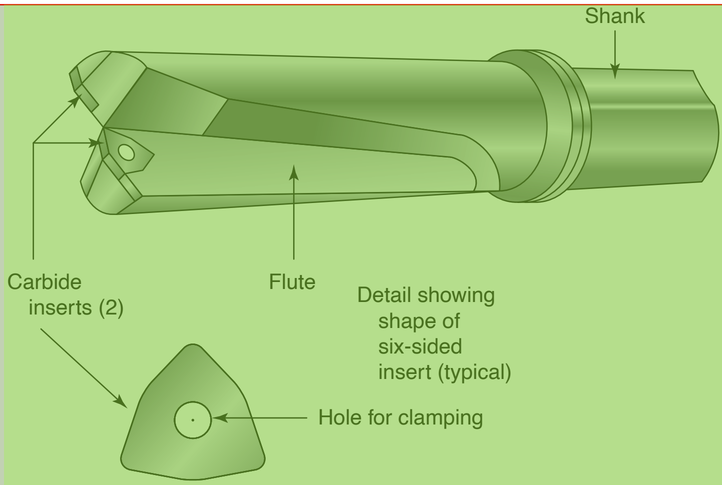


FIGURE 22.13 Straight-flute drill that uses indexable inserts.

the drill to allow the chips to be extracted from the hole. As the chip is formed it is forced through the flutes to the work surface. Friction makes matters worse in two ways. In addition to the usual friction in metal cutting between the chip and the rake face of the cutting edge, friction also results from rubbing between the outside diameter of the drill bit and the newly formed hole. This increases the temperature of the drill and work. Delivery of cutting fluid to the drill point to reduce the friction and heat is difficult because the chips are flowing in the opposite direction. Because of chip removal and heat, a twist drill is normally limited to a hole depth of about four times its diameter. Some twist drills are designed with internal holes running their lengths, through which cutting fluid can be pumped to the hole near the drill point, thus delivering the fluid directly to the cutting operation. An alternative approach with twist drills that do not have fluid holes is to use a "pecking" procedure during the drilling operation. In this procedure, the drill is periodically withdrawn from the hole to clear the chips before proceeding deeper.

Twist drills are normally made of high-speed steel. The geometry of the drill is fabricated before heat treatment, and then the outer shell of the drill (cutting edges and friction surfaces) is hardened while retaining an inner core that is relatively tough. Grinding is used to sharpen the cutting edges and shape the drill point.

Although twist drills are the most common hole-making tools, other drill types are also available. ***Straight-flute drills*** operate like twist drills except that the flutes for chip removal are straight along the length of the tool rather than spiraled. The simpler design of the straight-flute drill permits carbide tips to be used as the cutting edges, either as brazed or indexable inserts. Figure 22.13 illustrates the straight-flute indexable-insert drill. The cemented carbide inserts allow higher cutting speeds and greater production rates than HSS twist drills. However, the inserts limit how small the drills can be made. Thus, the diameter range of commercially available indexable-insert drills runs from about 16 mm (0.625 in) to about 127 mm (5 in) [9].

A straight-flute drill designed for deep-hole drilling is the ***gun drill***, shown in Figure 22.14. Whereas the twist drill is usually limited to a depth-to-diameter ratio of 4:1, and the straight-flute drill to about 3:1, the gun drill can cut holes up to 125 times its diameter. As shown in the figure, the gun drill has a carbide cutting edge, a single flute for chip removal, and a coolant hole running its complete length. In the typical gun drilling operation, the work rotates around the stationary drill (opposite

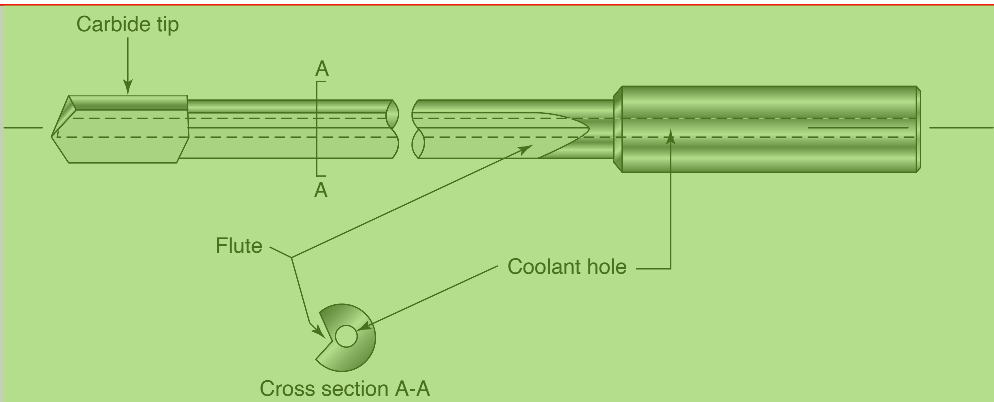


FIGURE 22.14 Gun drill.

of most drilling operations), and the coolant flows into the cutting process and out of the hole along the flute, carrying the chips with it. Gun drills range in diameter from less than 2 mm (0.075 in) to about 50 mm (2 in).

It was previously mentioned that twist drills are available with diameters up to 75 mm (3 in). Twist drills that large are uncommon because so much metal is required in the drill bit. An alternative for large diameter holes is the ***spade drill***, illustrated in Figure 22.15. Standard sizes range from 25 to 152 mm (1–6 in). The interchangeable drill bit is held in a toolholder, which provides rigidity during cutting. The mass of the spade drill is much less than a twist drill of the same diameter.

More information on hole-making tools can be found in references [3] and [9].

Milling Cutters Classification of milling cutters is closely associated with the milling operations described in Section 21.4.1. The major types of milling cutters are the following:

- ***Plain milling cutters.*** These are used for peripheral or slab milling. As Figures 21.17(a) and 21.18(a) indicate, they are cylinder shaped with several rows of

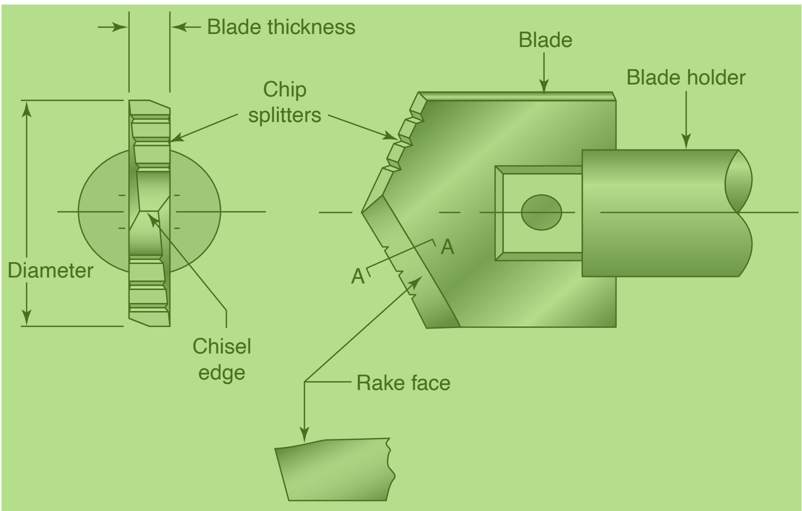


FIGURE 22.15 Spade drill.

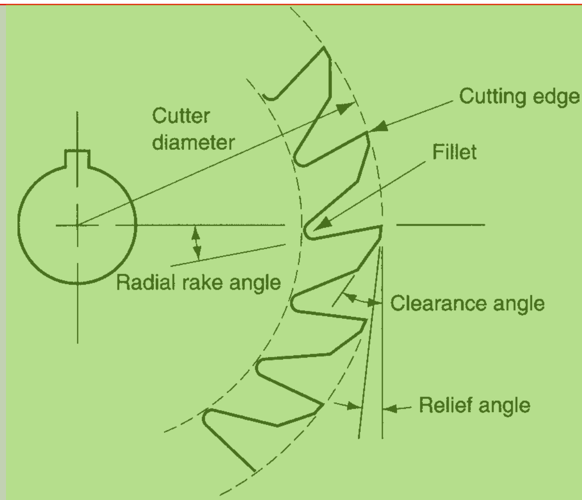


FIGURE 22.16 Tool geometry elements of an 18-tooth plain milling cutter.

teeth. The cutting edges are usually oriented at a helix angle (as in the figures) to reduce impact on entry into the work, and these cutters are called ***helical milling cutters***. Tool geometry elements of a plain milling cutter are shown in Figure 22.16.

➢ ***Form milling cutters***. These are peripheral milling cutters in which the cutting edges have a special profile that is to be imparted to the work, as in Figure 21.18(e). An important application is in gear making, in which the form milling cutter is shaped to cut the slots between adjacent gear teeth, thereby leaving the geometry of the gear teeth.

➢ ***Face milling cutters***. These are designed with teeth that cut on both the periphery as well as the end of the cutter. Face milling cutters can be made of HSS, as in Figure 21.17(b), or they can be designed to use cemented carbide inserts. Figure 22.17 shows a four-tooth face-milling cutter that uses inserts.

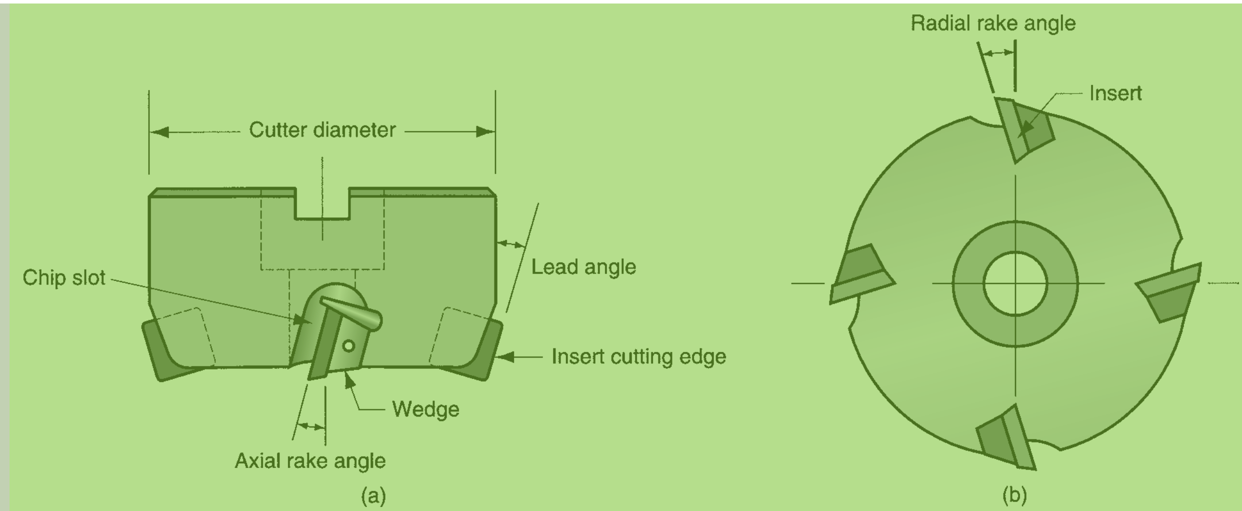


FIGURE 22.17 Tool geometry elements of a four-tooth face milling cutter: (a) side view and (b) bottom view.

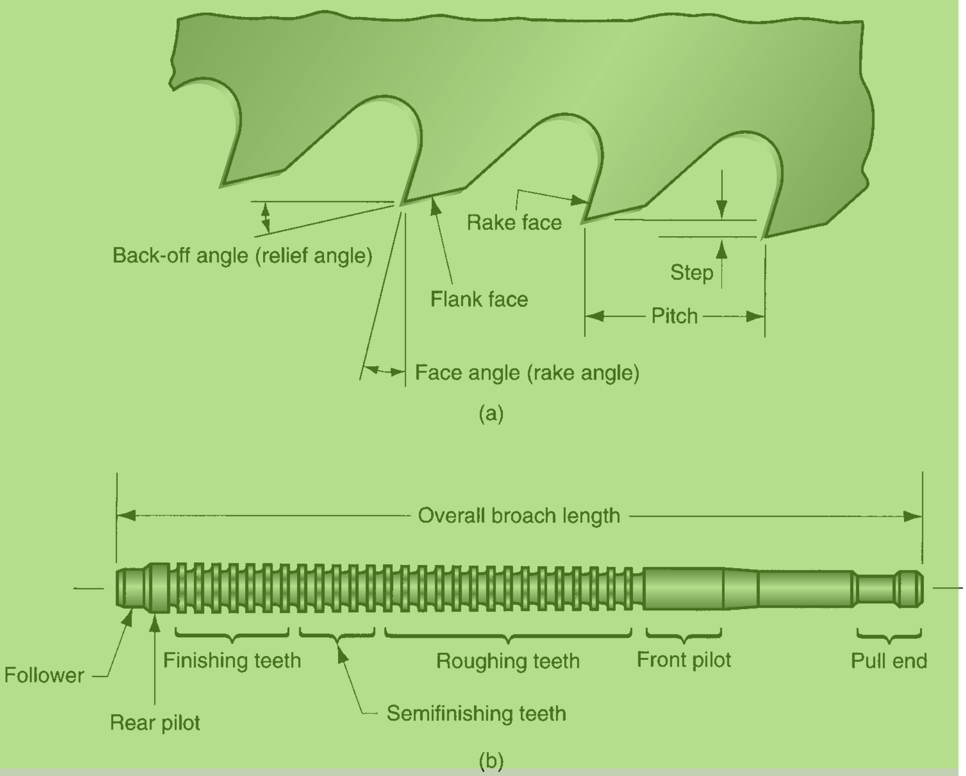


FIGURE 22.18 The broach: (a) terminology of the tooth geometry, and (b) a typical broach used for internal broaching.

➢ ***End milling cutters***. As shown in Figure 21.20(c), an end milling cutter looks like a drill bit, but close inspection indicates that it is designed for primary cutting with its peripheral teeth rather than its end. (A drill bit cuts only on its end as it penetrates into the work.) End mills are designed with square ends, ends with radii, and ball ends. End mills can be used for face milling, profile milling and pocketing, cutting slots, engraving, surface contouring, and die sinking.

Broaches The terminology and geometry of the broach are illustrated in Figure 22.18. The broach consists of a series of distinct cutting teeth along its length. Feed is accomplished by the increased step between successive teeth on the broach. This feeding action is unique among machining operations, because most operations accomplish feeding by a relative feed motion that is carried out by either the tool or the work. The total material removed in a single pass of the broach is the cumulative result of all the steps in the tool. The speed motion is accomplished by the linear travel of the tool past the work surface. The shape of the cut surface is determined by the contour of the cutting edges on the broach, particularly the final cutting edge. Owing to its complex geometry and the low speeds used in broaching, most broaches are made of HSS. In broaching of certain cast irons, the cutting edges are cemented carbide inserts either brazed or mechanically held in place on the broaching tool.

Saw Blades For each of the three sawing operations (Section 21.5.3), the saw blades possess certain common features, including tooth form, tooth spacing, and tooth set, as seen in Figure 22.19. ***Tooth form*** is concerned with the geometry of each cutting tooth. Rake angle, clearance angle, tooth spacing, and other features of geometry are

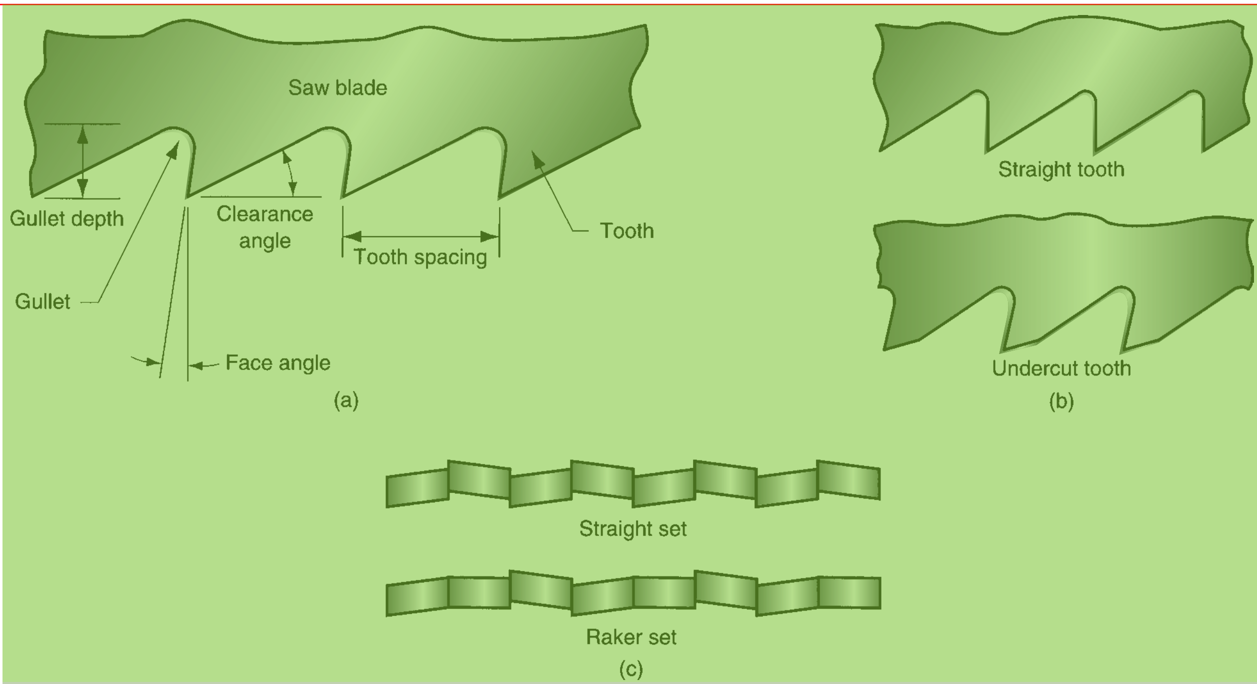


FIGURE 22.19 Features of saw blades: (a) nomenclature for saw blade geometries, (b) two common tooth forms, and (c) two types of tooth set.

shown in Figure 22.19(a). ***Tooth spacing*** is the distance between adjacent teeth on the saw blade. This parameter determines the size of the teeth and the size of the gullet between teeth. The gullet allows space for the formation of the chip by the adjacent cutting tooth. Different tooth forms are appropriate for different work materials and cutting situations. Two forms commonly used in hacksaw and bandsaw blades are shown in Figure 22.19(b). The ***tooth set*** permits the kerf cut by the saw blade to be wider than the width of the blade itself; otherwise the blade would bind against the walls of the slit made by the saw. Two common tooth sets are illustrated in Figure 22.19(c).

22.4 Cutting Fluids

A ***cutting fluid*** is any liquid or gas that is applied directly to the machining operation to improve cutting performance. Cutting fluids address two main problems: (1) heat generation at the shear zone and friction zone, and (2) friction at the tool–chip and tool–work interfaces. In addition to removing heat and reducing friction, cutting fluids provide additional benefits, such as washing away chips (especially in grinding and milling), reducing the temperature of the work part for easier handling, reducing cutting forces and power requirements, improving dimensional stability of the work part, and improving surface finish.

22.4.1 TYPES OF CUTTING FLUIDS

A variety of cutting fluids are commercially available. It is appropriate to discuss them first according to function and then to classify them according to chemical formulation.

Cutting Fluid Functions There are two general categories of cutting fluids, corresponding to the two main problems they are designed to address: coolants and lubricants. ***Coolants*** are cutting fluids designed to reduce the effects of heat in the machining operation. They have a limited effect on the amount of heat energy generated in cutting; instead, they carry away the heat that is generated, thereby reducing the temperature of tool and workpiece. This helps to prolong the life of the cutting tool. The capacity of a cutting fluid to reduce temperatures in machining depends on its thermal properties. Specific heat and thermal conductivity are the most important properties (Section 4.2.1). Water has high specific heat and thermal conductivity relative to other liquids, which is why water is used as the base in coolant-type cutting fluids. These properties allow the coolant to draw heat away from the operation, thereby reducing the temperature of the cutting tool.

Coolant-type cutting fluids seem to be most effective at relatively high cutting speeds in which heat generation and high temperatures are problems. They are most effective on tool materials that are most susceptible to temperature failures, such as high-speed steels, and are used frequently in turning and milling operations in which large amounts of heat are generated.

Lubricants are usually oil-based fluids (because oils possess good lubricating qualities) formulated to reduce friction at the tool–chip and tool–work interfaces. Lubricant cutting fluids operate by ***extreme pressure lubrication***, a special form of lubrication that involves formation of thin solid salt layers on the hot, clean metal surfaces through chemical reaction with the lubricant. Compounds of sulfur, chlorine, and phosphorous in the lubricant cause the formation of these surface layers, which act to separate the two metal surfaces (i.e., chip and tool). These extreme pressure films are significantly more effective in reducing friction in metal cutting than conventional lubrication, which is based on the presence of liquid films between the two surfaces.

Lubricant-type cutting fluids are most effective at lower cutting speeds. They tend to lose their effectiveness at high speeds, above about 120 m/min (400 ft/min), because the motion of the chip at these speeds prevents the cutting fluid from reaching the tool–chip interface. In addition, high cutting temperatures at these speeds cause the oils to vaporize before they can lubricate. Machining operations such as drilling and tapping usually benefit from lubricants. In these operations, built-up edge formation is retarded, and torque on the tool is reduced.

Although the principal purpose of a lubricant is to reduce friction, it also reduces the temperature in the operation through several mechanisms. First, the specific heat and thermal conductivity of the lubricant help to remove heat from the operation, thereby reducing temperatures. Second, because friction is reduced, the heat generated from friction is also reduced. Third, a lower coefficient of friction means a lower friction angle. According to Merchant's equation, Equation 21.16, a lower friction angle causes the shear plane angle to increase, hence reducing the amount of heat energy generated in the shear zone.

There is typically an overlapping effect between the two types of cutting fluids. Coolants are formulated with ingredients that help reduce friction. And lubricants have thermal properties that, although not as good as those of water, act to remove heat from the cutting operation. Cutting fluids (both coolants and lubricants) manifest their effect on the Taylor tool life equation through higher C values. Increases of 10% to 40% are typical. The slope n is not significantly affected.

Chemical Formulation of Cutting Fluids There are four categories of cutting fluids according to chemical formulation: (1) cutting oils, (2) emulsified oils,

(3) semichemical fluids, and (4) chemical fluids. All of these cutting fluids provide both coolant and lubricating functions. The cutting oils are most effective as lubricants, whereas the other three categories are more effective as coolants because they are primarily water.

Cutting oils are based on oil derived from petroleum, animal, marine, or vegetable origin. Mineral oils (petroleum based) are the principal type because of their abundance and generally desirable lubricating characteristics. To achieve maximum lubricity, several types of oils are often combined in the same fluid. Chemical additives are also mixed with the oils to increase lubricating qualities. These additives contain compounds of sulfur, chlorine, and phosphorous, and are designed to react chemically with the chip and tool surfaces to form solid films (extreme pressure lubrication) that help to avoid metal-to-metal contact between the two.

Emulsified oils consist of oil droplets suspended in water. The fluid is made by blending oil (usually mineral oil) in water using an emulsifying agent to promote blending and stability of the emulsion. A typical ratio of water to oil is 30:1. Chemical additives based on sulfur, chlorine, and phosphorous are often used to promote extreme pressure lubrication. Because they contain both oil and water, the emulsified oils combine cooling and lubricating qualities in one cutting fluid.

Chemical fluids are chemicals in a water solution rather than oils in emulsion. The dissolved chemicals include compounds of sulfur, chlorine, and phosphorous, plus wetting agents. The chemicals are intended to provide some degree of lubrication to the solution. Chemical fluids provide good coolant qualities but their lubricating qualities are less than the other cutting fluid types. ***Semichemical fluids*** have small amounts of emulsified oil added to increase the lubricating characteristics of the cutting fluid. In effect, they are a hybrid class between chemical fluids and emulsified oils.

Table 22.8 presents a troubleshooting guide for machining problems related to the use of cutting fluids.

TABLE • 22.8 Troubleshooting Guide for Problems Related to Cutting Fluids.

Problem	Likely conditions and symptoms	Possible Changes in Cutting Fluid
Heat	Premature tool failure due to high temperature	Increase fluid flow rate
	Cutting speed too high for tool	If cutting oil, reduce viscosity level
	Chip adheres to rake face	If cutting oil, try emulsifiable oil
	Continuous cutting (e.g., turning, drilling)	If emulsifiable oil, increase water proportion
		If emulsifiable oil, try chemical or semichemical fluid
Wear	Low cutting speed	If emulsifiable oil, try cutting oil
	Rapid tool wear	If emulsifiable oil, increase oil proportion
	Work metal is high tensile strength steel or heat-resistant alloy	If chemical fluid, try emulsifiable oil
	Work metal is abrasive (e.g., sand casting)	Try fluid with chemically active additives for extreme pressure lubrication
Chatter	Vibration	If dry, try using a cutting fluid to address vibration problem through hydraulic dampening
	Inadequate rigidity of setup	If cutting fluid, use fluid with higher viscosity

22.4.2 APPLICATION OF CUTTING FLUIDS

Cutting fluids are applied to machining operations in various ways, and this section considers these application methods. Also considered is the problem of cutting-fluid contamination and what steps can be taken to address this problem.

Application Methods The most common method is ***flooding***, sometimes called flood-cooling because it is generally used with coolant-type cutting fluids. In flooding, a steady stream of fluid is directed at the tool–work or tool–chip interface of the machining operation. A second method of delivery is ***mist application***, primarily used for water-based cutting fluids. In this method the fluid is directed at the operation in the form of a high-speed mist carried by a pressurized air stream. Mist application is generally not as effective as flooding in cooling the tool. However, because of the high-velocity air stream, mist application may be more effective in delivering the cutting fluid to areas that are difficult to access by conventional flooding.

Manual application by means of a squirt can or paint brush is sometimes used for applying lubricants in tapping and other operations in which cutting speeds are low and friction is a problem. It is generally not preferred by most production machine shops because of its variability in application.

Cutting Fluid Filtration and Dry Machining Cutting fluids become contaminated over time with a variety of foreign substances, such as tramp oil (machine oil, hydraulic fluid, etc.), garbage (cigarette butts, food, etc.), small chips, molds, fungi, and bacteria. In addition to causing odors and health hazards, contaminated cutting fluids do not perform their lubricating function as well. Alternative ways of dealing with this problem are to (1) replace the cutting fluid at regular and frequent intervals (perhaps twice per month); (2) use a filtration system to continuously or periodically clean the fluid; or (3) dry machining, that is, machining without cutting fluids. Because of growing concern about environmental pollution and associated legislation, disposing old fluids has become both costly and contrary to the general public welfare.

Filtration systems are being installed in numerous machine shops today to solve the contamination problem. Advantages of these systems include (1) prolonged cutting fluid life between changes—instead of replacing the fluid once or twice per month, coolant lives of 1 year have been reported; (2) reduced fluid disposal cost, since disposal is much less frequent when a filter is used; (3) cleaner cutting fluid for better working environment and reduced health hazards; (4) lower machine tool maintenance; and (5) longer tool life. There are various types of filtration systems for filtering cutting fluids. For the interested reader, filtration systems and the benefits of using them are discussed in reference [19].

The third alternative is called ***dry machining***, meaning that no cutting fluid is used. Dry machining avoids the problems of cutting fluid contamination, disposal, and filtration, but can lead to problems of its own: (1) overheating the tool, (2) operating at lower cutting speeds and production rates to prolong tool life, and (3) absence of chip removal benefits in grinding and milling. Cutting-tool producers have developed certain grades of carbides and coated carbides for use in dry machining.