TABLE OF CONTENTS

MACHINING OPERATIONS

CUTTING SPEEDS AND FEEDS

SPEED AND FEED TABLES

ESTIMATING SPEEDS AND MACHINING POWER

MACHINING ECONOMETRICS

TABLE OF CONTENTS

SCREW MACHINE FEEDS AND SPEEDS

CUTTING FLUIDS

MACHINING NONFERROUS METALS

GRINDING FEEDS AND SPEEDS

GRINDING AND OTHER ABRASIVE PROCESSES

GRINDING AND OTHER ABRASIVE PROCESSES (Cont.)

GRINDING AND OTHER ABRASIVE PROCESSES (Cont.)

TABLE OF CONTENTS

ARBORS, CHUCKS, AND SPINDLES

KNURLS AND KNURLING

MACHINE TOOL ACCURACY

NUMERICAL CONTROL

NUMERICAL CONTROL PROGRAMMING

CAD/CAM

CUTTING SPEEDS AND FEEDS

Work Materials.—The large number of work materials that are commonly machined vary greatly in their basic structure and the ease with which they can be machined. Yet it is possible to group together certain materials having similar machining characteristics, for the purpose of recommending the cutting speed at which they can be cut. Most materials that are machined are metals and it has been found that the most important single factor influencing the ease with which a metal can be cut is its microstructure, followed by any cold work that may have been done to the metal, which increases its hardness. Metals that have a similar, but not necessarily the same microstructure, will tend to have similar machining characteristics. Thus, the grouping of the metals in the accompanying tables has been done on the basis of their microstructure.

With the exception of a few soft and gummy metals, experience has shown that harder metals are more difficult to cut than softer metals. Furthermore, any given metal is more difficult to cut when it is in a harder form than when it is softer. It is more difficult to penetrate the harder metal and more power is required to cut it. These factors in turn will generate a higher cutting temperature at any given cutting speed, thereby making it necessary to use a slower speed, for the cutting temperature must always be kept within the limits that can be sustained by the cutting tool without failure. Hardness, then, is an important property that must be considered when machining a given metal. Hardness alone, however, cannot be used as a measure of cutting speed. For example, if pieces of AISI 11L17 and AISI 1117 steel both have a hardness of 150 Bhn, their recommended cutting speeds for high-speed steel tools will be 140 fpm and 130 fpm, respectively. In some metals, two entirely different microstructures can produce the same hardness. As an example, a fine pearlite microstructure and a tempered martensite microstructure can result in the same hardness in a steel. These microstructures will not machine alike. For practical purposes, however, information on hardness is usually easier to obtain than information on microstructure; thus, hardness alone is usually used to differentiate between different cutting speeds for machining a metal. In some situations, the hardness of a metal to be machined is not known. When the hardness is not known, the material condition can be used as a guide.

The surface of ferrous metal castings has a scale that is more difficult to machine than the metal below. Some scale is more difficult to machine than others, depending on the foundry sand used, the casting process, the method of cleaning the casting, and the type of metal cast. Special electrochemical treatments sometimes can be used that almost entirely eliminate the effect of the scale on machining, although castings so treated are not frequently encountered. Usually, when casting scale is encountered, the cutting speed is reduced approximately 5 or 10 per cent. Difficult-to-machine surface scale can also be encountered when machining hot-rolled or forged steel bars.

Metallurgical differences that affect machining characteristics are often found within a single piece of metal. The occurrence of hard spots in castings is an example. Different microstructures and hardness levels may occur within a casting as a result of variations in the cooling rate in different parts of the casting. Such variations are less severe in castings that have been heat treated. Steel bar stock is usually harder toward the outside than toward the center of the bar. Sometimes there are slight metallurgical differences along the length of a bar that can affect its cutting characteristics.

Cutting Tool Materials.—The recommended cutting feeds and speeds in the accompanying tables are given for high-speed steel, coated and uncoated carbides, ceramics, cermets, and polycrystalline diamonds. More data are available for HSS and carbides because these materials are the most commonly used. Other materials that are used to make cutting tools are cemented oxides or ceramics, cermets, cast nonferrous alloys (Stellite), single-crystal diamonds, polycrystalline diamonds, and cubic boron nitride.

Carbon Tool Steel: It is used primarily to make the less expensive drills, taps, and reamers. It is seldom used to make single-point cutting tools. Hardening in carbon steels is very shallow, although some have a small amount of vanadium and chromium added to improve their hardening quality. The cutting speed to use for plain carbon tool steel should be approximately one-half of the recommended speed for high-speed steel.

High-Speed Steel: This designates a number of steels having several properties that enhance their value as cutting tool material. They can be hardened to a high initial or room-temperature hardness ranging from 63 Rc to 65 Rc for ordinary high-speed steels and up to 70 Rc for the so-called superhigh-speed steels. They can retain sufficient hardness at temperatures up to 1,000 to 1,100°F to enable them to cut at cutting speeds that will generate these tool temperatures, and they will return to their original hardness when cooled to room temperature. They harden very deeply, enabling high-speed steels to be ground to the tool shape from solid stock and to be reground many times without sacrificing hardness at the cutting edge. High-speed steels can be made soft by annealing so that they can be machined into complex cutting tools such as drills, reamers, and milling cutters and then hardened.

The principal alloying elements of high-speed steels are tungsten (W), molybdenum (Mo), chromium (Cr), vanadium (V), together with carbon (C). There are a number of grades of high-speed steel that are divided into two types: tungsten high-speed steels and molybdenum high-speed steels. Tungsten high-speed steels are designated by the prefix T before the number that designates the grade. Molybdenum high-speed steels are designated by the prefix letter M. There is little performance difference between comparable grades of tungsten or molybdenum high-speed steel.

The addition of 5 to 12 per cent cobalt to high-speed steel increases its hardness at the temperatures encountered in cutting, thereby improving its wear resistance and cutting efficiency. Cobalt slightly increases the brittleness of high-speed steel, making it susceptible to chipping at the cutting edge. For this reason, cobalt high-speed steels are primarily made into single-point cutting tools that are used to take heavy roughing cuts in abrasive materials and through rough abrasive surface scales.

The M40 series and T15 are a group of high-hardness or so-called super high-speed steels that can be hardened to 70 Rc; however, they tend to be brittle and difficult to grind. For cutting applications, they are usually heat treated to 67–68 Rc to reduce their brittleness and tendency to chip. The M40 series is appreciably easier to grind than T15. They are recommended for machining tough die steels and other difficult-to-cut materials; they are not recommended for applications where conventional high-speed steels perform well. High-speed steels made by the powder-metallurgy process are tougher and have an improved grindability when compared with similar grades made by the customary process. Tools made of these steels can be hardened about 1 Rc higher than comparable high-speed steels made by the customary process without a sacrifice in toughness. They are particularly useful in applications involving intermittent cutting and where tool life is limited by chipping. All these steels augment rather than replace the conventional high-speed steels.

Cemented Carbides: They are also called sintered carbides or simply carbides. They are harder than high-speed steels and have excellent wear resistance. Information on cemented carbides and other hard metal tools is included in the section *CEMENTED CARBIDES* starting on page 747.

Cemented carbides retain a very high degree of hardness at temperatures up to 1400°F and even higher; therefore, very fast cutting speeds can be used. When used at fast cutting speeds, they produce good surface finishes on the workpiece. Carbides are more brittle than high-speed steel and, therefore, must be used with more care.

Hundreds of grades of carbides are available and attempts to classify these grades by area of application have not been entirely successful.

There are four distinct types of carbides: 1) straight tungsten carbides; 2) crater-resistant carbides; 3) titanium carbides; and 4) coated carbides.

Straight Tungsten Carbide: This is the most abrasion-resistant cemented carbide and is used to machine gray cast iron, most nonferrous metals, and nonmetallic materials, where abrasion resistance is the primary criterion. Straight tungsten carbide will rapidly form a crater on the tool face when used to machine steel, which reduces the life of the tool. Titanium carbide is added to tungsten carbide in order to counteract the rapid formation of the crater. In addition, tantalum carbide is usually added to prevent the cutting edge from deforming when subjected to the intense heat and pressure generated in taking heavy cuts.

Crater-Resistant Carbides: These carbides, containing titanium and tantalum carbides in addition to tungsten carbide, are used to cut steels, alloy cast irons, and other materials that have a strong tendency to form a crater.

Titanium Carbides: These carbides are made entirely from titanium carbide and small amounts of nickel and molybdenum. They have an excellent resistance to cratering and to heat. Their high hot hardness enables them to operate at higher cutting speeds, but they are more brittle and less resistant to mechanical and thermal shock. Therefore, they are not recommended for taking heavy or interrupted cuts. Titanium carbides are less abrasion-resistant and not recommended for cutting through scale or oxide films on steel. Although the resistance to cratering of titanium carbides is excellent, failure caused by crater formation can sometimes occur because the chip tends to curl very close to the cutting edge, thereby forming a small crater in this region that may break through.

Coated Carbides: These are available only as indexable inserts because the coating would be removed by grinding. The principal coating materials are titanium carbide (TiC), titanium nitride (TiN), and aluminum oxide (Al_2O_3). A very thin layer (approximately 0.0002 in.) of coating material is deposited over a cemented carbide insert; the material below the coating is called the substrate. The overall performance of the coated carbide is limited by the substrate, which provides the required toughness and resistance to deformation and thermal shock. With an equal tool life, coated carbides can operate at higher cutting speeds than uncoated carbides. The increase may be 20 to 30 per cent and sometimes up to 50 per cent faster. Titanium carbide and titanium nitride coated carbides usually operate in the medium (200–800 fpm) cutting speed range, and aluminum oxide coated carbides are used in the higher (800–1600 fpm) cutting speed range.

Carbide Grade Selection: The selection of the best grade of carbide for a particular application is very important. An improper grade of carbide will result in a poor performance—it may even cause the cutting edge to fail before any significant amount of cutting has been done. Because of the many grades and the many variables that are involved, the carbide producers should be consulted to obtain recommendations for the application of their grades of carbide. A few general guidelines can be given that are useful to form an orientation. Metal cutting carbides usually range in hardness from about 89.5 Ra (Rockwell A Scale) to 93.0 Ra with the exception of titanium carbide, which has a hardness range of 90.5 Ra to 93.5 Ra. Generally, the harder carbides are more wear-resistant and more brittle, whereas the softer carbides are less wear-resistant but tougher. A choice of hardness must be made to suit the given application. The very hard carbides are generally used for taking light finishing cuts. For other applications, select the carbide that has the highest hardness with sufficient strength to prevent chipping or breaking. Straight tungsten carbide grades should always be used unless cratering is encountered. Straight tungsten carbides are used to machine gray cast iron, ferritic malleable iron, austenitic stainless steel, high-temperature alloys, copper, brass, bronze, aluminum alloys, zinc alloy die castings, and plastics. Crater-resistant carbides should be used to machine plain carbon steel, alloy steel, tool steel, pearlitic malleable iron, nodular iron, other highly alloyed cast irons, ferritic stainless steel, martensitic stainless steel, and certain high-temperature alloys. Titanium carbides are recommended for taking high-speed finishing and semifinishing cuts on steel, especially the low-carbon, low-alloy steels, which are less abrasive and have a strong tendency to form a crater. They are also used to take light cuts on alloy cast iron and on some high-nickel alloys. Nonferrous materials, such as some aluminum alloys and brass, that are essentially nonabrasive may also be machined with titanium carbides. Abrasive materials and others that should not be machined with titanium carbides include gray cast iron, titanium alloys, cobalt- and nickel-base superalloys, stainless steel, bronze, many aluminum alloys, fiberglass, plastics, and graphite. The feed used should not exceed about 0.020 inch per revolution.

Coated carbides can be used to take cuts ranging from light finishing to heavy roughing on most materials that can be cut with these carbides. The coated carbides are recommended for machining all free-machining steels, all plain carbon and alloy steels, tool steels, martensitic and ferritic stainless steels, precipitation-hardening stainless steels, alloy cast iron, pearlitic and martensitic malleable iron, and nodular iron. They are also recommended for taking light finishing and roughing cuts on austenitic stainless steels. Coated carbides should not be used to machine nickel- and cobalt-base superalloys, titanium and titanium alloys, brass, bronze, aluminum alloys, pure metals, refractory metals, and nonmetals such as fiberglass, graphite, and plastics.

Ceramic Cutting Tool Materials: These are made from finely powdered aluminum oxide particles sintered into a hard dense structure without a binder material. Aluminum oxide is also combined with titanium carbide to form a composite, which is called a cermet. These materials have a very high hot hardness enabling very high cutting speeds to be used. For example, ceramic cutting tools have been used to cut AISI 1040 steel at a cutting speed of 18,000 fpm with a satisfactory tool life. However, much lower cutting speeds, in the range of 1000 to 4000 fpm and lower, are more common because of limitations placed by the machine tool, cutters, and chucks. Although most applications of ceramic and cermet cutting tool materials are for turning, they have also been used successfully for milling. Ceramics and cermets are relatively brittle and a special cutting edge preparation is required to prevent chipping or edge breakage. This preparation consists of honing or grinding a narrow flat land, 0.002 to 0.006 inch wide, on the cutting edge that is made about 30 degrees with respect to the tool face. For some heavy-duty applications, a wider land is used. The setup should be as rigid as possible and the feed rate should not normally exceed 0.020 inch, although 0.030 inch has been used successfully. Ceramics and cermets are recommended for roughing and finishing operations on all cast irons, plain carbon and alloy steels, and stainless steels. Materials up to a hardness of 60 Rockwell C Scale can be cut with ceramic and cermet cutting tools. These tools should not be used to machine aluminum and aluminum alloys, magnesium alloys, titanium, and titanium alloys.

Cast Nonferrous Alloy: Cutting tools of this alloy are made from tungsten, tantalum, chromium, and cobalt plus carbon. Other alloying elements are also used to produce materials with high temperature and wear resistance. These alloys cannot be softened by heat treatment and must be cast and ground to shape. The room-temperature hardness of cast nonferrous alloys is lower than for high-speed steel, but the hardness and wear resistance is retained to a higher temperature. The alloys are generally marketed under trade names such as Stellite, Crobalt, and Tantung. The initial cutting speed for cast nonferrous tools can be 20 to 50 per cent greater than the recommended cutting speed for high-speed steel as given in the accompanying tables.

Diamond Cutting Tools: These are available in three forms: single-crystal natural diamonds shaped to a cutting edge and mounted on a tool holder on a boring bar; polycrystalline diamond indexable inserts made from synthetic or natural diamond powders that have been compacted and sintered into a solid mass, and chemically vapor-deposited diamond. Single-crystal and polycrystalline diamond cutting tools are very wear-resistant, and are recommended for machining abrasive materials that cause other cutting tool materials to wear rapidly. Typical of the abrasive materials machined with single-crystal and polycrystalline diamond tools and cutting speeds used are the following: fiberglass, 300 to 1000 fpm; fused silica, 900 to 950 fpm; reinforced melamine plastics, 350 to 1000 fpm; reinforced phenolic plastics, 350 to 1000 fpm; thermosetting plastics, 300 to 2000 fpm; Teflon, 600 fpm; nylon, 200 to 300 fpm; mica, 300 to 1000 fpm; graphite, 200 to 2000 fpm; babbitt bearing metal, 700 fpm; and aluminum-silicon alloys, 1000 to 2000 fpm. Another important application of diamond cutting tools is to produce fine surface finishes on soft nonferrous metals that are difficult to finish by other methods. Surface finishes of 1 to 2 microinches can be readily obtained with single-crystal diamond tools, and finishes down to 10 microinches can be obtained with polycrystalline diamond tools. In addition to babbitt and the aluminum-silicon alloys, other metals finished with diamond tools include: soft aluminum, 1000 to 2000 fpm; all wrought and cast aluminum alloys, 600 to 1500 fpm; copper, 1000 fpm; brass, 500 to 1000 fpm; bronze, 300 to 600 fpm; oilite bearing metal, 500 fpm; silver, gold, and platinum, 300 to 2500 fpm; and zinc, 1000 fpm. Ferrous alloys, such as cast iron and steel, should not be machined with diamond cutting tools because the high cutting temperatures generated will cause the diamond to transform into carbon.

Chemically Vapor-Deposited (CVD) Diamond: This is a new tool material offering performance characteristics well suited to highly abrasive or corrosive materials, and hard-to-machine composites. CVD diamond is available in two forms: thick-film tools, which are fabricated by brazing CVD diamond tips, approximately 0.020 inch (0.5 mm) thick, to carbide substrates; and thin-film tools, having a pure diamond coating over the rake and flank surfaces of a ceramic or carbide substrate.

CVD is pure diamond, made at low temperatures and pressures, with no metallic binder phase. This diamond purity gives CVD diamond tools extreme hardness, high abrasion resistance, low friction, high thermal conductivity, and chemical inertness. CVD tools are generally used as direct replacements for PCD (polycrystalline diamond) tools, primarily in finishing, semifinishing, and continuous turning applications of extremely wear-intensive materials. The small grain size of CVD diamond (ranging from less than 1 μm to 50 μm) yields superior surface finishes compared with PCD, and the higher thermal conductivity and better thermal and chemical stability of pure diamond allow CVD tools to operate at faster speeds without generating harmful levels of heat. The extreme hardness of CVD tools may also result in significantly longer tool life.

CVD diamond cutting tools are recommended for the following materials: aluminum and other ductile; nonferrous alloys such as copper, brass, and bronze; and highly abrasive composite materials such as graphite, carbon-carbon, carbon-filled phenolic, fiberglass, and honeycomb materials.

Cubic Boron Nitride (CBN): Next to diamond, CBN is the hardest known material. It will retain its hardness at a temperature of 1800°F and higher, making it an ideal cutting tool material for machining very hard and tough materials at cutting speeds beyond those possible with other cutting tool materials. Indexable inserts and cutting tool blanks made from this material consist of a layer, approximately 0.020 inch thick, of polycrystalline cubic boron nitride firmly bonded to the top of a cemented carbide substrate. Cubic boron nitride is recommended for rough and finish turning hardened plain carbon and alloy steels, hardened tool steels, hard cast irons, all hardness grades of gray cast iron, and superalloys. As a class, the superalloys are not as hard as hardened steel; however, their combination of high strength and tendency to deform plastically under the pressure of the cut, or gumminess, places them in the class of hard-to-machine materials. Conventional materials that can be readily machined with other cutting tool materials should not be machined with cubic boron nitride. Round indexable CBN inserts are recommended when taking severe cuts in order to provide maximum strength to the insert. When using square or triangular inserts, a large lead angle should be used, normally 15°, and whenever possible, 45°. A negative rake angle should always be used, which for most applications is negative 5°. The relief angle should be 5° to 9°. Although cubic boron nitride cutting tools can be used without a coolant, flooding the tool with a water-soluble type coolant is recommended.

Cutting Speed, Feed, Depth of Cut, Tool Wear, and Tool Life.—The cutting conditions that determine the rate of metal removal are the cutting speed, the feed rate, and the depth of cut. These cutting conditions and the nature of the material to be cut determine the power required to take the cut. The cutting conditions must be adjusted to stay within the power available on the machine tool to be used. Power requirements are discussed in Estimating Machining Power later in this section.

The cutting conditions must also be considered in relation to the tool life. Tool life is defined as the cutting time to reach a predetermined amount of wear, usually flank wear. Tool life is determined by assessing the time—the tool life—at which a given predetermined flank wear is reached (0.01, 0.015, 0.025, 0.03 inch, for example). This amount of wear is called the tool wear criterion, and its size depends on the tool grade used. Usually, a tougher grade can be used with a bigger flank wear, but for finishing operations, where close tolerances are required, the wear criterion is relatively small. Other wear criteria are a predetermined value of the machined surface roughness and the depth of the crater that develops on the rake face of the tool.

The ANSI standard, Specification For Tool Life Testing With Single-Point Tools (ANSI B94.55M-1985), defines the end of tool life as a given amount of wear on the flank of a tool. This standard is followed when making scientific machinability tests with single-point cutting tools in order to achieve uniformity in testing procedures so that results from different machinability laboratories can be readily compared. It is not practicable or necessary to follow this standard in the shop; however, it should be understood that the cutting conditions and tool life are related.

Tool life is influenced most by cutting speed, then by the feed rate, and least by the depth of cut. When the depth of cut is increased to about 10 times greater than the feed, a further increase in the depth of cut will have no significant effect on the tool life. This characteristic of the cutting tool performance is very important in determining the operating or cutting conditions for machining metals. Conversely, if the cutting speed or the feed is decreased, the increase in the tool life will be proportionately greater than the decrease in the cutting speed or the feed.

Tool life is reduced when either feed or cutting speed is increased. For example, the cutting speed and the feed may be increased if a shorter tool life is accepted; furthermore, the reduction in the tool life will be proportionately greater than the increase in the cutting speed or the feed. However, it is less well understood that a higher feed rate (feed/rev × speed) may result in a longer tool life if a higher feed/rev is used in combination with a lower cutting speed. This principle is well illustrated in the speed tables of this section, where two sets of feed and speed data are given (labeled *optimum* and *average*) that result in the same tool life. The *optimum* set results in a greater feed rate (i.e., increased productivity) although the feed/rev is higher and cutting speed lower than the *average* set. Complete instructions for using the speed tables and for estimating tool life are given in *How to Use the Feeds and Speeds Tables* starting on page 991.

Selecting Cutting Conditions.—The first step in establishing the cutting conditions is to select the depth of cut. The depth of cut will be limited by the amount of metal that is to be machined from the workpiece, by the power available on the machine tool, by the rigidity of the workpiece and the cutting tool, and by the rigidity of the setup. The depth of cut has the least effect upon the tool life, so the heaviest possible depth of cut should always be used.

The second step is to select the feed (feed/rev for turning, drilling, and reaming, or feed/tooth for milling). The available power must be sufficient to make the required depth of cut at the selected feed. The maximum feed possible that will produce an acceptable surface finish should be selected.

The third step is to select the cutting speed. Although the accompanying tables provide recommended cutting speeds and feeds for many materials, experience in machining a certain material may form the best basis for adjusting the given cutting speeds to a particular job. However, in general, the depth of cut should be selected first, followed by the feed, and last the cutting speed.

Table 1. Tool Troubleshooting Check List

Problem	Tool Material	Remedy
Excessive flank wear—Tool life too short	Carbide	1. Change to harder, more wear-resistant grade 2. Reduce the cutting speed 3. Reduce the cutting speed and increase the feed to maintain production 4. Reduce the feed 5. For work-hardenable materials—increase the feed 6. Increase the lead angle 7. Increase the relief angles
	HSS	1. Use a coolant 2. Reduce the cutting speed 3. Reduce the cutting speed and increase the feed to maintain production 4. Reduce the feed 5. For work-hardenable materials—increase the feed 6. Increase the lead angle 7. Increase the relief angle
Excessive cratering	Carbide	1. Use a crater-resistant grade 2. Use a harder, more wear-resistant grade 3. Reduce the cutting speed 4. Reduce the feed 5. Widen the chip breaker groove
	HSS	1. Use a coolant 2. Reduce the cutting speed 3. Reduce the feed 4. Widen the chip breaker groove
Cutting edge chipping	Carbide	1. Increase the cutting speed 2. Lightly hone the cutting edge 3. Change to a tougher grade 4. Use negative-rake tools 5. Increase the lead angle 6. Reduce the feed 7. Reduce the depth of cut 8. Reduce the relief angles 9. If low cutting speed must be used, use a high-additive EP cutting fluid
	HSS	1. Use a high additive EP cutting fluid 2. Lightly hone the cutting edge before using 3. Increase the lead angle 4. Reduce the feed 5. Reduce the depth of cut 6. Use a negative rake angle 7. Reduce the relief angles
	Carbide and HSS	1. Check the setup for cause if chatter occurs 2. Check the grinding procedure for tool overheating 3. Reduce the tool overhang
Cutting edge deformation	Carbide	1. Change to a grade containing more tantalum 2. Reduce the cutting speed 3. Reduce the feed
Poor surface finish	Carbide	1. Increase the cutting speed 2. If low cutting speed must be used, use a high additive EP cutting fluid 4. For light cuts, use straight titanium carbide grade 5. Increase the nose radius 6. Reduce the feed 7. Increase the relief angles 8. Use positive rake tools

Table 1. *(Continued)* **Tool Troubleshooting Check List**

Problem	Tool Material	Remedy
Poor surface finish *(Continued)*	HSS	1. Use a high additive EP cutting fluid 2. Increase the nose radius 3. Reduce the feed 4. Increase the relief angles 5. Increase the rake angles
	Diamond	1. Use diamond tool for soft materials
Notching at the depth of cut line	Carbide and HSS	1. Increase the lead angle 2. Reduce the feed

Cutting Speed Formulas.—Most machining operations are conducted on machine tools having a rotating spindle. Cutting speeds are usually given in feet or meters per minute and these speeds must be converted to spindle speeds, in revolutions per minute, to operate the machine. Conversion is accomplished by use of the following formulas:

For U.S. units:

$$N = \frac{12V}{\pi D} = 3.82\frac{V}{D} \text{ rpm}$$

For metric units:

$$N = \frac{1000V}{\pi D} = 318.3\frac{V}{D} \text{ rpm}$$

where N is the spindle speed in revolutions per minute (rpm); V is the cutting speed in feet per minute (fpm) for U.S. units and meters per minute (m/min) for metric units. In turning, D is the diameter of the workpiece; in milling, drilling, reaming, and other operations that use a rotating tool, D is the cutter diameter in inches for U.S. units and in millimeters for metric units. $\pi = 3.1417$.

Example: The cutting speed for turning a 4-inch (102-mm) diameter bar has been found to be 575 fpm (175.3 m/min). Using both the inch and metric formulas, calculate the lathe spindle speed.

$$N = \frac{12V}{\pi D} = \frac{12 \times 575}{3.1417 \times 4} = 549 \text{ rpm} \qquad N = \frac{1000V}{\pi D} = \frac{1000 \times 175.3}{3.1417 \times 102} = 547 \text{ rpm}$$

The small difference in the answers is due to rounding off the numbers and to the lack of precision of the inch–metric conversion.

When the cutting tool or workpiece diameter and the spindle speed in rpm are known, it is often necessary to calculate the cutting speed in feet or meters per minute. In this event, the following formulas are used.

For U.S. units:

$$V = \frac{\pi DN}{12} \text{ fpm}$$

For metric units:

$$V = \frac{\pi DN}{1000} \text{ m/min}$$

As in the previous formulas, N is the rpm and D is the diameter in inches for the U.S. unit formula and in millimeters for the metric formula.

Example: Calculate the cutting speed in feet per minute and in meters per minute if the spindle speed of a $\frac{3}{4}$-inch (19.05-mm) drill is 400 rpm.

$$V = \frac{\pi DN}{12} = \frac{\pi \times 0.75 \times 400}{12} = 78.5 \text{ fpm}$$

$$V = \frac{\pi DN}{1000} = \frac{\pi \times 19.05 \times 400}{1000} = 24.9 \text{ m/min}$$

Cutting Speeds and Equivalent RPM for Drills of Number and Letter Sizes

Size No.	Cutting Speed, Feet per Minute										
	30′	40′	50′	60′	70′	80′	90′	100′	110′	130′	150′
	Revolutions per Minute for Number Sizes										
1	503	670	838	1005	1173	1340	1508	1675	1843	2179	2513
2	518	691	864	1037	1210	1382	1555	1728	1901	2247	2593
4	548	731	914	1097	1280	1462	1645	1828	2010	2376	2741
6	562	749	936	1123	1310	1498	1685	1872	2060	2434	2809
8	576	768	960	1151	1343	1535	1727	1919	2111	2495	2879
10	592	790	987	1184	1382	1579	1777	1974	2171	2566	2961
12	606	808	1010	1213	1415	1617	1819	2021	2223	2627	3032
14	630	840	1050	1259	1469	1679	1889	2099	2309	2728	3148
16	647	863	1079	1295	1511	1726	1942	2158	2374	2806	3237
18	678	904	1130	1356	1582	1808	2034	2260	2479	2930	3380
20	712	949	1186	1423	1660	1898	2135	2372	2610	3084	3559
22	730	973	1217	1460	1703	1946	2190	2433	2676	3164	3649
24	754	1005	1257	1508	1759	2010	2262	2513	2764	3267	3769
26	779	1039	1299	1559	1819	2078	2338	2598	2858	3378	3898
28	816	1088	1360	1631	1903	2175	2447	2719	2990	3534	4078
30	892	1189	1487	1784	2081	2378	2676	2973	3270	3864	4459
32	988	1317	1647	1976	2305	2634	2964	3293	3622	4281	4939
34	1032	1376	1721	2065	2409	2753	3097	3442	3785	4474	5162
36	1076	1435	1794	2152	2511	2870	3228	3587	3945	4663	5380
38	1129	1505	1882	2258	2634	3010	3387	3763	4140	4892	5645
40	1169	1559	1949	2339	2729	3118	3508	3898	4287	5067	5846
42	1226	1634	2043	2451	2860	3268	3677	4085	4494	5311	6128
44	1333	1777	2221	2665	3109	3554	3999	4442	4886	5774	6662
46	1415	1886	2358	2830	3301	3773	4244	4716	5187	6130	7074
48	1508	2010	2513	3016	3518	4021	4523	5026	5528	6534	7539
50	1637	2183	2729	3274	3820	4366	4911	5457	6002	7094	8185
52	1805	2406	3008	3609	4211	4812	5414	6015	6619	7820	9023
54	2084	2778	3473	4167	4862	5556	6251	6945	7639	9028	10417
Size	Revolutions per Minute for Letter Sizes										
A	491	654	818	982	1145	1309	1472	1636	1796	2122	2448
B	482	642	803	963	1124	1284	1445	1605	1765	2086	2407
C	473	631	789	947	1105	1262	1420	1578	1736	2052	2368
D	467	622	778	934	1089	1245	1400	1556	1708	2018	2329
E	458	611	764	917	1070	1222	1375	1528	1681	1968	2292
F	446	594	743	892	1040	1189	1337	1486	1635	1932	2229
G	440	585	732	878	1024	1170	1317	1463	1610	1903	2195
H	430	574	718	862	1005	1149	1292	1436	1580	1867	2154
I	421	562	702	842	983	1123	1264	1404	1545	1826	2106
J	414	552	690	827	965	1103	1241	1379	1517	1793	2068
K	408	544	680	815	951	1087	1223	1359	1495	1767	2039
L	395	527	659	790	922	1054	1185	1317	1449	1712	1976
M	389	518	648	777	907	1036	1166	1295	1424	1683	1942
N	380	506	633	759	886	1012	1139	1265	1391	1644	1897
O	363	484	605	725	846	967	1088	1209	1330	1571	1813
P	355	473	592	710	828	946	1065	1183	1301	1537	1774
Q	345	460	575	690	805	920	1035	1150	1266	1496	1726
R	338	451	564	676	789	902	1014	1127	1239	1465	1690
S	329	439	549	659	769	878	988	1098	1207	1427	1646
T	320	426	533	640	746	853	959	1066	1173	1387	1600
U	311	415	519	623	727	830	934	1038	1142	1349	1557
V	304	405	507	608	709	810	912	1013	1114	1317	1520
W	297	396	495	594	693	792	891	989	1088	1286	1484
X	289	385	481	576	672	769	865	962	1058	1251	1443
Y	284	378	473	567	662	756	851	945	1040	1229	1418
Z	277	370	462	555	647	740	832	925	1017	1202	1387

For fractional drill sizes, use the following table.

Revolutions per Minute for Various Cutting Speeds and Diameters

Dia., Inches	Cutting Speed, Feet per Minute											
	40	50	60	70	80	90	100	120	140	160	180	200
	Revolutions per Minute											
1/4	611	764	917	1070	1222	1376	1528	1834	2139	2445	2750	3056
5/16	489	611	733	856	978	1100	1222	1466	1711	1955	2200	2444
3/8	408	509	611	713	815	916	1018	1222	1425	1629	1832	2036
7/16	349	437	524	611	699	786	874	1049	1224	1398	1573	1748
1/2	306	382	459	535	611	688	764	917	1070	1222	1375	1528
9/16	272	340	407	475	543	611	679	813	951	1086	1222	1358
5/8	245	306	367	428	489	552	612	736	857	979	1102	1224
11/16	222	273	333	389	444	500	555	666	770	888	999	1101
3/4	203	254	306	357	408	458	508	610	711	813	914	1016
13/16	190	237	284	332	379	427	474	569	664	758	853	948
7/8	175	219	262	306	349	392	438	526	613	701	788	876
15/16	163	204	244	285	326	366	407	488	570	651	733	814
1	153	191	229	267	306	344	382	458	535	611	688	764
1 1/16	144	180	215	251	287	323	359	431	503	575	646	718
1 1/8	136	170	204	238	272	306	340	408	476	544	612	680
1 3/16	129	161	193	225	258	290	322	386	451	515	580	644
1 1/4	123	153	183	214	245	274	306	367	428	490	551	612
1 5/16	116	146	175	204	233	262	291	349	407	466	524	582
1 3/8	111	139	167	195	222	250	278	334	389	445	500	556
1 7/16	106	133	159	186	212	239	265	318	371	424	477	530
1 1/2	102	127	153	178	204	230	254	305	356	406	457	508
1 9/16	97.6	122	146	171	195	220	244	293	342	390	439	488
1 5/8	93.9	117	141	165	188	212	234	281	328	374	421	468
1 11/16	90.4	113	136	158	181	203	226	271	316	362	407	452
1 3/4	87.3	109	131	153	175	196	218	262	305	349	392	436
1 7/8	81.5	102	122	143	163	184	204	244	286	326	367	408
2	76.4	95.5	115	134	153	172	191	229	267	306	344	382
2 1/8	72.0	90.0	108	126	144	162	180	216	252	288	324	360
2 1/4	68.0	85.5	102	119	136	153	170	204	238	272	306	340
2 3/8	64.4	80.5	96.6	113	129	145	161	193	225	258	290	322
2 1/2	61.2	76.3	91.7	107	122	138	153	184	213	245	275	306
2 5/8	58.0	72.5	87.0	102	116	131	145	174	203	232	261	290
2 3/4	55.6	69.5	83.4	97.2	111	125	139	167	195	222	250	278
2 7/8	52.8	66.0	79.2	92.4	106	119	132	158	185	211	238	264
3	51.0	63.7	76.4	89.1	102	114	127	152	178	203	228	254
3 1/8	48.8	61.0	73.2	85.4	97.6	110	122	146	171	195	219	244
3 1/4	46.8	58.5	70.2	81.9	93.6	105	117	140	164	188	211	234
3 3/8	45.2	56.5	67.8	79.1	90.4	102	113	136	158	181	203	226
3 1/2	43.6	54.5	65.5	76.4	87.4	98.1	109	131	153	174	196	218
3 5/8	42.0	52.5	63.0	73.5	84.0	94.5	105	126	147	168	189	210
3 3/4	40.8	51.0	61.2	71.4	81.6	91.8	102	122	143	163	184	205
3 7/8	39.4	49.3	59.1	69.0	78.8	88.6	98.5	118	138	158	177	197
4	38.2	47.8	57.3	66.9	76.4	86.0	95.6	115	134	153	172	191
4 1/4	35.9	44.9	53.9	62.9	71.8	80.8	89.8	108	126	144	162	180
4 1/2	34.0	42.4	51.0	59.4	67.9	76.3	84.8	102	119	136	153	170
4 3/4	32.2	40.2	48.2	56.3	64.3	72.4	80.4	96.9	113	129	145	161
5	30.6	38.2	45.9	53.5	61.1	68.8	76.4	91.7	107	122	138	153
5 1/4	29.1	36.4	43.6	50.9	58.2	65.4	72.7	87.2	102	116	131	145
5 1/2	27.8	34.7	41.7	48.6	55.6	62.5	69.4	83.3	97.2	111	125	139
5 3/4	26.6	33.2	39.8	46.5	53.1	59.8	66.4	80.0	93.0	106	120	133
6	25.5	31.8	38.2	44.6	51.0	57.2	63.6	76.3	89.0	102	114	127
6 1/4	24.4	30.6	36.7	42.8	48.9	55.0	61.1	73.3	85.5	97.7	110	122
6 1/2	23.5	29.4	35.2	41.1	47.0	52.8	58.7	70.4	82.2	93.9	106	117
6 3/4	22.6	28.3	34.0	39.6	45.3	50.9	56.6	67.9	79.2	90.6	102	113
7	21.8	27.3	32.7	38.2	43.7	49.1	54.6	65.5	76.4	87.4	98.3	109
7 1/4	21.1	26.4	31.6	36.9	42.2	47.4	52.7	63.2	73.8	84.3	94.9	105
7 1/2	20.4	25.4	30.5	35.6	40.7	45.8	50.9	61.1	71.0	81.4	91.6	102
7 3/4	19.7	24.6	29.5	34.4	39.4	44.3	49.2	59.0	68.9	78.7	88.6	98.4
8	19.1	23.9	28.7	33.4	38.2	43.0	47.8	57.4	66.9	76.5	86.0	95.6

Revolutions per Minute for Various Cutting Speeds and Diameters

Dia., Inches	Cutting Speed, Feet per Minute											
	225	250	275	300	325	350	375	400	425	450	500	550
	Revolutions per Minute											
1/4	3438	3820	4202	4584	4966	5348	5730	6112	6493	6875	7639	8403
5/16	2750	3056	3362	3667	3973	4278	4584	4889	5195	5501	6112	6723
3/8	2292	2546	2801	3056	3310	3565	3820	4074	4329	4584	5093	5602
7/16	1964	2182	2401	2619	2837	3056	3274	3492	3710	3929	4365	4802
1/2	1719	1910	2101	2292	2483	2675	2866	3057	3248	3439	3821	4203
9/16	1528	1698	1868	2037	2207	2377	2547	2717	2887	3056	3396	3736
5/8	1375	1528	1681	1834	1987	2139	2292	2445	2598	2751	3057	3362
11/16	1250	1389	1528	1667	1806	1941	2084	2223	2362	2501	2779	3056
3/4	1146	1273	1401	1528	1655	1783	1910	2038	2165	2292	2547	2802
13/16	1058	1175	1293	1410	1528	1646	1763	1881	1998	2116	2351	2586
7/8	982	1091	1200	1310	1419	1528	1637	1746	1855	1965	2183	2401
15/16	917	1019	1120	1222	1324	1426	1528	1630	1732	1834	2038	2241
1	859	955	1050	1146	1241	1337	1432	1528	1623	1719	1910	2101
1 1/16	809	899	988	1078	1168	1258	1348	1438	1528	1618	1798	1977
1 1/8	764	849	933	1018	1103	1188	1273	1358	1443	1528	1698	1867
1 3/16	724	804	884	965	1045	1126	1206	1287	1367	1448	1609	1769
1 1/4	687	764	840	917	993	1069	1146	1222	1299	1375	1528	1681
1 5/16	654	727	800	873	946	1018	1091	1164	1237	1309	1455	1601
1 3/8	625	694	764	833	903	972	1042	1111	1181	1250	1389	1528
1 7/16	598	664	730	797	863	930	996	1063	1129	1196	1329	1461
1 1/2	573	636	700	764	827	891	955	1018	1082	1146	1273	1400
1 9/16	550	611	672	733	794	855	916	978	1039	1100	1222	1344
1 5/8	528	587	646	705	764	822	881	940	999	1057	1175	1293
1 11/16	509	566	622	679	735	792	849	905	962	1018	1132	1245
1 3/4	491	545	600	654	709	764	818	873	927	982	1091	1200
1 13/16	474	527	579	632	685	737	790	843	895	948	1054	1159
1 7/8	458	509	560	611	662	713	764	815	866	917	1019	1120
1 15/16	443	493	542	591	640	690	739	788	838	887	986	1084
2	429	477	525	573	620	668	716	764	811	859	955	1050
2 1/8	404	449	494	539	584	629	674	719	764	809	899	988
2 1/4	382	424	468	509	551	594	636	679	721	764	849	933
2 3/8	362	402	442	482	522	563	603	643	683	724	804	884
2 1/2	343	382	420	458	496	534	573	611	649	687	764	840
2 5/8	327	363	400	436	472	509	545	582	618	654	727	800
2 3/4	312	347	381	416	451	486	520	555	590	625	694	763
2 7/8	299	332	365	398	431	465	498	531	564	598	664	730
3	286	318	350	381	413	445	477	509	541	572	636	700
3 1/8	274	305	336	366	397	427	458	488	519	549	611	672
3 1/4	264	293	323	352	381	411	440	470	499	528	587	646
3 3/8	254	283	311	339	367	396	424	452	481	509	566	622
3 1/2	245	272	300	327	354	381	409	436	463	490	545	600
3 5/8	237	263	289	316	342	368	395	421	447	474	527	579
3 3/4	229	254	280	305	331	356	382	407	433	458	509	560
3 7/8	221	246	271	295	320	345	369	394	419	443	493	542
4	214	238	262	286	310	334	358	382	405	429	477	525
4 1/4	202	224	247	269	292	314	337	359	383	404	449	494
4 1/2	191	212	233	254	275	297	318	339	360	382	424	466
4 3/4	180	201	221	241	261	281	301	321	341	361	402	442
5	171	191	210	229	248	267	286	305	324	343	382	420
5 1/4	163	181	199	218	236	254	272	290	308	327	363	399
5 1/2	156	173	190	208	225	242	260	277	294	312	347	381
5 3/4	149	166	182	199	215	232	249	265	282	298	332	365
6	143	159	174	190	206	222	238	254	270	286	318	349
6 1/4	137	152	168	183	198	213	229	244	259	274	305	336
6 1/2	132	146	161	176	190	205	220	234	249	264	293	322
6 3/4	127	141	155	169	183	198	212	226	240	254	283	311
7	122	136	149	163	177	190	204	218	231	245	272	299
7 1/4	118	131	144	158	171	184	197	210	223	237	263	289
7 1/2	114	127	139	152	165	178	190	203	216	229	254	279
7 3/4	111	123	135	148	160	172	185	197	209	222	246	271
8	107	119	131	143	155	167	179	191	203	215	238	262

Revolutions per Minute for Various Cutting Speeds and Diameters (Metric Units)

Dia., mm	Cutting Speed, Meters per Minute											
	5	6	8	10	12	16	20	25	30	35	40	45
	Revolutions per Minute											
5	318	382	509	637	764	1019	1273	1592	1910	2228	2546	2865
6	265	318	424	530	637	849	1061	1326	1592	1857	2122	2387
8	199	239	318	398	477	637	796	995	1194	1393	1592	1790
10	159	191	255	318	382	509	637	796	955	1114	1273	1432
12	133	159	212	265	318	424	531	663	796	928	1061	1194
16	99.5	119	159	199	239	318	398	497	597	696	796	895
20	79.6	95.5	127	159	191	255	318	398	477	557	637	716
25	63.7	76.4	102	127	153	204	255	318	382	446	509	573
30	53.1	63.7	84.9	106	127	170	212	265	318	371	424	477
35	45.5	54.6	72.8	90.9	109	145	182	227	273	318	364	409
40	39.8	47.7	63.7	79.6	95.5	127	159	199	239	279	318	358
45	35.4	42.4	56.6	70.7	84.9	113	141	177	212	248	283	318
50	31.8	38.2	51	63.7	76.4	102	127	159	191	223	255	286
55	28.9	34.7	46.3	57.9	69.4	92.6	116	145	174	203	231	260
60	26.6	31.8	42.4	53.1	63.7	84.9	106	133	159	186	212	239
65	24.5	29.4	39.2	49	58.8	78.4	98	122	147	171	196	220
70	22.7	27.3	36.4	45.5	54.6	72.8	90.9	114	136	159	182	205
75	21.2	25.5	34	42.4	51	68	84.9	106	127	149	170	191
80	19.9	23.9	31.8	39.8	47.7	63.7	79.6	99.5	119	139	159	179
90	17.7	21.2	28.3	35.4	42.4	56.6	70.7	88.4	106	124	141	159
100	15.9	19.1	25.5	31.8	38.2	51	63.7	79.6	95.5	111	127	143
110	14.5	17.4	23.1	28.9	34.7	46.2	57.9	72.3	86.8	101	116	130
120	13.3	15.9	21.2	26.5	31.8	42.4	53.1	66.3	79.6	92.8	106	119
130	12.2	14.7	19.6	24.5	29.4	39.2	49	61.2	73.4	85.7	97.9	110
140	11.4	13.6	18.2	22.7	27.3	36.4	45.5	56.8	68.2	79.6	90.9	102
150	10.6	12.7	17	21.2	25.5	34	42.4	53.1	63.7	74.3	84.9	95.5
160	9.9	11.9	15.9	19.9	23.9	31.8	39.8	49.7	59.7	69.6	79.6	89.5
170	9.4	11.2	15	18.7	22.5	30	37.4	46.8	56.2	65.5	74.9	84.2
180	8.8	10.6	14.1	17.7	21.2	28.3	35.4	44.2	53.1	61.9	70.7	79.6
190	8.3	10	13.4	16.8	20.1	26.8	33.5	41.9	50.3	58.6	67	75.4
200	8	39.5	12.7	15.9	19.1	25.5	31.8	39.8	47.7	55.7	63.7	71.6
220	7.2	8.7	11.6	14.5	17.4	23.1	28.9	36.2	43.4	50.6	57.9	65.1
240	6.6	8	10.6	13.3	15.9	21.2	26.5	33.2	39.8	46.4	53.1	59.7
260	6.1	7.3	9.8	12.2	14.7	19.6	24.5	30.6	36.7	42.8	49	55.1
280	5.7	6.8	9.1	11.4	13.6	18.2	22.7	28.4	34.1	39.8	45.5	51.1
300	5.3	6.4	8.5	10.6	12.7	17	21.2	26.5	31.8	37.1	42.4	47.7
350	4.5	5.4	7.3	9.1	10.9	14.6	18.2	22.7	27.3	31.8	36.4	40.9
400	4	4.8	6.4	8	9.5	12.7	15.9	19.9	23.9	27.9	31.8	35.8
450	3.5	4.2	5.7	7.1	8.5	11.3	14.1	17.7	21.2	24.8	28.3	31.8
500	3.2	3.8	5.1	6.4	7.6	10.2	12.7	15.9	19.1	22.3	25.5	28.6

Revolutions per Minute for Various Cutting Speeds and Diameters (Metric Units)

Dia., mm	Cutting Speed, Meters per Minute											
	50	55	60	65	70	75	80	85	90	95	100	200
	Revolutions per Minute											
5	3183	3501	3820	4138	4456	4775	5093	5411	5730	6048	6366	12,732
6	2653	2918	3183	3448	3714	3979	4244	4509	4775	5039	5305	10,610
8	1989	2188	2387	2586	2785	2984	3183	3382	3581	3780	3979	7958
10	1592	1751	1910	2069	2228	2387	2546	2706	2865	3024	3183	6366
12	1326	1459	1592	1724	1857	1989	2122	2255	2387	2520	2653	5305
16	995	1094	1194	1293	1393	1492	1591	1691	1790	1890	1989	3979
20	796	875	955	1034	1114	1194	1273	1353	1432	1512	1592	3183
25	637	700	764	828	891	955	1019	1082	1146	1210	1273	2546
30	530	584	637	690	743	796	849	902	955	1008	1061	2122
35	455	500	546	591	637	682	728	773	819	864	909	1818
40	398	438	477	517	557	597	637	676	716	756	796	1592
45	354	389	424	460	495	531	566	601	637	672	707	1415
50	318	350	382	414	446	477	509	541	573	605	637	1273
55	289	318	347	376	405	434	463	492	521	550	579	1157
60	265	292	318	345	371	398	424	451	477	504	530	1061
65	245	269	294	318	343	367	392	416	441	465	490	979
70	227	250	273	296	318	341	364	387	409	432	455	909
75	212	233	255	276	297	318	340	361	382	403	424	849
80	199	219	239	259	279	298	318	338	358	378	398	796
90	177	195	212	230	248	265	283	301	318	336	354	707
100	159	175	191	207	223	239	255	271	286	302	318	637
110	145	159	174	188	203	217	231	246	260	275	289	579
120	133	146	159	172	186	199	212	225	239	252	265	530
130	122	135	147	159	171	184	196	208	220	233	245	490
140	114	125	136	148	159	171	182	193	205	216	227	455
150	106	117	127	138	149	159	170	180	191	202	212	424
160	99.5	109	119	129	139	149	159	169	179	189	199	398
170	93.6	103	112	122	131	140	150	159	169	178	187	374
180	88.4	97.3	106	115	124	133	141	150	159	168	177	354
190	83.8	92.1	101	109	117	126	134	142	151	159	167	335
200	79.6	87.5	95.5	103	111	119	127	135	143	151	159	318
220	72.3	79.6	86.8	94	101	109	116	123	130	137	145	289
240	66.3	72.9	79.6	86.2	92.8	99.5	106	113	119	126	132	265
260	61.2	67.3	73.4	79.6	85.7	91.8	97.9	104	110	116	122	245
280	56.8	62.5	68.2	73.9	79.6	85.3	90.9	96.6	102	108	114	227
300	53.1	58.3	63.7	69	74.3	79.6	84.9	90.2	95.5	101	106	212
350	45.5	50	54.6	59.1	63.7	68.2	72.8	77.3	81.8	99.1	91	182
400	39.8	43.8	47.7	51.7	55.7	59.7	63.7	67.6	71.6	75.6	79.6	159
450	35.4	38.9	42.4	46	49.5	53.1	56.6	60.1	63.6	67.2	70.7	141
500	31.8	35	38.2	41.4	44.6	47.7	50.9	54.1	57.3	60.5	63.6	127

SPEED AND FEED TABLES

How to Use the Feeds and Speeds Tables

Introduction to the Feed and Speed Tables.—The principal tables of feed and speed values are listed in the table below. In this section, Tables 1 through 9 give data for turning, Tables 10 through 15e give data for milling, and Tables 17 through 23 give data for reaming, drilling, threading.

The materials in these tables are categorized by description, and Brinell hardness number (Bhn) range or material condition. So far as possible, work materials are grouped by similar machining characteristics. The types of cutting tools (HSS end mill, for example) are identified in one or more rows across the tops of the tables. Other important details concerning the use of the tables are contained in the footnotes to Tables 1, 10 and 17. Information concerning specific cutting tool grades is given in notes at the end of each table.

Principal Feeds and Speeds Tables

Feeds and Speeds for Turning
Table 1. Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels
Table 2. Cutting Feeds and Speeds for Turning Tool Steels
Table 3. Cutting Feeds and Speeds for Turning Stainless Steels
Table 4a. Cutting Feeds and Speeds for Turning Ferrous Cast Metals
Table 4b. Cutting Feeds and Speeds for Turning Ferrous Cast Metals
Table 5c. Cutting-Speed Adjustment Factors for Turning with HSS Tools
Table 5a. Turning-Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle
Table 5b. Tool Life Factors for Turning with Carbides, Ceramics, Cermets, CBN, and Polycrystalline Diamond
Table 6. Cutting Feeds and Speeds for Turning Copper Alloys
Table 7. Cutting Feeds and Speeds for Turning Titanium and Titanium Alloys
Table 8. Cutting Feeds and Speeds for Turning Light Metals
Table 9. Cutting Feeds and Speeds for Turning Superalloys
Feeds and Speeds for Milling
Table 10. Cutting Feeds and Speeds for Milling Aluminum Alloys
Table 11. Cutting Feeds and Speeds for Milling Plain Carbon and Alloy Steels
Table 12. Cutting Feeds and Speeds for Milling Tool Steels
Table 13. Cutting Feeds and Speeds for Milling Stainless Steels
Table 14. Cutting Feeds and Speeds for Milling Ferrous Cast Metals
Table 15a. Recommended Feed in Inches per Tooth (ft) for Milling with High Speed Steel Cutters
Table 15b. End Milling (Full Slot) Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle
Table 15c. End, Slit, and Side Milling Speed Adjustment Factors for Radial Depth of Cut
Table 15d. Face Milling Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle
Table 15e. Tool Life Adjustment Factors for Face Milling, End Milling, Drilling, and Reaming
Table 16. Cutting Tool Grade Descriptions and Common Vendor Equivalents
Feeds and Speeds for Drilling, Reaming, and Threading
Table 17. Feeds and Speeds for Drilling, Reaming, and Threading Plain Carbon and Alloy Steels
Table 18. Feeds and Speeds for Drilling, Reaming, and Threading Tool Steels
Table 19. Feeds and Speeds for Drilling, Reaming, and Threading Stainless Steels
Table 20. Feeds and Speeds for Drilling, Reaming, and Threading Ferrous Cast Metals
Table 21. Feeds and Speeds for Drilling, Reaming, and Threading Light Metals
Table 22. Feed and Diameter Speed Adjustment Factors for HSS Twist Drills and Reamers
Table 23. Feeds and Speeds for Drilling and Reaming Copper Alloys

Each of the cutting speed tables in this section contains two distinct types of cutting speed data. The speed columns at the left of each table contain traditional Handbook cutting speeds for use with high-speed steel (HSS) tools. For many years, this extensive collection of cutting data has been used successfully as starting speed values for turning, milling, drilling, and reaming operations. Instructions and adjustment factors for use with these speeds are given in Table 5c (feed and depth-of-cut factors) for turning, and in Table 15a (feed, depth of cut, and cutter diameter) for milling. Feeds for drilling and reaming are discussed in Using the Feed and Speed Tables for Drilling, Reaming, and Threading. With traditional speeds and feeds, tool life may vary greatly from material to material, making it very difficult to plan efficient cutting operations, in particular for setting up unattended jobs on CNC equipment where the tool life must exceed cutting time, or at least be predictable so that tool changes can be scheduled. This limitation is reduced by using the combined feed/speed data contained in the remaining columns of the speed tables.

The combined feed/speed portion of the speed tables gives two sets of feed and speed data for each material represented. These feed/speed pairs are the *optimum* and *average* data (identified by *Opt.* and *Avg.*); the *optimum* set is always on the left side of the column and the *average* set is on the right. The *optimum* feed/speed data are approximate values of feed and speed that achieve minimum-cost machining by combining a high productivity rate with low tooling cost at a fixed tool life. The *average* feed/speed data are expected to achieve approximately the same tool life and tooling costs, but productivity is usually lower, so machining costs are higher. The data in this portion of the tables are given in the form of two numbers, of which the first is the feed in thousandths of an inch per revolution (or per tooth, for milling) and the second is the cutting speed in feet per minute. For example, the feed/speed set 15⁄215 represents a feed of 0.015 in./rev at a speed of 215 fpm. Blank cells in the data tables indicate that feed/speed data for these materials were not available at the time of publication.

Generally, the feed given in the *optimum* set should be interpreted as the maximum safe feed for the given work material and cutting tool grade, and the use of a greater feed may result in premature tool wear or tool failure before the end of the expected tool life. The primary exception to this rule occurs in milling, where the feed may be greater than the *optimum* feed if the radial depth of cut is less than the value established in the table footnote; this topic is covered later in the milling examples. Thus, except for milling, the speed and tool life adjustment tables, to be discussed later, do not permit feeds that are greater than the *optimum* feed. On the other hand, the speed and tool life adjustment factors often result in cutting speeds that are well outside the given *optimum* to *average* speed range.

The combined feed/speed data in this section were contributed by Dr. Colding of Colding International Corp., Ann Arbor, MI. The speed, feed, and tool life calculations were made by means of a special computer program and a large database of cutting speed and tool life testing data. The COMP computer program uses tool life equations that are extensions of the F. W. Taylor tool life equation, first proposed in the early 1900s. The Colding tool life equations use a concept called equivalent chip thickness (*ECT*), which simplifies cutting speed and tool life predictions, and the calculation of cutting forces, torque, and power requirements. *ECT* is a basic metal cutting parameter that combines the four basic turning variables (depth of cut, lead angle, nose radius, and feed per revolution) into one basic parameter. For other metal cutting operations (milling, drilling, and grinding, for example), *ECT* also includes additional variables such as the number of teeth, width of cut, and cutter diameter. The *ECT* concept was first presented in 1931 by Prof. R. Woxen, who showed that equivalent chip thickness is a basic metal cutting parameter for high-speed cutting tools. Dr. Colding later extended the theory to include other tool materials and metal cutting operations, including grinding.

The equivalent chip thickness is defined by $ECT = A/CEL$, where A is the cross-sectional area of the cut (approximately equal to the feed times the depth of cut), and CEL is the cutting edge length or tool contact rubbing length. *ECT* and several other terms related to tool geometry are illustrated in Figs. 1 and 2. Many combinations of feed, lead angle, nose radius and cutter diameter, axial and radial depth of cut, and numbers of teeth can give the same value of *ECT*. However, for a constant cutting speed, no matter how the depth of cut, feed, or lead angle, etc., are varied, if a constant value of *ECT* is maintained, the tool life will also remain constant. A constant value of *ECT* means that a constant cutting speed gives a constant tool life and an increase in speed results in a reduced tool life. Likewise, if *ECT* were increased and cutting speed were held constant, as illustrated in the generalized cutting speed vs. *ECT* graph that follows, tool life would be reduced.

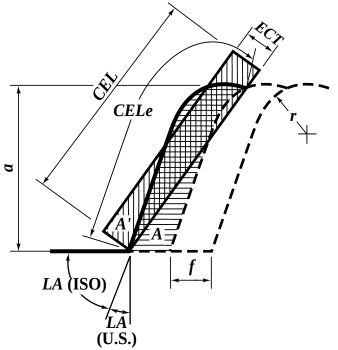


Fig. 1. Cutting Geometry, Equivalent Chip Thickness, and Cutting Edge Length

a = depth of cut
$A = A'$ = chip cross-sectional area
$CEL = CELe$ = engaged cutting edge length
ECT = equivalent chip thickness = A'/CEL
f = feed/rev
r = nose radius
LA = lead angle (U.S.)
$LA(ISO)$ = 90–LA

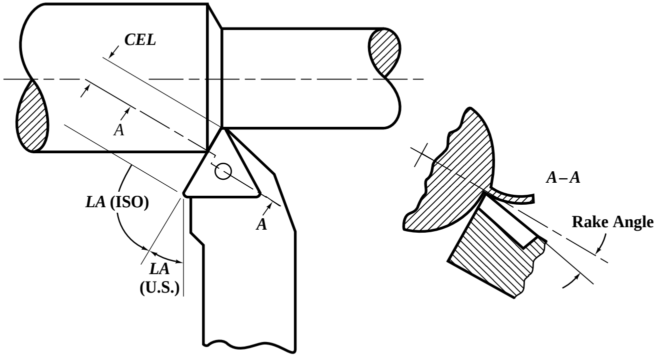


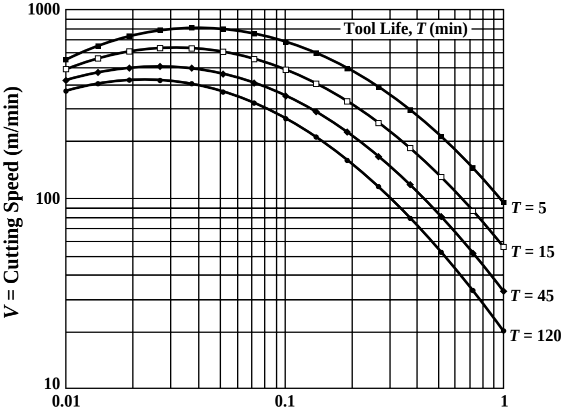
Fig. 2. Cutting Geometry for Turning

In the tables, the *optimum* feed/speed data have been calculated by COMP to achieve a fixed tool life based on the maximum *ECT* that will result in successful cutting, without premature tool wear or early tool failure. The same tool life is used to calculate the *average* feed/speed data, but these values are based on one-half of the maximum *ECT*. Because the data are not linear except over a small range of values, both *optimum* and *average* sets are required to adjust speeds for feed, lead angle, depth of cut, and other factors.

Tool life is the most important factor in a machining system, so feeds and speeds cannot be selected as simple numbers, but must be considered with respect to the many parameters that influence tool life. The accuracy of the combined feed/speed data presented is believed to be very high. However, machining is a variable and complicated process and use of the feed and speed tables requires the user to follow the instructions carefully to achieve good predictability. The results achieved, therefore, may vary due to material condition, tool material, machine setup, and other factors, and cannot be guaranteed.

The feed values given in the tables are valid for the standard tool geometries and fixed depths of cut that are identified in the table footnotes. If the cutting parameters and tool geometry established in the table footnotes are maintained, turning operations using either the *optimum* or *average* feed/speed data (Tables 1 through 9) should achieve a constant tool life of approximately 15 minutes; tool life for milling, drilling, reaming, and threading data (Tables 10 through 14 and Tables 17 through 22) should be approximately 45 minutes. The reason for the different economic tool lives is the higher tooling cost associated with milling-drilling operations than for turning. If the cutting parameters or tool geometry are different from those established in the table footnotes, the same tool life (15 or 45 minutes) still may be maintained by applying the appropriate speed adjustment factors, or tool life may be increased or decreased using tool life adjustment factors. The use of the speed and tool life adjustment factors is described in the examples that follow.

Both the *optimum* and *average* feed/speed data given are reasonable values for effective cutting. However, the *optimum* set with its higher feed and lower speed (always the left entry in each table cell) will usually achieve greater productivity. In Table 1, for example, the two entries for turning 1212 free-machining plain carbon steel with uncoated carbide are 17⁄805 and 8⁄1075. These values indicate that a feed of 0.017 in./rev and a speed of 805 ft/min, or a feed of 0.008 in./rev and a speed of 1075 ft/min can be used for this material. The tool life, in each case, will be approximately 15 minutes. If one of these feed and speed pairs is assigned an arbitrary cutting time of 1 minute, then the relative cutting time of the second pair to the first is equal to the ratio of their respective feed × speed products. Here, the same amount of material that can be cut in 1 minute, at the higher feed and lower speed (17⁄805), will require 1.6 minutes at the lower feed and higher speed (8⁄1075) because $17 \times 805/(8 \times 1075) = 1.6$ minutes.



Cutting Speed versus Equivalent Chip Thickness with Tool Life as a Parameter

Speed and Feed Tables for Turning.—Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $\frac{3}{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text.

Examples Using the Feed and Speed Tables for Turning: The examples that follow give instructions for determining cutting speeds for turning. In general, the same methods are also used to find cutting speeds for milling, drilling, reaming, and threading, so reading through these examples may bring some additional insight to those other metalworking processes as well. The first step in determining cutting speeds is to locate the work material in the left column of the appropriate table for turning, milling, or drilling, reaming, and threading.

Example 1, Turning: Find the cutting speed for turning SAE 1074 plain carbon steel of 225 to 275 Brinell hardness, using an uncoated carbide insert, a feed of 0.015 in./rev, and a depth of cut of 0.1 inch.

In Table 1, feed and speed data for two types of uncoated carbide tools are given, one for hard tool grades, the other for tough tool grades. In general, use the speed data from the tool category that most closely matches the tool to be used because there are often significant differences in the speeds and feeds for different tool grades. From the uncoated carbide hard grade values, the *optimum* and *average* feed/speed data given in Table 1 are 17⁄615 and 8⁄815, or 0.017 in./rev at 615 ft/min and 0.008 in./rev at 815 ft/min. Because the selected feed (0.015 in./rev) is different from either of the feeds given in the table, the cutting speed must be adjusted to match the feed. The other cutting parameters to be used must also be compared with the general tool and cutting parameters given in the speed tables to determine if adjustments need to be made for these parameters as well. The general tool and cutting parameters for turning, given in the footnote to Table 1, are depth of cut = 0.1 inch, lead angle = 15°, and tool nose radius = $\frac{3}{64}$ inch.

Table 5a is used to adjust the cutting speeds for turning (from Tables 1 through 9) for changes in feed, depth of cut, and lead angle. The new cutting speed V is found from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the *optimum* speed from the table (always the lower of the two speeds given), and F_f and F_d are the adjustment factors from Table 5a for feed and depth of cut, respectively.

To determine the two factors F_f and F_d, calculate the ratio of the selected feed to the *optimum* feed, 0.015⁄0.017 = 0.9, and the ratio of the two given speeds V_{avg} and V_{opt}, 815⁄615 = 1.35 (approximately). The feed factor $F_d = 1.07$ is found in Table 5a at the intersection of the feed ratio row and the speed ratio column. The depth-of-cut factor $F_d = 1.0$ is found in the same row as the feed factor in the column for depth of cut = 0.1 inch and lead angle = 15°, or for a tool with a 45° lead angle, $F_d = 1.18$. The final cutting speed for a 15° lead angle is $V = V_{opt} \times F_f \times F_d = 615 \times 1.07 \times 1.0 = 658$ fpm. Notice that increasing the lead angle tends to permit higher cutting speeds; such an increase is also the general effect of increasing the tool nose radius, although nose radius correction factors are not included in this table. Increasing lead angle also increases the radial pressure exerted by the cutting tool on the workpiece, which may cause unfavorable results on long, slender workpieces.

Example 2, Turning: For the same material and feed as the previous example, what is the cutting speed for a 0.4-inch depth of cut and a 45° lead angle?

As before, the feed is 0.015 in./rev, so F_f is 1.07, but $F_d = 1.03$ for depth of cut equal to 0.4 inch and a 45° lead angle. Therefore, $V = 615 \times 1.07 \times 1.03 = 676$ fpm. Increasing the lead angle from 15° to 45° permits a much greater (four times) depth of cut, at the same feed and nearly constant speed. Tool life remains constant at 15 minutes. *(Continued on page 1005)*

Table 1. Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels

Material AISI/SAE Designation	Brinell Hardness	Tool Material															
		HSS		Uncoated Carbide				Coated Carbide				Ceramic					
				Hard		Tough		Hard		Tough		Hard		Tough		Cermet	
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining plain carbon steels (resulfurized): 1212, 1213, 1215	100–150	150	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	805	1075	405	555	1165	1295	850	1200	3340	4985	1670	2500	1610	2055
	150–200	160	**f**	17	8	36	17	28	13	28	13	15	8	15	8	7	3
			s	745	935	345	470	915	1130	785	1110	1795	2680	1485	2215	1490	1815
1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	100–150	130															
	150–200	120	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	730	990	300	430	1090	1410	780	1105	1610	2780	1345	2005	1355	1695
1132, 1137, 1139, 1140, 1144, 1146, 1151	175–225	120	**f**	17	8	36	17	17	8	28	13	13	7	13	7		
			s	615	815	300	405	865	960	755	960	1400	1965	1170	1640		
	275–325	75															
	325–375	50	**f**	17	8	36	17	17	8	28	13	10	5	10	5		
			s	515	685	235	340	720	805	650	810	1430	1745	1070	1305		
	375–425	40															
(Leaded): 11L17, 11L18, 12L13, 12L14	100–150	140															
	150–200	145	**f**	17	8	36	17	28	13	28	13	15	8	15	8	7	3
			s	745	935	345	470	915	1130	785	1110	1795	2680	1485	2215	1490	1815
	200–250	110	**f**	17	8	36	17	17	8	28	13	13	7	13	7		
			s	615	815	300	405	865	960	755	960	1400	1965	1170	1640		
Plain carbon steels: 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	100–125	120	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	805	1075	405	555	1165	1295	850	1200	3340	4985	1670	2500	1610	2055
	125–175	110	**f**	17	8	36	17	28	13	28	13	15	8	15	8	7	3
			s	745	935	345	470	915	1130	785	1110	1795	2680	1485	2215	1490	1815
	175–225	90		17	8	36	17	17	8	28	13	13	7	13	7		
				615	815	300	405	865	960	755	960	1400	1965	1170	1640		
	225–275	70	**f**														
			s														

Table 1. *(Continued)* Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels

Material AISI/SAE Designation	Brinell Hardness	HSS		Tool Material													
				Uncoated Carbide				Coated Carbide				Ceramic					
				Hard		Tough		Hard		Tough		Hard		Tough		Cermet	
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Plain carbon steels *(continued)*: 1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	125–175	100	**f** **s**	17 745	8 935	36 345	17 470	28 915	13 1130	28 785	13 1110	15 1795	8 2680	15 1485	8 2215	7 1490	3 1815
	175–225	85	**f** **s**	17 615	8 815	36 300	17 405	17 865	8 960	28 755	13 960	13 1400	7 1965	13 1170	7 1640		
	225–275	70															
	275–325	60	**f** **s**	17 515	8 685	36 235	17 340	17 720	8 805	28 650	13 810	10 1430	5 1745	10 1070	5 1305		
	325–375	40															
	375–425	30															
Plain carbon steels *(continued)*: 1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125–175	100	**f** **s**	17 730	8 990	36 300	17 430	17 1090	8 1410	28 780	13 1105	15 1610	8 2780	15 1345	8 2005	7 1355	3 1695
	175–225	80															
	225–275	65	**f** **s**	17 615	8 815	36 300	17 405	17 865	8 960	28 755	13 960	13 1400	7 1965	13 1170	7 1640	7 1365	3 1695
	275–325	50	**f** **s**	17 515	8 685	36 235	17 340	17 720	8 805	28 650	13 810	10 1430	5 1745	10 1070	5 1305		
	325–375	35															
	375–425	30															
Free-machining alloy steels, (resulfurized): 4140, 4150	175–200	110	**f** **s**	17 525	8 705	36 235	17 320	17 505	8 525	28 685	13 960	15 1490	8 2220	15 1190	8 1780	7 1040	3 1310
	200–250	90															
	250–300	65	**f** **s**	17 355	8 445	36 140	17 200	17 630	8 850	28 455	13 650	10 1230	5 1510	10 990	5 1210	7 715	3 915
	300–375	50	**f** **s**	17 330	8 440	36 125	17 175	17 585	8 790	28 125	13 220	8 1200	4 1320	8 960	4 1060	7 575	3 740
	375–425	40															

Table 1. *(Continued)* Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels

Material AISI/SAE Designation	Brinell Hardness	Tool Material															
		HSS		Uncoated Carbide				Coated Carbide				Ceramic				Cermet	
				Hard		Tough		Hard		Tough		Hard		Tough			
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining alloy steels: (leaded): 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40	150–200	120	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	730	990	300	430	1090	1410	780	1105	1610	2780	1345	2005	1355	1695
	200–250	100	**f**	17	8	36	17	17	8	28	13	13	7	13	7	7	3
			s	615	815	300	405	865	960	755	960	1400	1965	1170	1640	1355	1695
	250–300 300–375 375–425	75 55 50	**f**	17	8	36	17	17	8	28	13	10	5	10	5		
			s	515	685	235	340	720	805	650	810	1430	1745	1070	1305		
Alloy steels: 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125–175 175–225	100 90	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	525	705	235	320	505	525	685	960	1490	2220	1190	1780	1040	1310
	225–275	70	**f**	17	8	36	1	17	8	28	13	10	5	10	5	7	3
			s	355	445	140	200	630	850	455	650	1230	1510	990	1210	715	915
	275–325	60	**f**	17	8	36	17	17	8	28	13	9	5	8	5	7	3
			s	330	440	135	190	585	790	240	350	1230	1430	990	1150	655	840
	325–35 375–425	50 30 (20)	**f**	17	8	36	17	17	8	28	13	8	4	8	4	7	3
			s	330	440	125	175	585	790	125	220	1200	1320	960	1060	575	740
Alloy steels: 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30 E51100, E52100 use (HSS Speeds)	175–225	85 (70)	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	525	705	235	320	505	525	685	960	1490	2220	1190	1780	1020	1310
	225–275	70 (65)	**f**	17	8	36	17	17	8	28	13	10	5	10	5	7	3
			s	355	445	140	200	630	850	455	650	1230	1510	990	1210	715	915
	275–325	60 (50)	**f**	17	8	36	17	17	8	28	13	9	5	8	5	7	3
			s	330	440	135	190	585	790	240	350	1230	1430	990	1150	655	840
	325–375 375–425	40 (30) 30 (20)	**f**	17	8	36	17	17	8	28	13	8	4	8	4	7	3
			s	330	440	125	175	585	790	125	220	1200	1320	960	1060	575	740

Table 1. *(Continued)* Cutting Feeds and Speeds for Turning Plain Carbon and Alloy Steels

Material AISI/SAE Designation	Brinell Hardness	HSS Speed (fpm)	Tool Material: f = feed (0.001 in./rev), s = speed (ft/min)	Uncoated Carbide Hard *Opt.*	Uncoated Carbide Hard *Avg.*	Uncoated Carbide Tough *Opt.*	Uncoated Carbide Tough *Avg.*	Coated Carbide Hard *Opt.*	Coated Carbide Hard *Avg.*	Coated Carbide Tough *Opt.*	Coated Carbide Tough *Avg.*	Ceramic Hard *Opt.*	Ceramic Hard *Avg.*	Ceramic Tough *Opt.*	Ceramic Tough *Avg.*	Cermet *Opt.*	Cermet *Avg.*
Ultra-high-strength steels (not ASI): AMS alloys 6421 (98B37 Mod.), 6422 (98BV40), 6424, 6427, 6428, 6430, 6432, 6433, 6434, 6436, and 6442; 300M and D6ac	220–300	65															
	300–350	50	**f**	17	8	36	17	20	10	28	13			10	5	7	3
			s	220	295	100	150	355	525	600	865			660	810	570	740
	350–400	35	**f**	17	8	36	17	17	8	28	13			8	4	7	3
			s	165	185	55	105	325	350	175	260			660	730	445	560
	43–48 Rc	25															
	48–52 Rc	10	**f**			17	8					7	3	10	5		
			s			55†	90					385	645	270	500		
Maraging steels (not AISI): 18% Ni, Grades 200, 250, 300, and 350	250–325	60	**f**	17	8	36	17	20	10	28	13	660	810	10	5	7	3
			s	220	295	100	150	355	525	600	865			570	740		
	50–52 Rc	10	**f**			17	8					7	3	10	5		
			s			55†	90					385‡	645	270	500		
Nitriding steels (not AISI): Nitralloy 125, 135, 135 Mod., 225, and 230, Nitralloy N, Nitralloy EZ, Nitrex 1	200–250	70	**f**	17	8	36	17	17	8	28	13	15	8	15	8	7	3
			s	525	705	235	320	505	525	685	960	1490	2220	1190	1780	1040	1310
	300–350	30	**f**	17	8	36	17	17	8	28	13	8	4	8	4	7	3
			s	330	440	125	175	585	790	125	220	1200	1320	960	1060	575	740

Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $\frac{3}{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text.

The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbides, hard = 17, tough = 19, † = 15; coated carbides, hard = 11, tough = 14; ceramics, hard = 2, tough = 3, ‡ = 4; cermet = 7 .

Table 2. Cutting Feeds and Speeds for Turning Tool Steels

		Tool Material															
		Uncoated		Uncoated Carbide				Coated Carbide				Ceramic					
		HSS		Hard		Tough		Hard		Tough		Hard		Tough		Cermet	
Material AISI Designation	Brinell Hardness	Speed (fpm)	**f** = feed (0.001 in./rev), **s** = speed (ft/min)														
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Water hardening: W1, W2, W5	150–200	100	**f** **s**	17 455	8 610	36 210	17 270	17 830	8 1110	28 575	13 805	13 935	7 1310	13 790	7 1110	7 915	3 1150
Shock resisting: S1, S2, S5, S6, S7	175–225	70															
Cold work, oil hardening: O1, O2, O6, O7	175–225	70															
Cold work, high carbon, high chromium: D2, D3, D4, D5, D7	200–250	45															
Cold work, air hardening: A2, A3, A8, A9, A10	200–250	70	**f** **s**	17 445	8 490	36 170	17 235	17 705	8 940	28 515	13 770	13 660	7 925	13 750	7 1210	7 1150	3 1510
A4, A6	200–250	55															
A7	225–275	45															
Hot work, chromium type: H10, H11, H12, H13, H14, H19	150–200	80															
	200–250	65															
	325–375	50	**f** **s**	17 165	8 185	36 55	17 105	17 325	8 350	28 175	13 260			8 660	4 730	7 445	3 560
	48–50 Rc	20	**f** **s**			17 55†	8 90					7 385‡	3 645	10 270	5 500		
	50–52 Rc	10															
	52–56 Rc	—															
Hot work, tungsten type: H21, H22, H23, H24, H25, H26	150–200	60	**f** **s**	17 445	8 490	36 170	17 235	17 705	8 940	28 515	13 770	13 660	7 925	13 750	7 1210	7 1150	3 1510
	200–250	50															
Hot work, molybdenum type: H41, H42, H43	150–200	55															
	200–250	45															
Special purpose, low alloy: L2, L3, L6	150–200	75	**f** **s**	17 445	8 610	36 210	17 270	17 830	8 1110	28 575	13 805	13 935	7 1310	13 790	7 1110	7 915	3 1150
Mold: P2, P3, P4, P5, P6, P26, P21	100–150	90	**f** **s**	17 445	8 610	36 210	17 270	17 830	8 1110	28 575	13 805	13 935	7 1310	13 790	7 1110	7 915	3 1150
	150–200	80															
High-speed steel: M1, M2, M6, M10, T1, T2,T6	200–250	65	**f** **s**	17 445	8 490	36 170	17 235	17 705	8 940	28 515	13 770	13 660	7 925	13 750	7 1210	7 1150	3 1510
M3-1, M4 M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8	225–275	55															
T15, M3-2	225–275	45															

Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $\frac{3}{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text.The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbides, hard = 17, tough = 19, † = 15; coated carbides, hard = 11, tough = 14; ceramics, hard = 2, tough = 3, ‡ = 4; cermet = 7.

Table 3. Cutting Feeds and Speeds for Turning Stainless Steels

Material	Brinell Hardness	Tool Material											
		Uncoated HSS	Uncoated Carbide					Coated Carbide				Cermet	
			Hard			Tough		Hard		Tough			
		Speed (fpm)	**f** = feed (0.001 in./rev), **s** = speed (ft/min)										
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining stainless steel (Ferritic): 430F, 430FSe	135–185	110	**f** **s**	20 480	10 660	36 370	17 395	17 755	8 945	28 640	13 810	7 790	3 995
(Austenitic): 203EZ, 303, 303Se, 303MA, 303Pb, 303Cu, 303 Plus X	135–185 225–275	100 80	**f** **s**	13 520	7 640	36 310	17 345			28 625	13 815	7 695	3 875
(Martensitic): 416, 416Se, 416 Plus X, 420F, 420FSe, 440F, 440FSe	135–185 185–240	110 100	**f** **s**	13 520	7 640	36 310				28 625	13 815	7 695	3 875
	275–325 375–425	60 30	**f** **s**	13 210	7 260	36 85	17 135			28 130	13 165		
Stainless steels (Ferritic): 405, 409 429, 430, 434, 436, 442, 446, 502	135–185	90	**f** **s**	20 480	10 660	36 370	17 395	17 755	8 945	28 640	13 810	7 790	3 995
(Austenitic): 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135–185 225–275	75 65	**f** **s**	13 520	7 640	36 310	17 345			28 625	13 165	7 695	3 875
(Austenitic): 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	135–185	70											
(Martensitic): 403, 410, 420, 501	135–175 175–225	95 85											
	275–325 375–425	55 35	**f** **s**	13 210	7 260	36 85	17 135			28 130	13 165	13 200†	7 230
(Martensitic): 414, 431, Greek Ascoloy, 440A, 440B, 440C	225–275 275–325 375–425	55–60 45–50 30											
(Precipitation hardening):15 -5PH, 17-4PH, 17-7PH, AF-71, 17-14CuMo, AFC-77, AM-350, AM-355, AM-362, Custom 455, HNM, PH13-8, PH14-8Mo, PH15-7Mo, Stainless W	150–200 275–325	60 50	**f** **s**	13 520	7 640	36 310	17 345			28 625	13 815	13 695	7 875
	325–375 375–450	40 25	**f** **s**	13 195	7 240	36 85	17 155						

See footnote to Table 1 for more information. The combined feed/speed data in this table are based on tool grades (identified in Table Table 16) as follows: uncoated carbides, hard = 17, tough = 19; coated carbides, hard = 11, tough = 14; cermet = 7, † = 18.

Table 4a. Cutting Feeds and Speeds for Turning Ferrous Cast Metals

Material	Brinell Hardness	Tool Material															
				Uncoated Carbide		Coated Carbide				Ceramic							
		HSS		Tough		Hard		Tough		Hard		Tough		Cermet		CBN	
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Gray Cast Iron																	
ASTM Class 20	120–150	120	**f** **s**	28 240	13 365	28 665	13 1040	28 585	13 945	15 1490	8 2220	15 1180	8 1880	8 395	4 510	24 8490	11 36380
ASTM Class 25	160–200	90															
ASTM Class 30, 35, and 40	190–220	80															
ASTM Class 45 and 50	220–260	60	**f** **s**	28 160	13 245	28 400	13 630	28 360	13 580	11 1440	6 1880	11 1200	6 1570	8 335	4 420	24 1590	11 2200
ASTM Class 55 and 60	250–320	35															
ASTM Type 1, 1b, 5 (Ni resist)	100–215	70	**f** **s**	28 110	13 175			28 410	13 575	15 1060	8 1590	15 885	8 1320	8 260	4 325		
ASTM Type 2, 3, 6 (Ni resist)	120–175	65															
ASTM Type 2b, 4 (Ni resist)	150–250	50															
Malleable Iron																	
(Ferritic): 32510, 35018	110–160	130	**f** **s**	28 180	13 280	28 730	13 940	28 660	13 885	15 1640	8 2450	15 1410	8 2110				
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160–200	95	**f** **s**	28 125	13 200	28 335	13 505	28 340	13 510	13 1640	7 2310	13 1400	7 1970				
	200–240	75															
(Martensitic): 53004, 60003, 60004	200–255	70	**f** **s**	28 100	13 120			28 205	13 250	11 1720	6 2240	11 1460	6 1910				
(Martensitic): 70002, 70003	220–260	60															
(Martensitic): 80002	240–280	50															
(Martensitic): 90001	250–320	30															

Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $\frac{3}{64}$ inch. Use Table 5a to adjust the given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text.

The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbides, tough = 15; Coated carbides, hard = 11, tough = 14; ceramics, hard = 2, tough = 3; cermet = 7; CBN = 1.

Table 4b. Cutting Feeds and Speeds for Turning Ferrous Cast Metals

		Tool Material															
		Uncoated HSS		Uncoated Carbide				Coated Carbide				Ceramic				Cermet	
				Hard		Tough		Hard		Tough		Hard		Tough			
	Brinell Hardness	Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)													
Material				*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Nodular (Ductile) Iron																	
(Ferritic): 60-40-18, 65-45-12	140–190	100	**f** **s**			28 200	13 325	28 490	13 700	28 435	13 665	15 970	8 1450	15 845	8 1260	8 365	4 480
(Ferritic-Pearlitic): 80-55-06	190–225	80	**f** **s**			28 130	13 210	28 355	13 510	28 310	13 460	11 765	6 995	11 1260	6 1640	8 355	4 445
	225–260	65															
(Pearlitic-Martensitic): 100-70-03	240–300	45															
(Martensitic): 120-90-02	270–330	30	**f** **s**			28 40	13 65			28 145	13 175	10 615	5 750	10 500	5 615	8 120	4 145
	300–400	15															
Cast Steels																	
(Low-carbon): 1010, 1020	100–150	110	**f** **s**	17 370	8 490	36 230	17 285	17 665	8 815	28 495	13 675	15 2090	8 3120			7 625	3 790
	125–175	100															
(Medium-carbon): 1030, 1040, 1050	175–225	90	**f** **s**	17 370	8 490	36 150	17 200	17 595	8 815	28 410	13 590	15 1460	8 2170			7 625	3 790
	225–300	70															
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150–200	90															
	200–250	80															
	250–300	60															
(Medium-carbon alloy): 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175–225	80	**f** **s**	17 310	8 415	36 115	17 150	17 555	8 760			15 830	8 1240				
	225–250	70															
	250–300	55	**f** **s**			28 70†	13 145					15 445	8 665				
	300–350	45															
	350–400	30	**f** **s**			28 115†	13 355			28 335	13 345			15 955	8 1430		

The combined feed/speed data in this table are based on tool grades (identified in Table 16) as shown: uncoated carbides, hard = 17; tough = 19, † = 15; coated carbides, hard = 11; tough = 14; ceramics, hard = 2; tough = 3; cermet = 7. Also, see footnote to Table 4a.

Table 5a. Turning-Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle

Ratio of Chosen Feed to *Optimum* Feed	Ratio of the two cutting speeds given in the tables V_{avg}/V_{opt}							Depth of Cut and Lead Angle									
								1 in. (25.4 mm)		0.4 in. (10.2 mm)		0.2 in. (5.1 mm)		0.1 in. (2.5 mm)		0.04 in. (1.0 mm)	
	1.00	1.10	1.25	1.35	1.50	1.75	2.00	15°	45°	15°	45°	15°	45°	15°	45°	15°	45°
	Feed Factor, F_f							Depth of Cut and Lead Angle Factor, F_d									
1.00	1.0	1.0	1.0	1.0	1.0	1.0	1.0	0.74	1.0	0.79	1.03	0.85	1.08	1.0	1.18	1.29	1.35
0.90	1.00	1.02	1.05	1.07	1.09	1.10	1.12	0.75	1.0	0.80	1.03	0.86	1.08	1.0	1.17	1.27	1.34
0.80	1.00	1.03	1.09	1.10	1.15	1.20	1.25	0.77	1.0	0.81	1.03	0.87	1.07	1.0	1.15	1.25	1.31
0.70	1.00	1.05	1.13	1.22	1.22	1.32	1.43	0.77	1.0	0.82	1.03	0.87	1.08	1.0	1.15	1.24	1.30
0.60	1.00	1.08	1.20	1.25	1.35	1.50	1.66	0.78	1.0	0.82	1.03	0.88	1.07	1.0	1.14	1.23	1.29
0.50	1.00	1.10	1.25	1.35	1.50	1.75	2.00	0.78	1.0	0.82	1.03	0.88	1.07	1.0	1.14	1.23	1.28
0.40	1.00	1.09	1.28	1.44	1.66	2.03	2.43	0.78	1.0	0.84	1.03	0.89	1.06	1.0	1.13	1.21	1.26
0.30	1.00	1.06	1.32	1.52	1.85	2.42	3.05	0.81	1.0	0.85	1.02	0.90	1.06	1.0	1.12	1.18	1.23
0.20	1.00	1.00	1.34	1.60	2.07	2.96	4.03	0.84	1.0	0.89	1.02	0.91	1.05	1.0	1.10	1.15	1.19
0.10	1.00	0.80	1.20	1.55	2.24	3.74	5.84	0.88	1.0	0.91	1.01	0.92	1.03	1.0	1.06	1.10	1.12

Use with Tables 1 through 9. Not for HSS tools. Tables 1 through 9 data, except for HSS tools, are based on depth of cut = 0.1 inch, lead angle = 15 degrees, and tool life = 15 minutes. For other depths of cut, lead angles, or feeds, use the two feed/speed pairs from the tables and calculate the ratio of desired (new) feed to *optimum* feed (largest of the two feeds given in the tables), and the ratio of the two cutting speeds (V_{avg}/V_{opt}). Use the value of these ratios to find the feed factor F_f at the intersection of the feed ratio row and the speed ratio column in the left half of the table. The depth-of-cut factor F_d is found in the same row as the feed factor in the right half of the table under the column corresponding to the depth of cut and lead angle. The adjusted cutting speed can be calculated from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the smaller (*optimum*) of the two speeds from the speed table (from the left side of the column containing the two feed/speed pairs). See the text for examples.

Table 5b. Tool Life Factors for Turning with Carbides, Ceramics, Cermets, CBN, and Polycrystalline Diamond

Tool Life, *T* (minutes)	Turning with Carbides: Workpiece < 300 Bhn			Turning with Carbides: Workpiece > 300 Bhn; Turning with Ceramics: Any Hardness			Turning with Mixed Ceramics: Any Workpiece Hardness		
	f_s	f_m	f_l	f_s	f_m	f_l	f_s	f_m	f_l
15	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0
45	0.86	0.81	0.76	0.80	0.75	0.70	0.89	0.87	0.84
90	0.78	0.71	0.64	0.70	0.63	0.56	0.82	0.79	0.75
180	0.71	0.63	0.54	0.61	0.53	0.45	0.76	0.72	0.67

Except for HSS speed tools, feeds and speeds given in Tables 1 through 9 are based on 15-minute tool life. To adjust speeds for another tool life, multiply the cutting speed for 15-minute tool life V_{15} by the tool life factor from this table according to the following rules: for small feeds where feed $\le \frac{1}{2}f_{opt}$, the cutting speed for desired tool life is $V_T = f_s \times V_{15}$; for medium feeds where $\frac{1}{2}f_{opt} <$ feed $< \frac{3}{4}f_{opt}$, $V_T = f_m \times V_{15}$; and for larger feeds where $\frac{3}{4}f_{opt} \le$ feed $\le f_{opt}$, $V_T = f_l \times V_{15}$. Here, f_{opt} is the largest (*optimum*) feed of the two feed/speed values given in the speed tables.

Table 5c. Cutting-Speed Adjustment Factors for Turning with HSS Tools

Feed in.	Feed mm	Feed Factor F_f	Depth of Cut in.	Depth of Cut mm	Depth-of-Cut Factor F_d
0.002	0.05	1.50	0.005	0.13	1.50
0.003	0.08	1.50	0.010	0.25	1.42
0.004	0.10	1.50	0.016	0.41	1.33
0.005	0.13	1.44	0.031	0.79	1.21
0.006	0.15	1.34	0.047	1.19	1.15
0.007	0.18	1.25	0.062	1.57	1.10
0.008	0.20	1.18	0.078	1.98	1.07
0.009	0.23	1.12	0.094	2.39	1.04
0.010	0.25	1.08	0.100	2.54	1.03
0.011	0.28	1.04	0.125	3.18	1.00
0.012	0.30	1.00	0.150	3.81	0.97
0.013	0.33	0.97	0.188	4.78	0.94
0.014	0.36	0.94	0.200	5.08	0.93
0.015	0.38	0.91	0.250	6.35	0.91
0.016	0.41	0.88	0.312	7.92	0.88
0.018	0.46	0.84	0.375	9.53	0.86
0.020	0.51	0.80	0.438	11.13	0.84
0.022	0.56	0.77	0.500	12.70	0.82
0.025	0.64	0.73	0.625	15.88	0.80
0.028	0.71	0.70	0.688	17.48	0.78
0.030	0.76	0.68	0.750	19.05	0.77
0.032	0.81	0.66	0.812	20.62	0.76
0.035	0.89	0.64	0.938	23.83	0.75
0.040	1.02	0.60	1.000	25.40	0.74
0.045	1.14	0.57	1.250	31.75	0.73
0.050	1.27	0.55	1.250	31.75	0.72
0.060	1.52	0.50	1.375	34.93	0.71

For use with HSS tool data only from Tables 1 through 9. Adjusted cutting speed $V = V_{HSS} \times F_f \times F_d$, where V_{HSS} is the tabular speed for turning with high-speed tools.

Example 3, Turning: Determine the cutting speed for turning 1055 steel of 175 to 225 Brinell hardness using a hard ceramic insert, a 15° lead angle, a 0.04-inch depth of cut and 0.0075 in./rev feed.

The two feed/speed combinations given in Table 5a for 1055 steel are 15⁄1610 and 8⁄2780, corresponding to 0.015 in./rev at 1610 fpm and 0.008 in./rev at 2780 fpm, respectively. In Table 5a, the feed factor $F_f = 1.75$ is found at the intersection of the row corresponding to feed/$f_{opt} = 7.5/15 = 0.5$ and the column corresponding to $V_{avg}/V_{opt} = 2780/1610 = 1.75$ (approximately). The depth-of-cut factor $F_d = 1.23$ is found in the same row, under the column heading for a depth of cut = 0.04 inch and lead angle = 15°. The adjusted cutting speed is $V = 1610 \times 1.75 \times 1.23 = 3466$ fpm.

Example 4, Turning: The cutting speed for 1055 steel calculated in Example 3 represents the speed required to obtain a 15-minute tool life. Estimate the cutting speed needed to obtain a tool life of 45, 90, and 180 minutes using the results of Example 3.

To estimate the cutting speed corresponding to another tool life, multiply the cutting speed for 15-minute tool life V_{15} by the adjustment factor from the Table 5b, Tool Life Factors for Turning. This table gives three factors for adjusting tool life based on the feed used, f_s for feeds less than or equal to $\frac{1}{2} f_{opt}$, $\frac{3}{4}{}_{fm}$ for midrange feeds between $\frac{1}{2}$ and $\frac{3}{4} f_{opt}$ and f_l for large feeds greater than or equal to $\frac{3}{4} f_{opt}$ and less than f_{opt}. In Example 3, f_{opt} is 0.015 in./rev and the selected feed is 0.0075 in./rev = $\frac{1}{2} f_{opt}$. The new cutting speeds for the various tool lives are obtained by multiplying the cutting speed for 15-minute tool life V_{15} by the factor for small feeds f_s from the column for turning with ceramics in Table 5b. These calculations, using the cutting speed obtained in Example 3, follow.

Tool Life	*Cutting Speed*
15 min	$V_{15} = 3466$ fpm
45 min	$V_{45} = V_{15} \times 0.80 = 2773$ fpm
90 min	$V_{90} = V_{15} \times 0.70 = 2426$ fpm
180 min	$V_{180} = V_{15} \times 0.61 = 2114$ fpm

Depth of cut, feed, and lead angle remain the same as in Example 3. Notice, increasing the tool life from 15 to 180 minutes, a factor of 12, reduces the cutting speed by only about one-third of the V_{15} speed.

Table 6. Cutting Feeds and Speeds for Turning Copper Alloys

Group 1
Architectural bronze (C38500); Extra-high-headed brass (C35600); Forging brass (C37700); Free-cutting phosphor bronze, B2 (C54400); Free-cutting brass (C36000); Free-cutting Muntz metal (C37000); High-leaded brass (C33200; C34200); High-leaded brass tube (C35300); Leaded commercial bronze (C31400); Leaded naval brass (C48500); Medium-leaded brass (C34000)
Group 2
Aluminum brass, arsenical (C68700); Cartridge brass, 70% (C26000); High-silicon bronze, B (C65500); Admiralty brass (inhibited) (C44300, C44500); Jewelry bronze, 87.5% (C22600); Leaded Muntz metal (C36500, C36800); Leaded nickel silver (C79600); Low brass, 80% (C24000); Low-leaded brass (C33500); Low-silicon bronze, B (C65100); Manganese bronze, A (C67500); Muntz metal, 60% (C28000); Nickel silver, 55-18 (C77000); Red brass, 85% (C23000); Yellow brass (C26800)
Group 3
Aluminum bronze, D (C61400); Beryllium copper (C17000, C17200, C17500); Commercial-bronze, 90% (C22000); Copper nickel, 10% (C70600); Copper nickel, 30% (C71500); Electrolytic tough pitch copper (C11000); Guilding, 95% (C21000); Nickel silver, 65-10 (C74500); Nickel silver, 65-12 (C75700); Nickel silver, 65-15 (C75400); Nickel silver, 65-18 (C75200); Oxygen-free copper (C10200) ; Phosphor bronze, 1.25% (C50200); Phosphor bronze, 10% D (C52400) Phosphor bronze, 5% A (C51000); Phosphor bronze, 8% C (C52100); Phosphorus deoxidized copper (C12200)

Wrought Alloys Description and UNS Alloy Numbers	Material Condition	HSS	Uncoated Carbide			Polycrystalline Diamond	
		Speed (fpm)	**f** = feed (0.001 in./rev), **s** = speed (ft/min)				
				Opt.	*Avg.*	*Opt.*	*Avg.*
Group 1	A	300	**f**	28	13		
	CD	350	**s**	1170	1680		
Group 2	A	200	**f**	28	13		
	CD	250	**s**	715	900		
Group 3	A	100	**f**	28	13	7	13
	CD	110	**s**	440	610	1780	2080

Abbreviations designate: A, annealed; CD, cold drawn.

The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide, 15; diamond, 9. See the footnote to Table 7.

Table 7. Cutting Feeds and Speeds for Turning Titanium and Titanium Alloys

Material	Brinell Hardness	Tool Material			
		HSS	Uncoated Carbide (Tough)		
		Speed (fpm)	**f** = feed (0.001 in./rev), **s** = speed (ft/min)		
				Opt.	*Avg.*
Commercially Pure and Low Alloyed					
99.5Ti, 99.5Ti-0.15Pd	110–150	100–105	**f** **s**	28 55	13 190
99.1Ti, 99.2Ti, 99.2Ti-0.15Pd, 98.9Ti-0.8Ni-0.3Mo	180–240	85–90	**f** **s**	28 50	13 170
99.0 Ti	250–275	70	**f** **s**	20 75	10 210
Alpha Alloys and Alpha-Beta Alloys					
5Al-2.5Sn, 8Mn, 2Al-11Sn-5Zr-1Mo, 4Al-3Mo-1V, 5Al-6Sn-2Zr-1Mo, 6Al-2Sn-4Zr-2Mo, 6Al-2Sn-4Zr-6Mo, 6Al-2Sn-4Zr-2Mo-0.25Si	300–350	50	**f** **s**	17 95	8 250
6Al-4V	310–350	40			
6Al-6V-2Sn, Al-4Mo,	320–370	30			
8V-5Fe-IAl	320–380	20			
6Al-4V, 6Al-2Sn-4Zr-2Mo, 6Al-2Sn-4Zr-6Mo, 6Al-2Sn-4Zr-2Mo-0.25Si	320–380	40			
4Al-3Mo-1V, 6Al-6V-2Sn, 7Al-4Mo	375–420	20			
I Al-8V-5Fe	375–440	20			
Beta Alloys					
13V-11Cr-3Al, 8Mo-8V-2Fe-3Al, 3Al-8V-6Cr-4Mo-4Zr, 11.5Mo-6ZR-4.5Sn {	275–350	25	**f** **s**	17 55	8 150
	375–440	20			

The speed recommendations for turning with HSS (high-speed steel) tools may be used as starting speeds for milling titanium alloys, using Table 15a to estimate the feed required. Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $^3/_{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide, 15.

Table 8. Cutting Feeds and Speeds for Turning Light Metals

Material Description	Material Condition	Tool Material					
		HSS	Uncoated Carbide (Tough)			Polycrystalline Diamond	
		Speed (fpm)	**f** = feed (0.001 in./rev), **s** = speed (ft/min)				
				Opt.	*Avg.*	*Opt.*	*Avg.*
All wrought and cast magnesium alloys	A, CD, ST, and A	800					
All wrought aluminum alloys, including 6061-T651, 5000, 6000, and 7000 series	CD	600	**f** **s**	36 2820	17 4570		
	ST and A	500					
All aluminum sand and permanent mold casting alloys	AC	750					
	ST and A	600					
Aluminum Die-Casting Alloys							
Alloys 308.0 and 319.0	—	—	**f** **s**	36 865	17 1280	11 5890[a]	8 8270
Alloys 390.0 and 392.0	AC	80	**f** **s**	24 2010	11 2760	8 4765	4 5755
	ST and A	60					
Alloy 413	—	—	**f** **s**	32 430	15 720	10 5085	5 6570
All other aluminum die-casting alloys including alloys 360.0 and 380.0	ST and A	100					
	AC	125	**f** **s**	36 630	17 1060	11 7560	6 9930

[a] The feeds and speeds for turning Al alloys 308.0 and 319.0 with (polycrystalline) diamond tooling represent an expected tool life T = 960 minutes = 16 hours; corresponding feeds and speeds for 15-minute tool life are 11⁄28600 and 6⁄37500.

Abbreviations for material condition: A, annealed; AC, as cast; CD, cold drawn; and ST and A, solution treated and aged, respectively. Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the HSS speeds for other feeds and depths of cut. The combined feed/speed data are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $\frac{3}{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. The data are based on tool grades (identified in Table 16) as follows: uncoated carbide, 15; diamond, 9.

Table 9. Cutting Feeds and Speeds for Turning Superalloys

	Tool Material										
	HSS Turning			Uncoated Carbide		Ceramic					
	Rough	Finish		Tough		Hard		Tough		CBN	
			f = feed (0.001 in./rev), s = speed (ft/min)								
Material Description	Speed (fpm)			Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.
T-D Nickel	70–80	80–100									
Discalloy	15–35	35–40									
19-9DL, W-545	25–35	30–40	f	24	11					20	10
16-25-6, A-286, Incoloy 800, 801, and 802, V-57	30–35	35–40	s	90	170					365	630
Refractaloy 26	15–20	20–25	f	20	10					20	10
J1300	15–25	20–30	s	75	135					245	420
Inconel 700 and 702, Nimonic 90 and 95	10–12	12–15	f s	20 75	10 125	11 1170	6 2590	11 405	6 900	20 230	10 400
S-816, V-36	10–15	15–20									
S-590	10–20	15–30									
Udimet 630		20–25									
N-155	15–20	15–25									
Air Resist 213; Hastelloy B, C, G and X (wrought); Haynes 25 and 188; J1570; M252 (wrought); Mar-M905 and M918; Nimonic 75 and 80		20–25									
CW-12M; Hastelloy B and C (cast); N-12M	8–12	10–15									
Rene 95 (Hot Isostatic Pressed)	—	—									
HS 6, 21, 2, 31 (X 40), 36, and 151; Haynes 36 and 151; Mar-M302, M322, and M509, WI-52	10–12	10–15	f s	28 20	13 40	11 895	6 2230	10 345	5 815	20 185	10 315
Rene 41	10–15	12–20									
Incoloy 901	10–20	20–35									
Waspaloy	10–30	25–35									
Inconel 625, 702, 706, 718 (wrought), 721, 722, X750, 751, 901, 600, and 604	15–20	20–35									
AF2-1DA, Unitemp 1753	8–10	10–15									
Colmonoy, Inconel 600, 718, K-Monel, Stellite	—	—									
Air Resist 13 and 215, FSH-H14, Nasa C-W-Re, X-45	10–12	10–15	f s	28 15	13 15	11 615	6 1720	10 290	5 700	20 165	10 280
Udimet 500, 700, and 710	10–15	12–20									
Astroloy	5–10	5–15									
Mar-M200, M246, M421, and Rene 77, 80, and 95 (forged)	8–10	10–12 10–15									
B-1900, GMR-235 and 235D, IN 100 and 738, Inconel 713C and 718 (cast), M252 (cast)		8–10									

The speed recommendations for rough turning may be used as starting values for milling and drilling with HSS tools. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 15; ceramic, hard = 4, tough = 3; CBN = 1.

Speeds for HSS (high-speed steel) tools are based on a feed of 0.012 inch/rev and a depth of cut of 0.125 inch; use Table 5c to adjust the given speeds for other feeds and depths of cut. The combined feed/speed data in the remaining columns are based on a depth of cut of 0.1 inch, lead angle of 15 degrees, and nose radius of $^3\!/_{64}$ inch. Use Table 5a to adjust given speeds for other feeds, depths of cut, and lead angles; use Table 5b to adjust given speeds for increased tool life up to 180 minutes. Examples are given in the text.

Speed and Feed Tables for Milling.—Tables 10 through 14 give feeds and speeds for milling. The data in the first speed column can be used with high-speed steel tools using the feeds given in Table 15a; these are the same speeds contained in previous editions of the Handbook. The remaining data in Tables 10 through 14 are combined feeds and speeds for end, face, and slit, slot, and side milling that use the speed adjustment factors given in Tables 15b, 15c, and 15d. Tool life for the combined feed/speed data can also be adjusted using the factors in Table 15e. Table 16 lists cutting tool grades and vendor equivalents.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $^3\!/_{64}$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $^3\!/_4$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter D of 4.0 inch, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes.

Using the Feed and Speed Tables for Milling: The basic feed for milling cutters is the feed per tooth (f), which is expressed in inches per tooth. There are many factors to consider in selecting the feed per tooth and no formula is available to resolve these factors. Among the factors to consider are the cutting tool material; the work material and its hardness; the width and the depth of the cut to be taken; the type of milling cutter to be used and its size; the surface finish to be produced; the power available on the milling machine; and the rigidity of the milling machine, the workpiece, the workpiece setup, the milling cutter, and the cutter mounting.

The cardinal principle is to always use the maximum feed that conditions will permit. Avoid, if possible, using a feed that is less than 0.001 inch per tooth because such low feeds reduce the tool life of the cutter. When milling hard materials with small-diameter end mills, such small feeds may be necessary, but otherwise use as much feed as possible. Harder materials in general will require lower feeds than softer materials. The width and the depth of cut also affect the feeds. Wider and deeper cuts must be fed somewhat more slowly than narrow and shallow cuts. A slower feed rate will result in a better surface finish; however, always use the heaviest feed that will produce the surface finish desired. Fine chips produced by fine feeds are dangerous when milling magnesium because spontaneous combustion can occur. Thus, when milling magnesium, a fast feed that will produce a relatively thick chip should be used. Cutting stainless steel produces a work-hardened layer on the surface that has been cut. Thus, when milling this material, the feed should be large enough to allow each cutting edge on the cutter to penetrate below the work-hardened layer produced by the previous cutting edge. The heavy feeds recommended for face milling cutters are to be used primarily with larger cutters on milling machines having an adequate amount of power. For smaller face milling cutters, start with smaller feeds and increase as indicated by the performance of the cutter and the machine.

When planning a milling operation that requires a high cutting speed and a fast feed, always check to determine if the power required to take the cut is within the capacity of the milling machine. Excessive power requirements are often encountered when milling with cemented carbide cutters. The large metal removal rates that can be attained require a high horsepower output. An example of this type of calculation is given in the section on Machining Power that follows this section. If the size of the cut must be reduced in order to stay within the power capacity of the machine, start by reducing the cutting speed rather than the feed in inches per tooth.

The formula for calculating the table feed rate, when the feed in inches per tooth is known, is as follows:

$$f_m = f_t n_t N$$

where f_m = milling machine table feed rate in inches per minute (ipm)

f_t = feed in inch per tooth (ipt)

n_t = number of teeth in the milling cutter

N = spindle speed of the milling machine in revolutions per minute (rpm)

Example: Calculate the feed rate for milling a piece of AISI 1040 steel having a hardness of 180 Bhn. The cutter is a 3-inch diameter high-speed steel plain or slab milling cutter with 8 teeth. The width of the cut is 2 inches, the depth of cut is 0.062 inch, and the cutting speed from Table 11 is 85 fpm. From Table 15a, the feed rate selected is 0.008 inch per tooth.

$$N = \frac{12V}{\pi D} = \frac{12 \times 85}{3.14 \times 3} = 108 \text{ rpm}$$

$$f_m = f_t n_t N = 0.008 \times 8 \times 108$$

$$= 7 \text{ ipm (approximately)}$$

Example 1, Face Milling: Determine the cutting speed and machine operating speed for face milling an aluminum die casting (alloy 413) using a 4-inch polycrystalline diamond cutter, a 3-inch width of cut, a 0.10-inch depth of cut, and a feed of 0.006 inch/tooth.

Table 10 gives the feeds and speeds for milling aluminum alloys. The feed/speed pairs for face milling die cast alloy 413 with polycrystalline diamond (PCD) are 8⁄2320 (0.008 in./tooth feed at 2320 fpm) and 4⁄4755 (0.004 in./tooth feed at 4755 fpm). These speeds are based on an axial depth of cut of 0.10 inch, an 8-inch cutter diameter *D*, a 6-inch radial depth (width) of cut *ar*, with the cutter approximately centered above the workpiece, i.e., eccentricity is low, as shown in Fig. 3. If the preceding conditions apply, the given feeds and speeds can be used without adjustment for a 45-minute tool life. The given speeds are valid for all cutter diameters if a radial depth of cut to cutter diameter ratio (*ar*/*D*) of ¾ is maintained (i.e., $\frac{6}{8} = \frac{3}{4}$). However, if a different feed or axial depth of cut is required, or if the *ar*/*D* ratio is not equal to ¾, the cutting speed must be adjusted for the conditions. The adjusted cutting speed *V* is calculated from $V = V_{opt} \times F_f \times F_d \times F_{ar}$, where V_{opt} is the lower of the two speeds given in the speed table, and F_f, F_d, and F_{ar} are adjustment factors for feed, axial depth of cut, and radial depth of cut, respectively, obtained from Table 15d (face milling); except, when cutting near the end or edge of the workpiece as in Fig. 4, Table 15c (side milling) is used to obtain F_f.

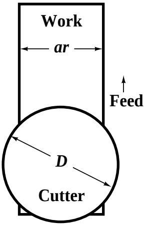


Fig. 3.

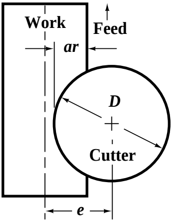


Fig. 4.

In this example, the cutting conditions match the standard conditions specified in the speed table for radial depth of cut to cutter diameter (3 in./4 in.), and depth of cut (0.01 in), but the desired feed of 0.006 in./tooth does not match either of the feeds given in the speed table (0.004 or 0.008). Therefore, the cutting speed must be adjusted for this feed. As with turning, the feed factor F_f is determined by calculating the ratio of the desired feed f to maximum feed f_{opt} from the speed table, and from the ratio V_{avg}/V_{opt} of the two speeds given in the speed table. The feed factor is found at the intersection of the feed ratio row and the speed ratio column in Table 15d. The speed is then obtained using the following equation:

$$\frac{\text{Chosen feed}}{Optimum\text{ feed}} = \frac{f}{f_{opt}} = \frac{0.006}{0.008} = 0.75 \qquad \frac{Average\text{ speed}}{Optimum\text{ speed}} = \frac{V_{avg}}{V_{opt}} = \frac{4755}{2320} \approx 2.0$$

$$F_f = (1.25 + 1.43)/2 = 1.34 \qquad F_d = 1.0 \qquad F_{ar} = 1.0$$

$$V = 2320 \times 1.34 \times 1.0 \times 1.0 = 3109 \text{ fpm, and } 3.82 \times 3109/4 = 2970 \text{ rpm}$$

Example 2, End Milling: What cutting speed should be used for cutting a full slot (i.e., a slot cut from the solid, in one pass, that is the same width as the cutter) in 5140 steel with hardness of 300 Bhn using a 1-inch diameter coated carbide (insert) 0° lead angle end mill, a feed of 0.003 in./tooth, and a 0.2-inch axial depth of cut?

The feed and speed data for end milling 5140 steel, Brinell hardness = 275–325, with a coated carbide tool are given in Table 11 as 15⁄80 and 8⁄240 for *optimum* and *average* sets, respectively. The speed adjustment factors for feed and depth of cut for full slot (end milling) are obtained from Table 15b. The calculations are the same as in the previous examples: $f/f_{opt} = 3/15 = 0.2$ and $V_{avg}/V_{opt} = 240/80 = 3.0$, therefore, $F_f = 6.86$ and $F_d = 1.0$. The cutting speed for a 45-minute tool life is $V = 80 \times 6.86 \times 1.0 = 548.8$, approximately 550 ft/min.

Example 3, End Milling: What cutting speed should be used in Example 2 if the radial depth of cut *ar* is 0.02 inch and axial depth of cut is 1 inch?

In end milling, when the radial depth of cut is less than the cutter diameter (as in Fig. 4), first obtain the feed factor F_f from Table 15c, then the axial depth of cut and lead angle factor F_d from Table 15b. The radial depth of cut to cutter diameter ratio *ar*/*D* is used in Table 15c to determine the maximum and minimum feeds that guard against tool failure at high feeds and against premature tool wear caused by the tool rubbing against the work at very low feeds. The feed used should be selected so that it falls within the minimum to maximum feed range, and then the feed factor F_f can be determined from the feed factors at minimum and maximum feeds, F_{f1} and F_{f2} as explained below.

The maximum feed f_{max} is found in Table 15c by multiplying the *optimum* feed from the speed table by the maximum feed factor that corresponds to the *ar*/*D* ratio, which in this instance is 0.02⁄1 = 0.02; the minimum feed f_{min} is found by multiplying the *optimum* feed by the minimum feed factor. Thus, $f_{max} = 4.5 \times 0.015 = 0.0675$ in./tooth and $f_{min} = 3.1 \times 0.015 = 0.0465$ in./tooth. If a feed between these maximum and minimum values is selected, 0.050 in./tooth for example, then for $ar/D = 0.02$ and $V_{avg}/V_{opt} = 3.0$, the feed factors at maximum and minimum feeds are $F_{f1} = 7.90$ and $F_{f2} = 7.01$, respectively, and by interpolation, $F_f = 7.01 + (0.050 - 0.0465)(0.0675 - 0.0465) \times (7.90 - 7.01) = 7.16$, approximately 7.2.

The depth of cut factor F_d is obtained from Table 15b, using f_{max} from Table 15c instead of the *optimum* feed f_{opt} for calculating the feed ratio (chosen feed/*optimum* feed). In this example, the feed ratio = chosen feed/f_{max} = 0.050⁄0.0675 = 0.74, so the feed factor is $F_d = 0.93$ for a depth of cut = 1.0 inch and 0° lead angle. Therefore, the final cutting speed is $80 \times 7.2 \times 0.93 = 587$ ft/min. Notice that f_{max} obtained from Table 15c was used instead of the *optimum* feed from the speed table, in determining the feed ratio needed to find F_d.

Slit Milling.—The tabular data for slit milling is based on an 8-tooth, 10-degree helix angle cutter with a width of 0.4 inch, a diameter *D* of 4.0 inch, and a depth of cut of 0.6 inch. The given feeds and speeds are valid for any diameters and tool widths, as long as sufficient machine power is available. Adjustments to cutting speeds for other feeds and depths of cut are made using Table 15c or 15d, depending on the orientation of the cutter to the work, as illustrated in Case 1 and Case 2 of Fig. 5. The situation illustrated in Case 1 is approximately equivalent to that illustrated in Fig. 3, and Case 2 is approximately equivalent to that shown in Fig. 4.

Case 1: If the cutter is fed directly into the workpiece, i.e., the feed is perpendicular to the surface of the workpiece, as in cutting off, then Table 15d (face milling) is used to adjust speeds for other feeds. The depth of cut portion of Table 15d is not used in this case ($F_d = 1.0$), so the adjusted cutting speed $V = V_{opt} \times F_f \times F_{ar}$. In determining the factor F_{ar} from Table 15d, the radial depth of cut *ar* is the length of cut created by the portion of the cutter engaged in the work.

Case 2: If the cutter feed is parallel to the surface of the workpiece, as in slotting or side milling, then Table 15c (side milling) is used to adjust the given speeds for other feeds. In Table 15c, the cutting depth (slot depth, for example) is the radial depth of cut *ar* that is used to determine maximum and minimum allowable feed/tooth and the feed factor F_f. These minimum and maximum feeds are determined in the manner described previously, however, the axial depth of cut factor F_d is not required. The adjusted cutting speed, valid for cutters of any thickness (width), is given by $V = V_{opt} \times F_f$.

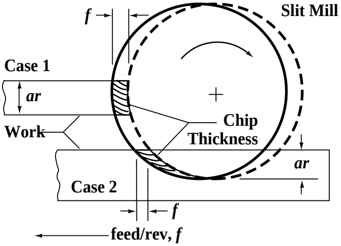


Fig. 5. Determination of Radial Depth of Cut or in Slit Milling

Table 10. Cutting Feeds and Speeds for Milling Aluminum Alloys

Material	Material Condition*		End Milling				Face Milling				Slit Milling			
			HSS		Indexable Insert Uncoated Carbide		Indexable Insert Uncoated Carbide		Polycrystalline Diamond		HSS		Indexable Insert Uncoated Carbide	
			f = feed (0.001 in./tooth), **s** = speed (ft/min)											
			Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
All wrought aluminum alloys, 6061-T651, 5000, 6000, 7000 series All aluminum sand and permanent mold casting alloys	CD ST and A CD ST and A	**f**	15	8	15	8	39	20	8	4	16	8	39	20
		s	165	850	620	2020	755	1720	3750	8430	1600	4680	840	2390
Aluminum Die-Casting Alloys														
Alloys 308.0 and 319.0	—	**f**	15	8	15	8	39	20			16	8	39	20
		s	30	100	620	2020	755	1720			160	375	840	2390
Alloys 360.0 and 380.0	—	**f**	15	8	15	8	39	20	8	4	16	8	39	20
		s	30	90	485	1905	555	1380	3105	7845	145	355	690	2320
Alloys 390.0 and 392.0	—	**f**					39	20						
		s					220	370						
Alloy 413 All other aluminum die-casting alloys	— ST and A	**f**			15	8	39	20	8	4			39	20
		s			355	1385	405	665	2320	4755			500	1680
All other aluminum die-casting alloys	AC	**f**	15	8	15	8	39	20	8	4	16	8	39	20
		s	30	90	485	1905	555	1380	3105	7845	145	335	690	2320

Abbreviations designate: A, annealed; AC, as cast; CD, cold drawn; and ST and A, solution treated and aged, respectively.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $^{3}/_{64}$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $^{3}/_{4}$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter *D* of 4.0 inch, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 15; diamond = 9.

Table 11. Cutting Feeds and Speeds for Milling Plain Carbon and Alloy Steels

		HSS		End Milling						Face Milling				Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)													
Material	Brinell Hardness	Speed (fpm)		*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining plain carbon steels (resulfurized): 1212, 1213, 1215	100–150	140	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
			s	45	125	465	735	800	1050	225	335	415	685	265	495	525	830
	150–200	130	**f**	7	4							39	20				
			s	35	100							215	405				
(Resulfurized): 1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	100–150	130	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
	150–200	115	**s**	30	85	325	565	465	720	140	220	195	365	170	350	245	495
	175–225	115	**f**	7	4									39	20		
			s	30	85									185	350		
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151	275–325	70															
	325–375	45	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
			s	25	70	210	435	300	560	90	170	175	330	90	235	135	325
	375–425	35															
(Leaded): 11L17, 11L18, 12L13, 12L14	100–150	140	**f**	7	4							39	20				
	150–200	130	**s**	35	100							215	405				
	200–250	110	**f**	7	4							39	20				
			s	30	85							185	350				
Plain carbon steels: 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	100–125	110	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
			s	45	125	465	735	800	1050	225	335	415	685	265	495	525	830
	125–175	110	**f**	7	4							39	20				
			s	35	100							215	405				
	175–225	90	**f**	7	4							39	20				
	225–275	65	**s**	30	85							185	350				

Table 11. *(Continued)* Cutting Feeds and Speeds for Milling Plain Carbon and Alloy Steels

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling				Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Plain carbon steels: 1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	125–175	100	**f**	7	4							39	20				
			s	35	100							215	405				
	175–225	85	**f**	7	4							39	20				
	225–275	70	**s**	30	85							185	350				
	275–325	55	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
	325–375	35	**s**	25	70	210	435	300	560	90	170	175	330	90	235	135	325
	375–425	25															
Plain carbon steels: 1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125–175	90	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
	175–225	75	**s**	30	85	325	565	465	720	140	220	195	365	170	350	245	495
	225–275	60	**f**	7	4							39	20				
			s	30	85							185	350				
	275–325	45	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
	325–375	30	**s**	25	70	210	435	300	560	90	170	175	330	90	235	135	325
	375–425	15															
Free-machining alloy steels (Resulfurized): 4140, 4150	175–200	100	**f**	15	8	15	8	15	8			39	20	39	20	7	4
	200–250	90	**s**	7	30	105	270	270	450			295	475	135	305	25	70
	250–300	60	**f**	15	8	15	8	15	8			39	20	39	20	7	4
			s	6	25	50	175	85	255			200	320	70	210	25	70
	300–375	45	**f**	15	8	15	8	15	8			39	20				
	375–425	35	**s**	5	20	40	155	75	225			175	280				

Table 11. *(Continued)* Cutting Feeds and Speeds for Milling Plain Carbon and Alloy Steels

Material	Brinell Hardness	HSS Speed (fpm)		End Milling HSS		End Milling Uncoated Carbide		End Milling Coated Carbide		Face Milling Uncoated Carbide		Face Milling Coated Carbide		Slit Milling Uncoated Carbide		Slit Milling Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining alloy steels (Leaded): 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40	150–200	115	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
			s	30	85	325	565	465	720	140	220	195	365	170	350	245	495
	200–250	95	**f**	7	4							39	20				
			s	30	85							185	350				
	250–300 300–375 375–425	70 50 40	**f**	7	4	7	4	7	4	39	20	39	20	39	20	39	20
			s	25	70	210	435	300	560	90	170	175	330	90	235	135	325
Alloy steels: 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125–175 175–225	100 90	**f**	15	8	15	8	15	8			39	20	39	20	39	20
			s	7	30	105	270	220	450			295	475	135	305	265	495
	225–275	60	**f**	15	8	15	8	15	8			39	20	39	20	39	20
			s	6	25	50	175	85	255			200	320	70	210	115	290
	275–325	50	**f**	15	8	15	8	15	8			39	20				
			s	5	20	45	170	80	240			190	305				
	325–375 375–425	40 25	**f**	15	8	15	8	15	8			39	20				
			s	5	20	40	155	75	225			175	280				
Alloy steels: 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30 E51100, E52100: use (HSS speeds)	175–225	75 (65)	**f**	15	8	15	8	15	8			39	20	39	20	39	20
			s	5	30	105	270	220	450			295	475	135	305	265	495
	225–275	60	**f**	15	8	15	8	15	8			39	20	39	20	39	20
			s	5	25	50	175	85	255			200	320	70	210	115	290
	275–325	50 (40)	**f**	15	8	15	8	15	8			39	20				
			s	5	25	45	170	80	240			190	305				
	325–375 375–425	35 (30) 20	**f**	15	8	15	8	15	8			39	20				
			s	5	20	40	155	75	225			175	280				

Table 11. *(Continued)* Cutting Feeds and Speeds for Milling Plain Carbon and Alloy Steels

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling				Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)													
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Ultra-high-strength steels (not AISI): AMS 6421 (98B37 Mod.), 6422 (98BV40), 6424, 6427, 6428, 6430, 6432, 6433, 6434, 6436, and 6442; 300M, D6ac	220–300 300–350	60 45	**f**			8	4	8	4								
			s			165	355	300	480								
	350–400	20	**f**	8	4	8	4					39	20	39	20		
			s	15	45	150	320					130	235	75	175		
	43–52 Rc	—	**f**			5	3							39	20		
			s			20†	55							5	15		
Maraging steels (not AISI): 18% Ni Grades 200, 250, 300, and 350	250–325	50	**f**			8	4	8	4								
			s			165	355	300	480								
	50–52 Rc	—	**f**			5	3							39	20		
			s			20†	55							5	15		
Nitriding steels (not AISI): Nitralloy 125, 135, 135 Mod., 225, and 230, Nitralloy N, Nitralloy EZ, Nitrex 1	200–250	60	**f**	15	8	15	8	15	8			39	20	39	20	39	20
			s	7	30	105	270	220	450			295	475	135	305	265	495
	300–350	25	**f**	15	8	15	8	15	8			39	20				
			s	5	20	40	155	75	225			175	280				

For HSS (high-speed steel) tools in the first speed column only, use Table 15a for recommended feed in inches per tooth and depth of cut.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $\frac{3}{64}$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $\frac{3}{4}$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter *D* of 4.0 inches, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: end and slit milling uncoated carbide = 20 except † = 15; face milling uncoated carbide = 19; end, face, and slit milling coated carbide = 10.

Table 12. Cutting Feeds and Speeds for Milling Tool Steels

				End Milling						Face Milling				Slit Milling			
		HSS		HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		CBN		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)													
Material	Brinell Hardness	Speed (fpm)		*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Water hardening: W1, W2, W5	150–200	85	**f** **s**	8 25	4 70	8 235	4 455	8 405	4 635	39 235	20 385			39 115	20 265	39 245	20 445
Shock resisting: S1, S2, S5, S6, S7	175–225	55															
Cold work, oil hardening: O1, O2, O6, O7	175–225	50															
Cold work, high carbon, high chromium: D2, D3, D4, D5, D7	200–250	40															
Cold work, air hardening: A2, A3, A8, A9, A10	200–250	50	**f** **s**							39 255	20 385						
A4, A6	200–250	45															
A7	225–275	40															
Hot work, chromium type: H10, H11, H12, H13, H14, H19	150–200	60															
	200–250	50															
	325–375	30	**f** **s**	8 15	4 45	8 150	4 320			39 130	20 235			39 75	20 175		
	48–50 Rc	—	**f** **s**			5 20†	3 55					39 50	20 135	39 5†	20 15		
	50–52 Rc	—															
	52–56 Rc	—															
Hot work, tungsten and molybdenum types: H21, H22, H23, H24, H25, H26, H41, H42, H43	150–200	55	**f** **s**							39 255	20 385						
	200–250	45															
Special-purpose, low alloy: L2, L3, L6	150–200	65	**f** **s**	8 25	4 70	8 235	4 455	8 405	4 635	39 235	20 385			39 115	20 265	39 245	20 445
Mold: P2, P3, P4, P5, P6 P20, P21	100–150	75															
	150–200	60															
High-speed steel: M1, M2, M6, M10, T1, T2, T6	200–250	50	**f** **s**							39 255	20 385						
M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8	225–275	40															
T15, M3-2	225–275	30															

For HSS (high-speed steel) tools in the first speed column only, use Table 15a for recommended feed in inches per tooth and depth of cut.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $\frac{3}{64}$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $\frac{3}{4}$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter *D* of 4.0 inches, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 20, † = 15; coated carbide = 10; CBN = 1.

Table 13. Cutting Feeds and Speeds for Milling Stainless Steels

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling		Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)											
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining stainless steels (Ferritic): 430F, 430FSe	135–185	110	**f** **s**	7 30	4 80	7 305	4 780	7 420	4 1240	39 210	20 385	39 120	20 345	39 155	20 475
(Austenitic): 203EZ, 303, 303Se, 303MA, 303Pb, 303Cu, 303 Plus X	135–185	100	**f** **s**	7 20	4 55	7 210	4 585					39 75	20 240		
	225–275	80													
(Martensitic): 416, 416Se, 416 Plus X, 420F, 420FSe, 440F, 440FSe	135–185	110													
	185–240	100													
	275–325	60													
	375–425	30													
Stainless steels (Ferritic): 405, 409, 429, 430, 434, 436, 442, 446, 502	135–185	90	**f** **s**	7 30	4 80	7 305	4 780	7 420	4 1240	39 210	20 385	39 120	20 345	39 155	20 475
(Austenitic): 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135–185	75	**f** **s**	7 20	4 55	7 210	4 585					39 75	20 240		
	225–275	65													
(Austenitic): 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	135–185	70													
(Martensitic): 403, 410, 420, 501	135–175	95													
	175–225	85													
	275–325	55													
	375–425	35													

Table 13. *(Continued)* Cutting Feeds and Speeds for Milling Stainless Steels

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling		Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)											
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Stainless Steels (Martensitic): 414, 431, Greek Ascoloy, 440A, 440B, 440C	225–275	55–60													
	275–325	45–50													
	375–425	30													
(Precipitation hardening): 15-5PH, 17-4PH, 17-7PH, AF-71, 17-14CuMo, AFC-77, AM-350, AM-355, AM-362, Custom 455, HNM, PH13-8, PH14-8Mo, PH15-7Mo, Stainless W	150–200	60	**f**	7	4	7	4					39	20		
			s	20	55	210	585					75	240		
	275–325	50													
	325–375	40													
	375–450	25													

For HSS (high-speed steel) tools in the first speed column only, use Table 15a for recommended feed in inches per tooth and depth of cut.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $3/64$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $3/4$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter *D* of 4.0 inch, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 20; coated carbide = 10.

Table 14. Cutting Feeds and Speeds for Milling Ferrous Cast Metals

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling								Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Ceramic		CBN		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)																	
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Gray Cast Iron																					
ASTM Class 20	120–150	100	**f** **s**	5 35	3 90	5 520	3 855			39 140	20 225	39 285	20 535	39 1130	20 1630	39 200	20 530	39 205	20 420		
ASTM Class 25	160–200	80																			
ASTM Class 30, 35, and 40	190–220	70																			
ASTM Class 45 and 50	220–260	50	**f** **s**	5 30	3 70	5 515	3 1100			39 95	20 160	39 185	20 395	39 845	20 1220	39 150	20 400	39 145	20 380		
ASTM Class 55 and 60	250–320	30																			
ASTM Type 1, 1b, 5 (Ni resist)	100–215	50																			
ASTM Type 2, 3, 6 (Ni resist)	120–175	40																			
ASTM Type 2b, 4 (Ni resist)	150–250	30																			
Malleable Iron																					
(Ferritic): 32510, 35018	110–160	110	**f** **s**	5 30	3 70	5 180	3 250			39 120	20 195	39 225	20 520	39 490	20 925			39 85	20 150		
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160–200	80	**f** **s**	5 25	3 65	5 150	3 215			39 90	20 150	39 210	20 400	39 295	20 645			39 70	20 125		
	200–240	65																			
(Martensitic): 53004, 60003, 60004	200–255	55																			
(Martensitic): 70002, 70003	220–260	50																			
(Martensitic): 80002	240–280	45																			
(Martensitic): 90001	250–320	25																			
Nodular (Ductile) Iron																					
(Ferritic): 60-40-18, 65-45-12	140–190	75	**f** **s**	7 15	4 35	7 125	4 240			39 100	20 155	39 120	20 255	39 580	20 920			39 60	20 135		
(Ferritic-Pearlitic): 80-55-06	190–225	60	**f** **s**	7 10	4 30	7 90	4 210			39 95	20 145	39 150	20 275	39 170	20 415			39 40	20 100		
	225–260	50																			
(Pearlitic-Martensitic): 100-70-03	240–300	40																			
(Martensitic): 120-90-02	270–330	25																			

Table 14. *(Continued)* Cutting Feeds and Speeds for Milling Ferrous Cast Metals

Material	Brinell Hardness	HSS Speed (fpm)		End Milling						Face Milling								Slit Milling			
				HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Ceramic		CBN		Uncoated Carbide		Coated Carbide	
				f = feed (0.001 in./tooth), **s** = speed (ft/min)																	
				Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Cast Steels																					
(Low carbon): 1010, 1020	100–150	100	**f**	7	4	7	4	7	4			39	20					39	20	39	20
	125–175	95	**s**	25	70	245†	410	420	650			265‡	430					135†	260	245	450
(Medium carbon): 1030, 1040 1050	175–225	80																			
	225–300	60																			
			f	7	4	7	4	7	4			39	20					39	20	39	20
			s	20	55	160†	400	345	560			205‡	340					65†	180	180	370
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150–200	85																			
	200–250	75																			
	250–300	50																			
(Medium-carbon alloy): 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175–225	70	**f**	7	4	7	4											39	20		
	225–250	65	**s**	15	45	120†	310											45†	135		
	250–300	50	**f**							39	20										
	300–350	30	**s**							25	40										

For HSS (high-speed steel) tools in the first speed column only, use Table 15a for recommended feed in inches per tooth and depth of cut.

End Milling: Table data for end milling are based on a 3-tooth, 20-degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot). Use Table 15b to adjust speeds for other feeds and axial depths of cut, and Table 15c to adjust speeds if the radial depth of cut is less than the tool diameter. Speeds are valid for all tool diameters.

Face Milling: Table data for face milling are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $\frac{3}{64}$-inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $\frac{3}{4}$). These speeds are valid if the cutter axis is above or close to the center line of the workpiece (eccentricity is small). Under these conditions, use Table 15d to adjust speeds for other feeds and axial and radial depths of cut. For larger eccentricity (i.e., when the cutter axis to workpiece center line offset is one half the cutter diameter or more), use the end and side milling adjustment factors (Tables 15b and 15c) instead of the face milling factors.

Slit and Slot Milling: Table data for slit milling are based on an 8-tooth, 10-degree helix angle tool with a cutter width of 0.4 inch, diameter *D* of 4.0 inches, and a depth of cut of 0.6 inch. Speeds are valid for all tool diameters and widths. See the examples in the text for adjustments to the given speeds for other feeds and depths of cut.

Tool life for all tabulated values is approximately 45 minutes; use Table 15e to adjust tool life from 15 to 180 minutes. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 15 except † = 20; end and slit milling coated carbide = 10; face milling coated carbide = 11 except ‡ = 10. ceramic = 6; CBN = 1.

Table 15a. Recommended Feed in Inches per Tooth (f_t) for Milling with High Speed Steel Cutters

Material	Hard-ness, HB	End Mills							Plain or Slab Mills	Form Relieved Cutters	Face Mills and Shell End Mills	Slotting and Side Mills
		Depth of Cut, .250 in			Depth of Cut, .050 in							
		Cutter Diam., in			Cutter Diam., in							
		½	¾	1 and up	¼	½	¾	1 and up				
		Feed per Tooth, inch										
Free-machining plain carbon steels	100–185	.001	.003	.004	.001	.002	.003	.004	.003–.008	.005	.004–.012	.002–.008
Plain carbon steels, AISI 1006 to 1030; 1513 to 1522 {	100–150	.001	.003	.003	.001	.002	.003	.004	.003–.008	.004	.004–.012	.002–.008
	150–200	.001	.002	.003	.001	.002	.002	.003	.003–.008	.004	.003–.012	.002–.008
AISI 1033 to 1095; 1524 to 1566 {	120–180	.001	.003	.003	.001	.002	.003	.004	.003–.008	.004	.004–.012	.002–.008
	180–220	.001	.002	.003	.001	.002	.002	.003	.003–.008	.004	.003–.012	.002–.008
	220–300	.001	.002	.002	.001	.001	.002	.003	.002–.006	.003	.002–.008	.002–.006
Alloy steels having less than 3% carbon. Typical examples: AISI 4012, 4023, 4027, 4118, 4320 4422, 4427, 4615, 4620, 4626, 4720, 4820, 5015, 5120, 6118, 8115, 8620 8627, 8720, 8820, 8822, 9310, 93B17	125–175	.001	.003	.003	.001	.002	.003	.004	.003–.008	.004	.004–.012	.002–.008
	175–225	.001	.002	.003	.001	.002	.003	.003	.003–.008	.004	.003–.012	.002–.008
	225–275	.001	.002	.003	.001	.001	.002	.003	.002–.006	.003	.003–.008	.002–.006
	275–325	.001	.002	.002	.001	.001	.002	.002	.002–.005	.003	.002–.008	.002–.005
Alloy steels having 3% carbon or more. Typical examples: AISI 1330, 1340, 4032, 4037, 4130, 4140, 4150, 4340, 50B40, 50B60, 5130, 51B60, 6150, 81B45, 8630, 8640, 86B45, 8660, 8740, 94B30	175–225	.001	.002	.003	.001	.002	.003	.004	.003–.008	.004	.003–.012	.002–.008
	225–275	.001	.002	.003	.001	.001	.002	.003	.002–.006	.003	.003–.010	.002–.006
	275–325	.001	.002	.002	.001	.001	.002	.003	.002–.005	.003	.002–.008	.002–.005
	325–375	.001	.002	.002	.001	.001	.002	.002	.002–.004	.002	.002–.008	.002–.005
Tool steel	150–200	.001	.002	.002	.001	.002	.003	.003	.003–.008	.004	.003–.010	.002–.006
	200–250	.001	.002	.002	.001	.002	.002	.003	.002–.006	.003	.003–.008	.002–.005
Gray cast iron	120–180	.001	.003	.004	.002	.003	.004	.004	.004–.012	.005	.005–.016	.002–.010
	180–225	.001	.002	.003	.001	.002	.003	.003	.003–.010	.004	.004–.012	.002–.008
	225–300	.001	.002	.002	.001	.001	.002	.002	.002–.006	.003	.002–.008	.002–.005
Free malleable iron	110–160	.001	.003	.004	.002	.003	.004	.004	.003–.010	.005	.005–.016	.002–.010

Table 15a. *(Continued)* **Recommended Feed in Inches per Tooth (f_t) for Milling with High Speed Steel Cutters**

Material*(Continued)*	Hard-ness, HB	End Mills							Plain or Slab Mills	Form Relieved Cutters	Face Mills and Shell End Mills	Slotting and Side Mills
		Depth of Cut, .250 in			Depth of Cut, .050 in							
		Cutter Diam., in			Cutter Diam., in							
		½	¾	1 and up	¼	½	¾	1 and up				
		Feed per Tooth, inch										
Pearlitic-Martensitic malleable iron	160–200	.001	.003	.004	.001	.002	.003	.004	.003–.010	.004	.004–.012	.002–.018
	200–240	.001	.002	.003	.001	.002	.003	.003	.003–.007	.004	.003–.010	.002–.006
	240–300	.001	.002	.002	.001	.001	.002	.002	.002–.006	.003	.002–.008	.002–.005
Cast steel	100–180	.001	.003	.003	.001	.002	.003	.004	.003–.008	.004	.003–.012	.002–.008
	180–240	.001	.002	.003	.001	.002	.003	.003	.003–.008	.004	.003–.010	.002–.006
	240–300	.001	.002	.002	.005	.002	.002	.002	.002–.006	.003	.003–.008	.002–.005
Zinc alloys (die castings)	…	.002	.003	.004	.001	.003	.004	.006	.003–.010	.005	.004–.015	.002–.012
Copper alloys (brasses & bronzes)	100–150	.002	.004	.005	.002	.003	.005	.006	.003–.015	.004	.004–.020	.002–.010
	150–250	.002	.003	.004	.001	.003	.004	.005	.003–.015	.004	.003–.012	.002–.008
Free cutting brasses & bronzes	80–100	.002	.004	.005	.002	.003	.005	.006	.003–.015	.004	.004–.015	.002–.010
Cast aluminum alloys—as cast	…	.003	.004	.005	.002	.004	.005	.006	.005–.016	.006	.005–.020	.004–.012
Cast aluminum alloys—hardened	…	.003	.004	.005	.002	.003	.004	.005	.004–.012	.005	.005–.020	.004–.012
Wrought aluminum alloys— cold drawn	…	.003	.004	.005	.002	.003	.004	.005	.004–.014	.005	.005–.020	.004–.012
Wrought aluminum alloys—hardened	…	.002	.003	.004	.001	.002	.003	.004	.003–.012	.004	.005–.020	.004–.012
Magnesium alloys	…	.003	.004	.005	.003	.004	.005	.007	.005–.016	.006	.008–.020	.005–.012
Ferritic stainless steel	135–185	.001	.002	.003	.001	.002	.003	.003	.002–.006	.004	.004–.008	.002–.007
Austenitic stainless steel	135–185	.001	.002	.003	.001	.002	.003	.003	.003–.007	.004	.005–.008	.002–.007
	185–275	.001	.002	.003	.001	.002	.002	.002	.003–.006	.003	.004–.006	.002–.007
Martensitic stainless steel	135–185	.001	.002	.002	.001	.002	.003	.003	.003–.006	.004	.004–.010	.002–.007
	185–225	.001	.002	.002	.001	.002	.002	.003	.003–.006	.004	.003–.008	.002–.007
	225–300	.0005	.002	.002	.0005	.001	.002	.002	.002–.005	.003	.002–.006	.002–.005
Monel	100–160	.001	.003	.004	.001	.002	.003	.004	.002–.006	.004	.002–.008	.002–.006

Table 15b. End Milling (Full Slot) Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle

Cutting Speed, $V = V_{opt} \times F_f \times F_d$																	
Ratio of Chosen Feed to *Optimum* Feed	Ratio of the two cutting speeds (*average*/*optimum*) given in the tables V_{avg}/V_{opt}							Depth of Cut and Lead Angle									
								1 in	(25.4 mm)	0.4 in	(10.2 mm)	0.2 in	(5.1 mm)	0.1 in	(2.4 mm)	0.04 in	(1.0 mm)
	1.00	1.25	1.50	2.00	2.50	3.00	4.00	0°	45°	0°	45°	0°	45°	0°	45°	0°	45°
	Feed Factor, F_f							Depth of Cut and Lead Angle Factor, F_d									
1.00	1.0	1.0	1.0	1.0	1.0	1.0	1.0	0.91	1.36	0.94	1.38	1.00	0.71	1.29	1.48	1.44	1.66
0.90	1.00	1.06	1.09	1.14	1.18	1.21	1.27	0.91	1.33	0.94	1.35	1.00	0.72	1.26	1.43	1.40	1.59
0.80	1.00	1.12	1.19	1.31	1.40	1.49	1.63	0.92	1.30	0.95	1.32	1.00	0.74	1.24	1.39	1.35	1.53
0.70	1.00	1.18	1.30	1.50	1.69	1.85	2.15	0.93	1.26	0.95	1.27	1.00	0.76	1.21	1.35	1.31	1.44
0.60	1.00	1.20	1.40	1.73	2.04	2.34	2.89	0.94	1.22	0.96	1.25	1.00	0.79	1.18	1.28	1.26	1.26
0.50	1.00	1.25	1.50	2.00	2.50	3.00	4.00	0.95	1.17	0.97	1.18	1.00	0.82	1.14	1.21	1.20	1.21
0.40	1.00	1.23	1.57	2.29	3.08	3.92	5.70	0.96	1.11	0.97	1.12	1.00	0.86	1.09	1.14	1.13	1.16
0.30	1.00	1.14	1.56	2.57	3.78	5.19	8.56	0.98	1.04	0.99	1.04	1.00	0.91	1.04	1.07	1.05	1.09
0.20	1.00	0.90	1.37	2.68	4.49	6.86	17.60	1.00	0.85	1.00	0.95	1.00	0.99	0.97	0.93	0.94	0.88
0.10	1.00	0.44	0.80	2.08	4.26	8.00	20.80	1.05	0.82	1.00	0.81	1.00	1.50	0.85	0.76	0.78	0.67

For HSS (high-speed steel) tool speeds in the first speed column of Tables 10 through 14, use Table 15a to determine appropriate feeds and depths of cut.

Cutting feeds and speeds for end milling given in Tables 11 through 14 (except those for high-speed steel in the first speed column) are based on milling a 0.20-inch deep full slot (i.e., radial depth of cut = end mill diameter) with a 1-inch diameter, 20-degree helix angle, 0-degree lead angle end mill. For other depths of cut (axial), lead angles, or feed, use the two feed/speed pairs from the tables and calculate the ratio of desired (new) feed to *optimum* feed (largest of the two feeds are given in the tables), and the ratio of the two cutting speeds (V_{avg}/V_{opt}). Find the feed factor F_f at the intersection of the feed ratio row and the speed ratio column in the left half of the Table. The depth of cut factor F_d is found in the same row as the feed factor, in the right half of the table under the column corresponding to the depth of cut and lead angle. The adjusted cutting speed can be calculated from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the smaller (*optimum*) of the two speeds from the speed table (from the left side of the column containing the two feed/speed pairs). See the text for examples.

If the radial depth of cut is less than the cutter diameter (i.e., for cutting less than a full slot), the feed factor F_f in the previous equation and the maximum feed f_{max} must be obtained from Table 15c. The axial depth of cut factor F_d can then be obtained from this table using f_{max} in place of the *optimum* feed in the feed ratio. Also see the footnote to Table 15c.

Table 15c. End, Slit, and Side Milling Speed Adjustment Factors for Radial Depth of Cut

Ratio of Radial Depth of Cut to Diameter	Cutting Speed, $V = V_{opt} \times F_f \times F_d$													
	Maximum Feed/Tooth Factor	V_{avg}/V_{opt}						Maximum Feed/Tooth Factor	V_{avg}/V_{opt}					
		1.25	1.50	2.00	2.50	3.00	4.00		1.25	1.50	2.00	2.50	3.00	4.00
		Feed Factor F_f at Maximum Feed per Tooth, F_{f1}							Feed Factor F_f at Minimum Feed per Tooth, F_{f2}					
1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	0.70	1.18	1.30	1.50	1.69	1.85	2.15
0.75	1.00	1.15	1.24	1.46	1.54	1.66	1.87	0.70	1.24	1.48	1.93	2.38	2.81	3.68
0.60	1.00	1.23	1.40	1.73	2.04	2.34	2.89	0.70	1.24	1.56	2.23	2.95	3.71	5.32
0.50	1.00	1.25	1.50	2.00	2.50	3.00	4.00	0.70	1.20	1.58	2.44	3.42	4.51	6.96
0.40	1.10	1.25	1.55	2.17	2.83	3.51	4.94	0.77	1.25	1.55	2.55	3.72	5.08	8.30
0.30	1.35	1.20	1.57	2.28	3.05	3.86	5.62	0.88	1.23	1.57	2.64	4.06	5.76	10.00
0.20	1.50	1.14	1.56	2.57	3.78	5.19	8.56	1.05	1.40	1.56	2.68	4.43	6.37	11.80
0.10	2.05	0.92	1.39	2.68	4.46	6.77	13.10	1.44	0.92	1.29	2.50	4.66	7.76	17.40
0.05	2.90	0.68	1.12	2.50	4.66	7.75	17.30	2.00	0.68	1.12	2.08	4.36	8.00	20.80
0.02	4.50	0.38	0.71	1.93	4.19	7.90	21.50	3.10	0.38	0.70	1.38	3.37	7.01	22.20

This table is for side milling, end milling when the radial depth of cut (width of cut) is less than the tool diameter (i.e., less than full slot milling), and slit milling when the feed is parallel to the work surface (slotting). The radial depth of cut to diameter ratio is used to determine the recommended maximum and minimum values of feed/tooth, which are found by multiplying the feed/tooth factor from the appropriate column above (maximum or minimum) by feed_{opt} from the speed tables. For example, given two feed/speed pairs 7⁄15 and 4⁄45 for end milling cast, medium-carbon, alloy steel, and a radial depth of cut to diameter ratio ar/D of 0.10 (a 0.05-inch width of cut for a ½-inch diameter end mill, for example), the maximum feed $f_{max} = 2.05 \times 0.007 = 0.014$ in./tooth and the minimum feed $f_{min} = 1.44 \times 0.007 = 0.010$ in./tooth. The feed selected should fall in the range between f_{min} and f_{max}. The feed factor F_d is determined by interpolating between the feed factors F_{f1} and F_{f2} corresponding to the maximum and minimum feed per tooth, at the appropriate ar/D and speed ratio. In the example given, $ar/D = 0.10$ and $V_{avg}/V_{opt} = 45/15 = 3$, so the feed factor F_{f1} at the maximum feed per tooth is 6.77, and the feed factor F_{f2} at the minimum feed per tooth is 7.76. If a working feed of 0.012 in./tooth is chosen, the feed factor F_f is half way between 6.77 and 7.76 or by formula, $F_f = F_{f1} + (\text{feed} - f_{min})/(f_{max} - f_{min}) \times (f_{f2} - f_{f1}) = 6.77 + (0.012 - 0.010)/(0.014 - 0.010) \times (7.76 - 6.77) = 7.27$. The cutting speed is $V = V_{opt} \times F_f \times F_d$, where F_d is the depth of cut and lead angle factor from Table 15b that corresponds to the feed ratio (chosen feed)/f_{max}, not the ratio (chosen feed)/*optimum* feed. For a feed ratio = 0.012⁄0.014 = 0.86 (chosen feed/f_{max}), depth of cut = 0.2 inch and lead angle = 45°, the depth of cut factor F_d in Table 15b is between 0.72 and 0.74. Therefore, the final cutting speed for this example is $V = V_{opt} \times F_f \times F_d = 15 \times 7.27 \times 0.73 = 80$ ft/min.

Slit and Side Milling: This table only applies when feed is parallel to the work surface, as in slotting. If feed is perpendicular to the work surface, as in cutting off, obtain the required speed-correction factor from Table 15d (face milling). The minimum and maximum feeds/tooth for slit and side milling are determined in the manner described above, however, the axial depth of cut factor F_d is not required. The adjusted cutting speed, valid for cutters of any thickness (width), is given by $V = V_{opt} \times F_f$. Examples are given in the text.

Table 15d. Face Milling Speed Adjustment Factors for Feed, Depth of Cut, and Lead Angle

Cutting Speed $V = V_{opt} \times F_f \times F_d \times F_{ar}$																								
Ratio of Chosen Feed to *Optimum* Feed	Ratio of the two cutting speeds (*average*/*optimum*) given in the tables V_{avg}/V_{opt}							Depth of Cut, inch (mm), and Lead Angle										Ratio of Radial Depth of Cut/Cutter Diameter, *ar*/*D*						
								1 in (25.4 mm)		0.4 in (10.2 mm)		0.2 in (5.1 mm)		0.1 in (2.4 mm)		0.04 in (1.0 mm)								
	1.00	1.10	1.25	1.35	1.50	1.00	2.00	15°	45°	15°	45°	15°	45°	15°	45°	15°	45°	1.00	0.75	0.50	0.40	0.30	0.20	0.10
	Feed Factor, F_f							Depth of Cut Factor, F_d										Radial Depth of Cut Factor, F_{ar}						
1.00	1.0	1.0	1.0	1.0	1.0	1.0	1.0	0.78	1.11	0.94	1.16	0.90	1.10	1.00	1.29	1.47	1.66	0.72	1.00	1.53	1.89	2.43	3.32	5.09
0.90	1.00	1.02	1.05	1.07	1.09	1.10	1.12	0.78	1.10	0.94	1.16	0.90	1.09	1.00	1.27	1.45	1.58	0.73	1.00	1.50	1.84	2.24	3.16	4.69
0.80	1.00	1.03	1.09	1.10	1.15	1.20	1.25	0.80	1.10	0.94	1.14	0.91	1.08	1.00	1.25	1.40	1.52	0.75	1.00	1.45	1.73	2.15	2.79	3.89
0.70	1.00	1.05	1.13	1.22	1.22	1.32	1.43	0.81	1.09	0.95	1.14	0.91	1.08	1.00	1.24	1.39	1.50	0.75	1.00	1.44	1.72	2.12	2.73	3.77
0.60	1.00	1.08	1.20	1.25	1.35	1.50	1.66	0.81	1.09	0.95	1.13	0.92	1.08	1.00	1.23	1.38	1.48	0.76	1.00	1.42	1.68	2.05	2.61	3.52
0.50	1.00	1.10	1.25	1.35	1.50	1.75	2.00	0.81	1.09	0.95	1.13	0.92	1.08	1.00	1.23	1.37	1.47	0.76	1.00	1.41	1.66	2.02	2.54	3.39
0.40	1.00	1.09	1.28	1.44	1.66	2.03	2.43	0.82	1.08	0.95	1.12	0.92	1.07	1.00	1.21	1.34	1.43	0.78	1.00	1.37	1.60	1.90	2.34	2.99
0.30	1.00	1.06	1.32	1.52	1.85	2.42	3.05	0.84	1.07	0.96	1.11	0.93	1.06	1.00	1.18	1.30	1.37	0.80	1.00	1.32	1.51	1.76	2.10	2.52
0.20	1.00	1.00	1.34	1.60	2.07	2.96	4.03	0.86	1.06	0.96	1.09	0.94	1.05	1.00	1.15	1.24	1.29	0.82	1.00	1.26	1.40	1.58	1.79	1.98
0.10	1.00	0.80	1.20	1.55	2.24	3.74	5.84	0.90	1.04	0.97	1.06	0.96	1.04	1.00	1.10	1.15	1.18	0.87	1.00	1.16	1.24	1.31	1.37	1.32

For HSS (high-speed steel) tool speeds in the first speed column, use Table 15a to determine appropriate feeds and depths of cut.

Tabular feeds and speeds data for face milling in Tables 11 through 14 are based on a 10-tooth, 8-inch diameter face mill, operating with a 15-degree lead angle, $\frac{3}{64}$-inch cutter insert nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches (i.e., width of cut to cutter diameter ratio = $\frac{3}{4}$). For other depths of cut (radial or axial), lead angles, or feed, calculate the ratio of desired (new) feed to *optimum* feed (largest of the two feeds given in the speed table), and the ratio of the two cutting speeds (V_{avg}/V_{opt}). Use these ratios to find the feed factor F_f at the intersection of the feed ratio row and the speed ratio column in the left third of the table. The depth of cut factor F_d is found in the same row as the feed factor, in the center third of the table, in the column corresponding to the depth of cut and lead angle. The radial depth of cut factor F_{ar} is found in the same row as the feed factor, in the right third of the table, in the column corresponding to the radial depth of cut to cutter diameter ratio *ar*/*D*. The adjusted cutting speed can be calculated from $V = V_{opt} \times F_f \times F_d \times F_{ar}$, where V_{opt} is the smaller (*optimum*) of the two speeds from the speed table (from the left side of the column containing the two feed/speed pairs).

The cutting speeds as calculated above are valid if the cutter axis is centered above or close to the center line of the workpiece (eccentricity is small). For larger eccentricity (i.e., the cutter axis is offset from the center line of the workpiece by about one-half the cutter diameter or more), use the adjustment factors from Tables 15b and 15c (end and side milling) instead of the factors from this table. Use Table 15e to adjust end and face milling speeds for increased tool life up to 180 minutes.

Slit and Slot Milling: Tabular speeds are valid for all tool diameters and widths. Adjustments to the given speeds for other feeds and depths of cut depend on the circumstances of the cut. *Case 1:* If the cutter is fed directly into the workpiece, i.e., the feed is perpendicular to the surface of the workpiece, as in cutting off, then this table (face milling) is used to adjust speeds for other feeds. The depth of cut factor is not used for slit milling ($F_d = 1.0$), so the adjusted cutting speed $V = V_{opt} \times F_f \times F_{ar}$. For determining the factor F_{ar}, the radial depth of cut *ar* is the length of cut created by the portion of the cutter engaged in the work. *Case 2:* If the cutter is fed parallel to the surface of the workpiece, as in slotting, then Tables 15b and 15c are used to adjust the given speeds for other feeds. See Fig. 5.

Table 15e. Tool Life Adjustment Factors for Face Milling, End Milling, Drilling, and Reaming

Tool Life, T (minutes)	Face Milling with Carbides and Mixed Ceramics			End Milling with Carbides and HSS			Twist Drilling and Reaming with HSS		
	f_s	f_m	f_l	f_s	f_m	f_l	f_s	f_m	f_l
15	1.69	1.78	1.87	1.10	1.23	1.35	1.11	1.21	1.30
45	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
90	0.72	0.70	0.67	0.94	0.89	0.83	0.93	0.89	0.85
180	0.51	0.48	0.45	0.69	0.69	0.69	0.87	0.80	0.72

The feeds and speeds given in Tables 11 through 14 and Tables 17 through 23 (except for HSS speeds in the first speed column) are based on a 45-minute tool life. To adjust the given speeds to obtain another tool life, multiply the adjusted cutting speed for the 45-minute tool life V_{45} by the tool life factor from this table according to the following rules: for small feeds, where feed $\le \frac{1}{2} f_{opt}$, the cutting speed for the desired tool life T is $V_T = f_s \times V_{15}$; for medium feeds, where $\frac{1}{2} f_{opt} <$ feed $< \frac{3}{4} f_{opt}$, $V_T = f_m \times V_{15}$; and for larger feeds, where $\frac{3}{4} f_{opt} \le$ feed $\le f_{opt}$, $V_T = f_l \times V_{15}$. Here, f_{opt} is the largest (*optimum*) feed of the two feed/speed values given in the speed tables or the maximum feed f_{max} obtained from Table 15c, if that table was used in calculating speed adjustment factors.

Table 16. Cutting Tool Grade Descriptions and Common Vendor Equivalents

Grade Description	Tool Identification Code	Approximate Vendor Equivalents			
		Sandvik Coromant	Kennametal	Seco	Valenite
Cubic boron nitride	1	CB50	KD050	CBN20	VC721
Ceramics	2	CC620	K060	480	—
	3	CC650	K090	480	Q32
	4 (Whiskers)	CC670	KYON2500	—	—
	5 (Sialon)	CC680	KYON2000	480	—
	6	CC690	KYON3000	—	Q6
Cermets	7	CT515	KT125	CM	VC605
	8	CT525	KT150	CR	VC610
Polycrystalline	9	CD10	KD100	PAX20	VC727
Coated carbides	10	GC-A	—	—	—
	11	GC3015	KC910	TP100	SV310
	12	GC235	KC9045	TP300	SV235
	13	GC4025	KC9025	TP200	SV325
	14	GC415	KC950	TP100	SV315
Uncoated carbides	15	H13A	K8, K4H	883	VC2
	16	S10T	K420, K28	CP20	VC7
	17	S1P	K45	CP20	VC7
	18	S30T	—	CP25	VC5
	19	S6	K21, K25	CP50	VC56
	20	SM30	KC710	CP25	VC35M

See Table 2 on page 753 and the section *Cemented Carbides and Other Hard Materials* for more detailed information on cutting tool grades.

The identification codes in column two correspond to the grade numbers given in the footnotes to Tables 1 to 4b, 6 to 14, and 17 to 23.

Using the Feed and Speed Tables for Drilling, Reaming, and Threading.—The first two speed columns in Tables 17 through 23 give traditional Handbook speeds for drilling and reaming. The following material can be used for selecting feeds for use with the traditional speeds.

The remaining columns in Tables 17 through 23 contain combined feed/speed data for drilling, reaming, and threading, organized in the same manner as in the turning and milling tables. Operating at the given feeds and speeds is expected to result in a tool life of approximately 45 minutes, except for indexable insert drills, which have an expected tool life of approximately 15 minutes per edge. Examples of using this data follow.

Adjustments to HSS drilling speeds for feed and diameter are made using Table 22; Table 5a is used for adjustments to indexable insert drilling speeds, where one-half the drill diameter *D* is used for the depth of cut. Tool life for HSS drills, reamers, and thread chasers and taps may be adjusted using Table 15e and for indexable insert drills using Table 5b.

The feed for drilling is governed primarily by the size of the drill and by the material to be drilled. Other factors that also affect selection of the feed are the workpiece configuration, the rigidity of the machine tool and the workpiece setup, and the length of the chisel edge. A chisel edge that is too long will result in a very significant increase in the thrust force, which may cause large deflections to occur on the machine tool and drill breakage.

For ordinary twist drills, the feed rate used is 0.001 to 0.003 in /rev for drills smaller than ⅛ in, 0.002 to 0.006 in./rev for ⅛- to ¼-in drills; 0.004 to 0.010 in./rev for ¼- to ½-in drills; 0.007 to 0.015 in./rev for ½- to 1-in drills; and, 0.010 to 0.025 in./rev for drills larger than 1 inch.

The lower values in the feed ranges should be used for hard materials such as tool steels, superalloys, and work-hardening stainless steels; the higher values in the feed ranges should be used to drill soft materials such as aluminum and brass.

Example 1, Drilling: Determine the cutting speed and feed for use with HSS drills in drilling 1120 steel.

Table 15a gives two sets of feed and speed parameters for drilling 1120 steel with HSS drills. These sets are 16⁄50 and 8⁄95, i.e., 0.016 in./rev feed at 50 ft/min and 0.008 in./rev at 95 fpm, respectively. These feed/speed sets are based on a 0.6-inch diameter drill. Tool life for either of the given feed/speed settings is expected to be approximately 45 minutes.

For different feeds or drill diameters, the cutting speeds must be adjusted and can be determined from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the minimum speed for this material given in the speed table (50 fpm in this example) and F_f and F_d are the adjustment factors for feed and diameter, respectively, found in Table 22.

Table 17. Feeds and Speeds for Drilling, Reaming, and Threading Plain Carbon and Alloy Steels

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Coated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining plain carbon steels (Resulfurized): 1212, 1213, 1215 {	100–150	120	80	**f** **s**	21 55	11 125	8 310	4 620	36 140	18 185	83 140	20 185
	150–200	125	80									
(Resulfurized): 1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211 {	100–150	110	75	**f** **s**	16 50	8 95	8 370	4 740	27 105	14 115	83 90	20 115
	150–200	120	80									
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151 {	175–225	100	65	**f** **s**			8 365	4 735				
	275–325	70	45									
	325–375	45	30									
	375–425	35	20									
(Leaded): 11L17, 11L18, 12L13, 12L14 {	100–150	130	85									
	150–200	120	80									
	200–250	90	60	**f** **s**			8 365	4 735				
Plain carbon steels: 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514 {	100–125	100	65	**f** **s**	21 55	11 125	8 310	4 620	36 140	18 185	83 140	20 185
	125–175	90	60									
	175–225	70	45	**f** **s**			8 365	4 735				
	225–275	60	40									
Plain carbon steels*:* 1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541 {	125–175	90	60	**f** **s**			8 365	4 735				
	175–225	75	50									
	225–275	60	40									
	275–325	50	30									
	325–375	35	20									
	375–425	25	15									

Table 17. *(Continued)* Feeds and Speeds for Drilling, Reaming, and Threading Plain Carbon and Alloy Steels

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Coated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Plain carbon steels *(Continued):* 1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125–175	85	55	**f**	16	8	8	4	27	14	83	20
	175–225	70	45	**s**	50	95	370	740	105	115	90	115
	225–275	50	30	**f** **s**			8 365	4 735				
	275–325	40	25									
	325–375	30	20									
	375–425	15	10									
Free-machining alloy steels (Resulfurized): 4140, 4150	175–200	90	60	**f**	16	8	8	4	26	13	83	20
	200–250	80	50	**s**	75	140	410	685	150	160	125	160
	250–300	55	30	**f** **s**			8 355	4 600				
	300–375	40	25	**f**			8	4				
	375–425	30	15	**s**			310	525				
(Leaded): 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40	150–200	100	65	**f** **s**	16 50	8 95	8 370	4 740	27 105	14 115	83 90	20 115
	200–250	90	60	**f** **s**			8 365	4 735				
	250–300	65	40									
	300–375	45	30									
	375–425	30	15									
Alloy steels: 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125–175	85	55	**f**	16	8	8	4	26	13	83	20
	175–225	70	45	**s**	75	140	410	685	150	160	125	160
	225–275	55	35	**f** **s**			8 355	4 600				
	275–325	50	30	**f** **s**	11 50	6 85	8 335	4 570	19 95	10 135	83 60	20 95
	325–375	35	25	**f**			8	4				
	375–425	25	15	**s**			310	525				

Table 17. *(Continued)* **Feeds and Speeds for Drilling, Reaming, and Threading Plain Carbon and Alloy Steels**

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Coated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Alloy steels: 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30 E51100, E52100: use (HSS speeds)	175–225	75 (60)	50 (40)	**f**	16	8	8	4	26	13	83	20
				s	75	140	410	685	150	160	125	160
	225–275	60 (50)	40 (30)	**f**			8	4				
				s			355	600				
	275–325	45 (35)	30 (25)	**f**	11	6	8	4	19	10	83	20
				s	50	85	335	570	95	135	60	95
	325–375	30 (30)	15 (20)	**f**			8	4				
	375–425	20 (20)	15 (10)	**s**			310	525				
Ultra-high-strength steels (not AISI): AMS 6421 (98B37 Mod.), 6422 (98BV40), 6424, 6427, 6428, 6430, 6432, 6433, 6434, 6436, and 6442; 300M, D6ac	220–300	50	30	**f**			8	4				
	300–350	35	20	**s**			325	545				
	350–400	20	10	**f**			8	4				
				s			270	450				
Maraging steels (not AISI): 18% Ni Grade 200, 250, 300, and 350	250–325	50	30	**f**			8	4				
				s			325	545				
Nitriding steels (not AISI): Nitralloy 125, 135, 135 Mod., 225, and 230, Nitralloy N, Nitralloy EZ, Nitrex I	200–250	60	40	**f**	16	8	8	4	26	13	83	20
				s	75	140	410	685	150	160	125	160
	300–350	35	20	**f**			8	4				
				s			310	525				

The two leftmost speed columns in this table contain traditional Handbook speeds for drilling and reaming with HSS steel tools. The section Feed Rates for Drilling and Reaming contains useful information concerning feeds to use in conjunction with these speeds.

HSS Drilling and Reaming: The combined feed/speed data for drilling are based on a 0.60-inch diameter HSS drill with standard drill point geometry (2-flute with 118° tip angle). Speed adjustment factors in Table 22 are used to adjust drilling speeds for other feeds and drill diameters. Examples of using this data are given in the text. The given feeds and speeds for reaming are based on an 8-tooth, $^{25}/_{32}$-inch diameter, 30° lead angle reamer, and a 0.008-inch radial depth of cut. For other feeds, the correct speed can be obtained by interpolation using the given speeds if the desired feed lies in the recommended range (between the given values of *optimum* and *average* feed). If a feed lower than the given *average* value is chosen, the speed should be maintained at the corresponding *average* speed (i.e., the highest of the two speed values given). The cutting speeds for reaming do not require adjustment for tool diameters for standard ratios of radical depth of cut to reamer diameter (i.e., $f_d =$ 1.00). Speed adjustment factors to modify tool life are found in Table 15e.

Indexable Insert Drilling: The feed/speed data for indexable insert drilling are based on a tool with two cutting edges, an insert nose radius of $\frac{3}{64}$ inch, a 10-degree lead angle, and diameter $D = 1$ inch. Adjustments to cutting speed for feed and depth of cut are made using Table 5aAdjustment Factors) using a depth of cut of $D/2$, or one-half the drill diameter. Expected tool life at the given feeds and speeds is approximately 15 minutes for short hole drilling (i.e., where maximum hole depth is about $2D$ or less). Speed adjustment factors to increase tool life are found in Table 5b.

Tapping and Threading: The data in this column are intended for use with thread chasers and for tapping. The feed used for tapping and threading must be equal to the lead (feed = lead = pitch) of the thread being cut. The two feed/speed pairs given for each material, therefore, are representative speeds for two thread pitches, 12 and 50 threads per inch ($1/0.083 = 12$, and $1/0.020 = 50$). Tool life is expected to be approximately 45 minutes at the given feeds and speeds. When cutting fewer than 12 threads per inch (pitch ≥ 0.08 inch), use the lower (*optimum*) speed; for cutting more than 50 threads per inch (pitch ≤ 0.02 inch), use the larger (*average*) speed; and, in the intermediate range between 12 and 50 threads per inch, interpolate between the given *average* and *optimum* speeds.

The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: coated carbide = 10.

Example 2, Drilling: If the 1120 steel of Example 1 is to be drilled with a 0.60-inch drill at a feed of 0.012 in./rev, what is the cutting speed in ft/min? Also, what spindle rpm of the drilling machine is required to obtain this cutting speed?

To find the feed factor F_d in Table 22, calculate the ratio of the desired feed to the *optimum* feed and the ratio of the two cutting speeds given in the speed tables. The desired feed is 0.012 in./rev and the *optimum* feed, as explained above is 0.016 in./rev, therefore, $\text{feed}/f_{opt} = 0.012/0.016 = 0.75$ and $V_{avg}/V_{opt} = 95/50 = 1.9$, approximately 2.

The feed factor F_f is found at the intersection of the feed ratio row and the speed ratio column. $F_f = 1.40$ corresponds to about halfway between 1.31 and 1.50, which are the feed factors that correspond to $V_{avg}/V_{opt} = 2.0$ and feed/f_{opt} ratios of 0.7 and 0.8, respectively. F_d, the diameter factor, is found on the same row as the feed factor (halfway between the 0.7 and 0.8 rows, for this example) under the column for drill diameter = 0.60 inch. Because the speed table values are based on a 0.60-inch drill diameter, $F_d = 1.0$ for this example, and the cutting speed is $V = V_{opt} \times F_f \times F_d = 50 \times 1.4 \times 1.0 = 70$ ft/min. The spindle speed in rpm is $N = 12 \times V/(\pi \times D) = 12 \times 70/(3.14 \times 0.6) = 445$ rpm.

Example 3, Drilling: Using the same material and feed as in the previous example, what cutting speeds are required for 0.079-inch and 4-inch diameter drills? What machine rpm is required for each?

Because the feed is the same as in the previous example, the feed factor is $F_f = 1.40$ and does not need to be recalculated. The diameter factors are found in Table 22 on the same row as the feed factor for the previous example (about halfway between the diameter factors corresponding to feed/f_{opt} values of 0.7 and 0.8) in the column corresponding to drill diameters 0.079 and 4.0 inches, respectively. Results of the calculations are summarized below.

Drill diameter = 0.079 inch	*Drill diameter = 4.0 inches*
$F_f = 1.40$	$F_f = 1.40$
$F_d = (0.34 + 0.38)/2 = 0.36$	$F_d = (1.95 + 1.73)/2 = 1.85$
$V = 50 \times 1.4 \times 0.36 = 25.2$ fpm	$V = 50 \times 1.4 \times 1.85 = 129.5$ fpm
$12 \times 25.2/(3.14 \times 0.079) = 1219$ rpm	$12 \times 129.5/(3.14 \times 4) = 124$ rpm

Drilling Difficulties: A drill split at the web is evidence of too much feed or insufficient lip clearance at the center due to improper grinding. Rapid wearing away of the extreme outer corners of the cutting edges indicates that the speed is too high. A drill chipping or breaking out at the cutting edges indicates that either the feed is too heavy or the drill has been ground with too much lip clearance. Nothing will “check” a high-speed steel drill quicker than to turn a stream of cold water on it after it has been heated while in use. It is equally bad to plunge it in cold water after the point has been heated in grinding. The small checks or cracks resulting from this practice will eventually chip out and cause rapid wear or breakage. Insufficient speed in drilling small holes with hand feed greatly increases the risk of breakage, especially at the moment the drill is breaking through the farther side of the work, due to the operator's inability to gage the feed when the drill is running too slowly.

Small drills have heavier webs and smaller flutes in proportion to their size than do larger drills, so breakage due to clogging of chips in the flutes is more likely to occur. When drilling holes deeper than three times the diameter of the drill, it is advisable to withdraw the drill (peck feed) at intervals to remove the chips and permit coolant to reach the tip of the drill.

Drilling Holes in Glass: The simplest method of drilling holes in glass is to use a standard, tungsten-carbide-tipped masonry drill of the appropriate diameter, in a gun-drill. The edges of the carbide in contact with the glass should be sharp. Kerosene or other liquid may be used as a lubricant, and a light force is maintained on the drill until just before the point breaks through. The hole should then be started from the other side if possible, or a very light force applied for the remainder of the operation, to prevent excessive breaking of material from the sides of the hole. As the hard particles of glass are abraded, they accumulate and act to abrade the hole, so it may be advisable to use a slightly smaller drill than the required diameter of the finished hole.

Alternatively, for holes of medium and large size, use brass or copper tubing, having an outside diameter equal to the size of hole required. Revolve the tube at a peripheral speed of about 100 feet per minute, and use carborundum (80 to 100 grit) and light machine oil between the end of the pipe and the glass. Insert the abrasive under the drill with a thin piece of soft wood, to avoid scratching the glass. The glass should be supported by a felt or rubber cushion, not much larger than the hole to be drilled. If practicable, it is advisable to drill about halfway through, then turn the glass over, and drill down to meet the first cut. Any fin that may be left in the hole can be removed with a round second-cut file wetted with turpentine.

Smaller-diameter holes may also be drilled with triangular-shaped cemented carbide drills that can be purchased in standard sizes. The end of the drill is shaped into a long tapering triangular point. The other end of the cemented carbide bit is brazed onto a steel shank. A glass drill can be made to the same shape from hardened drill rod or an old three-cornered file. The location at which the hole is to be drilled is marked on the workpiece. A dam of putty or glazing compound is built up on the work surface to contain the cutting fluid, which can be either kerosene or turpentine mixed with camphor. Chipping on the back edge of the hole can be prevented by placing a scrap plate of glass behind the area to be drilled and drilling into the backup glass. This procedure also provides additional support to the workpiece and is essential for drilling very thin plates. The hole is usually drilled with an electric hand drill. When the hole is being produced, the drill should be given a small circular motion using the point as a fulcrum, thereby providing a clearance for the drill in the hole.

Very small round or intricately shaped holes and narrow slots can be cut in glass by the ultrasonic machining process or by the abrasive jet cutting process.

Table 18. Feeds and Speeds for Drilling, Reaming, and Threading Tool Steels

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Uncoated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Water hardening: W1, W2, W5	150–200	85	55	**f** **s**	15 45	7 85	8 360	4 605	24 90	12 95	83 75	20 95
Shock resisting: S1, S2, S5, S6, S7	175–225	50	35									
Cold work (oil hardening): O1, O2, O6, O7	175–225	45	30									
(High carbon, high chromium): D2, D3, D4, D5, D7	200–250	30	20									
(Air hardening): A2, A3, A8, A9, A10	200–250	50	35									
A4, A6	200–250	45	30									
A7	225–275	30	20									
Hot work (chromium type): H10, H11, H12, H13, H14, H19	150–200	60	40									
	200–250	50	30									
	325–375	30	20	**f** **s**			8 270	4 450				
(Tungsten type): H21, H22, H23, H24, H25, H26	150–200	55	35	**f** **s**	15 45	7 85	8 360	4 605	24 90	12 95	83 75	20 95
	200–250	40	25									
(Molybdenum type): H41, H42, H43	150–200	45	30									
	200–250	35	20									
Special-purpose, low alloy: L2, L3, L6	150–200	60	40									
Mold steel: P2, P3, P4, P5, P6	100–150	75	50									
P20, P21	150–200	60	40									
High-speed steel: M1, M2, M6, M10, T1, T2, T6	200–250	45	30									
M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8	225–275	35	20									
T15, M3-2	225–275	25	15									

See the footnote to Table 17 for instructions concerning the use of this table. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: coated carbide = 10.

Table 19. Feeds and Speeds for Drilling, Reaming, and Threading Stainless Steels

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Coated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Free-machining stainless steels (Ferritic): 430F, 430FSe	135–185	90	60	**f** **s**	15 25	7 45	8 320	4 540	24 50	12 50	83 40	20 51
(Austenitic): 203EZ, 303, 303Se, 303MA, 303Pb, 303Cu, 303 Plus X	135–185	85	55	**f** **s**	15 20	7 40	8 250	4 425	24 40	12 40	83 35	20 45
	225–275	70	45									
(Martensitic): 416, 416Se, 416 Plus X, 420F, 420FSe, 440F, 440FSe	135–185	90	60									
	185–240	70	45									
	275–325	40	25									
	375–425	20	10									
Stainless steels (Ferritic): 405, 409, 429, 430, 434	135–185	65	45	**f** **s**	15 25	7 45	8 320	4 540	24 50	12 50	83 40	20 51
(Austenitic): 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135–185	55	35	**f** **s**	15 20	7 40	8 250	4 425	24 40	12 40	83 35	20 45
	225–275	50	30									
(Austenitic): 302B, 309, 309S, 310, 310S, 314, 316	135–185	50	30									
(Martensitic): 403, 410, 420, 501	135–175	75	50									
	175–225	65	45									
	275–325	40	25									
	375–425	25	15									
(Martensitic): 414, 431, Greek Ascoloy	225–275	50	30									
	275–325	40	25									
	375–425	25	15									
(Martensitic): 440A, 440B, 440C	225–275	45	30									
	275–325	40	25									
	375–425	20	10									
(Precipitation hardening): 15–5PH, 17–4PH, 17–7PH, AF–71, 17–14CuMo, AFC–77, AM–350, AM–355, AM–362, Custom 455, HNM, PH13–8, PH14–8Mo, PH15–7Mo, Stainless W	150–200	50	30	**f** **s**	15 20	7 40	8 250	4 425	24 40	12 40	83 35	20 45
	275–325	45	25									
	325–375	35	20									
	375–450	20	10									

See the footnote to Table 17 for instructions concerning the use of this table. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: coated carbide = 10.

Table 20. Feeds and Speeds for Drilling, Reaming, and Threading Ferrous Cast Metals

Material	Brinell Hardness	Drilling	Reaming		Drilling						Reaming		Threading	
		HSS			HSS		Indexable Carbide Insert				HSS		HSS	
							Uncoated		Coated					
		Speed (fpm)			f = feed (0.001 in./rev), s = speed (ft/min)									
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
ASTM Class 20	120–150	100	65	**f** **s**	16 80	8 90	11 85	6 180	11 235	6 485	26 85	13 65	83 90	20 80
ASTM Class 25	160–200	90	60											
ASTM Class 30, 35, and 40	190–220	80	55											
ASTM Class 45 and 50	220–260	60	40	**f** **s**	13 50	6 50	11 70	6 150	11 195	6 405	21 50	10 30	83 55	20 45
ASTM Class 55 and 60	250–320	30	20											
ASTM Type 1, 1b, 5 (Ni resist)	100–215	50	30											
ASTM Type 2, 3, 6 (Ni resist)	120–175	40	25											
ASTM Type 2b, 4 (Ni resist)	150–250	30	20											
Malleable Iron														
(Ferritic): 32510, 35018	110–160	110	75	**f** **s**	19 80	10 100	11 85	6 180	11 270	6 555	30 95	16 80	83 100	20 85
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160–200	80	55	**f** **s**	14 65	7 65			11 235	6 485	22 65	11 45	83 70	20 60
	200–240	70	45											
(Martensitic): 53004, 60003, 60004	200–255	55	35											
(Martensitic): 70002, 70003	220–260	50	30											
(Martensitic): 80002	240–280	45	30											
(Martensitic): 90001	250–320	25	15											
Nodular (Ductile) Iron														
(Ferritic): 60-40-18, 65-45-12	140–190	100	65	**f** **s**	17 70	9 80	11 85	6 180	11 235	6 485	28 80	14 60	83 80	20 70

Table 20. *(Continued)* Feeds and Speeds for Drilling, Reaming, and Threading Ferrous Cast Metals

Material	Brinell Hardness	Drilling	Reaming	Drilling							Reaming		Threading	
		HSS			HSS		Indexable Carbide Insert				HSS		HSS	
							Uncoated		Coated					
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)										
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
(Martensitic): 120-90-02	270–330	25	15											
	330–400	10	5											
(Ferritic-Pearlitic): 80-55-06	190–225	70	45	**f** **s**	13 60	6 60	11 70	6 150	11 195	6 405	21 55	11 40	83 60	20 55
	225–260	50	30											
(Pearlitic-Martensitic): 100-70-03	240–300	40	25											
Cast Steels														
(Low carbon): 1010, 1020	100–150	100	65	**f** **s**	18 35	9 70					29 75	15 85	83 65	20 85
(Medium carbon): 1030, 1040, 1050	125–175	90	60	**f** **s**	15 35	7 60			8 195†	4 475	24 65	12 70	83 55	20 70
	175–225	70	45											
	225–300	55	35											
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150–200	75	50											
	200–250	65	40											
	250–300	50	30											
(Medium-carbon alloy): 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175–225	70	45	**f** s					8 130†	4 315				
	225–250	60	35											
	250–300	45	30											
	300–350	30	20											
	350–400	20	10											

See the footnote to Table 17 for instructions concerning the use of this table. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated = 15; coated carbide = 11, † = 10.

Table 21. Feeds and Speeds for Drilling, Reaming, and Threading Light Metals

Material	Brinell Hardness	Drilling	Reaming		Drilling				Reaming		Threading	
		HSS			HSS		Indexable Insert Uncoated Carbide		HSS		HSS	
		Speed (fpm)			**f** = feed (0.001 in./rev), **s** = speed (ft/min)							
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
All wrought aluminum alloys, 6061-T651, 5000, 6000, 7000 series	CD ST and A	400 350	400 350	**f** **s**	31 390	16 580	11 3235	6 11370	52 610	26 615	83 635	20 565
All aluminum sand and permanent mold casting alloys	AC ST and A	500 350	500 350									
Aluminum Die-Casting Alloys												
Alloys 308.0 and 319.0	—	—	—	**f** **s**	23 110	11 145	11 945	6 3325	38 145	19 130	83 145	20 130
Alloys 360.0 and 380.0	—	—	—	**f** **s**	27 90	14 125	11 855	6 3000	45 130	23 125	83 130	20 115
Alloys 390.0 and 392.0 {	AC ST and A	300 70	300 70									
Alloys 413	ST and A	— 45	— 40	**f** **s**	24 65	12 85	11 555	6 1955	40 85	20 80	83 85	20 80
All other aluminum die-casting alloys {	AC	125	100	**f** **s**	27 90	14 125	11 855	6 3000	45 130	23 125	83 130	20 115
Magnesium Alloys												
All wrought magnesium alloys	A,CD,ST and A	500	500									
All cast magnesium alloys	A,AC, ST and A	450	450									

Abbreviations designate: A, annealed; AC, as cast; CD, cold drawn; and ST and A, solution treated and aged, respectively. See the footnote to Table 17 for instructions concerning the use of this table. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows; uncoated carbide = 15.

Table 22. Feed and Diameter Speed Adjustment Factors for HSS Twist Drills and Reamers

	Cutting Speed, $V = V_{opt} \times F_f \times F_d$															
Ratio of Chosen Feed to *Optimum* Feed	Ratio of the two cutting speeds (*average*/*optimum*) given in the tables V_{avg}/V_{opt}							Tool Diameter								
	1.00	1.25	1.50	2.00	2.50	3.00	4.00	0.08 in (2 mm)	0.15 in (4 mm)	0.25 in (6 mm)	0.40 in (10 mm)	0.60 in (15 mm)	1.00 in (25 mm)	2.00 in (50 mm)	3.00 in (75 mm)	4.00 in (100 mm)
	Feed Factor, F_f							Diameter Factor, F_d								
1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	0.30	0.44	0.56	0.78	1.00	1.32	1.81	2.11	2.29
0.90	1.00	1.06	1.09	1.14	1.18	1.21	1.27	0.32	0.46	0.59	0.79	1.00	1.30	1.72	1.97	2.10
0.80	1.00	1.12	1.19	1.31	1.40	1.49	1.63	0.34	0.48	0.61	0.80	1.00	1.27	1.64	1.89	1.95
0.70	1.00	1.15	1.30	1.50	1.69	1.85	2.15	0.38	0.52	0.64	0.82	1.00	1.25	1.52	1.67	1.73
0.60	1.00	1.23	1.40	1.73	2.04	2.34	2.89	0.42	0.55	0.67	0.84	1.00	1.20	1.46	1.51	1.54
0.50	1.00	1.25	1.50	2.00	2.50	3.00	5.00	0.47	0.60	0.71	0.87	1.00	1.15	1.30	1.34	1.94
0.40	1.00	1.23	1.57	2.29	3.08	3.92	5.70	0.53	0.67	0.77	0.90	1.00	1.10	1.17	1.16	1.12
0.30	1.00	1.14	1.56	2.57	3.78	5.19	8.56	0.64	0.76	0.84	0.94	1.00	1.04	1.02	0.96	0.90
0.20	1.00	0.90	1.37	2.68	4.49	6.86	17.60	0.83	0.92	0.96	1.00	1.00	0.96	0.81	0.73	0.66
0.10	1.00	1.44	0.80	2.08	4.36	8.00	20.80	1.29	1.26	1.21	1.11	1.00	0.84	0.60	0.46	0.38

This table is specifically for use with the combined feed/speed data for HSS twist drills in Tables 17 through 23; use Tables 5a and 5b to adjust speed and tool life for indexable insert drilling with carbides. The combined feed/speed data for HSS twist drilling are based on a 0.60-inch diameter HSS drill with standard drill point geometry (2-flute with 118° tip angle). To adjust the given speeds for different feeds and drill diameters, use the two feed/speed pairs from the tables and calculate the ratio of desired (new) feed to *optimum* feed (largest of the two feeds from the speed table), and the ratio of the two cutting speeds $V_{\mathrm{avg}}/V_{\mathrm{opt}}$. Use the values of these ratios to find the feed factor F_f at the intersection of the feed ratio row and the speed ratio column in the left half of the table. The diameter factor F_d is found in the same row as the feed factor, in the right half of the table, under the column corresponding to the drill diameter. For diameters not given, interpolate between the nearest available sizes. The adjusted cutting speed can be calculated from $V = V_{opt} \times F_f \times F_d$, where V_{opt} is the smaller (*optimum*) of the two speeds from the speed table (from the left side of the column containing the two feed/speed pairs). Tool life using the selected feed and the adjusted speed should be approximately 45 minutes. Speed adjustment factors to modify tool life are found in Table 15e.

Table 23. Feeds and Speeds for Drilling and Reaming Copper Alloys

Group 1
Architectural bronze(C38500); Extra-high-leaded brass (C35600); Forging brass (C37700); Free-cutting phosphor bronze (B-2) (C54400); Free-cutting brass (C36000); Free-cutting Muntz metal (C37000); High-leaded brass (C33200, C34200); High-leaded brass tube (C35300); Leaded commercial bronze (C31400); Leaded naval brass (C48500); Medium-leaded brass (C34000)
Group 2
Aluminum brass, arsenical (C68700); Cartridge brass, 70% (C26000); High-silicon bronze, B (C65500); Admiralty brass (inhibited) (C44300, C44500); Jewelry bronze, 87.5% (C22600); Leaded Muntz metal (C36500, C36800); Leaded nickel silver (C79600); Low brass, 80% (C24000); Low-leaded brass (C33500); Low-silicon bronze, B (C65100); Manganese bronze, A (C67500); Muntz metal, 60% (C28000); Nickel silver, 55–18 (C77000); Red brass, 85% (C23000); Yellow brass (C26800)
Group 3
Aluminum bronze, D (C61400); Beryllium copper (C17000, C17200, C17500); Commercial bronze, 90% (C22000); Copper nickel, 10% (C70600); Copper nickel, 30% (C71500);Electrolytic tough-pitch copper (C11000); Gilding, 95% (C21000); Nickel silver, 65–10 (C74500); Nickel silver, 65–12 (C75700); Nickel silver, 65–15 (C75400); Nickel silver, 65–18 (C75200); Oxygen-free copper (C10200); Phosphor bronze, 1.25% (C50200); Phosphor bronze, 10% D (C52400); Phosphor bronze, 5% A (C51000); Phosphor bronze, 8% C (C52100); Phosphorus deoxidized copper (C12200)

Alloy Description and UNS Alloy Numbers	Material Condition	Drilling	Reaming	Drilling					Reaming	
		HSS		HSS			Indexable Insert Uncoated Carbide		HSS	
		Speed (fpm)		**f** = feed (0.001 in./rev), **s** = speed (ft/min)						
					Opt.	*Avg.*	*Opt.*	*Avg.*	*Opt.*	*Avg.*
Wrought Alloys										
Group 1	A	160	160	**f**	21	11	11	6	36	18
	CD	175	175	**s**	210	265	405	915	265	230
Group 2	A	120	110	**f**	24	12	11	6	40	20
	CD	140	120	**s**	100	130	205	455	130	120
Group 3	A	60	50	**f**	23	11	11	6	38	19
	CD	65	60	**s**	155	195	150	340	100	175

Abbreviations designate: A, annealed; CD, cold drawn. The two leftmost speed columns in this table contain traditional Handbook speeds for HSS steel tools. The text contains information concerning feeds to use in conjunction with these speeds.

HSS Drilling and Reaming: The combined feed/speed data for drilling and Table 22 are used to adjust drilling speeds for other feeds and drill diameters. Examples are given in the text. The given feeds and speeds for reaming are based on an 8-tooth, $^{25}/_{32}$-inch diameter, 30° lead angle reamer, and a 0.008-inch radial depth of cut. For other feeds, the correct speed can be obtained by interpolation using the given speeds if the desired feed lies in the recommended range (between the given values of *optimum* and *average* feed). The cutting speeds for reaming do not require adjustment for tool diameter as long as the radial depth of cut does not become too large. Speed adjustment factors to modify tool life are found in Table 15e.

Indexable Insert Drilling: The feed/speed data for indexable insert drilling are based on a tool with two cutting edges, an insert nose radius of $^{3}/_{64}$ inch, a 10-degree lead angle, and diameter *D* of 1 inch. Adjustments for feed and depth of cut are made using Table 5a (Turning Speed Adjustment Factors) using a depth of cut of D/2, or one-half the drill diameter. Expected tool life at the given feeds and speeds is 15 minutes for short hole drilling (i.e., where hole depth is about *2D* or less). Speed adjustment factors to increase tool life are found in Table 5b. The combined feed/speed data in this table are based on tool grades (identified in Table 16) as follows: uncoated carbide = 15.

Using the Feed and Speed Tables for Tapping and Threading.—The feed used in tapping and threading is always equal to the pitch of the screw thread being formed. The threading data contained in the tables for drilling, reaming, and threading (Tables 17 through 23) are primarily for tapping and thread chasing, and do not apply to thread cutting with single-point tools.

The threading data in Tables 17 through 23 give two sets of feed (pitch) and speed values, for 12 and 50 threads/inch, but these values can be used to obtain the cutting speed for any other thread pitches. If the desired pitch falls between the values given in the tables, i.e., between 0.020 inch (50 tpi) and 0.083 inch (12 tpi), the required cutting speed is obtained by interpolation between the given speeds. If the pitch is less than 0.020 inch (more than 50 tpi), use the *average* speed, i.e., the largest of the two given speeds. For pitches greater than 0.083 inch (fewer than 12 tpi), the *optimum* speed should be used. Tool life using the given feed/speed data is intended to be approximately 45 minutes, and should be about the same for threads between 12 and 50 threads per inch.

Example: Determine the cutting speed required for tapping 303 stainless steel with a $\frac{1}{2}$–20 coated HSS tap.

The two feed/speed pairs for 303 stainless steel, in Table 19, are 83⁄35 (0.083 in./rev at 35 fpm) and 20⁄45 (0.020 in./rev at 45 fpm). The pitch of a $\frac{1}{2}$–20 thread is 1⁄20 = 0.05 inch, so the required feed is 0.05 in./rev. Because 0.05 is between the two given feeds (Table 19), the cutting speed can be obtained by interpolation between the two given speeds as follows:

$$V = 35 + \frac{0.05 - 0.02}{0.083 - 0.02}(45 - 35) = 40 \text{ fpm}$$

The cutting speed for coarse-pitch taps must be lower than for fine-pitch taps with the same diameter. Usually, the difference in pitch becomes more pronounced as the diameter of the tap becomes larger and slight differences in the pitch of smaller-diameter taps have little significant effect on the cutting speed. Unlike all other cutting tools, the feed per revolution of a tap cannot be independently adjusted—it is always equal to the lead of the thread and is always greater for coarse pitches than for fine pitches. Furthermore, the thread form of a coarse-pitch thread is larger than that of a fine-pitch thread; therefore, it is necessary to remove more metal when cutting a coarse-pitch thread.

Taps with a long chamfer, such as starting or tapper taps, can cut faster in a short hole than short chamfer taps, such as plug taps. In deep holes, however, short chamfer or plug taps can run faster than long chamfer taps. Bottoming taps must be run more slowly than either starting or plug taps. The chamfer helps to start the tap in the hole. It also functions to involve more threads, or thread form cutting edges, on the tap in cutting the thread in the hole, thus reducing the cutting load on any one set of thread form cutting edges. In so doing, more chips and thinner chips are produced that are difficult to remove from deeper holes. Shortening the chamfer length causes fewer thread form cutting edges to cut, thereby producing fewer and thicker chips that can easily be disposed of. Only one or two sets of thread form cutting edges are cut on bottoming taps, causing these cutting edges to assume a heavy cutting load and produce very thick chips.

Spiral-pointed taps can operate at a faster cutting speed than taps with normal flutes. These taps are made with supplementary angular flutes on the end that push the chips ahead of the tap and prevent the tapped hole from becoming clogged with chips. They are used primarily to tap open or through holes although some are made with shorter supplementary flutes for tapping blind holes.

The tapping speed must be reduced as the percentage of full thread to be cut is increased. Experiments have shown that the torque required to cut a 100 per cent thread form is more than twice that required to cut a 50 per cent thread form. An increase in the percentage of full thread will also produce a greater volume of chips.

The tapping speed must be lowered as the length of the hole to be tapped is increased. More friction must be overcome in turning the tap and more chips accumulate in the hole.

It will be more difficult to apply the cutting fluid at the cutting edges and to lubricate the tap to reduce friction. This problem becomes greater when the hole is being tapped in a horizontal position.

Cutting fluids have a very great effect on the cutting speed for tapping. Although other operating conditions when tapping frequently cannot be changed, a free selection of the cutting fluid usually can be made. When planning the tapping operation, the selection of a cutting fluid warrants a very careful consideration and perhaps an investigation.

Taper threaded taps, such as pipe taps, must be operated at a slower speed than straight thread taps with a comparable diameter. All the thread form cutting edges of a taper threaded tap that are engaged in the work cut and produce a chip, but only those cutting edges along the chamfer length cut on straight thread taps. Pipe taps often are required to cut the tapered thread from a straight hole, adding to the cutting burden.

The machine tool used for the tapping operation must be considered in selecting the tapping speed. Tapping machines and other machines that are able to feed the tap at a rate of advance equal to the lead of the tap, and that have provisions for quickly reversing the spindle, can be operated at high cutting speeds. On machines where the feed of the tap is controlled manually—such as on drill presses and turret lathes—the tapping speed must be reduced to allow the operator to maintain safe control of the operation.

There are other special considerations in selecting the tapping speed. Very accurate threads are usually tapped more slowly than threads with a commercial grade of accuracy. Thread forms that require deep threads for which a large amount of metal must be removed, producing a large volume of chips, require special techniques and slower cutting speeds. Acme, buttress, and square threads, therefore, are generally cut at lower speeds.

Cutting Speed for Broaching.—Broaching offers many advantages in manufacturing metal parts, including high production rates, excellent surface finishes, and close dimensional tolerances. These advantages are not derived from the use of high cutting speeds; they are derived from the large number of cutting teeth that can be applied consecutively in a given period of time, from their configuration and precise dimensions, and from the width or diameter of the surface that can be machined in a single stroke. Most broaching cutters are expensive in their initial cost and are expensive to sharpen. For these reasons, a long tool life is desirable, and to obtain a long tool life, relatively slow cutting speeds are used. In many instances, slower cutting speeds are used because of the limitations of the machine in accelerating and stopping heavy broaching cutters. At other times, the available power on the machine places a limit on the cutting speed that can be used; i.e., the cubic inches of metal removed per minute must be within the power capacity of the machine.

The cutting speeds for high-speed steel broaches range from 3 to 50 feet per minute, although faster speeds have been used. In general, the harder and more difficult to machine materials are cut at a slower cutting speed and those that are easier to machine are cut at a faster speed. Some typical recommendations for high-speed steel broaches are: AISI 1040, 10 to 30 fpm; AISI 1060, 10 to 25 fpm; AISI 4140, 10 to 25 fpm; AISI 41L40, 20 to 30 fpm; 201 austenitic stainless steel, 10 to 20 fpm; Class 20 gray cast iron, 20 to 30 fpm; Class 40 gray cast iron, 15 to 25 fpm; aluminum and magnesium alloys, 30 to 50 fpm; copper alloys, 20 to 30 fpm; commercially pure titanium, 20 to 25 fpm; alpha and beta titanium alloys, 5 fpm; and the superalloys, 3 to 10 fpm. Surface broaching operations on gray iron castings have been conducted at a cutting speed of 150 fpm, using indexable insert cemented carbide broaching cutters. In selecting the speed for broaching, the cardinal principle of the performance of all metal cutting tools should be kept in mind; i.e., increasing the cutting speed may result in a proportionately larger reduction in tool life, and reducing the cutting speed may result in a proportionately larger increase in the tool life. When broaching most materials, a suitable cutting fluid should be used to obtain a good surface finish and a better tool life. Gray cast iron can be broached without using a cutting fluid although some shops prefer to use a soluble oil.

ESTIMATING SPEEDS AND MACHINING POWER

Estimating Planer Cutting Speeds.—Whereas most planers of modern design have a means of indicating the speed at which the table is traveling, or cutting, many older planers do not. Thus, the following formulas are useful for planers that do not have a means of indicating the table or cutting speed. It is not practicable to provide a formula for calculating the exact cutting speed at which a planer is operating because the time to stop and start the table when reversing varies greatly. The formulas below will, however, provide a reasonable estimate.

$$V_c \cong S_c L$$

$$S_c \cong \frac{V_c}{L}$$

where V_c = cutting speed; fpm or m/min
S_c = number of cutting strokes per minute of planer table
L = length of table cutting stroke; ft or m

Cutting Speed for Planing and Shaping.—The traditional HSS cutting tool speeds in Tables 1 through 4b and Tables 6 through 9 can be used for planing and shaping. The feed and depth of cut factors in Tables 5c should also be used, as explained previously. Very often, other factors relating to the machine or the setup will require a reduction in the cutting speed used on a specific job.

Cutting Time for Turning, Boring, and Facing.—The time required to turn a length of metal can be determined by the following formula in which T = time in minutes, L = length of cut in inches, f = feed in inches per revolution, and N = lathe spindle speed in revolutions per minute.

$$T = \frac{L}{fN}$$

When making job estimates, the time required to load and to unload the workpiece on the machine, and the machine handling time, must be added to the cutting time for each length cut to obtain the floor-to-floor time.

Planing Time.—The approximate time required to plane a surface can be determined from the following formula in which T = time in minutes, L = length of stroke in feet, V_c = cutting speed in feet per minute, V_r = return speed in feet per minute; W = width of surface to be planed in inches, F = feed in inches, and 0.025 = approximate reversal time factor per stroke in minutes for most planers:

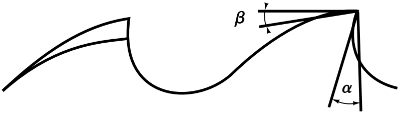
$$T = \frac{W}{F}\left[L \times \left(\frac{1}{V_c} + \frac{1}{V_r}\right) + 0.025\right]$$

Speeds for Metal-Cutting Saws.—The following speeds and feeds for solid-tooth, high-speed-steel, circular, metal-cutting saws are recommended by Saws International, Inc. (sfpm = surface feet per minute = 3.142 × blade diameter in inches × rpm of saw shaft ÷ 12).

Speeds for Turning Unusual Materials.—*Slate*, on account of its peculiarly stratified formation, is rather difficult to turn, but if handled carefully, can be machined in an ordinary lathe. The cutting speed should be about the same as for cast iron. A sheet of fiber or pressed paper should be interposed between the chuck or steadyrest jaws and the slate, to protect the latter. Slate rolls must not be centered and run on the tailstock. A satisfactory method of supporting a slate roll having journals at the ends is to bore a piece of lignum vitae to receive the turned end of the roll, and center it for the tailstock spindle.

Rubber can be turned at a peripheral speed of 200 feet per minute, although it is much easier to grind it with an abrasive wheel that is porous and soft. For cutting a rubber roll in

Speeds, Feeds, and Tooth Angles for Sawing Various Materials



α = Cutting angle
β = Relief angle

Materials	Front Rake Angle α (deg)	Back Rake Angle β (deg)	Stock Diameters (inches)			
			¼–¾	¾–1½	1½–2½	2½–3½
Aluminum	24	12	6500 sfpm 100 in./min	6200 sfpm 85 in./min	6000 sfpm 80 in./min	5000 sfpm 75 in./min
Light Alloys with Cu, Mg, and Zn	22	10	3600 sfpm 70 in./min	3300 sfpm 65 in./min	3000 sfpm 63 in./min	2600 sfpm 60 in./min
Light Alloys with High Si	20	8	650 sfpm 16 in./min	600 sfpm 16 in./min	550 sfpm 14 in./min	550 sfpm 12 in./min
Copper	20	10	1300 sfpm 24 in./min	1150 sfpm 24 in./min	1000 sfpm 22 in./min	800 sfpm 22 in./min
Bronze	15	8	1300 sfpm 24 in./min	1150 sfpm 24 in./min	1000 sfpm 22 in./min	800 sfpm 20 in./min
Hard Bronze	10	8	400 sfpm 6.3 in./min	360 sfpm 6 in./min	325 sfpm 5.5 in./min	300 sfpm 5.1 in./min
Cu-Zn Brass	16	8	2000 sfpm 43 in./min	2000 sfpm 43 in./min	1800 sfpm 39 in./min	1800 sfpm 35 in./min
Gray Cast Iron	12	8	82 sfpm 4 in./min	75 sfpm 4 in./min	72 sfpm 3.5 in./min	66 sfpm 3 in./min
Carbon Steel	20	8	160 sfpm 6.3 in./min	150 sfpm 5.9 in./min	150 sfpm 5.5 in./min	130 sfpm 5.1 in./min
Medium Hard Steel	18	8	100 sfpm 5.1 in./min	100 sfpm 4.7 in./min	80 sfpm 4.3 in./min	80 sfpm 4.3 in./min
Hard Steel	15	8	66 sfpm 4.3 in./min	66 sfpm 4.3 in./min	60 sfpm 4 in./min	57 sfpm 3.5 in./min
Stainless Steel	15	8	66 sfpm 2 in./min	63 sfpm 1.75 in./min	60 sfpm 1.75 in./min	57 sfpm 1.5 in./min

two, the ordinary parting tool should not be used, but a tool shaped like a knife; such a tool severs the rubber without removing any material.

Gutta percha can be turned as easily as wood, but the tools must be sharp and a good soap-and-water lubricant used.

Copper can be turned easily at 200 feet per minute.

Limestone such as is used in the construction of pillars for balconies, etc., can be turned at 150 feet per minute, and the formation of ornamental contours is quite easy. *Marble* is a treacherous material to turn. It should be cut with a tool such as would be used for brass, but at a speed suitable for cast iron. It must be handled very carefully to prevent flaws in the surface.

The foregoing speeds are for high-speed steel tools. Tools tipped with tungsten carbide are adapted for cutting various non-metallic products which cannot be machined readily with steel tools, such as slate, marble, synthetic plastic materials, etc. In drilling slate and marble, use flat drills; and for plastic materials, tungsten-carbide-tipped twist drills. Cutting speeds ranging from 75 to 150 feet per minute have been used for drilling slate (without coolant) and a feed of 0.025 inch per revolution for drills $\frac{3}{4}$ and 1 inch in diameter.

Estimating Machining Power.—Knowledge of the power required to perform machining operations is useful when planning new machining operations, for optimizing existing machining operations, and to develop specifications for new machine tools that are to be acquired. The available power on any machine tool places a limit on the size of the cut that it can take. When much metal must be removed from the workpiece it is advisable to estimate the cutting conditions that will utilize the maximum power on the machine. Many machining operations require only light cuts to be taken for which the machine obviously has ample power; in this event, estimating the power required is a wasteful effort. Conditions in different shops may vary and machine tools are not all designed alike, so some variations between the estimated results and those obtained on the job are to be expected. However, by using the methods provided in this section a reasonable estimate of the power required can be made, which will suffice in most practical situations.

The measure of power in customary inch units is the horsepower; in SI metric units it is the kilowatt, which is used for both mechanical and electrical power. The power required to cut a material depends upon the rate at which the material is being cut and upon an experimentally determined power constant, K_p, which is also called the unit horsepower, unit power, or specific power consumption. The power constant is equal to the horsepower required to cut a material at a rate of one cubic inch per minute; in SI metric units the power constant is equal to the power in kilowatts required to cut a material at a rate of one cubic centimeter per second, or 1000 cubic millimeters per second ($1\ \text{cm}^3 = 1000\ \text{mm}^3$). Different values of the power constant are required for inch and for metric units, which are related as follows: to obtain the SI metric power constant, multiply the inch power constant by 2.73; to obtain the inch power constant, divide the SI metric power constant by 2.73. Values of the power constant in Tables 24, 30, and 25 can be used for all machining operations except drilling and grinding. Values given are for sharp tools.

Table 24. Power Constants, K_p, for Ferrous Cast Metals, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units	Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units
Gray Cast Iron	100–120	0.28	0.76	Malleable Iron			
	120–140	0.35	0.96	Ferritic	150–175	0.42	1.15
	140–160	0.38	1.04	Pearlitic	175–200	0.57	1.56
	160–180	0.52	1.42		200–250	0.82	2.24
	180–200	0.60	1.64		250–300	1.18	3.22
	200–220	0.71	1.94				
	220–240	0.91	2.48	Cast Steel	150–175	0.62	1.69
					175–200	0.78	2.13
Alloy Cast Iron	150–175	0.30	0.82		200–250	0.86	2.35
	175–200	0.63	1.72	…	…	…	…
	200–250	0.92	2.51	…	…	…	…

The value of the power constant is essentially unaffected by the cutting speed, the depth of cut, and the cutting tool material. Factors that do affect the value of the power constant, and thereby the power required to cut a material, include the hardness and microstructure of the work material, the feed rate, the rake angle of the cutting tool, and whether the cutting edge of the tool is sharp or dull. Values are given in the power constant tables for different material hardness levels, whenever this information is available. Feed factors for the power constant are given in Table 25. All metal cutting tools wear but a worn cutting edge requires more power to cut than a sharp cutting edge.

Factors to provide for tool wear are given in Table 26. In this table, the extra-heavy-duty category for milling and turning occurs only on operations where the tool is allowed to wear more than a normal amount before it is replaced, such as roll turning. The effect of the rake angle usually can be disregarded. The rake angle for which most of the data in the power constant tables are given is positive 14 degrees. Only when the deviation from this angle is large is it necessary to make an adjustment. Using a rake angle that is more positive reduces the power required approximately 1 per cent per degree; using a rake angle that is more negative increases the power required; again approximately 1 per cent per degree.

Many indexable insert cutting tools are formed with an integral chip breaker or other cutting edge modifications, which have the effect of reducing the power required to cut a material. The extent of this effect cannot be predicted without a test of each design. Cutting fluids will also usually reduce the power required, when operating in the lower range of cutting speeds. Again, the extent of this effect cannot be predicted because each cutting fluid exhibits its own characteristics.

Table 25. Feed Factors, *C*, for Power Constants

Inch Units				SI Metric Units			
Feed in.[a]	*C*	Feed in.[a]	*C*	Feed mm[b]	*C*	Feed mm[b]	*C*
0.001	1.60	0.014	0.97	0.02	1.70	0.35	0.97
0.002	1.40	0.015	0.96	0.05	1.40	0.38	0.95
0.003	1.30	0.016	0.94	0.07	1.30	0.40	0.94
0.004	1.25	0.018	0.92	0.10	1.25	0.45	0.92
0.005	1.19	0.020	0.90	0.12	1.20	0.50	0.90
0.006	1.15	0.022	0.88	0.15	1.15	0.55	0.88
0.007	1.11	0.025	0.86	0.18	1.11	0.60	0.87
0.008	1.08	0.028	0.84	0.20	1.08	0.70	0.84
0.009	1.06	0.030	0.83	0.22	1.06	0.75	0.83
0.010	1.04	0.032	0.82	0.25	1.04	0.80	0.82
0.011	1.02	0.035	0.80	0.28	1.01	0.90	0.80
0.012	1.00	0.040	0.78	0.30	1.00	1.00	0.78
0.013	0.98	0.060	0.72	0.33	0.98	1.50	0.72

[a] Turning—in./rev; milling—in./tooth: planing and shaping—in./stroke; broaching—in./tooth.

[b] Turning—mm/rev; milling—mm/tooth: planing and shaping—mm/stroke; broaching—mm/tooth.

Table 26. Tool Wear Factors, *W*

Type of Operation		*W*
For all operations with sharp cutting tools		1.00
Turning:	Finish turning (light cuts)	1.10
	Normal rough and semifinish turning	1.30
	Extra-heavy-duty rough turning	1.60–2.00
Milling:	Slab milling	1.10
	End milling	1.10
	Light and medium face milling	1.10–1.25
	Extra-heavy-duty face milling	1.30–1.60
Drilling:	Normal drilling	1.30
	Drilling hard-to-machine materials and drilling with a very dull drill	1.50
Broaching:	Normal broaching	1.05–1.10
	Heavy-duty surface broaching	1.20–1.30

For planing and shaping, use values given for turning.

The machine tool transmits the power from the driving motor to the workpiece, where it is used to cut the material. The effectiveness of this transmission is measured by the machine tool efficiency factor, *E*. Average values of this factor are given in Table 28. Formulas for calculating the metal removal rate, *Q*, for different machining operations are given in Table 29. These formulas are used together with others given below. The following formulas can be used with either customary inch or with SI metric units.

$$P_c = K_p CQW \tag{1}$$

$$P_m = \frac{P_c}{E} = \frac{K_p CQW}{E} \tag{2}$$

where P_c = power at the cutting tool; hp, or kW
P_m = power at the motor; hp, or kW
K_p = power constant (see Tables 24, 30, and 25)
Q = metal removal rate; in. ^{3}/min. or cm^3/s (see Table 29)
C = feed factor for power constant (see Table 25)
W = tool wear factor (see Table 26)
E = machine tool efficiency factor (see Table 28)
V = cutting speed, fpm, or m/min
N = cutting speed, rpm
f = feed rate for turning; in./rev. or mm/rev
f = feed rate for planing and shaping; in./stroke, or mm/stroke
f_t = feed per tooth; in./tooth, or mm/tooth
f_m = feed rate; in./min. or mm/min
d_t = maximum depth of cut per tooth: in., or mm
d = depth of cut; in., or mm
n_t = number of teeth on milling cutter

Table 27. Power Constant, K_p, for High-Temperature Alloys, Tool Steel, Stainless Steel, and Nonferrous Metals, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p Metric Units	Material	Brinell Hardness Number	K_p Inch Units	K_p Metric Units
High-Temperature Alloys				Stainless Steel	150-175	0.60	1.64
A286	165	0.82	2.24		175-200	0.72	1.97
A286	285	0.93	2.54		200-250	0.88	2.40
Chromoloy	200	0.78	3.22	Zinc Die Cast Alloys	…	0.25	0.68
Chromoloy	310	1.18	3.00	Copper (pure)	…	0.91	2.48
Inco 700	330	1.12	3.06	Brass			
Inco 702	230	1.10	3.00	Hard	…	0.83	2.27
Hastelloy-B	230	1.10	3.00	Medium	…	0.50	1.36
M-252	230	1.10	3.00	Soft	…	0.25	0.68
M-252	310	1.20	3.28	Leaded	…	0.30	0.82
Ti-150A	340	0.65	1.77				
U-500	375	1.10	3.00	Bronze			
				Hard	…	0.91	2.48
Monel Metal	…	1.00	2.73	Medium	…	0.50	1.36
Tool Steel	175-200	0.75	2.05	Aluminum			
	200-250	0.88	2.40	Cast	…	0.25	0.68
	250-300	0.98	2.68	Rolled (hard)	…	0.33	0.90
	300-350	1.20	3.28				
	350-400	1.30	3.55	Magnesium Alloys	…	0.10	0.27

n_c = number of teeth engaged in work

w = width of cut; in., or mm

Table 28. Machine Tool Efficiency Factors, E

Type of Drive	E	Type of Drive	E
Direct Belt Drive	0.90	Geared Head Drive	0.70–0.80
Back Gear Drive	0.75	Oil-Hydraulic Drive	0.60–0.90

Example: A 180–200 Bhn AISI shaft is to be turned on a geared head lathe using a cutting speed of 350 fpm (107 m/min), a feed rate of 0.016 in./rev (0.40 mm/rev), and a depth of cut of 0.100 inch (2.54 mm). Estimate the power at the cutting tool and at the motor, using both the inch and metric data.

Inch units:

$K_p = 0.62$ (from Table 30)

$C = 0.94$ (from Table 25)

$W = 1.30$ (from Table 26)

$E = 0.80$ (from Table 28)

$Q = 12\ Vfd = 12 \times 350 \times 0.016 \times 0.100$ (from Table 29)

$Q = 6.72\ \text{in.}^3/\text{min}$

Table 29. Formulas for Calculating the Metal Removal Rate, *Q*

Operation	Metal Removal Rate For Inch Units Only Q = in.^{3}/min	Metal Removal Rate For SI Metric Units Only Q = cm^3/s
Single-Point Tools (Turning, Planing, and Shaping)	$12Vfd$	$\frac{V}{60}fd$
Milling	$f_m wd$	$\frac{f_m wd}{60,000}$
Surface Broaching	$12Vwn_c d_t$	$\frac{V}{60}un_c d_t$

$$P_c = K_p CQW = 0.62 \times 0.94 \times 6.72 \times 1.30 = 5 \text{ hp}$$

$$P_m = \frac{P_c}{E} = \frac{5}{0.80} = 6.25 \text{ hp}$$

SI metric units:

$K_p = 1.60$ (from Table 24)
$C = 0.94$ (from Table 25)
$W = 1.30$ (from Table 26)
$E = 0.80$ (from Table 30)

$$Q = \frac{V}{60}fd = \frac{107}{60} \times 0.40 \times 2.54 \text{ (from Table 29)}$$
$$= 1.81 \text{ cm}^3\text{/s}$$

$$P_c = K_p CQW = 1.69 \times 0.94 \times 1.81 \times 1.30 = 3.74 \text{ kW}$$

$$P_m = \frac{P_c}{E} = \frac{3.74}{0.80} = 4.675 \text{ kW}$$

Whenever possible the maximum power available on a machine tool should be used when heavy cuts must be taken.

The cutting conditions for utilizing the maximum power should be selected in the following order: 1) select the maximum depth of cut that can be used; 2) select the maximum feed rate that can be used; and 3) estimate the cutting speed that will utilize the maximum power available on the machine. This sequence is based on obtaining the longest tool life of the cutting tool and at the same time obtaining as much production as possible from the machine.

The life of a cutting tool is most affected by the cutting speed, then by the feed rate, and least of all by the depth of cut. The maximum metal removal rate that a given machine is capable of machining from a given material is used as the basis for estimating the cutting speed that will utilize all the power available on the machine.

Example: A 0.125 inch deep cut is to be taken on a 200–210 Bhn AISI 1050 steel part using a 10 hp geared head lathe. The feed rate selected for this job is 018 in./rev. Estimate the cutting speed that will utilize the maximum power available on the lathe.

$K_p = 0.85$ (From Table 30)
$C = 0.92$ (From Table 25)

$W = 1.30$ (From Table 26)

$E = 0.80$ (From Table 28)

$$Q_{max} = \frac{P_m E}{K_p C W} = \frac{10 \times 0.80}{0.85 \times 0.92 \times 1.30} \qquad \left(P_m = \frac{K_p C Q W}{E}\right)$$

$$= 7.87 \text{ in.}^3\text{/min}$$

$$V = \frac{Q_{max}}{12 f d} = \frac{7.87}{12 \times 0.018 \times 0.125} \qquad (Q = 12 V f d)$$

$$= 290 \text{ fpm}$$

Example: A 160-180 Bhn gray iron casting that is 6 inches wide is to have ⅛ inch stock removed on a 10 hp milling machine, using an 8 inch diameter, 10 tooth, indexable insert cemented carbide face milling cutter. The feed rate selected for this cutter is 0.012 in./tooth, and all the stock (0.125 in.) will be removed in one cut. Estimate the cutting speed that will utilize the maximum power available on the machine.

$K_p = 0.52$ (From Table 30)

$C = 1.00$ (From Table 25)

$W = 1.20$ (From Table 26)

$E = 0.80$ (From Table 27)

$$Q_{max} = \frac{P_m E}{K_p C W} = \frac{10 \times 0.80}{0.52 \times 1.00 \times 1.20} = 12.82 \text{ in.}^3\text{/min} \qquad \left(P_m = \frac{K_p C Q W}{E}\right)$$

$$f_m = \frac{Q_{max}}{w d} = \frac{12.82}{6 \times 0.125} = 17 \text{ in./min} \qquad (Q = f_m w d)$$

$$N = \frac{f_{max}}{f_t n_t} = \frac{17}{0.012 \times 10} = 140 \text{ rpm} \qquad (f_m = f_t n_t N)$$

$$V = \frac{\pi D N}{12} = \frac{\pi \times 8 \times 140}{12} = 293 \text{ fpm} \qquad \left(N = \frac{12 V}{\pi D}\right)$$

Estimating Drilling Thrust, Torque, and Power.—Although the lips of a drill cut metal and produce a chip in the same manner as the cutting edges of other metal cutting tools, the chisel edge removes the metal by means of a very complex combination of extrusion and cutting. For this reason a separate method must be used to estimate the power required for drilling. Also, it is often desirable to know the magnitude of the thrust and the torque required to drill a hole. The formulas and tabular data provided in this section are based on information supplied by the National Twist Drill Division of Regal-Beloit Corp. The values in Tables 31 through 34 are for sharp drills and the tool wear factors are given in Table 26. For most ordinary drilling operations 1.30 can be used as the tool wear factor. When drilling most difficult-to-machine materials and when the drill is allowed to become very dull, 1.50 should be used as the value of this factor. It is usually more convenient to measure the web thickness at the drill point than the length of the chisel edge; for this reason, the approximate w/d ratio corresponding to each c/d ratio for a correctly ground drill is provided in Table 32. For most standard twist drills the c/d ratio is 0.18, unless the drill has been ground short or the web has been thinned. The c/d ratio of split point drills is 0.03. The formulas given below can be used for spade drills, as well as for twist drills. Separate formulas are required for use with customary inch units and for SI metric units.

Table 30. Power Constants, K_p, for Wrought Steels, Using Sharp Cutting Tools

Material	Brinell Hardness Number	K_p Inch Units	K_p SI Metric Units
Plain Carbon Steels			
All Plain Carbon Steels	80–100	0.63	1.72
	100–120	0.66	1.80
	120–140	0.69	1.88
	140–160	0.74	2.02
	160–180	0.78	2.13
	180–200	0.82	2.24
	200–220	0.85	2.32
	220–240	0.89	2.43
	240–260	0.92	2.51
	260–280	0.95	2.59
	280–300	1.00	2.73
	300–320	1.03	2.81
	320–340	1.06	2.89
	340–360	1.14	3.11
Free Machining Steels			
AISI 1108, 1109, 1110, 1115, 1116, 1117, 1118, 1119, 1120, 1125, 1126, 1132	100–120	0.41	1.12
	120–140	0.42	1.15
	140–160	0.44	1.20
	160–180	0.48	1.31
	180–200	0.50	1.36
AISI 1137, 1138, 1139, 1140, 1141, 1144, 1145, 1146, 1148, 1151	180–200	0.51	1.39
	200–220	0.55	1.50
	220–240	0.57	1.56
	240–260	0.62	1.69
Alloy Steels			
AISI 4023, 4024, 4027, 4028, 4032, 4037, 4042, 4047, 4137, 4140, 4142, 4145, 4147, 4150, 4340, 4640, 4815, 4817, 4820, 5130, 5132, 5135, 5140, 5145, 5150, 6118, 6150, 8637, 8640, 8642, 8645, 8650, 8740	140–160	0.62	1.69
	160–180	0.65	1.77
	180–200	0.69	1.88
	200–220	0.72	1.97
	220–240	0.76	2.07
	240–260	0.80	2.18
	260–280	0.84	2.29
	280–300	0.87	2.38
	300–320	0.91	2.48
	320–340	0.96	2.62
	340–360	1.00	2.73
AISI 4130, 4320, 4615, 4620, 4626, 5120, 8615, 8617, 8620, 8622, 8625, 8630, 8720	140–160	0.56	1.53
	160–180	0.59	1.61
	180–200	0.62	1.69
	200–220	0.65	1.77
	220–240	0.70	1.91
	240–260	0.74	2.02
	260–280	0.77	2.10
	280–300	0.80	2.18
	300–320	0.83	2.27
	320–340	0.89	2.43
AISI 1330, 1335, 1340, E52100	160–180	0.79	2.16
	180–200	0.83	2.27
	200–220	0.87	2.38
	220–240	0.91	2.48
	240–260	0.95	2.59
	260–280	1.00	2.73

Table 31. Work Material Factor, K_d, for Drilling with a Sharp Drill

Work Material	Work Material Constant, K_d
AISI 1117 (Resulfurized free machining mild steel)	12,000
Steel, 200 Bhn	24,000
Steel, 300 Bhn	31,000
Steel, 400 Bhn	34,000
Cast Iron, 150 Bhn	14,000
Most Aluminum Alloys	7,000
Most Magnesium Alloys	4,000
Most Brasses	14,000
Leaded Brass	7,000
Austenitic Stainless Steel (Type 316)	24,000[a] for Torque
	35,000[a] for Thrust
Titanium Alloy T16A 4V 40R_c	18,000[a] for Torque
	29,000[a] for Thrust
René 41 40R_c	40,000[ab] min.
Hastelloy-C	30,000[a] for Torque
	37,000[a] for Thrust

[a] Values based upon a limited number of tests.

[b] Will increase with rapid wear.

Table 32. Chisel Edge Factors for Torque and Thrust

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

For drills of standard design, use $c/d = .18$.
For split point drills, use $c/d = .03$.
c/d = Length of Chisel Edge ÷ Drill Diameter.
w/d = Web Thickness at Drill Point ÷ Drill Diameter.

For inch units only:

$$T = 2k_d F_f F_T B W + K_d d^2 J W \qquad (3)$$

$$M = K_d F_f F_M A W \qquad (4)$$

$$P_c = MN/63.025 \qquad (5)$$

For SI metric units only:

$$T = 0.05\, K_d F_f F_T B W + 0.007\, K_d d^2 J W \tag{6}$$

$$M = \frac{K_d F_f F_M A W}{40{,}000} = 0.000025\, K_d F_f F_M A W \tag{7}$$

$$P_c = MN ⁄ 9550 \tag{8}$$

Use with either inch or metric units:

$$P_m = \frac{P_c}{E} \tag{9}$$

where P_c = Power at the cutter; hp, or kW

P_m = Power at the motor; hp, or kW

M = Torque; in. lb, or N.m

T = Thrust; lb, or N

K_d = Work material factor (See Table 31)

F_f = Feed factor (See Table 33)

F_T = Thrust factor for drill diameter (See Table 34)

F_M = Torque factor for drill diameter (See Table 34)

A = Chisel edge factor for torque (See Table 32)

B = Chisel edge factor for thrust (See Table 32)

J = Chisel edge factor for thrust (See Table 32)

W = Tool wear factor (See Table 26)

N = Spindle speed; rpm

E = Machine tool efficiency factor (See Table 28)

D = Drill diameter; in., or mm

c = Chisel edge length; in., or mm (See Table 32)

w = Web thickness at drill point; in., or mm (See Table 32)

Table 33. Feed Factors, F_f, for Drilling

Inch Units				SI Metric Units			
Feed, in./rev	F_f	Feed, in./rev	F_f	Feed, mm/rev	F_f	Feed, mm/rev	F_f
0.0005	0.0023	0.012	0.029	0.01	0.025	0.30	0.382
0.001	0.004	0.013	0.031	0.03	0.060	0.35	0.432
0.002	0.007	0.015	0.035	0.05	0.091	0.40	0.480
0.003	0.010	0.018	0.040	0.08	0.133	0.45	0.528
0.004	0.012	0.020	0.044	0.10	0.158	0.50	0.574
0.005	0.014	0.022	0.047	0.12	0.183	0.55	0.620
0.006	0.017	0.025	0.052	0.15	0.219	0.65	0.708
0.007	0.019	0.030	0.060	0.18	0.254	0.75	0.794
0.008	0.021	0.035	0.068	0.20	0.276	0.90	0.919
0.009	0.023	0.040	0.076	0.22	0.298	1.00	1.000
0.010	0.025	0.050	0.091	0.25	0.330	1.25	1.195

Table 34. Drill Diameter Factors: F_T for Thrust; F_M for Torque

Inch Units						SI Metric Units					
Drill Dia., in.	F_T	F_M	Drill Dia., in.	F_T	F_M	Drill Dia., mm	F_T	F_M	Drill Dia., mm	F_T	F_M
0.063	0.110	0.007	0.875	0.899	0.786	1.60	1.46	2.33	22.00	11.86	260.8
0.094	0.151	0.014	0.938	0.950	0.891	2.40	2.02	4.84	24.00	12.71	305.1
0.125	0.189	0.024	1.000	1.000	1.000	3.20	2.54	8.12	25.50	13.34	340.2
0.156	0.226	0.035	1.063	1.050	1.116	4.00	3.03	12.12	27.00	13.97	377.1
0.188	0.263	0.049	1.125	1.099	1.236	4.80	3.51	16.84	28.50	14.58	415.6
0.219	0.297	0.065	1.250	1.195	1.494	5.60	3.97	22.22	32.00	16.00	512.0
0.250	0.330	0.082	1.375	1.290	1.774	6.40	4.42	28.26	35.00	17.19	601.6
0.281	0.362	0.102	1.500	1.383	2.075	7.20	4.85	34.93	38.00	18.36	697.6
0.313	0.395	0.124	1.625	1.475	2.396	8.00	5.28	42.22	42.00	19.89	835.3
0.344	0.426	0.146	1.750	1.565	2.738	8.80	5.96	50.13	45.00	21.02	945.8
0.375	0.456	0.171	1.875	1.653	3.100	9.50	6.06	57.53	48.00	22.13	1062
0.438	0.517	0.226	2.000	1.741	3.482	11.00	6.81	74.90	50.00	22.86	1143
0.500	0.574	0.287	2.250	1.913	4.305	12.50	7.54	94.28	58.00	25.75	1493
0.563	0.632	0.355	2.500	2.081	5.203	14.50	8.49	123.1	64.00	27.86	1783
0.625	0.687	0.429	2.750	2.246	6.177	16.00	9.19	147.0	70.00	29.93	2095
0.688	0.741	0.510	3.000	2.408	7.225	17.50	9.87	172.8	76.00	31.96	2429
0.750	0.794	0.596	3.500	2.724	9.535	19.00	10.54	200.3	90.00	36.53	3293
0.813	0.847	0.689	4.000	3.031	12.13	20.00	10.98	219.7	100.00	39.81	3981

Example: A standard $\frac{7}{8}$ inch drill is to drill steel parts having a hardness of 200 Bhn on a drilling machine having an efficiency of 0.80. The spindle speed to be used is 350 rpm and the feed rate will be 0.008 in./rev. Calculate the thrust, torque, and power required to drill these holes:

$$K_d = 24{,}000 \text{ (From Table 31)}$$
$$F_f = 0.021 \text{ (From Table 33)}$$
$$F_T = 0.899 \text{ (From Table 34)}$$
$$F_M = 0.786 \text{ (From Table 34)}$$
$$A = 1.085 \text{ (From Table 32)}$$
$$B = 1.355 \text{ (From Table 32)}$$
$$J = 0.030 \text{ (From Table 32)}$$
$$W = 1.30 \text{ (From Table 26)}$$

$$\begin{aligned} T &= 2K_d F_f F_T BW + K_d d^2 JW \\ &= 2 \times 24{,}000 \times 0.21 \times 0.899 \times 1.355 \times 1.30 + 24{,}000 \times 0.875^2 \times 0.030 \times 1.30 \\ &= 2313 \text{ lb} \end{aligned}$$

$$\begin{aligned} M &= K_d F_f F_m AW \\ &= 24{,}000 \times 0.021 \times 0.786 \times 1.085 \times 1.30 = 559 \text{ in. lb} \end{aligned}$$

$$P_c = \frac{MN}{63{,}025} = \frac{559 \times 350}{63{,}025} = 3.1 \text{ hp} \qquad P_m = \frac{P_c}{E} = \frac{3.1}{0.80} = 3.9 \text{ hp}$$

Twist drills are generally the most highly stressed of all metal cutting tools. They must not only resist the cutting forces on the lips, but also the drill torque resulting from these forces and the very large thrust force required to push the drill through the hole. Therefore, often when drilling smaller holes, the twist drill places a limit on the power used and for very large holes, the machine may limit the power.

MACHINING ECONOMETRICS

Tool Wear And Tool Life Relationships

Tool wear.—Tool-life is defined as the cutting time to reach a predetermined wear, called the tool wear criterion. The size of tool wear criterion depends on the grade used, usually a tougher grade can be used at bigger flank wear. For finishing operations, where close tolerances are required, the wear criterion is relatively small. Other alternative wear criteria are a predetermined value of the surface roughness, or a given depth of the crater which develops on the rake face of the tool. The most appropriate wear criteria depends on cutting geometry, grade, and materials.

Tool-life is determined by assessing the time — the tool-life — at which a given predetermined flank wear is reached, 0.25, 0.4, 0.6, 0.8 mm etc. Fig. 1 depicts how flank wear varies with cutting time (approximately straight lines in a semi-logarithmic graph) for three combinations of cutting speeds and feeds. Alternatively, these curves may represent how variations of machinability impact on tool-life, when cutting speed and feed are constant. All tool wear curves will sooner or later bend upwards abruptly and the cutting edge will break, i.e., catastrophic failure as indicated by the white arrows in Fig. 1.

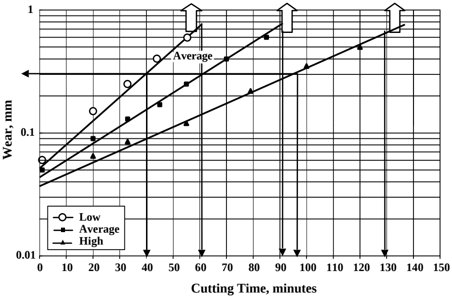


Fig. 1. Flank Wear as a Function of Cutting Time

The maximum deviation from the average tool-life 60 minutes in Fig. 1 is assumed to range between 40 and 95 minutes, i.e. −33% and +58% variation. The positive deviation from the average (longer than expected tool-life) is not important, but the negative one (shorter life) is, as the edge may break before the scheduled tool change after 60 minutes, when the flank wear is 0.6 mm.

It is therefore important to set the wear criterion at a safe level such that tool failures due to "normal" wear become negligible. This is the way machinability variations are mastered.

Equivalent Chip Thickness (*ECT*).—*ECT* combines the four basic turning variables, depth of cut, lead angle, nose radius and feed per revolution into one basic parameter. For all other metal cutting operations such as drilling, milling and grinding, additional variables such as number of teeth, width of cut, and cutter diameter are included in the parameter *ECT*. In turning, milling, and drilling, according to the *ECT* principle, when the product of feed times depth of cut is constant the tool-life is constant no matter how the depth of cut or feed is selected, provided that the cutting speed and cutting edge length are maintained constant. By replacing the geometric parameters with *ECT*, the number of tool-life tests to evaluate cutting parameters can be reduced considerably, by a factor of 4 in turning, and in milling by a factor of 7 because radial depth of cut, cutter diameter and number of teeth are additional parameters.

The introduction of the *ECT* concept constitutes a major simplification when predicting tool-life and calculating cutting forces, torque, and power. *ECT* was first presented in 1931 by Professor R. Woxen, who both theoretically and experimentally proved that *ECT* is a basic metal cutting parameter for high-speed cutting tools. Dr. Colding later proved that the concept also holds for carbide tools, and extended the calculation of *ECT* to be valid for cutting conditions when the depth of cut is smaller than the tool nose radius, or for round inserts. Colding later extended the concept to all other metal cutting operations, including the grinding process.

The definition of *ECT* is:

$$ECT = \frac{Area}{CEL} \text{ (mm or inch)}$$

where A = cross sectional area of cut (approximately = feed × depth of cut), (mm^2 or inch^2)

CEL = cutting edge length (tool contact rubbing length), (mm or inch), see Fig.9.

An exact value of *A* is obtained by the product of *ECT* and *CEL*. In turning, milling, and drilling, *ECT* varies between 0.05 and 1 mm, and is always less than the feed/rev or feed/tooth; its value is usually about 0.7 to 0.9 times the feed.

Example 1: For a feed of 0.8 mm/rev, depth of cut a = 3 mm, and a cutting edge length $CEL = 4\ \text{mm}^2$, the value of *ECT* is approximately $ECT = 0.8 \times 3 \div 4 = 0.6$ mm.

The product of *ECT*, *CEL*, and cutting speed *V* (m/min or ft/min) is equal to the metal removal rate, *MRR*, which is measured in terms of the volume of chips removed per minute:

$$\begin{aligned} MRR &= 1000V \times Area = 1000V \times ECT \times CEL \text{ mm}^3\text{/min} \\ &= V \times Area \text{ cm}^3\text{/min or inch}^3\text{/min} \end{aligned}$$

The specific metal removal rate *SMRR* is the metal removal rate per mm cutting edge length *CEL*, thus:

$$\begin{aligned} SMMR &= 1000V \times ECT \text{ mm}^3\text{/min/mm} \\ &= V \times ECT \text{ cm}^3\text{/min/mm or inch}^3\text{/min/inch} \end{aligned}$$

Example 2: Using above data and a cutting speed of *V* = 250 m/min specific metal removal rate becomes $SMRR = 0.6 \times 250 = 150$ (cm^3/min/mm).

ECT in Grinding: In grinding *ECT* is defined as in the other metal cutting processes, and is approximately equal to $ECT = Vw \times ar \div V$, where *Vw* is the work speed, *ar* is the depth of cut, and $A = Vw \times ar$. Wheel life is constant no matter how depth *ar*, or work speed *Vw*, is selected at *V* = constant (usually the influence of grinding contact width can be neglected). This translates into the same wheel life as long as the specific metal removal rate is constant, thus:

$$SMMR = 1000Vw \times ar \text{ mm}^3\text{/min/mm}$$

In grinding, *ECT* is much smaller than in the other cutting processes, ranging from about 0.0001 to 0.001 mm (0.000004 to 0.00004 inch). The grinding process is described in a separate chapter *GRINDING FEEDS AND SPEEDS* starting on page 1120.

Tool-life Relationships.—Plotting the cutting times to reach predetermined values of wear typically results in curves similar to those shown in Fig. 2 (cutting time versus cutting speed at constant feed per tooth) and Fig. 3 (cutting time versus feed per tooth at constant cutting speed). These tests were run in 1993 with mixed ceramics turn-milling hard steel, 82 R_C, at the Technische Hochschule Darmstadt.

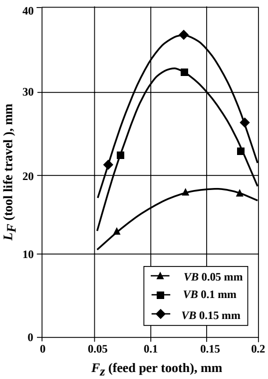


Fig. 2. Influence of feed per tooth on cutting time

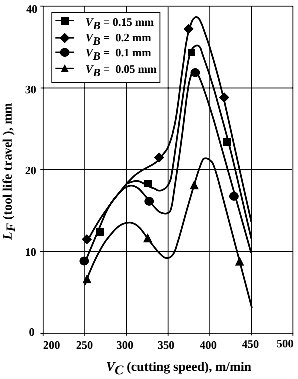


Fig. 3. Influence of cutting speed on tool-life

Tool-life has a maximum value at a particular setting of feed and speed. Economic and productive cutting speeds always occur on the right side of the curves in Figs. 2 and 4, which are called Taylor curves, represented by the so called Taylor's equation.

The variation of tool-life with feed and speed constitute complicated relationships, illustrated in Figs. 6a, 6b, and 6c.

Taylor's Equation.—Taylor's equation is the most commonly used relationship between tool-life *T*, and cutting speed *V*. It constitutes a straight line in a log-log plot, one line for each feed, nose radius, lead angle, or depth of cut, mathematically represented by:

$$V \times T^n = \mathrm{C} \tag{1a}$$

where $n =$ is the slope of the line

C = is a constant equal to the cutting speed for $T = 1$ minute

By transforming the equation to logarithmic axes, the Taylor lines become straight lines with slope = n. The constant C is the cutting speed on the horizontal (V) axis at tool-life T = 1 minute, expressed as follows

$$\ln V + n \times \ln T = \ln \mathrm{C} \tag{1b}$$

For different values of feed or *ECT*, log-log plots of Equation (1a) form approximately straight lines in which the slope decreases slightly with a larger value of feed or *ECT*. In practice, the Taylor lines are usually drawn parallel to each other, i.e., the slope n is assumed to be constant.

Fig. 4 illustrates the Taylor equation, tool-life *T* versus cutting speed *V*, plotted in log-log coordinates, for four values of *ECT* = 0.1, 0.25, 0.5 and 0.7 mm.

In Fig. 4, starting from the right, each *T–V* line forms a generally straight line that bends off and reaches its maximum tool-life, then drops off with decreasing speed (see also Figs. 2 and 3. When operating at short tool-lives, approximately when *T* is less than 5 minutes, each line bends a little so that the cutting speed for 1 minute life becomes less than the value calculated by constant C.

The Taylor equation is a very good approximation of the right hand side of the real tool-life curve (slightly bent). The portion of the curve to the left of the maximum tool-life gives shorter and shorter tool-lives when decreasing the cutting speed starting from the point of maximum tool-life. Operating at the maximum point of maximum tool-life, or to the left of it, causes poor surface finish, high cutting forces, and sometimes vibrations.

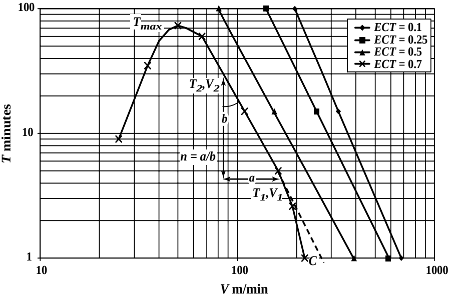


Fig. 4. Definition of slope *n* and constant C in Taylor's equation

Evaluation of Slope *n*, and Constant C.—When evaluating the value of the Taylor slope based on wear tests, care must be taken in selecting the tool-life range over which the slope is measured, as the lines are slightly curved.

The slope *n* can be found in three ways:

- Calculate *n* from the formula $n = (\ln C - \ln V)/\ln T$, reading the values of C and *V* for any value of *T* in the graph.
- Alternatively, using two points on the line, (V_1, T_1) and (V_2, T_2), calculate *n* using the relationship $V_1 \times T_1^n = V_2 \times T_2^n$. Then, solving for *n*,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)}$$

- Graphically, *n* may be determined from the graph by measuring the distances "a" and "b" using a mm scale, and *n* is the ratio of *a* and *b*, thus, *n* = a/b

Example: Using Fig. 4, and a given value of *ECT*= 0.7 mm, calculate the slope and constant of the Taylor line.

On the Taylor line for *ECT*= 0.7, locate points corresponding to tool-lives $T_1 = 15$ minutes and $T_2 = 60$ minutes. Read off the associated cutting speeds as, approximately, $V_1 = 110$ m/min and $V_2 = 65$ m/min.

The slope *n* is then found to be $n = \ln(110/65)/\ln(60/15) = 0.38$

The constant C can be then determined using the Taylor equation and either point (T_1, V_1) or point (T_2, V_2), with equivalent results, as follows:

$$C = V \times T^n = 110 \times 15^{0.38} = 65 \times 60^{0.38} = 308 \text{ m/min (1027 fpm)}$$

The Generalized Taylor Equation.—The above calculated slope and constant C define tool-life at one particular value of feed *f*, depth of cut *a*, lead angle *LA*, nose radius *r*, and other relevant factors.

The generalized Taylor equation includes these parameters and is written

$$T^n = A \times f^m \times a^p \times LA^q \times r^s \tag{2}$$

where A = area; and, *n*, *m*, *p*, *q*, and s = constants.

There are two problems with the generalized equation: 1) a great number of tests have to be run in order to establish the constants *n*, *m*, *p*, *q*, *s*, etc.; and 2) the accuracy is not very good because Equation (2) yields straight lines when plotted versus *f*, *a*, *LA*, and *r*, when in reality, they are parabolic curves..

The Generalized Taylor Equation using Equivalent Chip Thickness (ECT): Due to the compression of the aforementioned geometrical variables (f, a, LA, r, etc.) into ECT, Equation (2) can now be rewritten:

$$V \times T^n = A \times ECT^m \tag{3}$$

Experimental data confirms that the Equation (3) holds, approximately, within the range of the test data, but as soon as the equation is extended beyond the test results, the error can become very great because the *V–ECT* curves are represented as straight lines by Equation (3)and the real curves have a parabolic shape.

The Colding Tool-life Relationship.—This relationship contains 5 constants H, K, L, M, and N_0, which attain different values depending on tool grade, work material, and the type of operation, such as longitudinal turning versus grooving, face milling versus end milling, etc.

This tool-life relationship is proven to describe, with reasonable accuracy, how tool-life varies with ECT and cutting speed for any metal cutting and grinding operation. It is expressed mathematically as follows either as a generalized Taylor equation (4a), or, in logarithmic coordinates (4b):

$$V \times T^{(N_0 - L \times \ln ECT)} \times ECT^{\left(-\frac{H}{2M} + \frac{\ln ECT}{4M}\right)} = e^{\left(K - \frac{H}{4M}\right)} \tag{4a}$$

$$y = K - \frac{x - H}{4M} - z(N_0 - L_x) \tag{4b}$$

where $x = \ln ECT \quad y = \ln V \quad z = \ln T$

M = the vertical distance between the maximum point of cutting speed (ECT_H, V_H) for $T = 1$ minute and the speed V_G at point (ECT_G, V_G), as shown in Fig. 5.

$2M$ = the horizontal distance between point (ECT_H, V_G) and point (V_G, ECT_G)

H and K = the logarithms of the coordinates of the maximum speed point (ECT_H, V_H) at tool-life $T = 1$ minute, thus $H = \ln(ECT_H)$ and $K = \ln(V_H)$

N_0 and L = the variation of the Taylor slope n with ECT: $n = N_0 - L \times \ln(ECT)$

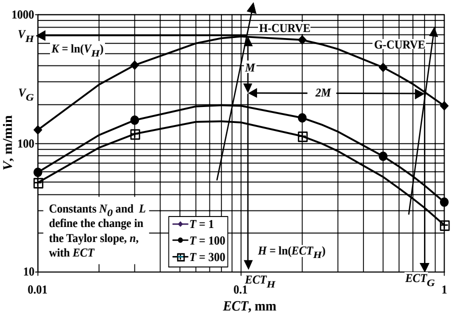


Fig. 5. Definitions of the constants H, K, L, M, and N_0 for tool-life equation in the V-ECT plane with tool-life constant

The constants L and N_0 are determined from the slopes n_1 and n_2 of two Taylor lines at ECT_1 and ECT_2, and the constant M from 3 *V–ECT* values at any constant tool-life. Constants H and K are then solved using the tool-life equation with the above-calculated values of L, N_0 and M.

The *G*- and *H*-curves.—The *G*-curve defines the longest possible tool-life for any given metal removal rate, *MRR*, or specific metal removal rate, *SMRR*. It also defines the point where the total machining cost is minimum, after the economic tool-life T_E, or optimal tool-life T_O, has been calculated, see *Optimization Models, Economic Tool-life when Feed is Constant* starting on page 1073.

The tool-life relationship is depicted in the 3 planes: *T–V*, where *ECT* is the plotted parameter (the Taylor plane); *T–ECT*, where *V* is plotted; and, *V–ECT*, where *T* is a parameter. The latter plane is the most useful because the optimal cutting conditions are more readily understood when viewing in the *V–ECT* plane. Figs. 6a, 6b, and 6c show how the tool-life curves look in these 3 planes in log-log coordinates.

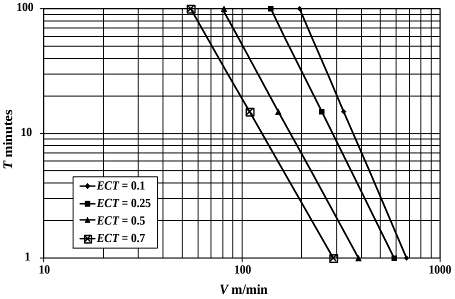


Fig. 6a. Tool-life vs. cutting sped *T–V*, *ECT* plotted

Fig. 6a shows the Taylor lines, and Fig. 6b illustrates how tool-life varies with *ECT* at different values of cutting speed, and shows the *H*-curve. Fig. 6c illustrates how cutting speed varies with *ECT* at different values of tool-life. The *H*- and *G*-curves are also drawn in Fig. 6c.

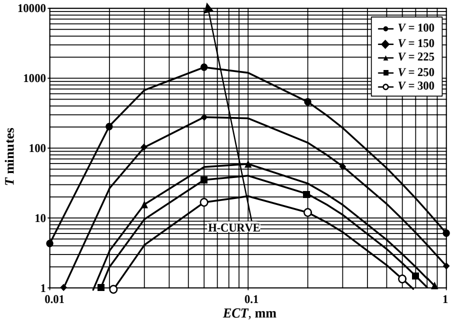


Fig. 6b. Tool-life vs. *ECT*, *T–ECT*, cutting speed plotted

A simple and practical method to ascertain that machining is not done to the left of the *H*-curve is to examine the chips. When *ECT* is too small, about 0.03-0.05 mm, the chips tend to become irregular and show up more or less as dust.

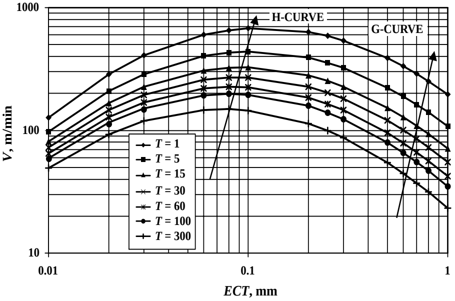


Fig. 6c. Cutting speed vs. *ECT*, *V–ECT*, tool-life plotted

The *V–ECT–T* Graph and the Tool-life Envelope.— The tool-life envelope, in Fig. 7, is an area laid over the *V–ECT–T* graph, bounded by the points A, B, C, D, and E, within which successful cutting can be realized. The H- and *G*-curves represent two borders, lines $\overline{\text{AE}}$ and $\overline{\text{BC}}$. The border curve, line $\overline{\text{AB}}$, shows a lower limit of tool-life, T_{MIN} = 5 minutes, and border curve, line $\overline{\text{DE}}$, represents a maximum tool-life, T_{MAX} = 300 minutes.

T_{MIN} is usually 5 minutes due to the fact that tool-life versus cutting speed does not follow a straight line for short tool-lives; it decreases sharply towards one minute tool-life. T_{MAX} varies with tool grade, material, speed and *ECT* from 300 minutes for some carbide tools to 10000 minutes for diamond tools or diamond grinding wheels, although systematic studies of maximum tool-lives have not been conducted.

Sometimes the metal cutting system cannot utilize the maximum values of the *V–ECT–T* envelope, that is, cutting at optimum *V–ECT* values along the *G*-curve, due to machine power or fixture constraints, or vibrations. Maximum ECT values, ECT_{MAX}, are related to the strength of the tool material and the tool geometry, and depend on the tool grade and material selection, and require a relatively large nose radius.

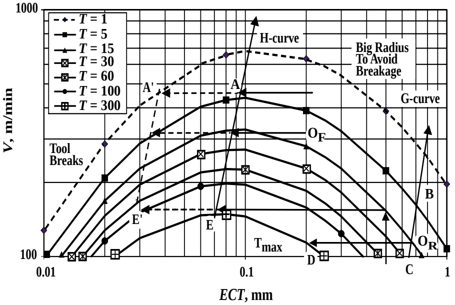


Fig. 7. Cutting speed vs. *ECT*, *V–ECT*, tool-life plotted

Minimum *ECT* values, ECT_{MIN}, are defined by the conditions at which surface finish suddenly deteriorates and the cutting edge begins rubbing rather than cutting. These conditions begin left of the *H*-curve, and are often accompanied by vibrations and built-up edges on the tool. If feed or *ECT* is reduced still further, excessive tool wear with sparks and tool breakage, or melting of the edge occurs. For this reason, values of *ECT* lower than approximately 0.03 mm should not be allowed. In Fig. 7, the ECT_{MIN} boundary is indicated by contour line $\overline{A'E'}$.

In milling the minimum feed/tooth depends on the ratio *ar*/*D*, of radial depth of cut *ar*, and cutter diameter *D*. For small *ar*/*D* ratios, the chip thickness becomes so small that it is necessary to compensate by increasing the feed/tooth. See *High-speed Machining Econometrics* starting on page 1085 for more on this topic.

Fig. 7 demonstrates, in principle, minimum cost conditions for roughing at point $\mathbf{O_R}$, and for finishing at point $\mathbf{O_F}$, where surface finish or tolerances have set a limit. Maintaining the speed at $\mathbf{O_R}$, 125 m/min, and decreasing feed reaches a maximum tool-life = 300 minutes at $ECT = 0.2$, and a further decrease of feed will result in shorter lives.

Similarly, starting at point X ($V = 150$, $ECT = 0.5$, $T = 15$) and reducing feed, the *H*-curve will be reached at point E ($ECT = 0.075$, $T = 300$). Continuing to the left, tool-life will decrease and serious troubles occur at point E′ ($ECT = 0.03$).

Starting at point $\mathbf{O_F}$ ($V = 300$, $ECT = 0.2$, $T = 15$) and reducing feed the *H*-curve will be reached at point E ($ECT = 0.08$, $T = 15$). Continuing to the left, life will decrease and serious troubles occur at $ECT = 0.03$.

Starting at point X ($V = 400$, $ECT = 0.2$, $T = 5$) and reducing feed the *H*-curve will be reached at point E ($ECT = 0.09$, $T = 7$). Continuing to the left, life will decrease and serious troubles occur at point A′ ($ECT = 0.03$), where $T = 1$ minute.

Cutting Forces and Chip Flow Angle.—There are three cutting forces, illustrated in Fig. 8, that are associated with the cutting edge with its nose radius *r*, depth of cut *a*, lead angle *LA*, and feed per revolution *f*, or in milling feed per tooth f_z. There is one drawing for roughing and one for finishing operations.

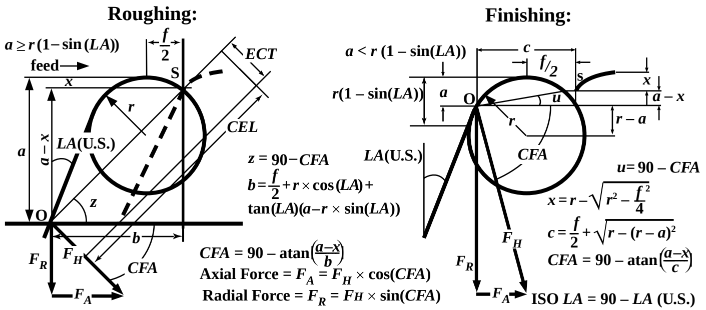


Fig. 8. Definitions of equivalent chip thickness, *ECT*, and chip flow angle, *CFA*.

The cutting force F_C, or tangential force, is perpendicular to the paper plane. The other two forces are the feed or axial force F_A, and the radial force F_R directed towards the work piece. The resultant of F_A and F_R is called F_H. When finishing, F_R is bigger than F_A, while in roughing F_A is usually bigger than F_R. The direction of F_H, measured by the chip flow angle *CFA*, is perpendicular to the rectangle formed by the cutting edge length *CEL* and *ECT* (the product of *ECT* and CEL constitutes the cross sectional area of cut, *A*). The important task of determining the direction of F_H, and calculation of F_A and F_R, are shown in the formulas given in the Fig. 8.

The method for calculating the magnitudes of F_H, F_A, and F_R is described in the following. The first thing is to determine the value of the cutting force F_C. Approximate formulas to calculate the tangential cutting force, torque and required machining power are found in the section *ESTIMATING SPEEDS AND MACHINING POWER* starting on page 1044.

Specific Cutting Force, Kc: The specific cutting force, or the specific energy to cut, Kc, is defined as the ratio between the cutting force F_C and the chip cross sectional area, A. thus, $Kc = F_C \div A$ N/mm^2.

The value of Kc decreases when ECT increases, and when the cutting speed V increases. Usually, Kc is written in terms of its value at $ECT = 1$, called Kc_1, and neglecting the effect of cutting speed, thus $Kc = Kc_1 \times ECT^B$, where B = slope in log-log coordinates

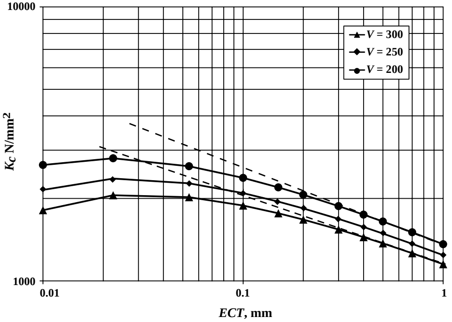


Fig. 9. Kc vs. ECT, cutting speed plotted

A more accurate relationship is illustrated in Fig. 9, where Kc is plotted versus ECT at 3 different cutting speeds. In Fig. 9, the two dashed lines represent the aforementioned equation, which each have different slopes, B. For the middle value of cutting speed, Kc varies with ECT from about 1900 to 1300 N/mm^2 when ECT increases from 0.1 to 0.7 mm. Generally the speed effect on the magnitude of Kc is approximately 5 to 15 percent when using economic speeds.

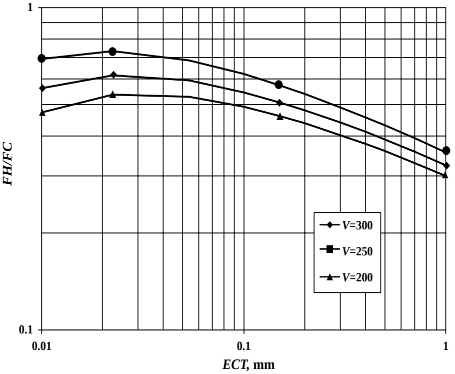


Fig. 10. F_H/F_C vs. ECT, cutting speed plotted

Determination of Axial, F_A, and Radial, F_R, Forces: This is done by first determining the resultant force F_H and then calculating F_A and F_R using the Fig. 8 formulas. F_H is derived from the ratio F_H/F_C, which varies with *ECT* and speed in a fashion similar to *Kc*. Fig. 10 shows how this relationship may vary.

As seen in Fig. 10, F_H/F_C is in the range 0.3 to 0.6 when *ECT* varies from 0.1 to 1 mm, and speed varies from 200 to 250 m/min using modern insert designs and grades. Hence, using reasonable large feeds F_H/F_C is around 0.3 – 0.4 and when finishing about 0.5 – 0.6.

Example: Determine F_A and F_R, based on the chip flow angle *CFA* and the cutting force F_C, in turning.

Using a value of $Kc = 1500$ N/mm^2 for roughing, when $ECT = 0.4$, and the cutting edge length $CEL = 5$ mm, first calculate the area $A = 0.4 \times 5 = 2$ mm^2. Then, determine the cutting force $F_C = 2 \times 1500 = 3000$ Newton, and an approximate value of $F_H = 0.5 \times 3000 = 1500$ Newton.

Using a value of $Kc = 1700$ N/mm^2 for finishing, when $ECT = 0.2$, and the cutting edge length $CEL = 2$ mm, calculate the area $A = 0.2 \times 2 = 0.4$ mm^2. The cutting force $F_C = 0.4 \times 1700 = 680$ Newton and an approximate value of $F_H = 0.35 \times 680 = 238$ Newton.

Fig. 8 can be used to estimate *CFA* for rough and finish turning. When the lead angle *LA* is 15 degrees and the nose radius is relatively large, an estimated value of the chip flow angle becomes about 30 degrees when roughing, and about 60 degrees in finishing. Using the formulas for F_A and F_R relative to F_H gives:

Roughing:

$F_A = F_H \times \cos(CFA) = 1500 \times \cos 30 = 1299$ Newton

$F_R = F_H \times \sin(CFA) = 1500 \times \sin 30 = 750$ Newton

Finishing:

$F_A = F_H \times \cos(CFA) = 238 \times \cos 60 = 119$ Newton

$F_R = F_H \times \sin(CFA) = 238 \times \sin 60 = 206$ Newton

The force ratio F_H/F_C also varies with the tool rake angle and increases with negative rakes. In grinding, F_H is much larger than the grinding cutting force F_C; generally F_H/F_C is approximately 2 to 4, because grinding grits have negative rakes of the order –35 to –45 degrees.

Forces and Tool-life.—Forces and tool life are closely linked. The ratio F_H/F_C is of particular interest because of the unique relationship of F_H/F_C with tool-life.

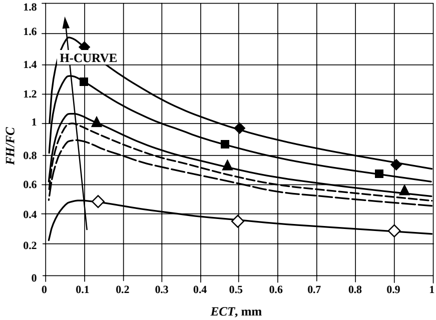


Fig. 11a. F_H/F_C vs. *ECT*

The results of extensive tests at Ford Motor Company are shown in Figs. 11a and 11b, where F_H/F_C and tool-life *T* are plotted versus *ECT* at different values of cutting speed *V*.

For any constant speed, tool-life has a maximum at approximately the same values of *ECT* as has the function F_H/F_C.

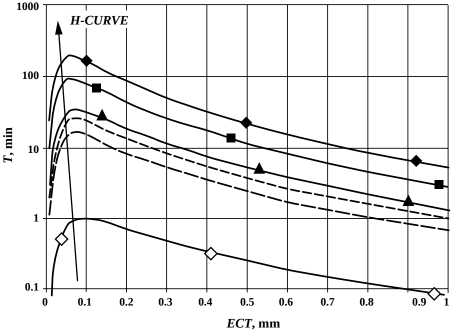


Fig. 11b. Tool-life vs. *ECT*

The Force Relationship: Similar tests performed elsewhere confirm that the F_H/F_C function can be determined using the 5 tool-life constants (H, K, M, L, N_0) introduced previously, and a new constant (L_F/L).

$$\ln\left(\frac{1}{a}\cdot\frac{F_H}{F_C}\right) = \frac{K - y - \dfrac{(x-H)^2}{4M}}{\dfrac{L_F}{L}(N_0 - Lx)} \qquad (5)$$

The constant a depends on the rake angle; in turning a is approximately 0.25 to 0.5 and L_F/L is 10 to 20. F_C attains it maximum values versus *ECT* along the *H*-curve, when the tool-life equation has maxima, and the relationships in the three force ratio planes look very similar to the tool-life functions shown in the tool-life planes in Figs. 6a, 6b, and 6c.

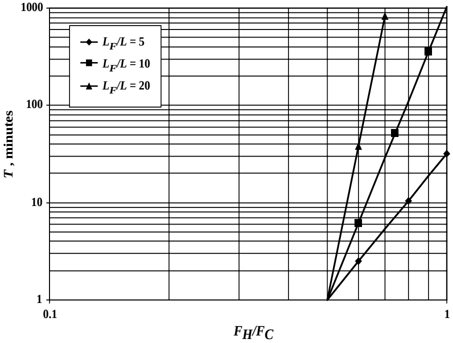


Fig. 12. Tool-life vs. F_H/F_C

Tool-life varies with F_H/F_C with a very simple formula according to Equation (5) as follows:

$$T = \left(\frac{F_H}{aF_C}\right)^{\frac{L_F}{L}}$$

where L is the constant in the tool-life equation, Equation (4a) or (4b), and L_F is the corresponding constant in the force ratio equation, Equation (5). In Fig. 12 this function is plotted for $a = 0.5$ and for $L_F/L = 5$, 10, and 20.

Accurate calculations of aforementioned relationships require elaborate laboratory tests, or better, the design of a special test and follow-up program for parts running in the ordinary production. A software machining program, such as Colding International Corp. *COMP* program can be used to generate the values of all 3 forces, torque and power requirements both for sharp and worn tools

Surface Finish R_a and Tool-life.—It is well known that the surface finish in turning decreases with a bigger tool nose radius and increases with feed; usually it is assumed that R_a increases with the square of the feed per revolution, and decreases inversely with increasing size of the nose radius. This formula, derived from simple geometry, gives rise to great errors. In reality, the relationship is more complicated because the tool geometry must taken into account, and the work material and the cutting conditions also have a significant influence.

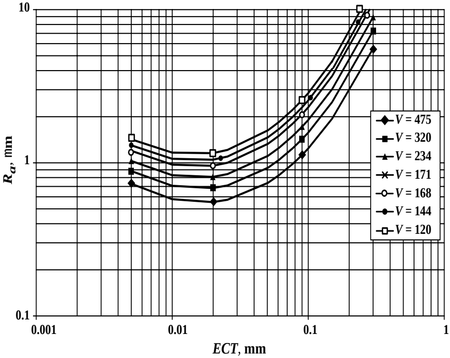


Fig. 13. R_a vs. *ECT*, nose radius *r* constant

Fig. 13 shows surface finish R_a versus *ECT* at various cutting speeds for turning cast iron with carbide tools and a nose radius $r = 1.2$ mm. Increasing the cutting speed leads to a smaller R_a value.

Fig. 14 shows how the finish improves when the tool nose radius, *r*, increases at a constant cutting speed (168 m/min) in cutting nodular cast iron.

In Fig. 15, R_a is plotted versus *ECT* with cutting speed *V* for turning a 4310 steel with carbide tools, for a nose radius $r = 1.2$ mm, illustrating that increasing the speed also leads to a smaller R_a value for steel machining.

A simple rule of thumb for the effect of increasing nose radius *r* on decreasing surface finish R_a, regardless of the ranges of *ECT* or speeds used, albeit within common practical values, is as follows. In finishing,

$$\frac{R_{a1}}{R_{a2}} = \left(\frac{r_2}{r_1}\right)^{0.5} \qquad (6)$$

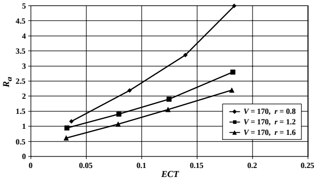


Fig. 14. R_a vs. *ECT*, cutting speed constant, nose radius *r* varies

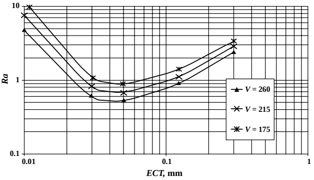


Fig. 15. R_a vs. *ECT*, cutting speed and nose radius *r* constant

In roughing, multiply the finishing values found using Equation (6) by 1.5, thus, $R_{a\,\text{(Rough)}} = 1.5 \times R_{a\,\text{(Finish)}}$ for each *ECT* and speed.

Example 1: Find the decrease in surface roughness resulting from a tool nose radius change from $r = 0.8$ mm to $r = 1.6$ mm in finishing. Also, find the comparable effect in roughing.

For finishing, using $r_2 = 1.6$ and $r_1 = 0.8$, $R_{a1}/R_{a2} = (1.6/0.8)^{0.5} = 1.414$, thus, the surface roughness using the larger tool radius is $R_{a2} = R_{a1} \div 1.414 = 0.7R_{a1}$

In roughing, at the same *ECT* and speed, $R_a = 1.5 \times R_{a2} = 1.5 \times 0.7R_{a1} = 1.05R_{a1}$

Example 2: Find the decrease in surface roughness resulting from a tool nose radius change from $r = 0.8$ mm to $r = 1.2$ mm

For finishing, using $r_2 = 1.2$ and $r_1 = 0.8$, $R_{a1}/R_{a2} = (1.2/0.8)^{0.5} = 1.224$, thus, the surface roughness using the larger tool radius is $R_{a2} = R_{a1} \div 1.224 = 0.82R_{a1}$

In roughing, at the same *ECT* and speed, $R_a = 1.5 \times R_{a2} = 1.5 \times 0.82R_{a1} = 1.23R_{a1}$

It is interesting to note that, at a given *ECT*, the R_a curves have a minimum, see Figs. 13 and 15, while tool-life shows a maximum, see Figs. 6b and 6c. As illustrated in Fig. 16, R_a increases with tool-life *T* when *ECT* is constant, in principle in the same way as does the force ratio.

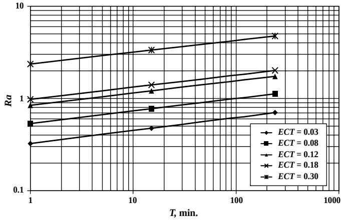


Fig. 16. R_a vs. *T*, holding *ECT* constant

The Surface Finish Relationship: R_a is determined using the same type of mathematical relationship as for tool-life and force calculations:

$$y = K_{Ra} - \frac{x - H_{Ra}{}^2}{4M_{Ra}} - (N_{0Ra} - L_{Ra})\ln(R_a)$$

where K_{RA}, H_{RA}, M_{RA}, N_{ORA}, and L_{RA} are the 5 surface finish constants.

Shape of Tool-life Relationships for Turning, Milling, Drilling and Grinding Operations—Overview.—A summary of the general shapes of tool-life curves (*V–ECT–T* graphs) for the most common machining processes, including grinding, is shown in double logarithmic coordinates in Fig. 17a through Fig. 17h.

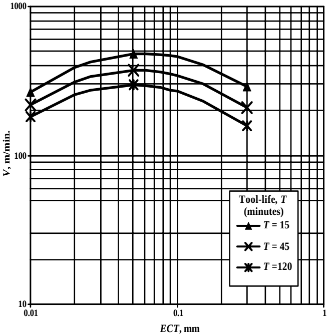


Fig. 17a. Tool-life for turning cast iron using coated carbide

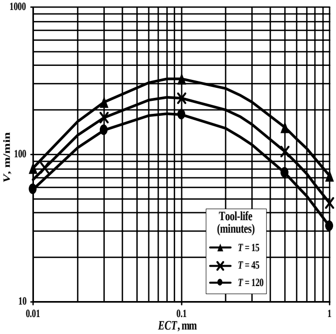


Fig. 17b. Tool-life for turning low-alloy steel using coated carbide

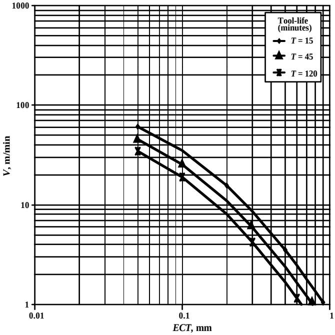


Fig. 17c. Tool-life for end-milling AISI 4140 steel using high-speed steel

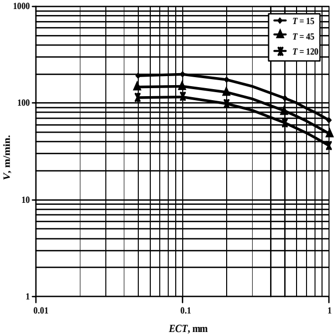


Fig. 17d. Tool-life for end-milling low-allow steel using uncoated carbide

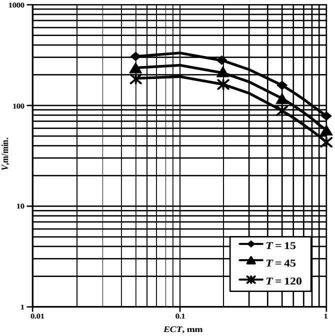


Fig. 17e. Tool-life for end-milling low-alloy steel using coated carbide

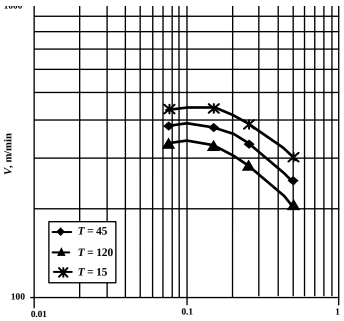


Fig. 17f. Tool-life for face-milling SAE 1045 steel using coated carbide

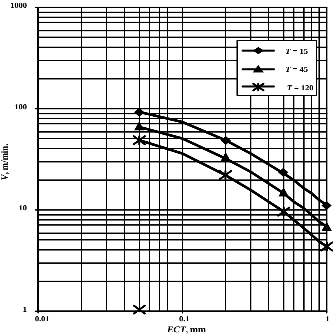


Fig. 17g. Tool-life for solid carbide drill

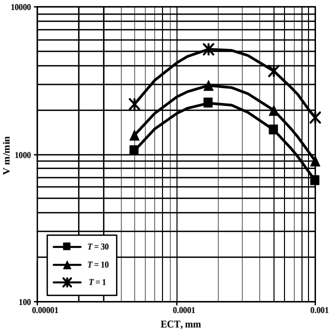


Fig. 17h. Wheel-life in grinding M4 tool-steel

Calculation Of Optimized Values Of Tool-life, Feed And Cutting Speed

Minimum Cost.—Global optimum is defined as the absolute minimum cost considering all alternative speeds, feeds and tool-lives, and refers to the determination of optimum tool-life T_O, feed f_O, and cutting speed V_O, for either minimum cost or maximum production rate. When using the tool-life equation, $T = f(V, ECT)$, determine the corresponding feed, for given values of depth of cut and operation geometry, from optimum equivalent chip thickness, ECT_O. Mathematically the task is to determine minimum cost, employing the cost function C_{TOT} = cost of machining time + tool changing cost + tooling cost. Minimum cost optima occur along the so-called *G*-curve, identified in Fig. 6c.

Another important factor when optimizing cutting conditions involves choosing the proper cost values for cost per edge C_E, replacement time per edge T_{RPL}, and not least, the hourly rate H_R that should be applied. H_R is defined as the portion of the hourly shop rate that is applied to the operations and machines in question. If optimizing all operations in the portion of the shop for which H_R is calculated, use the full rate; if only one machine is involved, apply a lower rate, as only a portion of the general overhead rate should be used, otherwise the optimum, and anticipated savings, are erroneous.

Production Rate.—The production rate is defined as the cutting time or the metal removal rate, corrected for the time required for tool changes, but neglecting the cost of tools.

The result of optimizing production rate is a shorter tool-life, higher cutting speed, and a higher feed compared to minimum cost optimization, and the tooling cost is considerably higher. Production rates optima also occur along the *G*-curve.

The Cost Function.—There are a number of ways the total machining cost C_{TOT} can be plotted, for example, versus feed, *ECT*, tool-life, cutting speed or other parameter. In Fig. 18a, cost for a face milling operation is plotted versus cutting time, holding feed constant, and using a range of tool-lives, *T*, varying from 1 to 240 minutes.

	t_c	C_{TOOL}	C_{TOT}	T	V
	5.85	0.487	0.569	1	598
	6.91	0.192	0.288	3	506
	7.47	0.125	0.228	5	468
	8.30	0.069	0.185	10	421
	8.83	0.049	0.172	15	396
Minimum cost	**9.81**	**0.027**	**0.164**	**30**	**356**
	10.91	0.015	0.167	60	321
	11.60	0.011	0.172	90	302
	12.12	0.008	0.177	120	289
	13.47	0.005	0.192	240	260

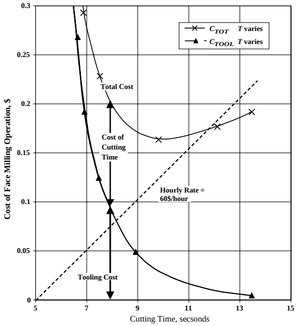


Fig. 18a. Variation of tooling cost C_{TOOL}, and total cost C_C, with cutting time t_c, including minimum cost cutting time

The tabulated values show the corresponding cutting speeds determined from the tool-life equation, and the influence of tooling on total cost. Tooling cost, C_{TOOL} = sum of tool cost + cost of replacing worn tools, decreases the longer the cutting time, while the total cost, C_{TOT}, has a minimum at around 10 seconds of cutting time. The dashed line in the graph represents the cost of machining time: the product of hourly rate H_R, and the cutting time t_c divided by 60. The slope of the line defines the value of H_R.

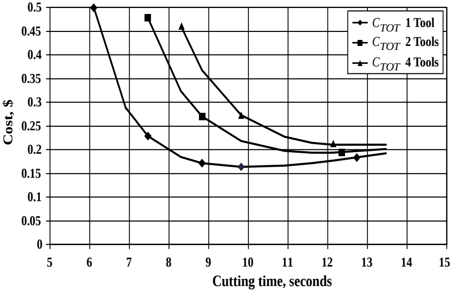


Fig. 18b. Total cost vs. cutting time for simultaneously cutting with 1, 2, and 4 tools

The cutting time for minimum cost varies with the ratio of tooling cost and H_R. Minimum cost moves towards a longer cutting time (longer tool-life) when either the price of the tooling increases, or when several tools cut simultaneously on the same part. In Fig. 18b, this is exemplified by running 2 and 4 cutters simultaneously on the same work piece, at the same feed and depth of cut, and with a similar tool as in Fig. 18a. As the tooling cost goes up 2 and 4 times, respectively, and H_R is the same, the total costs curves move up, but also moves to the right, as do the points of minimum cost and optimal cutting times. This means that going somewhat slower, with more simultaneously cutting tools, is advantageous.

Global Optimum.—Usually, global optimum occurs for large values of feed, heavy roughing, and in many cases the cutting edge will break trying to apply the large feeds required. Therefore, true optima cannot generally be achieved when roughing, in particular when using coated and wear resistant grades; instead, use the maximum values of feed, ECT_{max}, along the tool-life envelope, see Fig. 7.

As will be shown in the following, the first step is to determine the optimal tool-life T_O, and then determine the optimum values of feeds and speeds.

Optimum Tool-life T_O = 22 minutes

	fz	V_{22}	t_c, sec.	C_{TOOL}	C_{TOT}
	0.03	416	28.067	0.1067	0.4965
	0.08	397	11.017	0.0419	0.1949
	0.10	374	9.357	0.0356	0.1655
	0.17	301	6.831	0.0260	0.1208
	0.20	276	6.334	0.0241	0.1120
	0.40	171	5.117	0.0194	0.0905
Minimum Cost	**0.60**	**119**	**4.903**	**0.0186**	**0.0867**
	0.70	91	4.924	0.0187	0.0871

Maximum Production Rate, T = 5 minutes

fz	V_5	t_c	C_{TOOL}	C_{TOT}
0.7	163	3.569	0.059	0.109

T Varies between 1 and 240 minutes

f_z = 0.10 ECT= 0.26

	t_c secs.	C_{TOOL}	C_{TOT}	T	V
	5.850	0.487	0.569	1	598
	6.914	0.192	0.288	3	506
	7.473	0.125	0.228	5	468
	8.304	0.069	0.185	10	421
	8.832	0.049	0.172	15	396
Minimum Cost	**9.815**	**0.027**	**0.164**	**30**	**357**
	10.906	0.015	0.167	60	321
	11.600	0.011	0.172	90	302
	12.119	0.008	0.177	120	289
	13.467	0.005	0.192	240	260

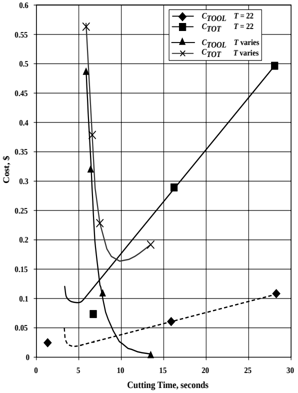


Fig. 19. Variation of tooling and total cost with cutting time, comparing global optimum with minimum cost at *fz* = 0.1 mm

The example in Fig. 19 assumes that T_O = 22 minutes and the feed and speed optima were calculated as f_O = 0.6 mm/tooth, V_O = 119 m/min, and cutting time t_{cO} = 4.9 secs.

The point of maximum production rate corresponds to f_O = 0.7 mm/tooth, V_O = 163 m/min, at tool-life T_O =5 minutes, and cutting time t_{cO} = 3.6 secs. The tooling cost is approximately 3 times higher than at minimum cost (0.059 versus 0.0186), while the piece cost is only slightly higher: $0.109 versus $0.087.

When comparing the global optimum cost with the minimum at feed = 0.1 mm/tooth the graph shows it to be less than half (0.087 versus 0.164), but also the tooling cost is about 1/3 lower (0.0186 versus 0.027). The reason why tooling cost is lower depends on the tooling cost term $t_c \times C_E/T$ (see *Calculation of Cost of Cutting and Grinding Operations* on page 1078). In this example, cutting times t_c= 4.9 and 9.81 seconds, at T = 22 and 30 minutes respectively, and the ratios are proportional to 4.9/22 = 0.222 and 9.81/30 = 0.327 respectively.

The portions of the total cost curve for shorter cutting times than at minimum corresponds to using feeds and speeds right of the *G*-curve, and those on the other side are left of this curve.

Optimization Models, Economic Tool-life when Feed is Constant.—Usually, optimization is performed versus the parameters tool-life and cutting speed, keeping feed at a constant value. The cost of cutting as function of cutting time is a straight line with the slope = H_R = hourly rate. This cost is independent of the values of tool change and tooling. Adding the cost of tool change and tooling, gives the variation of total cutting cost which shows a minimum with cutting time that corresponds to an economic tool-life, T_E. Economic tool-life represents a local optima (minimum cost) at a given constant value of feed, feed/tooth, or *ECT*.

Using the Taylor Equation: $V \times T = \mathrm{C}$ and differentiating C_{TOT} with respect to T yields:

Economic tool-life:

$T_E = T_V \times (1/n - 1)$, minutes

Economic cutting speed:

$V_E = \mathrm{C} / T_E{}^n$, m/min, or sfm

In these equations, n and C are constants in the Taylor equation for the given value of feed. Values of Taylor slopes, n, are estimated using the speed and feed Tables 1 through 23 starting on page 996 and handbook Table 5b on page 1004 for turning, and Table 15e on page 1028 for milling and drilling; and T_V is the equivalent tooling-cost time. $T_V = T_{RPL} + 60 \times C_E \div H_R$, minutes, where T_{RPL} = time for replacing a worn insert, or a set of inserts in a milling cutter or inserted drill, or a twist drill, reamer, thread chaser, or tap. T_V is described in detail, later; C_E = cost per edge, or set of edges, or cost per regrind including amortized price of tool; and H_R = hourly shop rate, or that rate that is impacted by the changes of cutting conditions .

In two dimensions, Fig. 20a shows how economic tool-life varies with feed per tooth. In this figure, the equivalent tooling-cost time T_V is constant, however the Taylor constant n varies with the feed per tooth.

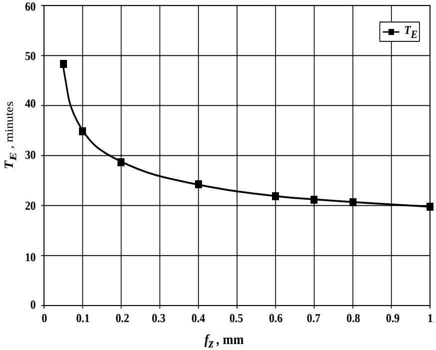


Fig. 20a. Economic tool-life, T_E vs. feed per tooth, *fz*

Economic tool-life increases with greater values of T_V, either when T_{RPL} is longer, or when cost per edge C_E is larger for constant H_R, or when H_R is smaller and T_{RPL} and C_E are unchanged. For example, when using an expensive machine (which makes H_R bigger) the value of T_V gets smaller, as does the economic tool-life, $T_E = T_V \times (1/n - 1)$. Reducing T_E results in an increase in the economic cutting speed, V_E. This means raising the cutting speed, and illustrates the importance, in an expensive system, of utilizing the equipment better by using more aggressive machining data.

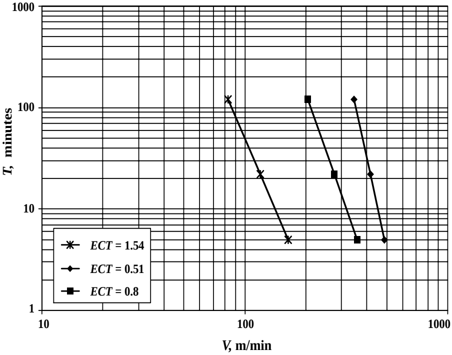


Fig. 20b. Tool-life vs. cutting speed, constant *ECT*

As shown in Fig. 20a for a face milling operation, economic tool-life T_E varies considerably with feed/tooth f_z, in spite of the fact that the Taylor lines have only slightly different slopes ($ECT = 0.51, 0.6, 1.54$), as shown in Fig. 20b. The calculation is based on the following cost data: $T_V = 6$, hourly shop rate H_R = \$60/hour, cutter diameter $D = 125$ mm with number of teeth $z = 10$, and radial depth of cut $ar = 40$ mm.

The conclusion relating to the determination of economic tool-life is that both hourly rate H_R and slope n must be evaluated with reasonable accuracy in order to arrive at good values. However, the method shown will aid in setting the trend for general machining economics evaluations.

Global Optimum, Graphical Method.—There are several ways to demonstrate in graphs how cost varies with the production parameters including optimal conditions. In all cases, tool-life is a crucial parameter.

Cutting time t_c is inversely proportional to the specific metal removal rate, $SMRR = V \times ECT$, thus, $1/t_c = V \times ECT$. Taking the log of both sides,

$$\ln V = -\ln ECT - \ln t_c + C \qquad (7)$$

where C is a constant.

Equation (7) is a straight line with slope (– 1) in the *V–ECT* graph when plotted in a log-log graph. This means that a constant cutting time is a straight 45-degree line in the *V–ECT* graph, when plotted in log-log coordinates with the same scale on both axis (a square graph).

The points at which the constant cutting time lines (at 45 degrees slope) are tangent to the tool-life curves define the *G*-curve, along which global optimum cutting occurs.

Note: If the ratio *a*/*CEL* is not constant when *ECT* varies, the constant cutting time lines are not straight, but the cutting time deviation is quite small in most cases.

In the *V–ECT* graph, Fig. 21, 45-degree lines have been drawn tangent to each tool-life curve: *T*=1, 5, 15, 30, 60, 100 and 300 minutes. The tangential points define the *G*-curve, and the 45-degree lines represent different constant cutting times: 1, 2, 3, 10 minutes, etc. Following one of these lines and noting the intersection points with the tool-life curves *T* = 1, 5, etc., many different speed and feed combinations can be found that will give the same cutting time. As tool-life gets longer (tooling cost is reduced), *ECT* (feed) increases but the cutting speed has to be reduced.

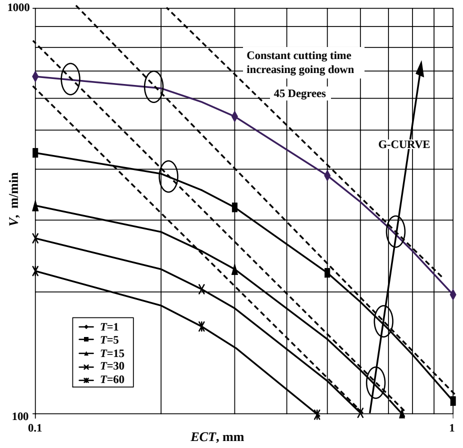


Fig. 21. Constant cutting time in the *V-ECT* plane, tool-life constant

Global Optimum, Mathematical Method.—Global optimization is the search for extremum of C_{TOT} for the three parameters: *T*, *ECT*, and *V*. The results, in terms of the tool-life equation constants, are:

Optimum tool-life:

$$T_O = T_V \times \left(\frac{1}{n_O} - 1\right)$$

$$n_O = 2M \times (L \times \ln T_O)^2 + 1 - N_0 + L \times (2M + H)$$

where n_O = slope at optimum *ECT*.

The same approach is used when searching for maximum production rate, but without the term containing tooling cost.

Optimum cutting speed:

$$V_O = e^{-M + K + (H \times L - N_0) \times \ln T_O + M \times L^2 \times (\ln T_O)^2}$$

Optimum *ECT*:

$$ECT_O = e^{H + 2M \times (L \times \ln(T_O) + 1)}$$

Global optimum is not reached when face milling for very large feeds, and C_{TOT} decreases continually with increasing feed/tooth, but can be reached for a cutter with many teeth, say 20 to 30. In end milling, global optimum can often be achieved for big feeds and for 3 to 8 teeth.

Determination Of Machine Settings And Calculation Of Costs

Based on the rules and knowledge presented in Chapters 1 and 2, this chapter demonstrates, with examples, how machining times and costs are calculated.

Additional formulas are given, and the speed and feed tables given in *SPEED AND FEED TABLES* starting on page 991 should be used. Finally the selection of feeds, speeds and tool-lives for optimized conditions are described with examples related to turning, end milling, and face milling.

There are an infinite number of machine settings available in the machine tool power train producing widely different results. In practice only a limited number of available settings are utilized. Often, feed is generally selected independently of the material being cut, however, the influence of material is critical in the choice of cutting speed. The tool-life is normally not known or directly determined, but the number of pieces produced before the change of worn tools is better known, and tool-life can be calculated using the formula for piece cutting time t_c given in this chapter.

It is well known that increasing feeds or speeds reduces the number of pieces cut between tool changes, but not how big are the changes in the basic parameter tool-life. Therefore, there is a tendency to select "safe" data in order to get a long tool-life. Another common practice is to search for a tool grade yielding a longer life using the current speeds and feeds, or a 10–20% increase in cutting speed while maintaining the current tool-life. The reason for this old-fashioned approach is the lack of knowledge about the opportunities the metal cutting process offers for increased productivity.

For example, when somebody wants to calculate the cutting time, he/she can select a value of the feed rate (product of feed and rpm), and easily find the cutting time by dividing cutting distance by the feed rate. The number of pieces obtained out of a tool is a guesswork, however. This problem is very common and usually the engineers find desired tool-lives after a number of trial and error runs using a variety of feeds and speeds. If the user is not well familiar with the material cut, the tool-life obtained could be any number of seconds or minutes, or the cutting edge might break.

There are an infinite number of feeds and speeds, giving the same feed rate, producing equal cutting time. The same cutting time per piece t_c is obtained independent of the selection of feed/rev f and cutting speed V, (or rpm), as long as the feed rate F_R remains the same: $F_R = f_1 \times \text{rpm}_1 = f_2 \times \text{rpm}_2 = f_3 \times \text{rpm}_3 \ldots$, etc. However, the number of parts before tool change N_{ch} will vary considerably including the tooling cost c_{tool} and the total cutting cost c_{tot}.

The dilemma confronting the machining-tool engineer or the process planner is how to set feeds and speeds for either desired cycle time, or number of parts between tool changes, while balancing the process versus other operations or balancing the total times in one cell with another. These problems are addressed in this section.

Nomenclature

f = feed/rev or tooth, mm f_E = economic feed f_O = optimum feed
T = tool-life, minutes T_E = economic tool-life T_O = optimum tool-life
V = cutting speed, m/min V_E = economic cutting speed V_O = optimum cutting speed, m/min

Similarly, economic and optimum values of:

c_{tool} = piece cost of tooling, \$ C_{TOOL} = cost of tooling per batch, \$
c_{tot} = piece total cost of cutting, \$ C_{TOT} = total cost of cutting per batch, \$
F_R = feed rate measured in the feeding direction, mm/rev
N = batch size
N_{ch} = number of parts before tool change
t_c = piece cutting time, minutes T_C = cutting time per batch, minutes
t_{cyc} = piece cycle time, minutes T_{CYC} = cycle time before tool change, minutes

t_i = idle time (tool “air” motions during cycle), minutes
z = cutter number of teeth

The following variables are used for calculating the per batch cost of cutting:

C_C = cost of cutting time per batch, $
C_{CH} = cost of tool changes per batch, $
C_E = cost per edge, for replacing or regrinding, $
H_R = hourly rate, $
T_V = equivalent tooling-cost time, minutes
T_{RPL} = time for replacing worn edge(s), or tool for regrinding, minutes

Note: In the list above, when two variables use the same name, one in capital letters and one lower case, T_C and t_c for example, the variable name in capital letters refers to batch processing and lowercase letters to per piece processing, such as $T_C = N_{ch} \times t_c$, $C_{TOT} = N_{ch} \times c_{tot}$, etc.

Formulas Valid For All Operation Types Including Grinding

Calculation of Cutting Time and Feed Rate

Feed Rate:

$F_R = f \times \text{rpm}$ (mm/min), where f is the feed in mm/rev along the feeding direction, rpm is defined in terms of work piece or cutter diameter D in mm, and cutting speed V in m/min, as follows:

$$\text{rpm} = \frac{1000V}{\pi D} = \frac{318V}{D}$$

Cutting time per piece:

Note: Constant cutting time is a straight 45-degree line in the *V–ECT* graph, along which tool-life varies considerably, as is shown in Chapter 2.

$$t_c = \frac{Dist}{F_R} = \frac{Dist}{f \times \text{rpm}} = \frac{Dist \times \pi D}{1000V \times f}$$

where the units of distance cut *Dist*, diameter *D*, and feed *f* are mm, and *V* is in m/min.

In terms of *ECT*, cutting time per piece, t_c, is as follows:

$$t_c = \frac{Dist \times \pi D}{1000V} \times \frac{a}{CEL \times ECT}$$

where a = depth of cut, because feed × cross sectional chip area = $f \times a = CEL \times ECT$.

Example 3, Cutting Time: Given $Dist$ = 105 mm, D = 100 mm, f = 0.3 mm, V = 300 m/min, rpm = 700, F_R = 210 mm/min, find the cutting time.

Cutting time = $t_c = 105 \times 3.1416 \times 100 \div (1000 \times 300 \times 0.3) = 0.366$ minutes = 22 seconds

Scheduling of Tool Changes

Number of parts before tool change:

$N_{ch} = T \div t_c$

Cycle time before tool change:

$T_{CYC} = N_{ch} \times (t_c + t_i)$, where $t_{cyc} = t_c + t_i$, where t_c = cutting time per piece, t_i = idle time per piece

Tool-life:

$T = N_{ch} \times t_c$

Example 4: Given tool-life T = 90 minutes, cutting time t_c = 3 minutes, and idle time t_i = 3 minutes, find the number of parts produced before a tool change is required and the time until a tool change is required.

Number of parts before tool change = $N_{ch} = 90/3 = 30$ parts.

Cycle time before tool change = $T_{CYC} = 30 \times (3 + 3) = 180$ minutes

Example 5: Given cutting time, $t_c = 1$ minute, idle time $t_i = 1$ minute, $N_{ch} = 100$ parts, calculate the tool-life T required to complete the job without a tool change, and the cycle time before a tool change is required.

Tool-life = $T = N_{ch} \times t_c = 100 \times 1 = 100$ minutes.

Cycle time before tool change = $T_{CYC} = 100 \times (1 + 1) = 200$ minutes.

Calculation of Cost of Cutting and Grinding Operations.—When machining data varies, the cost of cutting, tool changing, and tooling will change, but the costs of idle and slack time are considered constant.

Cost of Cutting per Batch:

$C_C = H_R \times T_C/60$

T_C = cutting time per batch = (number of parts) × t_c, minutes, or when determining time for tool change $T_{Cch} = N_{ch} \times t_c$ minutes = cutting time before tool change.

t_c = Cutting time/part, minutes

H_R = Hourly Rate

Cost of Tool Changes per Batch:

$$C_{CH} = \frac{H_R}{60} \times T_C \times \frac{T_{RPL}}{T} \qquad \frac{\$}{min} \cdot min = \$$$

where T = tool-life, minutes, and T_{RPL} = time for replacing a worn edge(s), or tool for regrinding, minutes

Cost of Tooling per Batch:

Including cutting tools and holders, but without tool changing costs,

$$C_{TOOL} = \frac{H_R}{60} \times T_C \times \frac{\dfrac{60 C_E}{H_R}}{T} \qquad \frac{\$}{min} \cdot min \cdot \frac{\dfrac{min}{hr} \cdot \$ \cdot \dfrac{hr}{\$}}{min} = \$$$

Cost of Tooling + Tool Changes per Batch:

Including cutting tools, holders, and tool changing costs,

$$(C_{TOOL} + C_{CH}) = \frac{H_R}{60} \times T_C \times \frac{T_{RPL} + \dfrac{60 C_E}{H_R}}{T}$$

Total Cost of Cutting per Batch:

$$C_{TOT} = \frac{H_R}{60} \times T_C \left(1 + \frac{T_{RPL} + \dfrac{60 C_E}{H_R}}{T} \right)$$

Equivalent Tooling-cost Time, T_V:

The two previous expressions can be simplified by using $T_V = T_{RPL} + \dfrac{60 C_E}{H_R}$

thus:

$$(C_{TOOL} + C_{CH}) = \frac{H_R}{60} \times T_C \times \frac{T_V}{T}$$

$$C_{TOT} = \frac{H_R}{60} \times T_C\left(1 + \frac{T_V}{T}\right)$$

C_E = cost per edge(s) is determined using two alternate formulas, depending on whether tools are reground or inserts are replaced:

Cost per Edge, Tools for Regrinding

$$C_E = \frac{\text{cost of tool} + (\text{number of regrinds} \times \text{cost/regrind})}{1 + \text{number of regrinds}}$$

Cost per Edge, Tools with Inserts:

$$C_E = \frac{\text{cost of insert(s)}}{\text{number of edges per insert}} + \frac{\text{cost of cutter body}}{\text{cutter body life in number of edges}}$$

Note: In practice allow for insert failures by multiplying the insert cost by 4/3, that is, assuming only 3 out of 4 edges can be effectively used.

Example 6, Cost per Edge–Tools for Regrinding: Use the data in the table below to calculate the cost per edge(s) C_E, and the equivalent tooling-cost time T_V, for a drill.

Time for cutter replacement T_{RPL}, minute	Cutter Price, $	Cost per regrind, $	Number of regrinds	Hourly shop rate, $	Batch size	Taylor slope, n	Economic cutting time, t_{cE} minute
1	40	6	5	50	1000	0.25	1.5

Using the cost per edge formula for reground tools, $C_E = (40 + 5 \times 6) \div (1 + 5) = \$6.80$

When the hourly rate is \$50/hr, $T_V = T_{RPL} + \frac{60C_E}{H_R} = 1 + \frac{60(6.8)}{50} = 9.16\text{ minutes}$

Calculate economic tool-life using $T_E = T_V \times \left(\frac{1}{n} - 1\right)$ thus, $T_E = 9.17 \times (1/0.25 - 1) = 9.16 \times 3 = 27.48$ minutes.

Having determined, elsewhere, the economic cutting time per piece to be t_{cE} = 1.5 minutes, for a batch size = 1000 calculate:

Cost of Tooling + Tool Change per Batch:

$$(C_{TOOL} + C_{CH}) = \frac{H_R}{60} \times T_C \times \frac{T_V}{T} = \frac{50}{60} \times 1000 \times 1.5 \times \frac{9.16}{27.48} = \$\ 417$$

Total Cost of Cutting per Batch:

$$C_{TOT} = \frac{H_R}{60} \times T_C\left(1 + \frac{T_V}{T}\right) = \frac{50}{60} \times 1000 \times 1.5 \times \left(1 + \frac{9.16}{27.48}\right) = \$\ 1617$$

Example 7, Cost per Edge–Tools with Inserts: Use data from the table below to calculate the cost of tooling and tool changes, and the total cost of cutting.

For face milling, multiply insert price by safety factor 4/3 then calculate the cost per edge: $C_E = 10 \times (5/3) \times (4/3) + 750/500 = 23.72$ per set of edges

When the hourly rate is \$50, equivalent tooling-cost time is $T_V = 2 + 23.72 \times 60/50 = 30.466$ minutes (first line in table below). The economic tool-life for Taylor slope n = 0.333 would be $T_E = 30.466 \times (1/0.333 - 1) = 30.466 \times 2 = 61$ minutes.

When the hourly rate is \$25, equivalent tooling-cost time is $T_V = 2 + 23.72 \times 60/25 = 58.928$ minutes (second line in table below). The economic tool-life for Taylor slope n = 0.333 would be $T_E = 58.928 \times (1/0.333 - 1) = 58.928 \times 2 = 118$ minutes.

Time for replacement of inserts T_{RPL}, minutes	Number of inserts	Price per insert	Edges per insert	Cutter Price	Edges per cutter	Cost per set of edges, C_E	Hourly shop rate	T_V minutes
Face mill								
2	10	5	3	750	500	23.72	50	30.466
2	10	5	3	750	500	23.72	25	58.928
End mill								
1	3	6	2	75	200	4.375	50	6.25
Turning								
1	1	5	3	50	100	2.72	30	6.44

With above data for the face mill, and after having determined the economic cutting time as $t_{cE} = 1.5$ minutes, calculate for a batch size = 1000 and $50 per hour rate:

Cost of Tooling + Tool Change per Batch:

$$(C_{TOOL} + C_{CH}) = \frac{H_R}{60} \times T_C \times \frac{T_V}{T} = \frac{50}{60} \times 1000 \times 1.5 \times \frac{30.466}{61} = \$\ 624$$

Total Cost of Cutting per Batch:

$$C_{TOT} = \frac{H_R}{60} \times T_C\left(1 + \frac{T_V}{T}\right) = \frac{50}{60} \times 1000 \times 1.5 \times \left(1 + \frac{30.466}{61}\right) = \$\ 1874$$

Similarly, at the $25/hour shop rate, $(C_{TOOL} + C_{CH})$ and C_{TOT} are $312 and $937, respectively.

Example 8, Turning: Production parts were run in the shop at feed/rev = 0.25 mm. One series was run with speed $V_1 = 200$ m/min and tool-life was $T_1 = 45$ minutes. Another was run with speed $V_2 = 263$ m/min and tool-life was $T_2 = 15$ minutes. Given idle time $t_i = 1$ minute, cutting distance $Dist = 1000$ mm, work diameter $D = 50$ mm.

First, calculate Taylor slope, *n*, using Taylor's equation $V_1 \times T_1^n = V_2 \times T_2^n$, as follows:

$$n = \ln\frac{V_1}{V_2} \div \ln\frac{T_2}{T_1} = \ln\frac{200}{263} \div \ln\frac{15}{45} = 0.25$$

Economic tool-life T_E is next calculated using the equivalent tooling-cost time T_V, as described previously. Assuming a calculated value of $T_V = 4$ minutes, then T_E can be calculated from

$$T_E = T_V \times \left(\frac{1}{n} - 1\right) = 4 \times \left(\frac{1}{0.25} - 1\right) = 12\ \text{minutes}$$

Economic cutting speed, V_E can be found using Taylor's equation again, this time using the economic tool-life, as follows,

$$V_{E1} \times (T_E)^n = V_2 \times (T_2)^n$$

$$V_{E1} = V_2 \times \left(\frac{T_2}{T_E}\right)^n = 263 \times \left(\frac{15}{12}\right)^{0.25} = 278\ \text{m/min}$$

Using the process data, the remaining economic parameters can be calculated as follows:

Economic spindle rpm, $\text{rpm}_E = (1000V_E)/(\pi D) = (1000 \times 278)/(3.1416 \times 50) = 1770$ rpm

Economic feed rate, $F_{RE} = f \times \text{rpm}_E = 0.25 \times 1770 = 443$ mm/min

Economic cutting time, $t_{cE} = Dist/F_{RE} = 1000/443 = 2.259$ minutes

Economic number of parts before tool change, $N_{chE} = T_E \div t_{cE} = 12 \div 2.259 = 5.31$ parts

Economic cycle time before tool change, $T_{CYCE} = N_{chE} \times (t_c + t_i) = 5.31 \times (2.259 + 1) = 17.3$ minutes.

Variation Of Tooling And Total Cost With The Selection Of Feeds And Speeds

It is a well-known fact that tool-life is reduced when either feed or cutting speed is increased. When a higher feed/rev is selected, the cutting speed must be decreased in order to maintain tool-life. However, a higher feed rate (feed rate = feed/rev × rpm, mm/min) can result in a longer tool-life if proper cutting data are applied. Optimized cutting data require accurate machinability databases and a computer program to analyze the options. Reasonably accurate optimized results can be obtained by selecting a large feed/rev or tooth, and then calculating the economic tool-life T_E. Because the cost versus feed or *ECT* curve is shallow around the true minimum point, i.e., the global optimum, the error in applying a large feed is small compared with the exact solution.

Once a feed has been determined, the economic cutting speed V_E can be found by calculating the Taylor slope, and the time/cost calculations can be completed using the formulas described in last section.

The remainder of this section contains examples useful for demonstrating the required procedures. Global optimum may or may not be reached, and tooling cost may or may not be reduced, compared to currently used data. However, the following examples prove that significant time and cost reductions are achievable in today's industry.

Note: Starting values of reasonable feeds in mm/rev can be found in the Handbook speed and feed tables, see *Principal Feeds and Speeds Tables* on page 991, by using the f_{avg} values converted to mm as follows: feed (mm/rev) = feed (inch/rev) × 25.4 (mm/inch), thus 0.001 inch/rev = 0.001× 25.4 = 0.0254 mm/rev. When using speed and feed Tables 1 through 23, where feed values are given in thousandths of inch per revolution, simply multiply the given feed by 25.4/1000 = 0.0254, thus feed (mm/rev) = feed (0.001 inch/rev) × 0.0254 (mm/ 0.001inch).

Example 9, Converting Handbook Feed Values From Inches to Millimeters: Handbook tables give feed values f_{opt} and f_{avg} for 4140 steel as 17 and 8 × (0.001 inch/rev) = 0.017 and 0.009 inch/rev, respectively. Convert the given feeds to mm/rev.

feed = 0.017 × 25.4 = 17 × 0.0254 = 0.4318 mm/rev

feed = 0.008 × 25.4 = 9 × 0.0254 = 0.2032 mm/rev

Example 10, Using Handbook Tables to Find the Taylor Slope and Constant: Calculate the Taylor slope and constant, using cutting speed data for 4140 steel in Table 1 starting on page 996, and for ASTM Class 20 grey cast iron using data from Table 4a on page 1002, as follows:

For the 175–250 Brinell hardness range, and the hard tool grade,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)} = \frac{\ln(525 / 705)}{\ln(15 / 45)} = 0.27 \qquad \mathrm{C} = V_1 \times (T_1)^n = 1467$$

For the 175–250 Brinell hardness range, and the tough tool grade,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)} = \frac{\ln(235 / 320)}{\ln(15 / 45)} = 0.28 \qquad \mathrm{C} = V_1 \times (T_1)^n = 1980$$

For the 300–425 Brinell hardness range, and the hard tool grade,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)} = \frac{\ln(330 / 440)}{\ln(15 / 45)} = 0.26 \qquad \mathrm{C} = V_1 \times (T_1)^n = 2388$$

For the 300–425 Brinell hardness range, and the tough tool grade,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)} = \frac{\ln(125 / 175)}{\ln(15 / 45)} = 0.31 \qquad \mathrm{C} = V_1 \times (T_1)^n = 1324$$

For ASTM Class 20 grey cast iron, using hard ceramic,

$$n = \frac{\ln(V_1 / V_2)}{\ln(T_2 / T_1)} = \frac{\ln(1490 / 2220)}{\ln(15 / 45)} = 0.36 \qquad \mathrm{C} = V_1 \times (T_1)^n = 5932$$

Selection of Optimized Data.—Fig. 22 illustrates cutting time, cycle time, number of parts before a tool change, tooling cost, and total cost, each plotted versus feed for a constant tool-life. Approximate minimum cost conditions can be determined using the formulas previously given in this section.

First, select a large feed/rev or tooth, and then calculate economic tool-life T_E, and the economic cutting speed V_E, and do all calculations using the time/cost formulas as described previously.

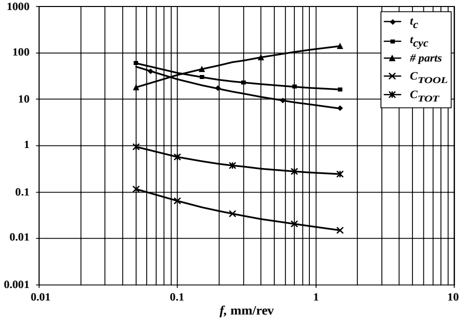


Fig. 22. Cutting time, cycle time, number of parts before tool change, tooling cost, and total cost vs. feed for tool-life = 15 minutes, idle time = 10 s, and batch size = 1000 parts

Example 11, Step by Step Procedure: Turning – Facing out: 1) Select a big feed/rev, in this case $f = 0.9$ mm/rev (0.035 inch/rev). A Taylor slope n is first determined using the Handbook tables and the method described in Example 10. In this example, use $n = 0.35$.

2) Calculate T_V from the tooling cost parameters:

If cost of insert = \$7.50; edges per insert = 2; cost of tool holder = \$100; life of holder = 100 insert sets; and for tools with inserts, allowance for insert failures = cost per insert by 4/3, assuming only 3 out of 4 edges can be effectively used.

Then, cost per edge = C_E is calculated as follows:

$$C_E = \frac{\text{cost of insert(s)}}{\text{number of edges per insert}} + \frac{\text{cost of cutter body}}{\text{cutter body life in number of edges}}$$

$$= \frac{7.50}{4 / 3 \times 2} + \frac{100}{100} = \$6.00$$

The time for replacing a worn edge of the facing insert $= T_{RPL} = 2.24$ minutes. Assuming an hourly rate $H_R = \$50$/hour, calculate the equivalent tooling-cost time T_V

$$T_V = T_{RPL} + 60 \times C_E / H_R = 2.24 + 60 \times 6/50 = 8.24 \text{ minutes.}$$

3) Determine economic tool-life T_E

$$T_E = T_V \times (1/n - 1) = T_E = T_V \times (1/n - 1) = 8.24 \times (1/\,0.35 - 1) = 15 \text{ minutes}$$

4) Determine economic cutting speed using the Handbook tables using the method shown in Example 10,

$$V_E = \mathrm{C} \times T_E{}^n \text{ m/min} = \mathrm{C} \times T_E{}^n = 280 \times 15^{-0.35} = 109 \text{ m/min}$$

5) Determine cost of tooling per batch (cutting tools, holders and tool changing) then total cost of cutting per batch:

$$C_{TOOL} = H_R \times T_C \times (C_E / T)/60$$

$$(C_{TOOL}+C_{CH}) = H_R \times T_C \times ((T_{RPL}+C_E/T)/60$$
$$C_{TOT} = H_R \times T_C(1 + (T_{RPL}+C_E)/T).$$

Example 12, Face Milling – Minimum Cost : This example demonstrates how a modern firm, using the formulas previously described, can determine optimal data. It is here applied to a face mill with 10 teeth, milling a 1045 type steel, and the radial depth versus the cutter diameter is 0.8. The *V–ECT–T* curves for tool-lives 5, 22, and 120 minutes for this operation are shown in Fig. 23a.

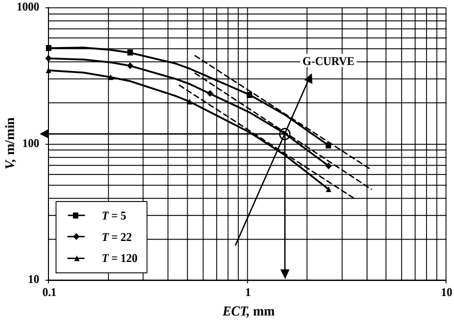


Fig. 23a. Cutting speed vs. ECT, tool-life constant

The global cost minimum occurs along the *G*-curve, see Fig. 6c and Fig. 23a, where the 45-degree lines defines this curve. Optimum *ECT* is in the range 1.5 to 2 mm.

For face and end milling operations, $ECT = z \times f_z \times ar/D \times aa/CEL \div \pi$. The ratio aa/CEL = 0.95 for lead angle $LA = 0$, and for $ar/D = 0.8$ and 10 teeth, using the formula to calculate the feed/tooth range gives for $ECT = 1.5$, $f_z = 0.62$ mm and for $ECT = 2$, $f_z = 0.83$ mm.

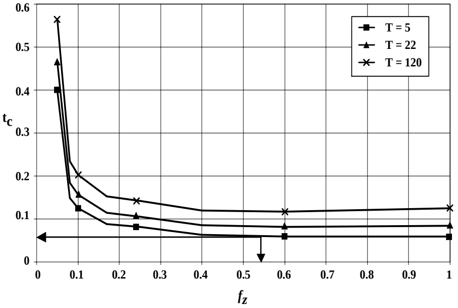


Fig. 23b. Cutting time per part vs. feed per tooth

Using computer simulation, the minimum cost occurs approximately where Fig. 23a indicates it should be. Total cost has a global minimum at f_z around 0.6 to 0.7 mm and a speed of around 110 m/min. *ECT* is about 1.9 mm and the optimal cutter life is $T_O = 22$ minutes. Because it may be impossible to reach the optimum feed value due to tool breakage, the maximum practical feed f_{max} is used as the optimal value. The difference in costs between a global optimum and a practical minimum cost condition is negligible, as shown in Figs. 23c and 23e. A summary of the results are shown in Figs. 23a through 23e, and Table 1.

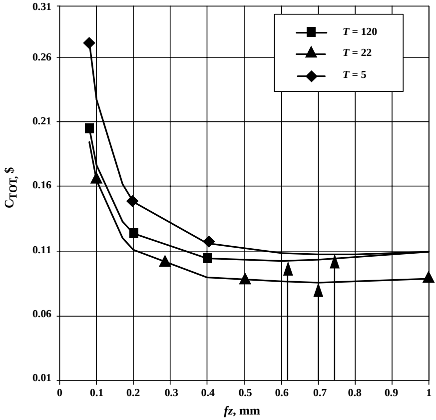


Fig. 23c. Total cost vs. feed/tooth

When plotting cutting time/part, t_c, versus feed/tooth, f_z, at $T = 5$, 22, 120 in Figs. 23b, tool-life $T = 5$ minutes yields the shortest cutting time, but total cost is the highest; the minimum occurs for f_z about 0.75 mm, see Figs. 23c. The minimum for $T = 120$ minutes is about 0.6 mm and for $T_O = 22$ minutes around 0.7 mm.

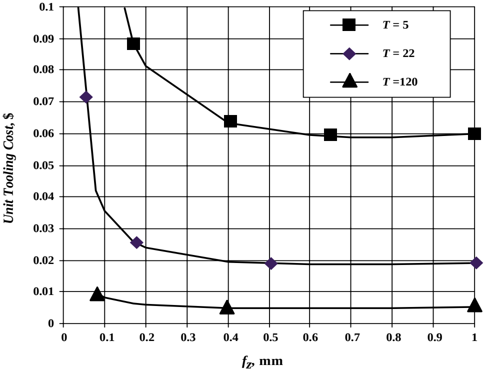


Fig. 23d. Tooling cost versus feed/tooth

Fig. 23d shows that tooling cost drop off quickly when increasing feed from 0.1 to 0.3 to 0.4 mm, and then diminishes slowly and is almost constant up to 0.7 to 0.8 mm/tooth. It is generally very high at the short tool-life 5 minutes, while tooling cost of optimal tool-life 22 minutes is about 3 times higher than when going slow at $T = 120$ minutes.

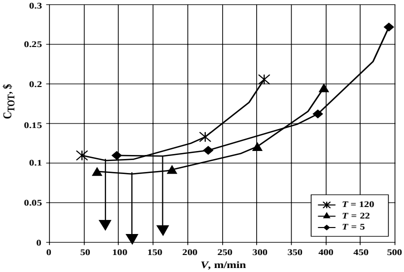


Fig. 23e. Total cost vs. cutting speed at 3 constant tool-lives, feed varies

The total cost curves in Fig. 24e. were obtained by varying feed and cutting speed in order to maintain constant tool-lives at 5, 22 and 120 minutes. Cost is plotted as a function of speed V instead of feed/tooth. Approximate optimum speeds are $V = 150$ m/min at $T = 5$ minutes, $V = 180$ m/min at $T = 120$ minutes, and the global optimum speed is $V_O = 110$ m/min for $T_O = 22$ minutes.

Table 1 displays the exact numerical values of cutting speed, tooling cost and total cost for the selected tool-lives of 5, 22, and 120 minutes, obtained from the software program.

Table 1. Face Milling, Total and Tooling Cost versus *ECT*, Feed/tooth *fz*, and Cutting Speed *V*, at Tool-lives 5, 22, and 120 minutes

		T = 5 minutes			*T* = 22 minutes			*T* = 120 minutes		
fz	*ECT*	*V*	C_{TOT}	C_{TOOL}	*V*	C_{TOT}	C_{TOOL}	*V*	C_{TOT}	C_{TOOL}
0.03	0.08	489	0.72891	0.39759	416	0.49650	0.10667	344	0.49378	0.02351
0.08	0.21	492	0.27196	0.14834	397	0.19489	0.04187	311	0.20534	0.00978
0.10	0.26	469	0.22834	0.12455	374	0.16553	0.03556	289	0.17674	0.00842
0.17	0.44	388	0.16218	0.08846	301	0.12084	0.02596	225	0.13316	0.00634
0.20	0.51	359	0.14911	0.08133	276	0.11204	0.02407	205	0.12466	0.00594
0.40	1.03	230	0.11622	0.06339	171	0.09051	0.01945	122	0.10495	0.00500
0.60	1.54	164	0.10904	0.05948	119	0.08672	0.01863	83	0.10301	0.00491
0.70	1.80	141	0.10802	0.05892	102	0.08665	0.01862	70	0.10393	0.00495
0.80	2.06	124	0.10800	0.05891	89	0.08723	0.01874	60	0.10547	0.00502
1.00	2.57	98	0.10968	0.05982	69	0.08957	0.01924	47	0.10967	0.00522

High-speed Machining Econometrics

High-speed Machining – No Mystery.—This section describes the theory and gives the basic formulas for any milling operation and high-speed milling in particular, followed by several examples on high-speed milling econometrics. These rules constitute the basis on which selection of milling feed factors is done. Selection of cutting speeds for general milling is done using the Handbook Table 10 through 14, starting on page 1013.

High-speed machining is no mystery to those having a good knowledge of metal cutting. Machining materials with very good machinability, such as low-alloyed aluminum, have for ages been performed at cutting speeds well below the speed values at which these materials should be cut. Operating at these low speeds often results in built-up edges and poor surface finish, because the operating conditions selected are on the wrong side of the Taylor curve, i.e. to the left of the *H*-curve representing maximum tool-life values (see Fig. 4 on page 1059).

In the 1950's it was discovered that cutting speed could be raised by a factor of 5 to 10 when hobbing steel with HSS cutters. This is another example of being on the wrong side of the Taylor curve.

One of the first reports on high-speed end milling using high-speed steel (HSS) and carbide cutters for milling 6061-T651 and A356-T6 aluminum was reported in a study funded by Defense Advanced Research Project Agency (DARPA). Cutting speeds of up to 4400 m/min (14140 fpm) were used. Maximum tool-lives of 20 through 40 minutes were obtained when the feed/tooth was 0.2 through 0.25 mm (0.008 to 0.01 inch), or measured in terms of *ECT* around 0.07 to 0.09 mm. Lower or higher feed/tooth resulted in shorter cutter lives. The same types of previously described curves, namely *T–ECT* curves with maximum tool-life along the *H*-curve, were produced.

When examining the influence of *ECT*, or feed/rev, or feed/tooth, it is found that too small values cause chipping, vibrations, and poor surface finish. This is caused by inadequate (too small) chip thickness, and as a result the material is not cut but plowed away or scratched, due to the fact that operating conditions are on the wrong (left) side of the tool-life versus *ECT* curve (*T-ECT* with constant speed plotted).

There is a great difference in the thickness of chips produced by a tooth traveling through the cutting arc in the milling process, depending on how the center of the cutter is placed in relation to the workpiece centerline, in the feed direction. Although end and face milling cut in the same way, from a geometry and kinematics standpoint they are in practice distinguished by the cutter center placement away from, or close to, the work centerline, respectively, because of the effect of cutter placement on chip thickness. This is the criteria used to distinguishing between the end and face milling processes in the following.

Depth of Cut/Cutter Diameter, ar/D is the ratio of the radial depth of cut *ar* and the cutter diameter *D*. In face milling when the cutter axis points approximately to the middle of the work piece axis, eccentricity is close to zero, as illustrated in Figs. 3 and 4, page 1011, and Fig. 5 on page 1012. In end milling, $ar/D = 1$ for full slot milling.

Mean Chip Thickness, hm is a key parameter that is used to calculate forces and power requirements in high-speed milling. If the mean chip thickness *hm* is too small, which may occur when feed/tooth is too small (this holds for all milling operations), or when *ar*/*D* decreases (this holds for ball nose as well as for straight end mills), then cutting occurs on the left (wrong side) of the tool-life versus *ECT* curve, as illustrated in Figs. 6b and 6c.

In order to maintain a given chip thickness in end milling, the feed/tooth has to be increased, up to 10 times for very small *ar*/*D* values in an extreme case with no run out and otherwise perfect conditions. A 10 times increase in feed/tooth results in 10 times bigger feed rates (F_R) compared to data for full slot milling (valid for $ar/D = 1$), yet maintain a given chip thickness. The cutter life at any given cutting speed will not be the same, however.

Increasing the number of teeth from say 2 to 6 increases equivalent chip thickness *ECT* by a factor of 3 while the mean chip thickness *hm* remains the same, but does not increase the feed rate to 30 (3×10) times bigger, because the cutting speed must be reduced. However, when the *ar*/*D* ratio matches the number of teeth, such that one tooth enters when the second tooth leaves the cutting arc, then $ECT = hm$. Hence, *ECT* is proportional to the number of teeth. Under ideal conditions, an increase in number of teeth *z* from 2 to 6 increases the feed rate by, say, 20 times, maintaining tool-life at a reduced speed. In practice about 5 times greater feed rates can be expected for small *ar*/*D* ratios (0.01 to 0.02), and up to 10 times with 3 times as many teeth. So, high-speed end milling is no mystery.

Chip Geometry in End and Face Milling.—Fig. 24 illustrates how the chip forming process develops differently in face and end milling, and how mean chip thickness *hm* varies with the angle of engagement *AE*, which depends on the *ar*/*D* ratio. The pertinent chip geometry formulas are given in the text that follows.

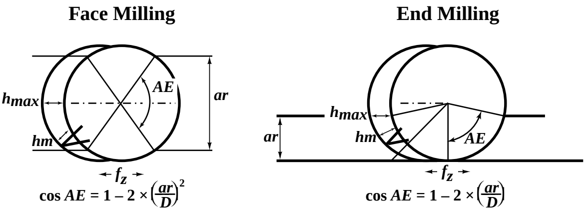


Fig. 24.

Comparison of face milling and end milling geometry

High-speed end milling refers to values of *ar*/*D* that are less than 0.5, in particular to *ar*/*D* ratios which are considerably smaller. When *ar*/*D* = 0.5 (*AE* = 90 degrees) and diminishing in end milling, the chip thickness gets so small that poor cutting action develops, including plowing or scratching. This situation is remedied by increasing the feed/tooth, as shown in Table 2a as an increasing f_z/f_{z0} ratio with decreasing *ar*/*D*. For end milling, the f_z/f_{z0} feed ratio is 1.0 for *ar*/*D* = 1 and also for *ar*/*D* = 0.5. In order to maintain the same *hm* as at *ar*/*D* = 1, the feed/tooth should be increased, by a factor of 6.38 when *ar*/*D* is 0.01 and by more than 10 when *ar*/*D* is less than 0.01. Hence high-speed end milling could be said to begin when *ar*/*D* is less than 0.5

In end milling, the ratio f_z/f_{z0} = 1 is set at *ar*/*D* = 1.0 (full slot), a common value in vendor catalogs and handbooks, for *hm* = 0.108 mm.

The face milling chip making process is exactly the same as end milling when face milling the side of a work piece and *ar*/*D* = 0.5 or less. However, when face milling close to and along the work centerline (eccentricity is close to zero) chip making is quite different, as shown in Fig. 24. When *ar*/*D* = 0.74 (*AE* = 95 degrees) in face milling, the f_z/f_{z0} ratio = 1 and increases up to 1.4 when the work width is equal to the cutter diameter (*ar*/*D* = 1). The face milling f_z/f_{z0} ratio continues to diminish when the *ar*/*D* ratio decreases below *ar*/*D* = 0.74, but very insignificantly, only about 11 percent when *ar*/*D* = 0.01.

In face milling f_z/f_{z0} = 1 is set at *ar*/*D* = 0.74, a common value recommended in vendor catalogs and handbooks, for *hm* = 0.151 mm.

Fig. 25 shows the variation of the feed/tooth-ratio in a graph for end and face milling.

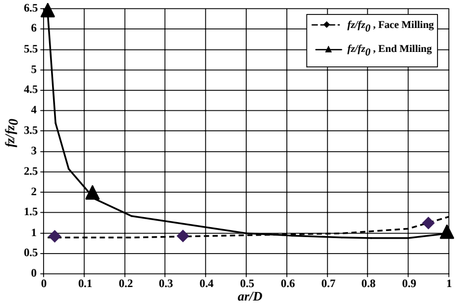


Fig. 25. Feed/tooth versus *ar*/*D* for face and end milling

Table 2a. Variation of Chip Thickness and f_z/f_{z0} with ar/D

	Face Milling					End Milling (straight)				
	ecentricity $e = 0$ $z = 8$ $f_{z0} = 0.017$ $\cos AE = 1 - 2 \times (ar/D)^2$					$z = 2$ $f_{z0} = 0.017$ $\cos AE = 1 - 2 \times (ar/D)$				
ar/D	AE	hm/f_z	hm	ECT/hm	f_z/f_{z0}	AE	hm/f_z	hm	ECT/hm	f_z/f_{z0}
1.0000	180.000	0.637	0.108	5.000	1.398	180.000	0.637	0.108	1.000	1.000
0.9000	128.316	0.804	0.137	3.564	1.107	143.130	0.721	0.122	0.795	0.884
0.8000	106.260	0.863	0.147	2.952	1.032	126.870	0.723	0.123	0.711	0.881
0.7355	94.702	0.890	0.151	2.631	1.000	118.102	0.714	0.122	0.667	0.892
0.6137	75.715	0.929	0.158	1.683	0.958	103.144	0.682	0.116	0.573	0.934
0.5000	60.000	0.162	0.932	0.216	0.202	90.000	0.674	0.115	0.558	1.000
0.3930	46.282	0.973	0.165	1.028	0.915	77.643	0.580	0.099	0.431	1.098
0.2170	25.066	0.992	0.169	0.557	0.897	55.528	0.448	0.076	0.308	1.422
0.1250	14.361	0.997	0.170	0.319	0.892	41.410	0.346	0.059	0.230	1.840
0.0625	7.167	0.999	0.170	0.159	0.891	28.955	0.247	0.042	0.161	2.574
0.0300	3.438	1.000	0.170	0.076	0.890	19.948	0.172	0.029	0.111	3.694
0.0100	1.146	1.000	0.170	0.025	0.890	11.478	0.100	0.017	0.064	6.377
0.0010	0.115	1.000	0.000	0.000	0.890	3.624	0.000	0.000	0.000	20.135

In Table 2a, a standard value $f_{z0} = 0.17$ mm/tooth (commonly recommended average feed) was used, but the f_z/f_{z0} values are independent of the value of feed/tooth, and the previously mentioned relationships are valid whether $f_{z0} = 0.17$ or any other value.

In both end and face milling, $hm = 0.108$ mm for $f_{z0} = 0.17$mm when $ar/D = 1$. When the f_z/f_{z0} ratio = 1, $hm = 0.15$ for face milling, and 0.108 in end milling both at $ar/D = 1$ and 0.5. The tabulated data hold for perfect milling conditions, such as, zero run-out and accurate sharpening of all teeth and edges.

Mean Chip Thickness *hm* and Equivalent Chip Thickness *ECT*.—The basic formula for equivalent chip thickness *ECT* for any milling process is:

$ECT = f_z \times z/\pi \times (ar/D) \times aa/CEL$, where f_z = feed/tooth, z = number of teeth, D = cutter diameter, ar = radial depth of cut, aa = axial depth of cut, and CEL = cutting edge length. As a function of mean chip thickness *hm*:

$ECT = hm \times (z/2) \times (AE/180)$, where AE = angle of engagement.

Both terms are exactly equal when one tooth engages as soon as the preceding tooth leaves the cutting section. Mathematically, $hm = ECT$ when $z = 360/AE$; thus:

for face milling, $AE = \arccos(1 - 2 \times (ar/D)^2)$

for end milling, $AE = \arccos(1 - 2 \times (ar/D))$

Calculation of Equivalent Chip Thickness (ECT) versus Feed/tooth and Number of teeth.: Table 2b is a continuation of Table 2a, showing the values of *ECT* for face and end milling for decreasing values ar/D, and the resulting *ECT* when multiplied by the f_z/f_{z0} ratio $f_{z0} = 0.17$ (based on $hm = 0.108$).

Small ar/D ratios produce too small mean chip thickness for cutting chips. In practice, minimum values of *hm* are approximately 0.02 through 0.04 mm for both end and face milling.

Formulas.— Equivalent chip thickness can be calculated for other values of f_z and z by means of the following formulas:

Face milling: $ECT_F = ECT_{0F} \times (z/8) \times (f_z/0.17) \times (aa/CEL)$

or, if ECT_F is known calculate f_z using:

$f_z = 0.17 \times (ECT_F/ECT_{0F}) \times (8/z) \times (CEL/aa)$

Table 2b. Variation of *ECT*, Chip Thickness and f_z/f_{z0} with *ar*/*D*

	Face Milling				End Milling (straight)			
ar/*D*	*hm*	fz/f_{z0}	*ECT*	ECT_0 corrected for fz/f_{z0}	*hm*	fz/f_{z0}	*ECT*	ECT_0 corrected for fz/f_{z0}
1.0000	0.108	1.398	0.411	0.575	0.108	1.000	0.103	0.103
0.9000	0.137	1.107	0.370	0.410	0.122	0.884	0.093	0.082
0.8080	0.146	1.036	0.332	0.344	0.123	0.880	0.083	0.073
0.7360	0.151	1.000	0.303	0.303	0.121	0.892	0.076	0.067
0.6137	0.158	0.958	0.252	0.242	0.116	0.934	0.063	0.059
0.5900	0.159	0.952	0.243	0.231	0.115	0.945	0.061	0.057
0.5000	0.162	0.932	0.206	0.192	0.108	1.000	0.051	0.051
0.2170	0.169	0.897	0.089	0.080	0.076	1.422	0.022	0.032
0.1250	0.170	0.892	0.051	0.046	0.059	1.840	0.013	0.024
0.0625	0.170	0.891	0.026	0.023	0.042	2.574	0.006	0.017
0.0300	0.170	0.890	0.012	0.011	0.029	3.694	0.003	0.011
0.0100	0.170	0.890	0.004	0.004	0.017	6.377	0.001	0.007
0.0010	0.170	0.890	0.002	0.002	0.005	20.135	0.001	0.005

In face milling, the approximate values of $aa/CEL = 0.95$ for lead angle $LA = 0°$ (90° in the metric system); for other values of *LA*, $aa/CEL = 0.95 \times \sin(LA)$, and $0.95 \times \cos(LA)$ in the metric system.

Example, Face Milling: For a cutter with $D = 250$ mm and $ar = 125$ mm, calculate ECT_F for $f_z = 0.1$, $z = 12$, and $LA = 30$ degrees. First calculate $ar/D = 0.5$, and then use Table 2b and find $ECT_{0F} = 0.2$.

Calculate ECT_F with above formula:

$ECT_F = 0.2 \times (12/8) \times (0.1/0.17) \times 0.95 \times \sin 30 = 0.084$ mm.

End milling: $ECT_E = ECT_{0E} \times (z/2) \times (f_z/0.17) \times (aa/CEL)$,

or if ECT_E is known calculate f_z from:

$f_z = 0.17 \times (ECT_E/ECT_{0E}) \times (2/z)) \times (CEL/aa)$

The approximate values of $aa/CEL = 0.95$ for lead angle $LA = 0°$ (90° in the metric system).

Example, High-speed End Milling: For a cutter with $D = 25$ mm and $ar = 3.125$ mm, calculate ECT_E for $f_z = 0.1$ and $z = 6$. First calculate $ar/D = 0.125$, and then use Table 2b and find $ECT_{0E} = 0.0249$.

Calculate ECT_E with above formula:

$ECT_E = 0.0249 \times (6/2) \times (0.1/0.17) \times 0.95 \times 1 = 0.042$ mm.

Example, High-speed End Milling: For a cutter with $D = 25$ mm and $ar = 0.75$ mm, calculate ECT_E for $f_z = 0.17$ and $z = 2$ and 6. First calculate $ar/D = 0.03$, and then use Table 2b and find $f_z/f_{z0} = 3.694$

Then, $f_z = 3.694 \times 0.17 = 0.58$ mm/tooth and $ECT_E = 0.0119 \times 0.95 = 0.0113$ mm and $0.0357 \times 0.95 = 0.0339$ mm for 2 and 6 teeth respectively. These cutters are marked HS2 and HS6 in Figs. 26a, 26d, and 26e.

Example, High-speed End Milling: For a cutter with $D = 25$ mm and $ar = 0.25$ mm, calculate ECT_E for $f_z = 0.17$ and $z = 2$ and 6. First calculate $ar/D = 0.01$, and then use Table 2b and find $ECT_{0E} = 0.0069$ and 0.0207 for 2 and 6 teeth respectively. When obtaining such small values of *ECT*, there is a great danger to be far on the left side of the *H*-curve, at least when there are only 2 teeth. Doubling the feed would be the solution if cutter design and material permit.

Example, Full Slot Milling: For a cutter with $D = 25$ mm and $ar = 25$ mm, calculate ECT_E for $f_z = 0.17$ and $z = 2$ and 6. First calculate $ar/D = 1$, and then use Table 2b and find $ECT_E =$

$0.108 \times 0.95 = 0.103$ and $3 \times 0.108 \times 0.95 = 0.308$ for 2 and 6 teeth, respectively. These cutters are marked SL2 and SL6 in Figs. 26a, 26d, and 26e.

Physics behind hm and ECT, Forces and Tool-life (T).— The ECT concept for all metal cutting and grinding operations says that the more energy put into the process, by increasing feed/rev, feed/tooth, or cutting speed, the life of the edge decreases. When increasing the number of teeth (keeping everything else constant) the work and the process are subjected to a higher energy input resulting in a higher rate of tool wear.

In high-speed milling when the angle of engagement AE is small the contact time is shorter compared to slot milling ($ar/D = 1$) but the chip becomes shorter as well. Maintaining the same chip thickness as in slot milling has the effect that the energy consumption to remove the chip will be different. Hence, maintaining a constant chip thickness is a good measure when calculating cutting forces (keeping speed constant), but not when determining tool wear. Depending on cutting conditions the wear rate can either increase or decrease, this depends on whether cutting occurs on the left or right side of the H-curve.

Fig. 26a shows an example of end milling of steel with coated carbide inserts, where cutting speed V is plotted versus ECT at 5, 15, 45 and 180 minutes tool-lives. Notice that the ECT values are independent of ar/D or number of teeth or feed/tooth, or whether f_z or f_{z0} is used, as long as the corresponding f_z/f_{z0}-ratio is applied to determine ECT_E. The result is one single curve per tool-life. Had cutting speed been plotted versus f_{z0}, ar/D, or z values (number of teeth), several curves would be required at each constant tool-life, one for each of these parameters This illustrates the advantage of using the basic parameter ECT rather than f_z, or hm, or ar/D on the horizontal axis.

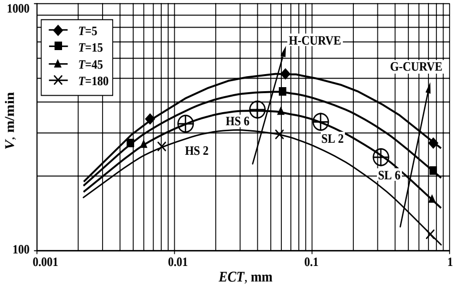


Fig. 26a. Cutting speed vs. ECT, tool-life plotted, for end milling

Example: The points (HS2, HS6) and (SL2, SL6) on the 45-minute curve in Fig. 26a relate to the previous high-speed and full slot milling examples for 2 and 6 teeth, respectively.

Running a slot at $f_{z0} = 0.17$ mm/tooth ($hm = 0.108$, $ECT_E = 0.103$ mm) with 2 teeth and for a tool-life 45 minutes, the cutting speed should be selected at $V = 340$ m/min at point SL2 and for six teeth ($hm = 0.108$ mm, $ECT_E = 0.308$) at $V = 240$ m/min at point SL6.

When high-speed milling for $ar/D = 0.03$ at $f_z = 3.394 \times 0.17 = 0.58$ mm/tooth = 0.58 mm/tooth, ECT is reduced to 0.011 mm ($hm = 0.108$) the cutting speed is 290 m/min to maintain $T = 45$ minutes, point HS2. This point is far to the left of the H-curve in Fig.26b, but if the number of teeth is increased to 6 ($ECT_E = 3 \times 0.103 = 0.3090$), the cutting speed is 360 m/min at $T = 45$ minutes and is close to the H-curve, point HS6. Slotting data using 6 teeth are on the right of this curve at point SL6, approaching the G-curve, but at a lower slotting speed of 240 m/min.

Depending on the starting f_z value and on the combination of cutter grade - work material, the location of the *H*-curve plays an important role when selecting high-speed end milling data.

Feed Rate and Tool-life in High-speed Milling, Effect of *ECT* and Number of Teeth.—Calculation of feed rate is done using the formulas in previously given:

Feed Rate:

$F_R = z \times f_z \times$ rpm, where $z \times f_z = f$ (feed/rev of cutter). Feed is measured along the feeding direction.

rpm = 1000 × V/3.1416/D, where D is diameter of cutter.

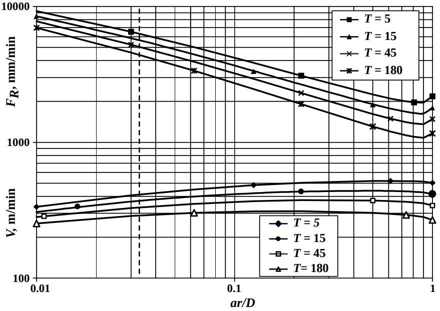


Fig. 26b. High speed feed rate and cutting speed versus *ar*/*D* at *T* = 5, 15, 45, and 180 minutes

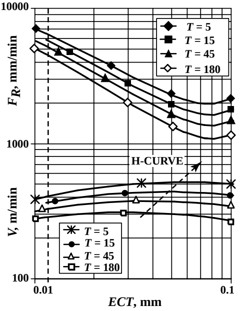


Fig. 26c. High speed feed rate and cutting speed versus *ECT*, *ar*/*D* plotted at *T* = 5, 15, 45, and 180 minutes

Fig. 26b shows the variation of feed rate F_R plotted versus *ar*/*D* for tool-lives 5, 15, 45 and 180 minutes with a 25 mm diameter cutter and 2 teeth. Fig. 26c shows the variation of feed rate F_R when plotted versus *ECT*. In both graphs the corresponding cutting speeds are also plotted. The values for *ar*/*D* = 0.03 in Fig. 26b correspond to *ECT* = 0.011 in Fig. 26c.

Feed rates have minimum around values of *ar*/*D* = 0.8 and *ECT*=0.75 and not along the *H*-curve. This is due to the fact that the f_z/f_{z0} ratio to maintain a mean chip thickness = 0.108 mm changes F_R in a different proportion than the cutting speed.

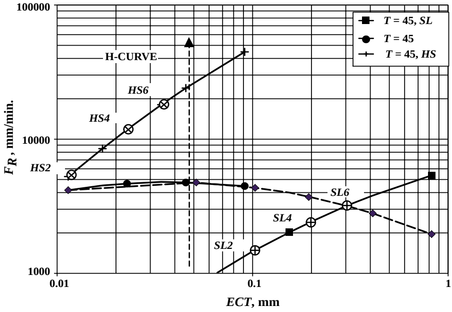


Fig. 26d. Feed rate versus *ECT* comparison of slot milling (*ar*/*D* = 1) and high-speed milling at (*ar*/*D* = 0.03) for 2, 4, and 6 teeth at *T* = 45 minutes

A comparison of feed rates for full slot ($ar/D = 1$) and high-speed end milling ($ar/D = 0.03$ and $f_z = 3.69 \times f_{z0} = 0.628$ mm) for tool-life 45 minutes is shown in Fig. 26d. The points SL2, SL4, SL6 and HS2, HS4, HS6, refer to 2, 4, and 6 teeth (2 to 6 teeth are commonly used in practice). Feed rate is also plotted versus number of teeth z in Fig. 26e, for up to 16 teeth, still at $f_z = 0.628$ mm.

Comparing the effect of using 2 versus 6 teeth in high-speed milling shows that feed rates increase from 5250 mm/min (413 ipm) up to 18000 mm/min (1417ipm) at 45 minutes tool-life. The effect of using 2 versus 6 teeth in full slot milling is that feed rate increases from 1480 mm/min (58 ipm) up to 3230 mm/min (127 ipm) at tool-life 45 minutes. If 16 teeth could be used at $ar/D = 0.03$, the feed rate increases to $F_R = 44700$ mm/min (1760 ipm), and for full slot milling $F_R = 5350$ mm/min (210 ipm).

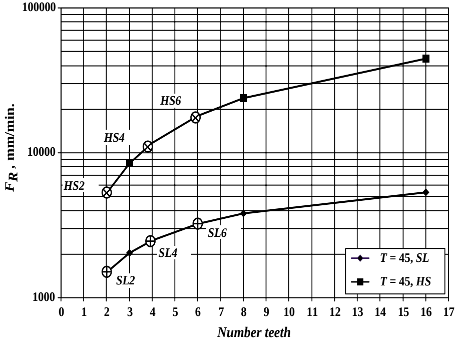


Fig. 26e. Feed rate versus number of teeth comparison of slot milling ($ar/D = 1$) and high-speed milling at ($ar/D = 0.03$) for 2, 4, and 6 teeth at $T = 45$ minutes

Comparing the feed rates that can be obtained in steel cutting with the one achieved in the earlier referred DARPA investigation, using HSS and carbide cutters milling 6061-T651 and A356-T6 aluminum, it is obvious that aluminium end milling can be run at 3 to 6 times higher feed rates. This requires 3 to 6 times higher spindle speeds (cutter diameter 25 mm, radial depth of cut $ar = 12.5$ mm, 2 teeth). Had these tests been run with 6 teeth, the feed rates would increase up to 150000-300000 mm/min, when feed/tooth = 3.4 × 0.25 = 0.8 mm/tooth at $ar/D = 0.03$.

Process Econometrics Comparison of High-speed and Slot End Milling .—When making a process econometrics comparison of high-speed milling and slot end milling use the formulas for total cost c_{tot} (*Determination Of Machine Settings And Calculation Of Costs* starting on page 1076). Total cost is the sum of the cost of cutting, tool changing, and tooling:

$$c_{tot} = H_R \times (Dist/F_R) \times (1 + T_V/T)/60$$

where $T_V = T_{RPL} + 60 \times C_E/H_R$ = equivalent tooling-cost time, minutes

T_{RPL} = replacement time for a set of edges or tool for regrinding

C_E = cost per edge(s)

H_R = hourly rate, $

Fig. 27. compares total cost c_{tot}, using the end milling cutters of the previous examples, for full slot milling with high-speed milling at ar/D =0.03, and versus ECT at T =45 minutes.

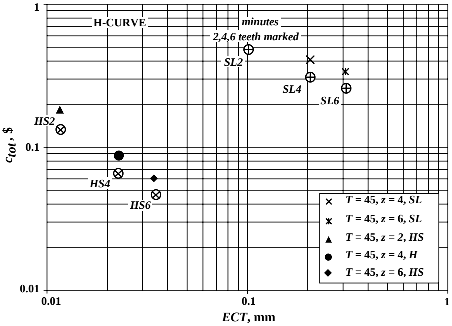


Fig. 27. Cost comparison of slot milling ($ar/D = 1$) and high-speed milling at ($ar/D = 0.03$) for 2, 4, and 6 teeth at $T = 45$ minutes

The feed/tooth for slot milling is $f_{z0} = 0.17$ and for high-speed milling at $ar/D = 0.03$ the feed is $f_z = 3.69 \times f_{z0} = 0.628$ mm.

The calculations for total cost are done according to above formula using tooling cost at $T_V = 6$, 10, and 14 minutes, for $z = 2$, 4, and 6 teeth respectively. The distance cut is $Dist$ = 1000 mm. Full slot milling costs are,

at feed rate $F_R = 3230$ and $z = 6$

$$c_{tot} = 50 \times (1000/3230) \times (1 + 14/45)/60 = \$0.338 \text{ per part}$$

at feed rate F_R =1480 and $z = 2$

$$c_{tot} = 50 \times (1000/1480) \times (1 + 6/45)/60 = \$0.638 \text{ per part}$$

High-speed milling costs,

at F_R=18000, z=6

$$c_{tot} = 50 \times (1000/18000) \times (1 + 14/45)/60 = \$0.0606 \text{ per part}$$

at F_R= 5250, z=2

$$c_{tot} = 50 \times (1000/5250) \times (1 + 6/45)/60 = \$0.208.$$

The cost reduction using high-speed milling compared to slotting is enormous. For high-speed milling with 2 teeth, the cost for high-speed milling with 2 teeth is 61 percent (0.208/0.338) of full slot milling with 6 teeth ($z = 6$). The cost for high-speed milling with 6 teeth is 19 percent (0.0638/0.338) of full slot for $z = 6$.

Aluminium end milling can be run at 3 to 6 times lower costs than when cutting steel. Costs of idle (non-machining) and slack time (waste) are not considered in the example. These data hold for perfect milling conditions such as zero run-out and accurate sharpening of all teeth and edges.

SCREW MACHINE FEEDS AND SPEEDS

Feeds and Speeds for Automatic Screw Machine Tools.—Approximate feeds and speeds for standard screw machine tools are given in the accompanying table.

Knurling in Automatic Screw Machines.—When knurling is done from the cross slide, it is good practice to feed the knurl gradually to the center of the work, starting to feed when the knurl touches the work and then passing off the center of the work with a quick rise of the cam. The knurl should also dwell for a certain number of revolutions, depending on the pitch of the knurl and the kind of material being knurled. See also *KNURLS AND KNURLING* starting on page 1211.

When two knurls are employed for spiral and diamond knurling from the turret, the knurls can be operated at a higher rate of feed for producing a spiral than they can for producing a diamond pattern. The reason for this is that in the first case the knurls work in the same groove, whereas in the latter case they work independently of each other.

Revolutions Required for Top Knurling.—The depth of the teeth and the feed per revolution govern the number of revolutions required for top knurling from the cross slide. If R is the radius of the stock, d is the depth of the teeth, c is the distance the knurl travels from the point of contact to the center of the work at the feed required for knurling, and r is the radius of the knurl; then

$$c = \sqrt{(R + r)^2 - (R + r - d)^2}$$

For example, if the stock radius R is $\frac{5}{32}$ inch, depth of teeth d is 0.0156 inch, and radius of knurl r is 0.3125 inch, then

$$\begin{aligned} c &= \sqrt{(0.1562 + 0.3125)^2 - (0.1562 + 0.3125 - 0.0156)^2} \\ &= 0.120 \text{ inch} = \text{cam rise required} \end{aligned}$$

Assume that it is required to find the number of revolutions to knurl a piece of brass $\frac{5}{16}$ inch in diameter using a 32 pitch knurl. The included angle of the teeth for brass is 90 degrees, the circular pitch is 0.03125 inch, and the calculated tooth depth is 0.0156 inch. The distance c (as determined in the previous example) is 0.120 inch. Referring to the accompanying table of feeds and speeds, the feed for top knurling brass is 0.005 inch per revolution. The number of revolutions required for knurling is, therefore, $0.120 \div 0.005 = 24$ revolutions. If conditions permit, the higher feed of 0.008 inch per revolution given in the table may be used, and 15 revolutions are then required for knurling.

Cams for Threading.—The table *Spindle Revolutions and Cam Rise for Threading* on page 1097 gives the revolutions required for threading various lengths and pitches and the corresponding rise for the cam lobe. To illustrate the use of this table, suppose a set of cams is required for threading a screw to the length of $\frac{3}{8}$ inch in a Brown & Sharpe machine. Assume that the spindle speed is 2400 revolutions per minute; the number of revolutions to complete one piece, 400; time required to make one piece, 10 seconds; pitch of the thread, $\frac{1}{32}$ inch or 32 threads per inch. By referring to the table, under 32 threads per inch, and opposite $\frac{3}{8}$ inch (length of threaded part), the number of revolutions required is found to be 15 and the rise required for the cam, 0.413 inch.

Approximate Cutting Speeds and Feeds for Standard Automatic Screw Machine Tools—Brown and Sharpe

	Cut		Material to be Machined						
			Brass[a]	Mild or Soft Steel			Tool Steel, 0.80–1.00% C		
	Width or Depth, Inches	Dia. of Hole, Inches	Feed, Inches per Rev.	Feed, Inches per Rev.	Surface Speed, Feet per Min.		Feed, Inches per Rev.	Surface Speed, Feet per Min.	
Tool					Carbon Tools	H.S.S. Tools		Carbon Tools	H.S.S. Tools
Boring tools	0.005	…	…	0.008	50	110	0.004	30	60
Box tools, roller rest Single chip finishing {	1⁄32	…	0.012	0.010	70	150	0.005	40	75
	1⁄16	…	0.010	0.008	70	150	0.004	40	75
	1⁄8	…	0.008	0.007	70	150	0.003	40	75
	3⁄16	…	0.008	0.006	70	150	0.002	40	75
	1⁄4	…	0.006	0.005	70	150	0.0015	40	75
Finishing	0.005	…	0.010	0.010	70	150	0.006	40	75
Center drills	…	Under 1⁄8	0.003	0.0015	50	110	0.001	30	75
	…	Over 1⁄8	0.006	0.0035	50	110	0.002	30	75
Cutoff tools { Angular	…	…	0.0015	0.0006	80	150	0.0004	50	85
Circular	3⁄64–1⁄8	…	0.0035	0.0015	80	150	0.001	50	85
Straight	1⁄16–1⁄8	…	0.0035	0.0015	80	150	0.001	50	85
Stock diameter under 1⁄8 in.	…	…	0.002	0.0008	80	150	0.0005	50	85
Dies { Button	…	…	…	…	30	…	…	14	…
Chaser	…	…	…	…	30	40	…	16	20
Drills, twist cut	…	0.02	0.0014	0.001	40	60	0.0006	30	45
	…	0.04	0.002	0.0014	40	60	0.0008	30	45
	…	1⁄16	0.004	0.002	40	60	0.0012	30	45
	…	3⁄32	0.006	0.0025	40	60	0.0016	30	45
	…	1⁄8	0.009	0.0035	40	75	0.002	30	60
	…	3⁄16	0.012	0.004	40	75	0.003	30	60
	…	1⁄4	0.014	0.005	40	75	0.003	30	60
	…	5⁄16	0.016	0.005	40	75	0.0035	30	60
	…	3⁄8–5⁄8	0.016	0.006	40	85	0.004	30	60
Form tools, circular	1⁄8	…	0.002	0.0009	80	150	0.0006	50	85
	1⁄4	…	0.002	0.0008	80	150	0.0005	50	85
	3⁄8	…	0.0015	0.0007	80	150	0.0004	50	85
	1⁄2	…	0.0012	0.0006	80	150	0.0004	50	85
	5⁄8	…	0.001	0.0005	80	150	0.0003	50	85
	3⁄4	…	0.001	0.0005	80	150	0.0003	50	85
	1	…	0.001	0.0004	80	150	…	…	…

Approximate Cutting Speeds and Feeds for Standard Automatic Screw Machine Tools—Brown and Sharpe *(Continued)*

Tool		Cut		Material to be Machined						
				Brass[a]	Mild or Soft Steel			Tool Steel, 0.80–1.00% C		
		Width or Depth, Inches	Dia. of Hole, Inches	Feed, Inches per Rev.	Feed, Inches per Rev.	Surface Speed, Feet per Min.		Feed, Inches per Rev.	Surface Speed, Feet per Min.	
						Carbon Tools	H.S.S. Tools		Carbon Tools	H.S.S. Tools
Hollow mills and balance turning tools {	Turned diam. under 5/32 in. {	1/32	…	0.012	0.010	70	150	0.008	40	85
		1/16	…	0.010	0.009	70	150	0.006	40	85
	Turned diam. over 5/32 in. {	1/32	…	0.017	0.014	70	150	0.010	40	85
		1/16	…	0.015	0.012	70	150	0.008	40	85
		1/8	…	0.012	0.010	70	150	0.008	40	85
		3/16	…	0.010	0.008	70	150	0.006	40	85
		1/4	…	0.009	0.007	70	150	0.0045	40	85
Knee tools		1/32	…	…	0.010	70	150	0.008	40	85
Knurling tools {	Turret {	On	…	0.020	0.015	150	…	0.010	105	…
		Off	…	0.040	0.030	150	…	0.025	105	…
	Side or swing {	…	…	0.004	0.002	150	…	0.002	105	…
		…	…	0.006	0.004	150	…	0.003	105	…
	Top {	…	…	0.005	0.003	150	…	0.002	105	…
		…	…	0.008	0.006	150	…	0.004	105	…
Pointing and facing tools		…	…	0.001	0.0008	70	150	0.0005	40	80
		…	…	0.0025	0.002	70	150	0.0008	40	80
Reamers and bits		0.003 – 0.004	1/8 or less	0.010 – 0.007	0.008 – 0.006	70	105	0.006 – 0.004	40	60
		0.004 – 0.008	1/8 or over	0.010	0.010	70	105	0.006 – 0.008	40	60
Recessing tools {	End cut {	…	… {	0.001	0.0006	70	150	0.0004	40	75
		…	…	0.005	0.003	70	150	0.002	40	75
	Inside cut	1/16–1/8	… {	0.0025	0.002	70	105	0.0015	40	60
			…	0.0008	0.0006	70	105	0.0004	40	60
Swing tools, forming		1/8	…	0.002	0.0007	70	150	0.0005	40	85
		1/4	…	0.0012	0.0005	70	150	0.0003	40	85
		3/8	…	0.001	0.0004	70	150	0.0002	40	85
		1/2	…	0.0008	0.0003	70	150	0.0002	40	85
Turning, straight and taper[b]		1/32	…	0.008	0.006	70	150	0.0035	40	85
		1/16	…	0.006	0.004	70	150	0.003	40	85
		1/8	…	0.005	0.003	70	150	0.002	40	85
		3/16	…	0.004	0.0025	70	150	0.0015	40	85
Taps		…	…	…	…	25	30	…	12	15

[a] Use maximum spindle speed on machine.

[b] For taper turning use feed slow enough for greatest depth depth of cut.

Spindle Revolutions and Cam Rise for Threading

Length of Threaded Portion, Inch	Number of Threads per Inch														
	80	72	64	56	48	40	36	32	30	28	24	20	18	16	14
	First Line: Revolutions of Spindle for Threading. Second Line: Rise on Cam for Threading, Inch														
1/16	9.50	9.00	8.50	8.00	6.00	5.50	5.50	5.00	5.00	5.00	3.00	…	…	…	…
	0.107	0.113	0.120	0.129	0.110	0.121	0.134	0.138	0.147	0.157	0.106	…	…	…	…
1/8	14.50	13.50	12.50	11.50	9.00	8.00	7.00	7.00	7.00	6.50	4.50	4.00	3.50	3.50	…
	0.163	0.169	0.176	0.185	0.165	0.176	0.171	0.193	0.205	0.204	0.159	0.170	0.165	0.186	…
3/16	19.50	18.00	16.50	15.00	12.00	10.50	10.00	9.00	8.50	8.50	6.00	5.50	5.00	4.50	4.00
	0.219	0.225	0.232	0.241	0.220	0.231	0.244	0.248	0.249	0.267	0.213	0.234	0.236	0.239	0.243
1/4	24.50	23.508	20.50	18.50	15.00	13.00	12.00	11.00	10.50	10.00	7.50	6.50	6.00	5.50	5.00
	0.276	0.294	0.288	0.297	0.275	0.286	0.293	0.303	0.308	0.314	0.266	0.276	0.283	0.292	0.304
5/16	29.50	27.00	24.50	22.00	18.00	15.50	14.50	13.00	12.50	12.00	9.00	8.00	7.00	6.50	6.00
	0.332	0.338	0.345	0.354	0.340	0.341	0.354	0.358	0.367	0.377	0.319	0.340	0.330	0.345	0.364
3/8	34.50	31.50	28.50	25.50	21.00	18.00	16.50	15.00	14.50	13.50	10.50	9.00	8.50	7.50	7.00
	0.388	0.394	0.401	0.410	0.385	0.396	0.403	0.413	0.425	0.424	0.372	0.383	0.401	0.398	0.425
7/16	39.50	36.00	32.50	29.00	24.00	20.50	19.00	17.00	16.00	15.50	12.00	10.50	9.50	8.50	7.50
	0.444	0.450	0.457	0.466	0.440	0.451	0.464	0.468	0.469	0.487	0.425	0.446	0.448	0.451	0.455
1/2	44.50	40.50	36.50	32.50	27.00	23.00	21.00	19.00	18.00	17.00	13.50	11.50	10.50	9.50	8.50
	0.501	0.506	0.513	0.522	0.495	0.506	0.513	0.523	0.528	0.534	0.478	0.489	0.496	0.504	0.516
9/16	49.50	45.00	40.50	36.00	30.00	25.50	23.50	21.00	20.00	19.00	15.00	13.00	11.50	10.50	9.50
	0.559	0.563	0.570	0.579	0.550	0.561	0.574	0.578	0.587	0.597	0.531	0.553	0.543	0.558	0.577
5/8	54.50	49.50	44.50	39.50	33.00	28.00	25.50	23.00	22.00	20.50	16.50	14.00	13.00	11.50	10.50
	0.613	0.619	0.626	0.635	0.605	0.616	0.623	0.633	0.645	0.644	0.584	0.595	0.614	0.611	0.637
11/16	59.50	54.00	48.50	43.00	36.00	30.50	28.00	25.00	23.50	22.50	18.00	15.50	14.00	12.50	11.00
	0.679	0.675	0.682	0.691	0.660	0.671	0.684	0.688	0.689	0.707	0.638	0.659	0.661	0.664	0.668
3/4	64.50	58.50	52.50	46.50	39.00	33.00	30.00	27.00	25.50	24.00	19.50	16.50	15.00	13.50	12.00
	0.726	0.731	0.738	0.747	0.715	0.726	0.733	0.743	0.748	0.754	0.691	0.701	0.708	0.717	0.728

Threading cams are often cut on a circular milling attachment. When this method is employed, the number of minutes the attachment should be revolved for each 0.001 inch rise, is first determined. As 15 spindle revolutions are required for threading and 400 for completing one piece, that part of the cam surface required for the actual threading operation equals 15 ÷ 400 = 0.0375, which is equivalent to 810 minutes of the circumference. The total rise, through an arc of 810 minutes is 0.413 inch, so the number of minutes for each 0.001 inch rise equals 810 ÷ 413 = 1.96 or, approximately, two minutes. If the attachment is graduated to read to five minutes, the cam will be fed laterally 0.0025 inch each time it is turned through five minutes of arc.

Practical Points on Cam and Tool Design.—The following general rules are given to aid in designing cams and special tools for automatic screw machines, and apply particularly to Brown and Sharpe machines:

1) Use the highest speeds recommended for the material used that the various tools will stand.

2) Use the arrangement of circular tools best suited for the class of work.

3) Decide on the quickest and best method of arranging the operations before designing the cams.

4) Do not use turret tools for forming when the cross-slide tools can be used to better advantage.

5) Make the shoulder on the circular cutoff tool large enough so that the clamping screw will grip firmly.

6) Do not use too narrow a cutoff blade.

7) Allow 0.005 to 0.010 inch for the circular tools to approach the work and 0.003 to 0.005 inch for the cutoff tool to pass the center.

8) When cutting off work, the feed of the cutoff tool should be decreased near the end of the cut where the piece breaks off.

9) When a thread is cut up to a shoulder, the piece should be grooved or necked to make allowance for the lead on the die. An extra projection on the forming tool and an extra amount of rise on the cam will be needed.

10) Allow sufficient clearance for tools to pass one another.

11) Always make a diagram of the cross-slide tools in position on the work when difficult operations are to be performed; do the same for the tools held in the turret.

12) Do not drill a hole the depth of which is more than 3 times the diameter of the drill, but rather use two or more drills as required. If there are not enough turret positions for the extra drills needed, make provision for withdrawing the drill clear of the hole and then advancing it into the hole again.

13) Do not run drills at low speeds. Feeds and speeds recommended in the table starting on page 1095 should be followed as far as is practicable.

14) When the turret tools operate farther in than the face of the chuck, see that they will clear the chuck when the turret is revolved.

15) See that the bodies of all turret tools will clear the side of the chute when the turret is revolved.

16) Use a balance turning tool or a hollow mill for roughing cuts.

17) The rise on the thread lobe should be reduced so that the spindle will reverse when the tap or die holder is drawn out.

18) When bringing another tool into position after a threading operation, allow clearance before revolving the turret.

19) Make provision to revolve the turret rapidly, especially when pieces are being made in from three to five seconds and when only a few tools are used in the turret. It is sometimes desirable to use two sets of tools.

20) When using a belt-shifting attachment for threading, clearance should be allowed, as it requires extra time to shift the belt.

21) When laying out a set of cams for operating on a piece that requires to be slotted, cross-drilled or burred, allowance should be made on the lead cam so that the transferring arm can descend and ascend to and from the work without coming in contact with any of the turret tools.

22) Always provide a vacant hole in the turret when it is necessary to use the transferring arm.

23) When designing special tools allow as much clearance as possible. Do not make them so that they will just clear each other, as a slight inaccuracy in the dimensions will often cause trouble.

24) When designing special tools having intricate movements, avoid springs as much as possible, and use positive actions.

Stock for Screw Machine Products.—The amount of stock required for the production of 1000 pieces on the automatic screw machine can be obtained directly from the table *Stock Required for Screw Machine Products*. To use this table, add to the length of the work the width of the cut-off tool blade; then the number of feet of material required for 1000 pieces can be found opposite the figure thus obtained, in the column headed "Feet per 1000 Parts." Screw machine stock usually comes in bars 10 feet long, and in compiling this table an allowance was made for chucking on each bar.

The table can be extended by using the following formula, in which

F = number of feet required for 1000 pieces

L = length of piece in inches

W = width of cut-off tool blade in inches

$$F = (L + W) \times 84$$

The amount to add to the length of the work, or the width of the cut-off tool, is given in the following, which is standard in a number of machine shops:

Diameter of Stock, Inches	Width of Cut-off Tool Blade, Inches
0.000–0.250	0.045
0.251–0.375	0.062
0.376–0.625	0.093
0.626–1.000	0.125
1.001–1.500	0.156

It is sometimes convenient to know the weight of a certain number of pieces, when estimating the price. The weight of round bar stock can be found by means of the following formulas, in which

W = weight in pounds

D = diameter of stock in inches

F = length in feet

For brass stock: $W = D^2 \times 2.86 \times F$

For steel stock: $W = D^2 \times 2.675 \times F$

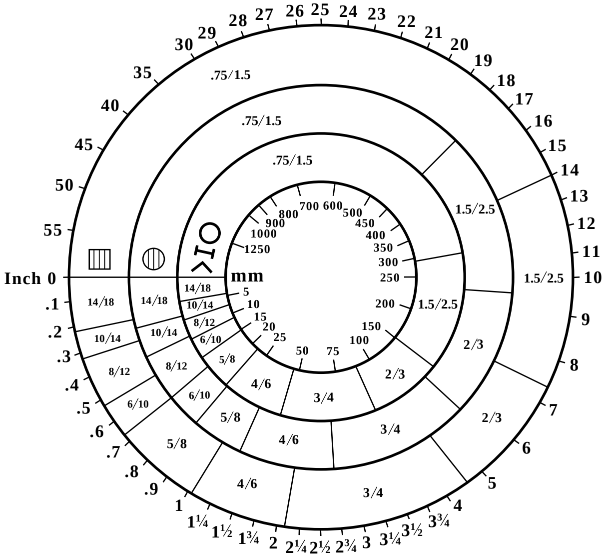
For iron stock: $W = D^2 \times 2.65 \times F$

Stock Required for Screw Machine Products

The table gives the amount of stock, in feet, required for 1000 pieces, when the length of the finished part plus the thickness of the cut-off tool blade is known. Allowance has been made for chucking. To illustrate, if length of cut-off tool and work equals 0.140 inch, 11.8 feet of stock is required for the production of 1000 parts.

Length of Piece and Cut-Off Tool	Feet per 1000 Parts	Length of Piece and Cut-Off Tool	Feet per 1000 Parts	Length of Piece and Cut-Off Tool	Feet per 1000 Parts	Length of Piece and Cut-Off Tool	Feet per 1000 Parts
0.050	4.2	0.430	36.1	0.810	68.1	1.380	116.0
0.060	5.0	0.440	37.0	0.820	68.9	1.400	117.6
0.070	5.9	0.450	37.8	0.830	69.7	1.420	119.3
0.080	6.7	0.460	38.7	0.840	70.6	1.440	121.0
0.090	7.6	0.470	39.5	0.850	71.4	1.460	122.7
0.100	8.4	0.480	40.3	0.860	72.3	1.480	124.4
0.110	9.2	0.490	41.2	0.870	73.1	1.500	126.1
0.120	10.1	0.500	42.0	0.880	73.9	1.520	127.7
0.130	10.9	0.510	42.9	0.890	74.8	1.540	129.4
0.140	11.8	0.520	43.7	0.900	75.6	1.560	131.1
0.150	12.6	0.530	44.5	0.910	76.5	1.580	132.8
0.160	13.4	0.540	45.4	0.920	77.3	1.600	134.5
0.170	14.3	0.550	46.2	0.930	78.2	1.620	136.1
0.180	15.1	0.560	47.1	0.940	79.0	1.640	137.8
0.190	16.0	0.570	47.9	0.950	79.8	1.660	139.5
0.200	16.8	0.580	48.7	0.960	80.7	1.680	141.2
0.210	17.6	0.590	49.6	0.970	81.5	1.700	142.9
0.220	18.5	0.600	50.4	0.980	82.4	1.720	144.5
0.230	19.3	0.610	51.3	0.990	83.2	1.740	146.2
0.240	20.2	0.620	52.1	1.000	84.0	1.760	147.9
0.250	21.0	0.630	52.9	1.020	85.7	1.780	149.6
0.260	21.8	0.640	53.8	1.040	87.4	1.800	151.3
0.270	22.7	0.650	54.6	1.060	89.1	1.820	152.9
0.280	23.5	0.660	55.5	1.080	90.8	1.840	154.6
0.290	24.4	0.670	56.3	1.100	92.4	1.860	156.3
0.300	25.2	0.680	57.1	1.120	94.1	1.880	158.0
0.310	26.1	0.690	58.0	1.140	95.8	1.900	159.7
0.320	26.9	0.700	58.8	1.160	97.5	1.920	161.3
0.330	27.7	0.710	59.7	1.180	99.2	1.940	163.0
0.340	28.6	0.720	60.5	1.200	100.8	1.960	164.7
0.350	29.4	0.730	61.3	1.220	102.5	1.980	166.4
0.360	30.3	0.740	62.2	1.240	104.2	2.000	168.1
0.370	31.1	0.750	63.0	1.260	105.9	2.100	176.5
0.380	31.9	0.760	63.9	1.280	107.6	2.200	184.9
0.390	32.8	0.770	64.7	1.300	109.2	2.300	193.3
0.400	33.6	0.780	65.5	1.320	110.9	2.400	201.7
0.410	34.5	0.790	66.4	1.340	112.6	2.500	210.1
0.420	35.3	0.800	67.2	1.360	114.3	2.600	218.5

Band Saw Blade Selection.—The primary factors to consider in choosing a saw blade are: the pitch, or the number of teeth per inch of blade; the tooth form; and the blade type (material and construction). Tooth pitch selection depends on the size and shape of the work, whereas tooth form and blade type depend on material properties of the workpiece and on economic considerations of the job.



Courtesy of American Saw and Manufacturing Company

The tooth selection chart above is a guide to help determine the best blade pitch for a particular job. The tooth specifications in the chart are standard variable-pitch blade sizes as specified by the Hack and Band Saw Association. The variable-pitch blades listed are designated by two numbers that refer to the approximate maximum and minimum tooth pitch. A 4⁄6 blade, for example, has a maximum tooth spacing of approximately ¼ inch and a minimum tooth spacing of about ⅙ inch. Blades are available, from most manufacturers, in sizes within about ±10 per cent of the sizes listed.

To use the chart, locate the length of cut in inches on the outside circle of the table (for millimeters use the inside circle) and then find the tooth specification that aligns with the length, on the ring corresponding to the material shape. The length of cut is the distance that any tooth of the blade is in contact with the work as it passes once through the cut. For cutting solid round stock, use the diameter as the length of cut and select a blade from the ring with the solid circle. When cutting angles, channels, I-beams, tubular pieces, pipe, and hollow or irregular shapes, the length of cut is found by dividing the cross-sectional area of the cut by the distance the blade needs to travel to finish the cut. Locate the length of cut on the outer ring (inner ring for mm) and select a blade from the ring marked with the angle, I-beam, and pipe sections.

Example: A 4-inch pipe with a 3-inch inside diameter is to be cut. Select a variable pitch blade for cutting this material.

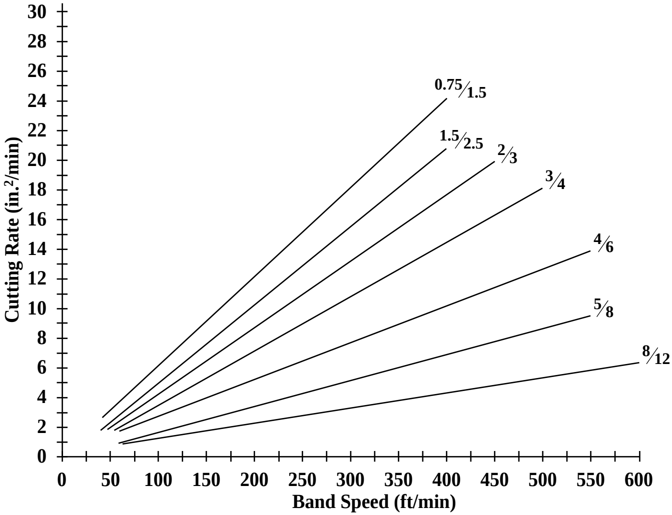
The area of the pipe is $\pi/4 \times (4^2 - 3^2) = 5.5$ in.^{2} The blade has to travel 4 inches to cut through the pipe, so the average length of cut is 5.5⁄4 = 1.4 inches. On the tooth selection wheel, estimate the location of 1.4 inches on the outer ring, and read the tooth specification from the ring marked with the pipe, angle, and I-beam symbols. The chart indicates that a 4⁄6 variable-pitch blade is the preferred blade for this cut.

Tooth Forms.—Band saw teeth are characterized by a tooth form that includes the shape, spacing (pitch), rake angle, and gullet capacity of the tooth. Tooth form affects the cutting efficiency, noise level, blade life, chip-carrying capacity, and the surface finish quality of the cut. The rake angle, which is the angle between the face of the tooth and a line perpendicular to the direction of blade travel, influences the cutting speed. In general, positive rake angles cut faster. The *standard tooth* form has conventional shape teeth, evenly spaced with deep gullets and a 0° rake angle. Standard tooth blades are used for general-purpose cutting on a wide variety of materials. The *skip tooth* form has shallow, widely spaced teeth arranged in narrow bands and a 0° rake angle. Skip tooth blades are used for cutting soft metals, wood, plastics, and composite materials. The *hook tooth* form is similar to the skip tooth, but has a positive rake angle and is used for faster cutting of large sections of soft metal, wood, and plastics, as well as for cutting some metals, such as cast iron, that form a discontinuous chip. The *variable-tooth* (variable-pitch) form has a conventional tooth shape, but the tips of the teeth are spaced a variable distance (pitch) apart. The variable pitch reduces vibration of the blade and gives smoother cutting, better surface finish, and longer blade life. The *variable positive tooth* form is a variable-pitch tooth with a positive rake angle that causes the blade to penetrate the work faster. The variable positive tooth blade increases production and gives the longest blade life.

Set is the angle that the teeth are offset from the straight line of a blade. The set affects the blade efficiency (i.e., cutting rate), chip-carrying ability, and quality of the surface finish. *Alternate set* blades have adjacent teeth set alternately one to each side. Alternate set blades, which cut faster but with a poorer finish than other blades, are especially useful for rapid rough cutting. A *raker set is* similar to the alternate set, but every few teeth, one of the teeth is set to the center, not to the side (typically every third tooth, but sometimes every fifth or seventh tooth). The raker set pattern cuts rapidly and produces a good surface finish. The *vari-raker set,* or variable raker, is a variable-tooth blade with a raker set. The vari-raker is quieter and produces a better surface finish than a raker set standard tooth blade. *Wavy set* teeth are set in groups, alternately to one side, then to the other. Both wavy set and vari-raker set blades are used for cutting tubing and other interrupted cuts, but the blade efficiency and surface finish produced are better with a vari-raker set blade.

Types of Blades.—The most important band saw blade types are carbon steel, bimetal, carbide tooth, and grit blades made with embedded carbide or diamond. *Carbon steel blades* have the lowest initial cost, but they may wear out faster. Carbon steel blades are used for cutting a wide variety of materials, including mild steels, aluminum, brass, bronze, cast iron, copper, lead, and zinc, as well as some abrasive materials such as cork, fiberglass, graphite, and plastics. *Bimetal blades* are made with a high-speed steel cutting edge that is welded to a spring steel blade back. Bimetal blades are stronger and last longer, and they tend to produce straighter cuts because the blade can be tensioned higher than carbon steel blades. Because bimetal blades last longer, the cost per cut is frequently lower than when using carbon steel blades. Bimetal blades are used for cutting all ferrous and nonferrous metals, a wide range of shapes of easy to moderately machinable material, and solids and heavy wall tubing with moderate to difficult machinability. *Tungsten carbide blades* are similar to bimetal blades but have tungsten carbide teeth welded to the blade back. The welded teeth of carbide blades have greater wear and high-temperature resistance than either carbon steel or bimetal blades and produce less tooth vibration, while giving smoother, straighter, faster, and quieter cuts requiring less feed force. Carbide blades are used on tough alloys such as cobalt, nickel- and titanium-based alloys, and for nonferrous materials such as aluminum castings, fiberglass, and graphite. The *carbide grit blade* has tungsten carbide grit metallurgically bonded to either a gulleted (serrated) or toothless steel band. The blades are made in several styles and grit sizes. Both carbide grit and diamond grit blades are used to cut materials that conventional (carbon and bimetal) blades are unable to cut such as: fiberglass, reinforced plastics, composite materials, carbon and graphite, aramid fibers, plastics, cast iron, stellites, high-hardness tool steels, and superalloys.

Band Saw Speed and Feed Rate.—The band speed necessary to cut a particular material is measured in feet per minute (fpm) or in meters per minute (m/min), and depends on material characteristics and size of the workpiece. Typical speeds for a bimetal blade cutting 4-inch material with coolant are given in the speed selection table that follows. For other size materials or when cutting without coolant, adjust speeds according to the instructions at the bottom of the table.



Cutting Rates for Band Saws

The feed or cutting rate, usually measured in square inches or square meters per minute, indicates how fast material is being removed and depends on the speed and pitch of the blade, not on the workpiece material. The graph above, based on material provided by American Saw and Mfg., gives approximate cutting rates (in.^{2}/min) for various variable-pitch blades and cutting speeds. Use the value from the graph as an initial starting value and then adjust the feed based on the performance of the saw. The size and character of the chips being produced are the best indicators of the correct feed force. Chips that are curly, silvery, and warm indicate the best feed rate and band speed. If the chips appear burned and heavy, the feed is too great, so reduce the feed rate, the band speed, or both. If the chips are thin or powdery, the feed rate is too low, so increase the feed rate or reduce the band speed. The actual cutting rate achieved during a cut is equal to the area of the cut divided by the time required to finish the cut. The time required to make a cut is equal to the area of the cut divided by the cutting rate in square inches per minute.

Bimetal Band Saw Speeds for Cutting 4-Inch Material with Coolant

Material	Category (AISI/SAE)	Speed (fpm)	Speed (m/min)
Aluminum Alloys	1100, 2011, 2017, 2024, 3003, 5052, 5086, 6061, 6063, 6101, 6262, 7075	500	152
Cast Iron	A536 (60-40-18)	360	110
	A47	300	91
	A220 (50005), A536 (80-55-06)	240	73
	A48 (20 ksi)	230	70
	A536 (100-70-03)	185	56
	A48 (40 ksi)	180	55
	A220 (60004)	170	52
	A436 (1B)	150	46
	A220 (70003)	145	44
	A436 (2)	140	43
	A220 (80002), A436 (2B)	125	38
	A536 (120-90-02)	120	37
	A220 (90001), A48 (60 ksi)	100	30
	A439 (D-2)	80	24
	A439 (D-2B)	60	18
Cobalt	WF-11	65	20
	Astroloy M	60	18
Copper	356, 360	450	137
	353	400	122
	187, 1452	375	114
	380, 544	350	107
	173, 932, 934	315	96
	330, 365	285	87
	623, 624	265	81
	230, 260, 272, 280, 464, 632, 655	245	75
	101, 102, 110, 122, 172, 17510, 182, 220, 510, 625, 706, 715	235	72
	630	230	70
	811	215	66
Iron Base Super Alloy	Pyromet X-15	120	37
	A286, Incoloy 800 and 801	90	27
Magnesium	AZ31B	900	274
Nickel	Nickel 200, 201, 205	85	26
Nickel Alloy	Inconel 625	100	30
	Incoloy 802, 804	90	27
	Monel R405	85	26
	20CB3	80	24
	Monel 400, 401	75	23
	Hastelloy B, B2, C, C4, C22, C276, F, G, G2, G3, G30, N, S, W, X, Incoloy 825, 926, Inconel 751, X750, Waspaloy	70	21
	Monel K500	65	20
	Incoloy 901, 903, Inconel 600, 718, Ni-Span-C902, Nimonic 263, Rene 41, Udimet 500	60	18
	Nimonic 75	50	15
Stainless Steel	416, 420	190	58
	203EZ, 430, 430F, 4302	150	46
	303, 303PB, 303SE, 410, 440F, 30323	140	43
	304	120	37
	414, 30403	115	35
	347	110	34
	316, 31603	100	30
	Greek Ascoloy	95	29
	18-18-2, 309, Ferralium	90	27
	15-5PH, 17-4PH, 17-7PH, 2205, 310, AM350, AM355, Custom 450, Custom 455, PH13-8Mo, PH14-8Mo, PH15-7Mo	80	24
	22-13-5, Nitronic 50, 60	60	18

Bimetal Band Saw Speeds for Cutting 4-Inch Material with Coolant *(Continued)*

Material	Category (AISI/SAE)	Speed (fpm)	Speed (m/min)
Steel	12L14	425	130
	1213, 1215	400	122
	1117	340	104
	1030	330	101
	1008, 1015, 1020, 1025	320	98
	1035	310	94
	1018, 1021, 1022, 1026, 1513, A242 Cor-Ten A	300	91
	1137	290	88
	1141, 1144, 1144 Hi Stress	280	85
	41L40	275	84
	1040, 4130, A242 Cor-Ten B, (A36 Shapes)	270	82
	1042, 1541, 4140, 4142	250	76
	8615, 8620, 8622	240	73
	W-1	225	69
	1044, 1045, 1330, 4340, E4340, 5160, 8630	220	67
	1345, 4145, 6150	210	64
	1060, 4150, 8640, A-6, O-1, S-1	200	61
	H-11, H-12, H-13, L-6, O-6	190	58
	1095	185	56
	A-2	180	55
	E9310	175	53
	300M, A-10, E52100, HY-80, HY-100	160	49
	S-5	140	43
	S-7	125	38
	M-1	110	34
	HP 9-4-20, HP 9-4-25	105	32
	M-2, M-42, T1	100	30
	D-2	90	27
	T-15	70	21
Titanium	Pure, Ti-3Al-8V-6Cr-4Mo-4Z, Ti-8Mo-8V-2Fe-3Al	80	24
	Ti-2Al-11Sn-5Zr-1Mo, Ti-5Al-2.5Sn, Ti-6Al-2Sn-4Zr-2Mo	75	23
	Ti-6Al-4V	70	21
	Ti-7Al-4Mo, Ti-8Al-1Mo-1V	65	20

The speed figures given are for 4-in. material (length of cut) using a 3⁄4 variable-tooth bimetal blade and cutting fluid. For cutting dry, reduce speed 30–50%; for carbon steel band saw blades, reduce speed 50%. For other cutting lengths: increase speed 15% for 1⁄4-in. material (10⁄14 blade); increase speed 12% for 3⁄4-in. material (6⁄10 blade); increase speed 10% for 1 1⁄4-in. material (4⁄6 blade); decrease speed 12% for 8-in. material (2⁄3 blade).

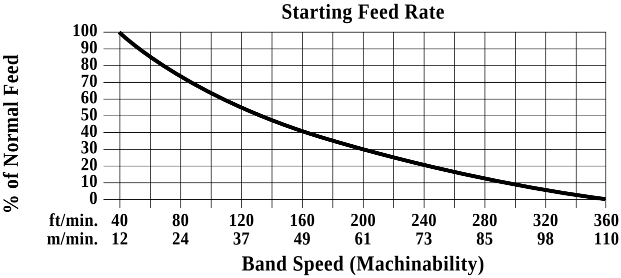
Table data are based on material provided by LENOX Blades, American Saw & Manufacturing Co.

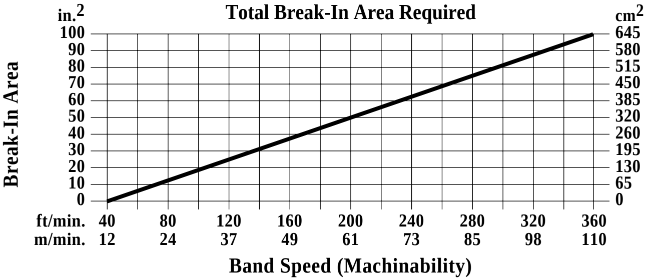
Example: Find the band speed, the cutting rate, and the cutting time if the 4-inch pipe of the previous example is made of 304 stainless steel.

The preceding blade speed table gives the band speed for 4-inch 304 stainless steel as 120 fpm (feet per minute). The average length of cut for this pipe (see the previous example) is 1.4 inches, so increase the band saw speed by about 10 per cent (see footnote on) to 130 fpm to account for the size of the piece. On the cutting rate graph above, locate the point on the 4⁄6 blade line that corresponds to the band speed of 130 fpm and then read the cutting rate from the left axis of the graph. The cutting rate for this example is approximately 4 in. ^{2}/min. The cutting time is equal to the area of the cut divided by the cutting rate, so cutting time = 5.5⁄4 = 1.375 minutes.

Band Saw Blade Break-In.— A new band saw blade must be broken in gradually before it is allowed to operate at its full recommended feed rate. Break-in relieves the blade of residual stresses caused by the manufacturing process so that the blade retains its cutting ability longer. Break-in requires starting the cut at the material cutting speed with a low feed rate and then gradually increasing the feed rate over time until enough material has been cut. A blade should be broken in with the material to be cut.

To break in a new blade, first set the band saw speed at the recommended cutting speed for the material and start the first cut at the feed indicated on the starting feed rate graph below. After the saw has penetrated the work to a distance equal to the width of the blade, increase the feed slowly. When the blade is about halfway through the cut, increase the feed again slightly and finish the cut without increasing the feed again. Start the next and each successive cut with the same feed rate that ended the previous cut, and increase the feed rate slightly again before the blade reaches the center of the cut. Repeat this procedure until the area cut by the new blade is equal to the total area required as indicated on the graph below. At the end of the break-in period, the blade should be cutting at the recommended feed rate, otherwise adjusted to that rate.





Cutting Fluids for Machining

The goal in all conventional metal-removal operations is to raise productivity and reduce costs by machining at the highest practical speed consistent with long tool life, fewest rejects, and minimum downtime, and with the production of surfaces of satisfactory accuracy and finish. Many machining operations can be performed “dry,” but the proper application of a cutting fluid generally makes possible: higher cutting speeds, higher feed rates, greater depths of cut, lengthened tool life, decreased surface roughness, increased dimensional accuracy, and reduced power consumption. Selecting the proper cutting fluid for a specific machining situation requires knowledge of fluid functions, properties, and limitations. Cutting fluid selection deserves as much attention as the choice of machine tool, tooling, speeds, and feeds.

To understand the action of a cutting fluid it is important to realize that almost all the energy expended in cutting metal is transformed into heat, primarily by the deformation of the metal into the chip and, to a lesser degree, by the friction of the chip sliding against the tool face. With these factors in mind it becomes clear that the primary functions of any cutting fluid are: cooling of the tool, workpiece, and chip; reducing friction at the sliding contacts; and reducing or preventing welding or adhesion at the contact surfaces, which forms the "built-up edge" on the tool. Two other functions of cutting fluids are flushing away chips from the cutting zone and protecting the workpiece and tool from corrosion.

The relative importance of the functions is dependent on the material being machined, the cutting tool and conditions, and the finish and accuracy required on the part. For example, cutting fluids with greater lubricity are generally used in low-speed machining and on most difficult-to-cut materials. Cutting fluids with greater cooling ability are generally used in high-speed machining on easier-to-cut materials.

Types of Cutting and Grinding Fluids.—In recent years a wide range of cutting fluids has been developed to satisfy the requirements of new materials of construction and new tool materials and coatings.

There are four basic types of cutting fluids; each has distinctive features, as well as advantages and limitations. Selection of the right fluid is made more complex because the dividing line between types is not always clear. Most machine shops try to use as few different fluids as possible and prefer fluids that have long life, do not require constant changing or modifying, have reasonably pleasant odors, do not smoke or fog in use, and, most important, are neither toxic nor cause irritation to the skin. Other issues in selection are the cost and ease of disposal.

The major divisions and subdivisions used in classifying cutting fluids are:

Cutting Oils, including straight and compounded mineral oils plus additives.

Water-Miscible Fluids, including emulsifiable oils; chemical or synthetic fluids; and semichemical fluids.

Gases.

Paste and Solid Lubricants.

Since the cutting oils and water-miscible types are the most commonly used cutting fluids in machine shops, discussion will be limited primarily to these types. It should be noted, however, that compressed air and inert gases, such as carbon dioxide, nitrogen, and Freon, are sometimes used in machining. Paste, waxes, soaps, graphite, and molybdenum disulfide may also be used, either applied directly to the workpiece or as an impregnant in the tool, such as in a grinding wheel.

Cutting Oils.—Cutting oils are generally compounds of mineral oil with the addition of animal, vegetable, or marine oils to improve the wetting and lubricating properties. Sulfur, chlorine, and phosphorous compounds, sometimes called extreme pressure (EP) additives, provide for even greater lubricity. In general, these cutting oils do not cool as well as water-miscible fluids.

Water-Miscible Fluids.—*Emulsions or soluble oils* are a suspension of oil droplets in water. These suspensions are made by blending the oil with emulsifying agents (soap and soaplike materials) and other materials. These fluids combine the lubricating and rust-prevention properties of oil with water's excellent cooling properties. Their properties are affected by the emulsion concentration, with "lean" concentrations providing better cooling but poorer lubrication, and with "rich" concentrations having the opposite effect. Additions of sulfur, chlorine, and phosphorus, as with cutting oils, yield "extreme pressure" (EP) grades.

Chemical fluids are true solutions composed of organic and inorganic materials dissolved in water. Inactive types are usually clear fluids combining high rust inhibition, high cooling, and low lubricity characteristics with high surface tension. Surface-active types include wetting agents and possess moderate rust inhibition, high cooling, and moderate lubricating properties with low surface tension. They may also contain chlorine and/or sulfur compounds for extreme pressure properties.

Semichemical fluids are combinations of chemical fluids and emulsions. These fluids have a lower oil content but a higher emulsifier and surface-active-agent content than emulsions, producing oil droplets of much smaller diameter. They possess low surface tension, moderate lubricity and cooling properties, and very good rust inhibition. Sulfur, chlorine, and phosphorus also are sometimes added.

Selection of Cutting Fluids for Different Materials and Operations.—The choice of a cutting fluid depends on many complex interactions including the machinability of the metal; the severity of the operation; the cutting tool material; metallurgical, chemical, and human compatibility; fluid properties, reliability, and stability; and finally cost. Other factors affect results. Some shops standardize on a few cutting fluids which have to serve all purposes. In other shops, one cutting fluid must be used for all the operations performed on a machine. Sometimes, a very severe operating condition may be alleviated by applying the "right" cutting fluid manually while the machine supplies the cutting fluid for other operations through its coolant system. Several voluminous textbooks are available with specific recommendations for the use of particular cutting fluids for almost every combination of machining operation and workpiece and tool material. In general, when experience is lacking, it is wise to consult the material supplier and/or any of the many suppliers of different cutting fluids for advice and recommendations. Another excellent source is the Machinability Data Center, one of the many information centers supported by the U.S. Department of Defense. While the following recommendations represent good practice, they are to serve as a guide only, and it is not intended to say that other cutting fluids will not, in certain specific cases, also be effective.

Steels: Caution should be used when using a cutting fluid on steel that is being turned at a high cutting speed with cemented carbide cutting tools. See *Application of Cutting Fluids to Carbides* later. Frequently this operation is performed dry. If a cutting fluid is used, it should be a soluble oil mixed to a consistency of about 1 part oil to 20 to 30 parts water. A sulfurized mineral oil is recommended for reaming with carbide tipped reamers although a heavy-duty soluble oil has also been used successfully.

The cutting fluid recommended for machining steel with high speed cutting tools depends largely on the severity of the operation. For ordinary turning, boring, drilling, and milling on medium and low strength steels, use a soluble oil having a consistency of 1 part oil to 10 to 20 parts water. For tool steels and tough alloy steels, a heavy-duty soluble oil having a consistency of 1 part oil to 10 parts water is recommended for turning and milling. For drilling and reaming these materials, a light sulfurized mineral-fatty oil is used. For tough operations such as tapping, threading, and broaching, a sulfochlorinated mineral-fatty oil is recommended for tool steels and high-strength steels, and a heavy sulfurized mineral-fatty oil or a sulfochlorinated mineral oil can be used for medium- and low-strength steels. Straight sulfurized mineral oils are often recommended for machining tough, stringy low carbon steels to reduce tearing and produce smooth surface finishes.

Stainless Steel: For ordinary turning and milling a heavy-duty soluble oil mixed to a consistency of 1 part oil to 5 parts water is recommended. Broaching, threading, drilling, and reaming produce best results using a sulfochlorinated mineral-fatty oil.

Copper Alloys: Most brasses, bronzes, and copper are stained when exposed to cutting oils containing active sulfur and chlorine; thus, sulfurized and sulfochlorinated oils should not be used. For most operations a straight soluble oil, mixed to 1 part oil and 20 to 25 parts water is satisfactory. For very severe operations and for automatic screw machine work a mineral-fatty oil is used. A typical mineral-fatty oil might contain 5 to 10 per cent lard oil with the remainder mineral oil.

Monel Metal: When turning this material, an emulsion gives a slightly longer tool life than a sulfurized mineral oil, but the latter aids in chip breakage, which is frequently desirable.

Aluminum Alloys: Aluminum and aluminum alloys are frequently machined dry. When a cutting fluid is used it should be selected for its ability to act as a coolant. Soluble oils mixed to a consistency of 1 part oil to 20 to 30 parts water can be used. Mineral oil-base cutting fluids, when used to machine aluminum alloys, are frequently cut back to increase their viscosity so as to obtain good cooling characteristics and to make them flow easily to cover the tool and the work. For example, a mineral-fatty oil or a mineral plus a sulfurized fatty oil can be cut back by the addition of as much as 50 per cent kerosene.

Cast Iron: Ordinarily, cast iron is machined dry. Some increase in tool life can be obtained or a faster cutting speed can be used with a chemical cutting fluid or a soluble oil mixed to consistency of 1 part oil and 20 to 40 parts water. A soluble oil is sometimes used to reduce the amount of dust around the machine.

Magnesium: Magnesium may be machined dry, or with an air blast for cooling. A light mineral oil of low acid content may be used on difficult cuts. Coolants containing water should not be used on magnesium because of the danger of releasing hydrogen caused by reaction of the chips with water. Proprietary water-soluble oil emulsions containing inhibitors that reduce the rate of hydrogen generation are available.

Grinding: Soluble oil emulsions or emulsions made from paste compounds are used extensively in precision grinding operations. For cylindrical grinding, 1 part oil to 40 to 50 parts water is used. Solution type fluids and translucent grinding emulsions are particularly suited for many fine-finish grinding applications. Mineral oil-base grinding fluids are recommended for many applications where a fine surface finish is required on the ground surface. Mineral oils are used with vitrified wheels but are not recommended for wheels with rubber or shellac bonds. Under certain conditions the oil vapor mist caused by the action of the grinding wheel can be ignited by the grinding sparks and explode. To quench the grinding spark a secondary coolant line to direct a flow of grinding oil below the grinding wheel is recommended.

Broaching: For steel, a heavy mineral oil such as sulfurized oil of 300 to 500 Saybolt viscosity at 100 degrees F can be used to provide both adequate lubricating effect and a dampening of the shock loads. Soluble oil emulsions may be used for the lighter broaching operations.

Cutting Fluids for Turning, Milling, Drilling and Tapping.—The following table, *Cutting Fluids Recommended for Machining Operations*, gives specific cutting oil recommendations for common machining operations.

Soluble Oils: Types of oils paste compounds that form emulsions when mixed with water: Soluble oils are used extensively in machining both ferrous and non-ferrous metals when the cooling quality is paramount and the chip-bearing pressure is not excessive. Care should be taken in selecting the proper soluble oil for precision grinding operations. Grinding coolants should be free from fatty materials that tend to load the wheel, thus affecting the finish on the machined part. Soluble coolants should contain rust preventive constituents to prevent corrosion.

Base Oils: Various types of highly sulfurized and chlorinated oils containing inorganic, animal, or fatty materials. This “base stock” usually is “cut back” or blended with a lighter oil, unless the chip-bearing pressures are high, as when cutting alloy steel. Base oils usually have a viscosity range of from 300 to 900 seconds at 100 degrees F.

Mineral Oils: This group includes all types of oils extracted from petroleum such as paraffin oil, mineral seal oil, and kerosene. Mineral oils are often blended with base stocks, but they are generally used in the original form for light machining operations on both free-machining steels and non-ferrous metals. The coolants in this class should be of a type that has a relatively high flash point. Care should be taken to see that they are nontoxic, so that they will not be injurious to the operator. The heavier mineral oils (paraffin oils) usually have a viscosity of about 100 seconds at 100 degrees F. Mineral seal oil and kerosene have a viscosity of 35 to 60 seconds at 100 degrees F.

Cutting Fluids Recommended for Machining Operations

Material to be Cut	Turning	Milling
Aluminum[a]	Mineral Oil with 10 Per cent Fat (or) Soluble Oil	Soluble Oil (96 Per Cent Water) (or) Mineral Seal Oil (or) Mineral Oil
Alloy Steels[b]	25 Per Cent Sulfur base Oil[b] with 75 Per Cent Mineral Oil	10 Per Cent Lard Oil with 90 Per Cent Mineral Oil
Brass	Mineral Oil with 10 Per Cent Fat	Soluble Oil (96 Per Cent Water)
Tool Steels and Low-carbon Steels	25 Per Cent Lard Oil with 75 Per Cent Mineral Oil	Soluble Oil
Copper	Soluble Oil	Soluble Oil
Monel Metal	Soluble Oil	Soluble Oil
Cast Iron[c]	Dry	Dry
Malleable Iron	Soluble Oil	Soluble Oil
Bronze	Soluble Oil	Soluble Oil
Magnesium[d]	10 Per Cent Lard Oil with 90 Per Cent Mineral Oil	Mineral Seal Oil
Material to be Cut	**Drilling**	**Tapping**
Aluminum[e]	Soluble Oil (75 to 90 Per Cent Water) (or) 10 Per Cent Lard Oil with 90 Per Cent Mineral Oil	Lard Oil (or) Sperm Oil (or) Wool Grease (or) 25 Per Cent Sulfur-base Oil[b] Mixed with Mineral Oil
Alloy Steels[b]	Soluble Oil	30 Per Cent Lard Oil with 70 Per Cent Mineral Oil
Brass	Soluble Oil (75 to 90 Per Cent Water) (or) 30 Per Cent Lard Oil with 70 Per Cent Mineral Oil	10 to 20 Per Cent Lard Oil with Mineral Oil
Tool Steels and Low-carbon Steels	Soluble Oil	25 to 40 Per Cent Lard Oil with Mineral Oil (or) 25 Per Cent Sulfur-base Oil[b] with 75 Per Cent Mineral Oil
Copper	Soluble Oil	Soluble Oil
Monel Metal	Soluble Oil	25 to 40 Per Cent Lard Oil Mixed with Mineral Oil (or) Sulfur-base Oil[b] Mixed with Mineral Oil
Cast Iron[c]	Dry	Dry (or) 25 Per Cent Lard Oil with 75 Per Cent Mineral Oil
Malleable Iron	Soluble Oil	Soluble Oil
Bronze	Soluble Oil	20 Per Cent Lard Oil with 80 Per Cent Mineral Oil
Magnesium[d]	60-second Mineral Oil	20 Per Cent Lard Oil with 80 Per Cent Mineral Oil

[a] In machining aluminum, several varieties of coolants may be used. For rough machining, where the stock removal is sufficient to produce heat, water soluble mixtures can be used with good results to dissipate the heat. Other oils that may be recommended are straight mineral seal oil; a 50–50 mixture of mineral seal oil and kerosene; a mixture of 10 per cent lard oil with 90 per cent kerosene; and a 100-second mineral oil cut back with mineral seal oil or kerosene.

[b] The sulfur-base oil referred to contains $4\frac{1}{2}$ per cent sulfur compound. Base oils are usually dark in color. As a rule, they contain sulfur compounds resulting from a thermal or catalytic refinery process. When so processed, they are more suitable for industrial coolants than when they have had such compounds as flowers of sulfur added by hand. The adding of sulfur compounds by hand to the coolant reservoir is of temporary value only, and the non-uniformity of the solution may affect the machining operation.

[c] A soluble oil or low-viscosity mineral oil may be used in machining cast iron to prevent excessive metal dust.

[d] When a cutting fluid is needed for machining magnesium, low or nonacid mineral seal or lard oils are recommended. Coolants containing water should not be used because of the fire danger when magnesium chips react with water, forming hydrogen gas.

[e] Sulfurized oils ordinarily are not recommended for tapping aluminum; however, for some tapping operations they have proved very satisfactory, although the work should be rinsed in a solvent right after machining to prevent discoloration.

Application of Cutting Fluids to Carbides.—Turning, boring, and similar operations on lathes using carbides are performed dry or with the help of soluble oil or chemical cutting fluids. The effectiveness of cutting fluids in improving tool life or by permitting higher cutting speeds to be used, is less with carbides than with high-speed steel tools. Furthermore, the effectiveness of the cutting fluid is reduced as the cutting speed is increased. Cemented carbides are very sensitive to sudden changes in temperature and to temperature gradients within the carbide. Thermal shocks to the carbide will cause thermal cracks to form near the cutting edge, which are a prelude to tool failure. An unsteady or interrupted flow of the coolant reaching the cutting edge will generally cause these thermal cracks. The flow of the chip over the face of the tool can cause an interruption to the flow of the coolant reaching the cutting edge even though a steady stream of coolant is directed at the tool. When a cutting fluid is used and frequent tool breakage is encountered, it is often best to cut dry. When a cutting fluid must be used to keep the workpiece cool for size control or to allow it to be handled by the operator, special precautions must be used. Sometimes applying the coolant from the front and the side of the tool simultaneously is helpful. On lathes equipped with overhead shields, it is very effective to apply the coolant from below the tool into the space between the shoulder of the work and the tool flank, in addition to applying the coolant from the top. Another method is not to direct the coolant stream at the cutting tool at all but to direct it at the workpiece above or behind the cutting tool.

The danger of thermal cracking is great when milling with carbide cutters. The nature of the milling operation itself tends to promote thermal cracking because the cutting edge is constantly heated to a high temperature and rapidly cooled as it enters and leaves the workpiece. For this reason, carbide milling operations should be performed dry.

Lower cutting-edge temperatures diminish the danger of thermal cracking. The cutting-edge temperatures usually encountered when reaming with solid carbide or carbide-tipped reamers are generally such that thermal cracking is not apt to occur except when reaming certain difficult-to-machine metals. Therefore, cutting fluids are very effective when used on carbide reamers. Practically every kind of cutting fluid has been used, depending on the job material encountered. For difficult surface-finish problems in holes, heavy duty soluble oils, sulfurized mineral-fatty oils, and sulfochlorinated mineral-fatty oils have been used successfully. On some work, the grade and the hardness of the carbide also have an effect on the surface finish of the hole.

Cutting fluids should be applied where the cutting action is taking place and at the highest possible velocity without causing splashing. As a general rule, it is preferable to supply from 3 to 5 gallons per minute for each single-point tool on a machine such as a turret lathe or automatic. The temperature of the cutting fluid should be kept below 110 degrees F. If the volume of fluid used is not sufficient to maintain the proper temperature, means of cooling the fluid should be provided.

Cutting Fluids for Machining Magnesium.—In machining magnesium, it is the general but not invariable practice in the United States to use a cutting fluid. In other places, magnesium usually is machined dry except where heat generated by high cutting speeds would not be dissipated rapidly enough without a cutting fluid. This condition may exist when, for example, small tools without much heat-conducting capacity are employed on automatics.

The cutting fluid for magnesium should be an anhydrous oil having, at most, a very low acid content. Various mineral-oil cutting fluids are used for magnesium.

Occupational Exposure To Metalworking Fluids

The term *metalworking fluids* (MWFs) describes coolants and lubricants used during the fabrication of products from metals and metal substitutes. These fluids are used to prolong the life of machine tools, carry away debris, and protect or treat the surfaces of the material being processed. MWFs reduce friction between the cutting tool and work surfaces, reduce wear and galling, protect surface characteristics, reduce surface adhesion or welding, carry away generated heat, and flush away swarf, chips, fines, and residues. Table 1 describes the four different classes of metal working fluids:

Table 1. Classes of Metalworking fluids (MWFs)

MWF	Description	Dilution factor
Straight oil (neat oil or cutting oil)	Highly refined petroleum oils (lubricant-base oils) or other animal, marine, vegetable, or synthetic oils used singly or in combination with or without additives. These are lubricants, or function to improve the finish on the metal cut, and prevent corrosion.	none
Soluble oil (emulsifiable oil)	Combinations of 30% to 85% highly refined, high-viscosity lubricant-base oils and emulsifiers that may include other performance additives. Soluble oils are diluted with water before use at ratios of parts water.	1 part concentrate to 5 to 40 parts water
Semisynthetic	Contain smaller amounts of severely refined lubricant-base oil (5 to 30% in the concentrate), a higher proportion of emulsifiers that may include other performance additives, and 30 to 50% water.	1 part concentrate to 10 to 40 parts water
Synthetic[a]	Contain no petroleum oils and may be water soluble or water dispersible. The simplest synthetics are made with organic and inorganic salts dissolved in water. Offer good rust protection and heat removal but usually have poor lubricating ability. May be formulated with other performance additives. Stable, can be made bioresistant.	1 part concentrate to 10 to 40 parts water

[a] Over the last several decades major changes in the U.S. machine tool industry have increased the consumption of MWFs. Specifically, the use of synthetic MWFs increased as tool and cutting speeds increased.

Occupational Exposures to Metal Working Fluids (MWFs).—Workers can be exposed to MWFs by inhalation of aerosols (mists) or by skin contact resulting in an increased risk of respiratory (lung) and skin disease. Health effects vary based on the type of MWF, route of exposure, concentration, and length of exposure.

Skin contact usually occurs when the worker dips his/her hands into the fluid, floods the machine tool, or handling parts, tools, equipment or workpieces coated with the fluid, without the use of personal protective equipment such as gloves and apron. Skin contact can also result from fluid splashing onto worker from the machine if guarding is absent or inadequate.

Inhalation exposures result from breathing MWF mist or aerosol. The amount of mist generated (and the severity of the exposure) depends on a variety of factors: the type of MWF and its application process; the MWF temperature; the specific machining or grinding operation; the presence of splash guarding; and the effectiveness of the ventilation system. In general, the exposure will be higher if the worker is in close proximity to the machine, the operation involves high tool speeds and deep cuts, the machine is not enclosed, or if ventilation equipment was improperly selected or poorly maintained. In addition, high-pressure and/or excessive fluid application, contamination of the fluid with tramp oils, and improper fluid selection and maintenance will tend to result in higher exposure.

Each MWF class consists of a wide variety of chemicals used in different combinations and the risk these chemicals pose to workers may vary because of different manufacturing processes, various degrees of refining, recycling, improperly reclaimed chemicals, different degrees of chemical purity, and potential chemical reactions between components.

Exposure to hazardous contaminants in MWFs may present health risks to workers. Contamination may occur from: process chemicals and ancillary lubricants inadvertently introduced; contaminants, metals, and alloys from parts being machined; water and cleaning agents used for routine housekeeping; and, contaminants from other environmental sources at the worksite. In addition, bacterial and fungal contaminants may metabolize and degrade the MWFs to hazardous end-products as well as produce endotoxins.

The improper use of biocides to manage microbial growth may result in potential health risks. Attempts to manage microbial growth solely with biocides may result in the emergence of biocide-resistant strains from complex interactions that may occur among different member species or groups within the population. For example, the growth of one species, or the elimination of one group of organisms may permit the overgrowth of another. Studies also suggest that exposure to certain biocides can cause either allergic or contact dermatitis.

Fluid Selection, Use, and Application.—The MWFs selected should be as nonirritating and nonsensitizing as possible while remaining consistent with operational requirements. Petroleum-containing MWFs should be evaluated for potential carcinogenicity using ASTM Standard D1687-95, “Determining Carcinogenic Potential of Virgin Base Oils in Metalworking Fluids”. If soluble oil or synthetic MWFs are used, ASTM Standard E1497-94, “Safe Use of Water-Miscible Metalworking Fluids” should be consulted for safe use guidelines, including those for product selection, storage, dispensing, and maintenance. To minimize the potential for nitrosamine formation, nitrate-containing materials should not be added to MWFs containing ethanolamines.

Many factors influence the generation of MWF mists, which can be minimized through the proper design and operation of the MWF delivery system. ANSI Technical Report B11 TR2-1997, “Mist Control Considerations for the Design, Installation and Use of Machine Tools Using Metalworking Fluids” provides directives for minimizing mist and vapor generation. These include minimizing fluid delivery pressure, matching the fluid to the application, using MWF formulations with low oil concentrations, avoiding contamination with tramp oils, minimizing the MWF flow rate, covering fluid reservoirs and return systems where possible, and maintaining control of the MWF chemistry. Also, proper application of MWFs can minimize splashing and mist generation. Proper application includes: applying MWFs at the lowest possible pressure and flow volume consistent with provisions for adequate part cooling, chip removal, and lubrication; applying MWFs at the tool/workpiece interface to minimize contact with other rotating equipment; ceasing fluid delivery when not performing machining; not allowing MWFs to flow over the unprotected hands of workers loading or unloading parts; and using mist collectors engineered for the operation and specific machine enclosures.

Properly maintained filtration and delivery systems provide cleaner MWFs, reduce mist, and minimize splashing and emissions. Proper maintenance of the filtration and delivery systems includes: the selection of appropriate filters; ancillary equipment such as chip handling operations, dissolved air-flotation devices, belt-skimmers, chillers or plate and frame heat exchangers, and decantation tanks; guard coolant return trenches to prevent dumping of floor wash water and other waste fluids; covering sumps or coolant tanks to prevent contamination with waste or garbage (e.g., cigarette butts, food, etc.); and, keeping the machine(s) clean of debris. Parts washing before machining can be an important part of maintaining cleaner MWFs.

Since all additives will be depleted with time, the MWF and additives concentrations should be monitored frequently so that components and additives can be made up as needed. The MWF should be maintained within the pH and concentration ranges recommended by the formulator or supplier. MWF temperature should be maintained at the lowest practical level to slow the growth of microorganisms, reduce water losses and changes in viscosity, and–in the case of straight oils–reduce fire hazards.

Fluid Maintenance.—Drums, tanks, or other containers of MWF concentrates should be stored appropriately to protect them from outdoor weather conditions and exposure to low or high temperatures. Extreme temperature changes may destabilize the fluid concentrates, especially in the case of concentrates mixed with water, and cause water to seep into unopened drums encouraging bacterial growth. MWFs should be maintained at as low a temperature as is practical. Low temperatures slow the growth of microorganisms, reduce water losses and change in viscosity, and in the case of straight oils, reduce the fire hazard risks.

To maintain proper MWF concentrations, neither water nor concentrate should be used to top off the system. The MWF mixture should be prepared by first adding the concentrate to the clean water (in a clean container) and then adding the emulsion to that mixture in the coolant tank. MWFs should be mixed just before use; large amounts should not be stored, as they may deteriorate before use.

Personal Protective Clothing: Personal protective clothing and equipment should always be worn when removing MWF concentrates from the original container, mixing and diluting concentrate, preparing additives (including biocides), and adding MWF emulsions, biocides, or other potentially hazardous ingredients to the coolant reservoir. Personal protective clothing includes eye protection or face shields, gloves, and aprons which do not react with but shed MWF ingredients and additives.

System Service: Coolant systems should be regularly serviced, and the machines should be rigorously maintained to prevent contamination of the fluids by tramp oils (e.g., hydraulic oils, gear box oils, and machine lubricants leaking from the machines or total loss slideway lubrication). Tramp oils can destabilize emulsions, cause pumping problems, and clog filters. Tramp oils can also float to the top of MWFs, effectively sealing the fluids from the air, allowing metabolic products such as volatile fatty acids, mercaptols, scatols, ammonia, and hydrogen sulfide are produced by the anaerobic and facultative anaerobic species growing within the biofilm to accumulate in the reduced state.

When replacing the fluids, thoroughly clean all parts of the system to inhibit the growth of microorganisms growing on surfaces. Some bacteria secrete layers of slime that may grow in stringy configurations that resemble fungal growth. Many bacteria secrete polymers of polysaccharide and/or protein, forming a glycocalyx which cements cells together much as mortar holds bricks. Fungi may grow as masses of hyphae forming mycelial mats. The attached community of microorganisms is called a biofilm and may be very difficult to remove by ordinary cleaning procedures.

Biocide Treatment: Biocides are used to maintain the functionality and efficacy of MWFs by preventing microbial overgrowth. These compounds are often added to the stock fluids as they are formulated, but over time the biocides are consumed by chemical and biological demands Biocides with a wide spectrum of biocidal activity should be used to suppress the growth of the widely diverse contaminant population. Only the concentration of biocide needed to meet fluid specifications should be used since overdosing could lead to skin or respiratory irritation in workers, and under-dosing could lead to an inadequate level of microbial control.

Ventilation Systems: The ventilation system should be designed and operated to prevent the accumulation or recirculation of airborne contaminants in the workplace. The ventilation system should include a positive means of bringing in at least an equal volume of air from the outside, conditioning it, and evenly distributing it throughout the exhausted area.

Exhaust ventilation systems function through suction openings placed near a source of contamination. The suction opening or exhaust hood creates and air motion sufficient to overcome room air currents and any airflow generated by the process. This airflow captures the contaminants and conveys them to a point where they can either be discharged or removed from the airstream. Exhaust hoods are classified by their position relative to the process as canopy, side draft, down draft or enclosure. ANSI Technical Report B11 TR 2-1997 contains guidelines for exhaust ventilation of machining and grinding operations. Enclosures are the only type of exhaust hood recommended by the ANSI committee. They consist of physical barriers between the process and the worker's environment. Enclosures can be further classified by the extent of the enclosure: close capture (enclosure of the point of operation, total enclosure (enclosure of the entire machine), or tunnel enclosure (continuous enclosure over several machines).

If no fresh make up air is introduced into the plant, air will enter the building through open doors and windows, potentially causing cross-contamination of all process areas. Ideally, all air exhausted from the building should be replaced by tempered air from an uncontaminated location. By providing a slight excess of make up air in relatively clean areas and s slight deficit of make up air in dirty areas, cross-contamination can be reduced. In addition, this air can be channeled directly to operator work areas, providing the cleanest possible work environment. Ideally, this fresh air should be supplied in the form of a low-velocity air shower (<100 ft/min to prevent interference with the exhaust hoods) directly above the worker.

Protective Clothing and Equipment: Engineering controls are used to reduce worker exposure to MWFs. But in the event of airborne exposures that exceed the NIOSH REL or dermal contact with the MWFs, the added protection of chemical protective clothing (CPC) and respirators should be provided. Maintenance staff may also need CPC because their work requires contact with MWFs during certain operations. All workers should be trained in the proper use and care of CPC. After any item of CPC has been in routine use, it should be examined to ensure that its effectiveness has not been compromised.

Selection of the appropriate respirator depends on the operation, chemical components, and airborne concentrations in the worker's breathing zone. Table 2. lists the NIOSH- recommended respiratory protection for workers exposed to MWF aerosol.

Table 2. Respiratory Protection for Workers Exposed to MWF Aerosols[*]

Concentration of MWF aerosol (mg/m^3)	Minimum respiratory protection[a]
#0.5 mg/m^3 (1 × REL)[b]	No respiratory protection required for healthy workers[c]
#5.0 mg/m^3 (10 × REL)	Any air-purifying, half-mask respirator including a disposable respirator[d,e] equipped with any P- or R-series particulate filter (P95, P99, P100, R95, R99, or R100) number
#12.5 mg/m^3 (25 × REL)	Any powered, air-purifying respirator equipped with a hood or helmet and a HEPA filter[f]

[a] Respirators with higher assigned protection factors (APFs) may be substituted for those with lower APFs [NIOSH 1987a].

[b] APF times the NIOSH REL for total particulate mass. The APF [NIOSH 1987b] is the minimum anticipated level of protection provided by each type of respirator.

[c] See text for recommendations regarding workers with asthma and for other workers affected by MWF aerosols.

[d] A respirator that should be discarded after the end of the manufacturer's recommended period of use or after a noticeable increase in breathing resistance or when physical damage, hygiene considerations, or other warning indicators render the respirator unsuitable for further use.

[e] An APF of 10 is assigned to disposable particulate respirators if they have been properly fitted.

[f] High-efficiency particulate air filter. When organic vapors are a potential hazard during metalworking operations, a combination particulate and organic vapor filter is necessary.

[*] Only NIOSH/MSHA-approved or NIOSH-approved (effective date July 10, 1995) respiratory equipment should be used.

MACHINING NONFERROUS METALS

Machining Aluminum.—Some of the alloys of aluminum have been machined successfully without any lubricant or cutting compound, but some form of lubricant is desirable to obtain the best results. For many purposes, a soluble cutting oil is good.

Tools for aluminum and aluminum alloys should have larger relief and rake angles than tools for cutting steel. For high-speed steel turning tools the following angles are recommended: relief angles, 14 to 16 degrees; back rake angle, 5 to 20 degrees; side rake angle, 15 to 35 degrees. For very soft alloys even larger side rake angles are sometimes used. High silicon aluminum alloys and some others have a very abrasive effect on the cutting tool. While these alloys can be cut successfully with high-speed-steel tools, cemented carbides are recommended because of their superior abrasion resistance. The tool angles recommended for cemented carbide turning tools are: relief angles, 12 to 14 degrees; back rake angle, 0 to 15 degrees; side rake angle, 8 to 30 degrees.

Cut-off tools and necking tools for machining aluminum and its alloys should have from 12 to 20 degrees back rake angle and the end relief angle should be from 8 to 12 degrees. Excellent threads can be cut with single-point tools in even the softest aluminum. Experience seems to vary somewhat regarding the rake angle for single-point thread cutting tools. Some prefer to use a rather large back and side rake angle although this requires a modification in the included angle of the tool to produce the correct thread contour. When both rake angles are zero, the included angle of the tool is ground equal to the included angle of the thread. Excellent threads have been cut in aluminum with zero rake angle thread-cutting tools using large relief angles, which are 16 to 18 degrees opposite the front side of the thread and 12 to 14 degrees opposite the back side of the thread. In either case, the cutting edges should be ground and honed to a keen edge. It is sometimes advisable to give the face of the tool a few strokes with a hone between cuts when chasing the thread to remove any built-up edge on the cutting edge.

Fine surface finishes are often difficult to obtain on aluminum and aluminum alloys, particularly the softer metals. When a fine finish is required, the cutting tool should be honed to a keen edge and the surfaces of the face and the flank will also benefit by being honed smooth. Tool wear is inevitable, but it should not be allowed to progress too far before the tool is changed or sharpened. A sulphurized mineral oil or a heavy-duty soluble oil will sometimes be helpful in obtaining a satisfactory surface finish. For best results, however, a diamond cutting tool is recommended. Excellent surface finishes can be obtained on even the softest aluminum and aluminum alloys with these tools.

Although ordinary milling cutters can be used successfully in shops where aluminum parts are only machined occasionally, the best results are obtained with coarse-tooth, large helix-angle cutters having large rake and clearance angles. Clearance angles up to 10 to 12 degrees are recommended. When slab milling and end milling a profile, using the peripheral teeth on the end mill, climb milling (also called down milling) will generally produce a better finish on the machined surface than conventional (or up) milling. Face milling cutters should have a large axial rake angle. Standard twist drills can be used without difficulty in drilling aluminum and aluminum alloys although high helix-angle drills are preferred. The wide flutes and high helix-angle in these drills helps to clear the chips. Sometimes split-point drills are preferred. Carbide tipped twist drills can be used for drilling aluminum and its alloys and may afford advantages in some production applications. Ordinary hand and machine taps can be used to tap aluminum and its alloys although spiral-fluted ground thread taps give superior results. Experience has shown that such taps should have a right-hand ground flute when intended to cut right-hand threads and the helix angle should be similar to that used in an ordinary twist drill.

Machining Magnesium.—Magnesium alloys are readily machined and with relatively low power consumption per cubic inch of metal removed. The usual practice is to employ high cutting speeds with relatively coarse feeds and deep cuts. Exceptionally fine finishes can be obtained so that grinding to improve the finish usually is unnecessary. The horsepower normally required in machining magnesium varies from 0.15 to 0.30 per cubic inch per minute. While this value is low, especially in comparison with power required for cast iron and steel, the total amount of power for machining magnesium usually is high because of the exceptionally rapid rate at which metal is removed.

Carbide tools are recommended for maximum efficiency, although high-speed steel frequently is employed. Tools should be designed so as to dispose of chips readily or without excessive friction, by employing polished chip-bearing surfaces, ample chip spaces, large clearances, and small contact areas. *Keen-edged tools should always be used.*

Feeds and Speeds for Magnesium: Speeds ordinarily range up to 5000 feet per minute for rough- and finish-turning, up to 3000 feet per minute for rough-milling, and up to 9000 feet per minute for finish-milling. For rough-turning, the following combinations of speed in feet per minute, feed per revolution, and depth of cut are recommended: Speed 300 to 600 feet per minute — feed 0.030 to 0.100 inch, depth of cut 0.5 inch; speed 600 to 1000 — feed 0.020 to 0.080, depth of cut 0.4; speed 1000 to 1500 — feed 0.010 to 0.060, depth of cut 0.3; speed 1500 to 2000 — feed 0.010 to 0.040, depth of cut 0.2; speed 2000 to 5000 — feed 0.010 to 0.030, depth of cut 0.15.

Lathe Tool Angles for Magnesium: The true or actual rake angle resulting from back and side rakes usually varies from 10 to 15 degrees. Back rake varies from 10 to 20, and side rake from 0 to 10 degrees. Reduced back rake may be employed to obtain better chip breakage. The back rake may also be reduced to from 2 to 8 degrees on form tools or other broad tools to prevent chatter.

Parting Tools: For parting tools, the back rake varies from 15 to 20 degrees, the front end relief 8 to 10 degrees, the side relief measured perpendicular to the top face 8 degrees, the side relief measured in the plane of the top face from 3 to 5 degrees.

Milling Magnesium: In general, the coarse-tooth type of cutter is recommended. The number of teeth or cutting blades may be one-third to one-half the number normally used; however, the two-blade fly-cutter has proved to be very satisfactory. As a rule, the land relief or primary peripheral clearance is 10 degrees followed by secondary clearance of 20 degrees. The lands should be narrow, the width being about $\frac{3}{64}$ to $\frac{1}{16}$ inch. The rake, which is positive, is about 15 degrees.

For rough-milling and speeds in feet per minute up to 900 — feed, inch per tooth, 0.005 to 0.025, depth of cut up to 0.5; for speeds 900 to 1500 — feed 0.005 to 0.020, depth of cut up to 0.375; for speeds 1500 to 3000 — feed 0.005 to 0.010, depth of cut up to 0.2.

Drilling Magnesium: If the depth of a hole is less than five times the drill diameter, an ordinary twist drill with highly polished flutes may be used. The included angle of the point may vary from 70 degrees to the usual angle of 118 degrees. The relief angle is about 12 degrees. The drill should be kept sharp and the outer corners rounded to produce a smooth finish and prevent burr formation. For deep hole drilling, use a drill having a helix angle of 40 to 45 degrees with large polished flutes of uniform cross-section throughout the drill length to facilitate the flow of chips. A pyramid-shaped “spur” or “pilot point” at the tip of the drill will reduce the “spiraling or run-off.”

Drilling speeds vary from 300 to 2000 feet per minute with feeds per revolution ranging from 0.015 to 0.050 inch.

Reaming Magnesium: Reamers up to 1 inch in diameter should have four flutes; larger sizes, six flutes. These flutes may be either parallel with the axis or have a negative helix angle of 10 degrees. The positive rake angle varies from 5 to 8 degrees, the relief angle from 4 to 7 degrees, and the clearance angle from 15 to 20 degrees.

Tapping Magnesium: Standard taps may be used unless Class 3B tolerances are required, in which case the tap should be designed for use in magnesium. A high-speed steel concentric type with a ground thread is recommended. The concentric form, which eliminates the radial thread relief, prevents jamming of chips while the tap is being backed out of the hole. The positive rake angle at the front may vary from 10 to 25 degrees and the “heel rake angle” at the back of the tooth from 3 to 5 degrees. The chamfer extends over two to three threads. For holes up to $\frac{1}{4}$ inch in diameter, two-fluted taps are recommended; for sizes from $\frac{1}{2}$ to $\frac{3}{4}$ inch, three flutes; and for larger holes, four flutes. Tapping speeds ordinarily range from 75 to 200 feet per minute, and mineral oil cutting fluid should be used.

Threading Dies for Magnesium: Threading dies for use on magnesium should have about the same cutting angles as taps. Narrow lands should be used to provide ample chip space. Either solid or self-opening dies may be used. The latter type is recommended when maximum smoothness is required. Threads may be cut at speeds up to 1000 feet per minute.

Grinding Magnesium: As a general rule, magnesium is ground dry. The highly inflammable dust should be formed into a sludge by means of a spray of water or low-viscosity mineral oil. Accumulations of dust or sludge should be avoided. For surface grinding, when a fine finish is desirable, a low-viscosity mineral oil may be used.

Machining Zinc Alloy Die-Castings.—Machining of zinc alloy die-castings is mostly done without a lubricant. For particular work, especially deep drilling and tapping, a lubricant such as lard oil and kerosene (about half and half) or a 50-50 mixture of kerosene and machine oil may be used to advantage. A mixture of turpentine and kerosene has been been found effective on certain difficult jobs.

Reaming: In reaming, tools with six straight flutes are commonly used, although tools with eight flutes irregularly spaced have been found to yield better results by one manufacturer. Many standard reamers have a land that is too wide for best results. A land about 0.015 inch wide is recommended but this may often be ground down to around 0.007 or even 0.005 inch to obtain freer cutting, less tendency to loading, and reduced heating.

Turning: Tools of high-speed steel are commonly employed although the application of Stellite and carbide tools, even on short runs, is feasible. For steel or Stellite, a positive top rake of from 0 to 20 degrees and an end clearance of about 15 degrees are commonly recommended. Where side cutting is involved, a side clearance of about 4 degrees minimum is recommended. With carbide tools, the end clearance should not exceed 6 to 8 degrees and the top rake should be from 5 to 10 degrees positive. For boring, facing, and other lathe operations, rake and clearance angles are about the same as for tools used in turning.

Machining Monel and Nickel Alloys.—These alloys are machined with high-speed steel and with cemented carbide cutting tools. High-speed steel lathe tools usually have a back rake of 6 to 8 degrees, a side rake of 10 to 15 degrees, and relief angles of 8 to 12 degrees. Broad-nose finishing tools have a back rake of 20 to 25 degrees and an end relief angle of 12 to 15 degrees. In most instances, standard commercial cemented-carbide tool holders and tool shanks can be used which provide an acceptable tool geometry. Honing the cutting edge lightly will help if chipping is encountered.

The most satisfactory tool materials for machining Monel and the softer nickel alloys, such as Nickel 200 and Nickel 230, are M2 and T5 for high-speed steel and crater resistant grades of cemented carbides. For the harder nickel alloys such as K Monel, Permanickel, Duranickel, and Nitinol alloys, the recommended tool materials are T15, M41, M42, M43, and for high-speed steel, M42. For carbides, a grade of crater resistant carbide is recommended when the hardness is less than 300 Bhn, and when the hardness is more than 300 Bhn, a grade of straight tungsten carbide will often work best, although some crater resistant grades will also work well.

A sulfurized oil or a water-soluble oil is recommended for rough and finish turning. A sulfurized oil is also recommended for milling, threading, tapping, reaming, and broaching. Recommended cutting speeds for Monel and the softer nickel alloys are 70 to 100 fpm for high-speed steel tools and 200 to 300 fpm for cemented carbide tools. For the harder nickel alloys, the recommended speed for high-speed steel is 40 to 70 fpm for a hardness up to 300 Bhn and for a higher hardness, 10 to 20 fpm; for cemented carbides, 175 to 225 fpm when the hardness is less than 300 Bhn and for a higher hardness, 30 to 70 fpm.

Nickel alloys have a high tendency to work harden. To minimize work hardening caused by machining, the cutting tools should be provided with adequate relief angles and positive rake angles. Furthermore, the cutting edges should be kept sharp and replaced when dull to prevent burnishing of the work surface. The depth of cut and feed should be sufficiently large to ensure that the tool penetrates the work without rubbing.

Machining Copper Alloys.—Copper alloys can be machined by tooling and methods similar to those used for steel, but at higher surface speeds. Machinability of copper alloys is discussed in the tables starting on page 526 and starting on page 533. Machinability is based on a rating of 100 per cent for the free-cutting alloy C35000, which machines with small, easily broken chips. As with steels, copper alloys containing lead have the best machining properties, with alloys containing tin, and lead, having machinability ratings of 80 and 70 per cent. Tellurium and sulphur are added to copper alloys to increase machinability with minimum effect on conductivity. Lead additions are made to facilitate machining, as their effect is to produce easily broken chips.

Copper alloys containing silicon, aluminum, manganese and nickel become progressively more difficult to machine, and produce long, stringy chips, the latter alloys having only 20 per cent of the machinability of the free-cutting alloys. Although copper is frequently machined dry, a cooling compound is recommended. Other lubricants that have been used include tallow for drilling, gasoline for turning, and beeswax for threading.

Machining Hard Rubber.—Tools suitable for steel may be used for hard rubber, with no top or side rake angles and 10 to 20 deg. clearance angles, of high speed steel or tungsten carbide. Without coolant, surface speeds of about 200 ft./min. are recommended for turning, boring and facing, and may be increased to 300 surface ft./min. with coolant.

Drilling of hard rubber requires high speed steel drills of 35 to 40 deg. helix angle to obtain maximum cutting speeds and drill life. Feed rates for drilling range up to 0.015 in./rev. Deep-fluted taps are best for threading hard rubber, and should be 0.002 to 0.005 in. oversize if close tolerances are to be held. Machine oil is used for a lubricant. Hard rubber may be sawn with band saws having 5 to 10 points per inch, running at about 3000 ft./min. or cut with abrasive wheels. Use of coolant in grinding rubber gives a smoother finish.

Piercing and blanking of sheet rubber is best performed with the rubber or dies heated. Straightening of the often-distorted blanks may be carried out by dropping them into a pan of hot water.

GRINDING FEEDS AND SPEEDS

Grinding data are scarcely available in handbooks, which usually recommend a small range of depths and work speeds at constant wheel speed, including small variations in wheel and work material composition. Wheel life or grinding stiffness are seldom considered.

Grinding parameter recommendations typically range as follows:

- Wheel speeds are usually recommended in the 1200 to 1800 m/min (4000 to 6000 fpm) range, or in rare cases up to 3600 m/min (12000 fpm)
- Work speeds are in the range 20 to 40 m/min (70 to 140 fpm); and, depths of cut of 0.01 to 0.025 mm (0.0004 to 0.001 inch) for roughing, and around 0.005 mm (.0002 in.) for finish grinding.
- Grit sizes for roughing are around 46 to 60 for easy-to-grind materials, and for difficult-to-grind materials higher such as 80 grit. In finishing, a smaller grit size (higher grit number) is recommended. Internal grinding grit sizes for small holes are approximately 100 to 320.
- Specific metal removal rate, *SMRR*, represents the rate of material removal per unit of wheel contact width and are commonly recommended from 200 to 500 mm^3/mm width/min (0.3 to 0.75 in^3/inch width/min).
- Grinding stiffness is a major variable in determining wheel-life and spark-out time. A typical value of system stiffness in outside-diameter grinding, for 10:1 length/diameter ratio, is approximately K_{ST} = 30–50 N/μm. System stiffness K_{ST} is calculated from the stiffness of the part, K_w and the machine and fixtures, K_m. Machine values can be obtained from manufacturers, or can be measured using simple equipment along with the part stiffness.
- Generally a lower wheel hardness (soft wheel) is recommended when the system stiffness is poor or when a better finish is desired.

Basic Rules

The wheel speed *V* and *equivalent chip thickness* $ECT = SMRR \div V \div 1000$ are the primary parameters that determine wheel-life, forces and surface finish in grinding. The following general rules and recommendations, using *ECT*, are based on extensive laboratory and industry tests both in Europe and USA. The relationships and shapes of curves pertaining to grinding tool-life, grinding time, and cost are similar to those of any metal cutting operation such as turning, milling and drilling.

In turning and milling, the *ECT* theory says that if the product of feed times depth of cut is constant, the tool-life is constant no matter how the depth of cut or feed is varied, provided that the cutting speed and cutting edge length are maintained constant.

In grinding, wheel-life *T* remains constant for constant cutting speed *V*, regardless of how depth of cut a_r or work speed V_w are selected as long as the specific metal removal rate $SMMR = V_w \times a_r$ is held constant (neglecting the influence of grinding contact width).

ECT is much smaller in grinding than in milling, ranging from about 0.0001 to 0.001 mm (0.000004 to 0.00004 inch). See the section *MACHINING ECONOMETRICS* starting on page 1056 for a detailed explanation of the role of *ECT* in conventional machining.

Wheel life *T* and *Grinding Ratio*.—A commonly used measure of relative wheel-life in grinding is the *grinding ratio* that is used to compare grindability when varying grinding wheel composition and work material properties under otherwise constant cutting conditions.

The *grinding ratio* is defined as the slope of the wear curve versus metal removal rate: *grinding ratio* = $MRR \div W^*$, where *MRR* is the metal removal rate, and W^* is the volume wheel wear at which the wheel has to be dressed. The grinding ratio is not a measure of wheel-life, but a relationship between *grinding ratio* and wheel-life *T* can be obtained from the formula $grinding\ ratio = SMRR \times T \div W^*$, where $SMRR$ (specific metal removal rate) is determined from $MRR = SMRR \times T$ or from $ECT = SMRR \div V \div 1000$.

Thus, $grinding\ ratio = 1000 \times ECT \times V \times T \div W^*$, and $T = grinding\ ratio \times W^* \div (1000 \times ECT \times V)$, provided that the wheel wear criterion W^* is valid for all data combinations.

Example 1: If W^* in one test is found to be 500 mm^3 for $ECT = 0.00033$ mm and $V = 3600$ m/min, and $grinding\ ratio = 10$, then wheel-life will vary with measured grinding ratios, wheel speed, and ECT as follows: $T = 500 \times grinding\ ratio \div (V \times ECT) = 4.2$ minutes.

In the remainder of this section the *grinding ratio* will not used, and wheel-life is expressed in terms of ECT or $SMRR$ and wheel speed V.

***ECT* in Grinding.—**In turning and milling, ECT is defined as the volume of chips removed per unit cutting edge length per revolution of the work or cutter. In milling specifically, ECT is defined as the ratio of (number of teeth $z \times$ feed per tooth $f_z \times$ radial depth of cut $a_r \times$ and axial depth of cut a_a) and (cutting edge length CEL divided by πD), where D is the cutter diameter, thus,

$$ECT = \frac{\pi D z f_z a_r a_a}{CEL}$$

In grinding, the same definition of ECT applies if we replace the number of teeth with the average number of grits along the wheel periphery, and replace the feed per tooth by the average feed per grit. This definition is not very practical, however, and ECT is better defined by the ratio of the specific metal removal rate $SMMR$, and the wheel speed V. Thus, $ECT = 1000 \times SMRR \div V$. Keeping ECT constant when varying $SMRR$ requires that the wheel speed must be changed proportionally.

In milling and turning ECT can also be redefined in terms of $SMRR$ divided by the work and the cutter speeds, respectively, because $SMRR$ is proportional to the feed rate F_R.

Work Speed and Depth of Cut Selection: Work speed V_w is determined by dividing $SMMR$ by the depth of cut a_r, or by using the graph in Fig. 1.

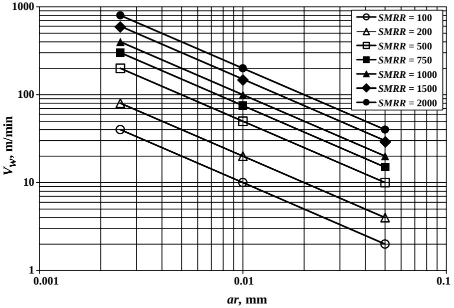


Fig. 1. Work speed V_w vs. depth of cut a_r

Referring to Fig. 1, for depths of cuts of 0.01 and 0.0025 mm, a specific metal removal rate $SMMR = 1000$ mm^3/mm width/min is achieved at work speeds of 100 and 400 m/min, respectively, and for $SMMR = 100$ mm^3/mm width/min at work speeds of 10 and 40 m/min, respectively.

Unfortunately, the common use of low values of work speed (20 to 40 m/min) in finishing cause thermal surface damage, disastrous in critical parts such as aircraft components. As the grains slide across the work they generate surface heat and fatigue-type loading may cause residual tensile stresses and severe surface cracks. Proper finish grinding conditions are obtained by increasing the work speed 5 to 10 times higher than the above recommendations indicate. These higher work speeds will create compressive stresses that are not detrimental to the surface. The by-product of higher work speeds is much higher *SMRR* values and thereby much shorter grinding times. Compressive stresses are also obtained by reducing the depth of cut a_r.

Wheel Life Relationships and Optimum Grinding Data.—Figs. 2a, 2b, and 2c show, in three planes, the 3-dimensional variation of wheel-life *T* with wheel speed *V* and *ECT* when grinding a hardened tool steel. Fig. 2a depicts wheel-life versus wheel speed (the *T–V* plane) with constant *ECT* appearing as approximately straight lines when plotted in log-log coordinates.

In grinding, the wheel-life variation follows curves similar to those obtained for conventional metal cutting processes, including a bend-off of the Taylor lines (*T–V* graph) towards shorter life and lower cutting speeds when a certain maximum life is achieved for each value of *ECT*. In the two other planes (*T–ECT*, and *V–ECT*) we usually find smooth curves in which the maximum values of wheel-life are defined by points along a curve called the *H*–curve.

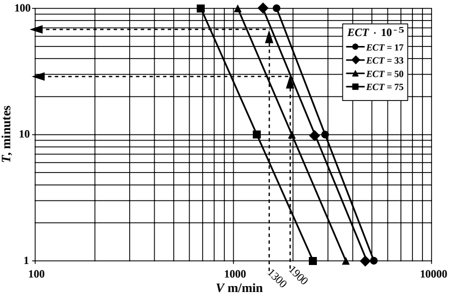


Fig. 2a. Taylor lines: *T* vs. *V*, *ECT* plotted for grinding M4 tool steel, hardness Rc 64

Example 2: The variation of *SMRR* = *V* × *ECT* × 1000 and wheel-life at various wheel speeds can be obtained from Fig. 2a. Using sample values of *ECT* = 33 × 10^{-5} mm and *V* = 1300 and 1900 m/min, *SMRR* = 1300 × 33 × 10^{-5} × 1000= 429, and 1900 × 33 × 10^{-5} × 1000 = 627 mm^3/mm width/min, respectively; the corresponding wheel lives are read off as approximately 70 and 30 minutes, respectively.

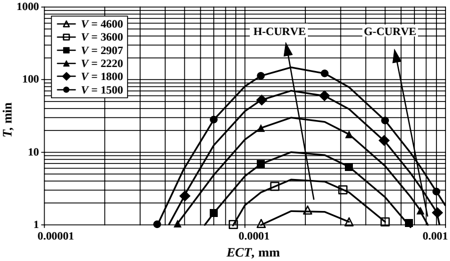


Fig. 2b. *T* vs. *ECT*, *V* plotted

Fig. 2b depicts wheel-life *T* versus *ECT* with constant wheel speed *V* shown as curves plotted in log-log coordinates, similar to those for the other cutting operations.

Example 3: Fig. 2b shows that maximum values of wheel-life occur along the *H*-curve. For the 3 speeds 1800, 2700, and 3600 m/min, maximum wheel lives are approximately 70, 14 and 4 minutes, respectively, at *ECT* around 17×10^{-5} through 20×10^{-5} mm along the *H*-curve. Left and right of the *H*-curve wheel-life is shorter.

Fig. 2c depicts wheel speed *V* versus *ECT* with wheel-life *T* parameter shown as curves in log-log coordinates, similar to those for the other cutting operations, with the characteristic *H*- and *G*-curves.

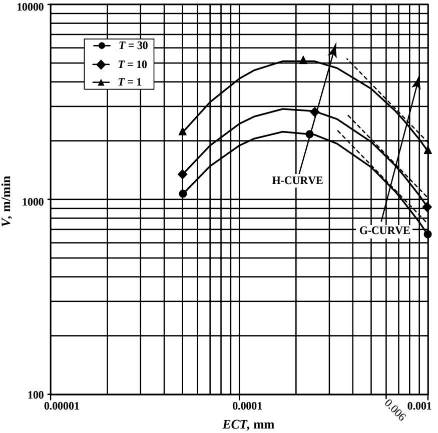


Fig. 2c. *V* vs. *ECT*, *T* plotted

Optimum grinding data for roughing occur along the *G*-curve, which is determined from the *V-ECT* graph by drawing 45-degree lines tangent to the *T*-curves, as shown in Fig. 2c, and drawing a line (the *G*-curve) through the points of tangency on the respective *T*-curves, thus the location and direction of the *G*-curve is determined. Globally optimum data correspond to the *T*-curve for which wheel-life is calculated using the corresponding equivalent tooling-cost time, T_V, calculated from $T_V = T_{RPL} + 60 \times C_E \div H_R$, minutes, where T_{RPL} is the time required to replace wheel, C_E = cost per wheel dressing = wheel cost + cost per dressing, and H_R is the hourly rate.

Minimum cost conditions occur along the *G*-curve; if optimum life T_O was determined at either 10 or 30 minutes then V_O = 1500 and 1100 m/min, respectively, and *ECT* is around $65 - 70 \times 10^{-5}$ mm in both cases. The corresponding optimum values of *SMRR* are $1000 \times 1500 \times 67 \times 10^{-5} = 1000$ and $1000 \times 1100 \times 67 \times 10^{-5} = 740$ mm^3/min/mm wheel contact width (1.5 to 1.1in^3/in/min).

Using Fig. 1 we find optimum work speeds for depths of cut a_r = 0.01 and 0.005 mm to be V_w = 100 and 75 m/min, and 200 and 150 m/min (330 and 250 fpm, and 660 and 500 fpm) respectively for 10- and 30-minute wheel-life.

These high work speeds are possible using proper dressing conditions, high system stiffness, good grinding fluid quality and wheel composition.

Fig. 3 shows the variation of specific metal removal rate with wheel speed for several materials and a range of *ECT*s at 10- and 30-minutes wheel-life. *ECT* decreases when moving to the left and down along each curve. The two curves for unhardened 1020 steel have the largest values of *SMRR*, and represent the most productive grinding conditions, while the heat resistant alloy Inconel yields the least productive grinding conditions. Each branch attains a maximum *SMRR* along the *G*-curve (compare with the same curve in the V–*ECT* graph, Fig. 2c) and a maximum speed region along the *H*-curve. When the *SMRR*-values are lower than the *H*-curve the *ECT* values for each branch decrease towards the bottom of the graph, then the speed for constant wheel-life must be reduced due to the fact that the *ECT* values are to the left of their respective *H*-curves in *V*–*ECT* graphs.

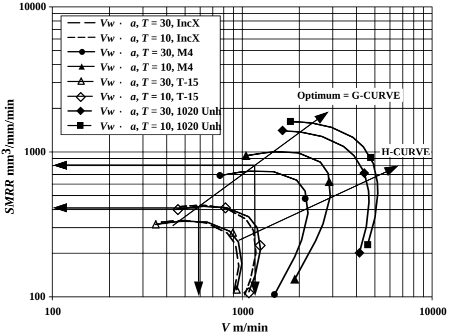


Fig. 3. Specific metal removal rate vs. cutting speed at T=10 and 30 minutes wheel life

In the figure, IncX is Inconel; M4, and T-15 are tool steels; and 1020 Unh is unhardened 1020 steel.

Surface Finish, *Ra*.—The finish is improved by decreasing the value of *ECT* as shown in Fig. 4, where *Ra* is plotted versus *ECT* at 3 different wheel lives 1,10 and 30 minutes at constant wheel speed. Because *ECT* is proportional to the depth of cut, a smaller depth of cut is favorable for reducing surface roughness when the work speed is constant.

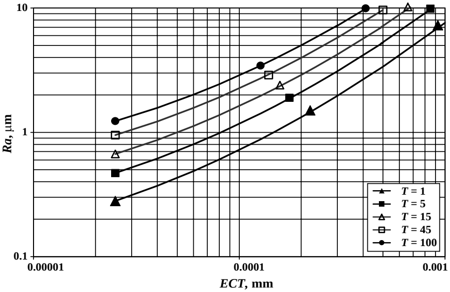


Fig. 4. Surface finish, *Ra* vs. *ECT*, wheel-life *T* plotted

In Fig. 5, *Ra* is plotted versus wheel-life at 5 different *ECT*'s. Both Figs. 4 and 5 illustrate that a shorter life improves the surface finish, which means that either an increased wheel speed (wheel-life decreases) at constant *ECT*, or a smaller *ECT* at constant speed (wheel-life increases), will result in an improved finish. For a required surface finish, *ECT* and wheel-life have to be selected appropriately in order to also achieve an optimum grinding time or cost. In cylindrical grinding a reduction of side feed f_s improves *Ra* as well.

In terms of specific metal removal rate, reducing *SMRR* will improve the surface finish R_A.

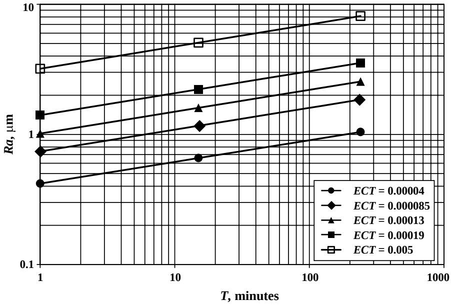


Fig. 5. Surface roughness, *Ra* vs. wheel life *T*, *ECT* plotted

Example 4, Specific Metal Removal Rates and Work Speeds in Rough and Finish Grinding: The tabulated values in the following table indicate that a decreasing *ECT* combined with a higher wheel speed for 10 minutes wheel-life will decrease the metal removal rate and thereby increasing the grinding time. This change is accompanied by a better finish in both roughing and finishing operations. Note the high work speeds when finishing.

	Tool Life $T = 10$ minutes		Roughing Depth $a_r = 0.025$ mm	Finishing Depth $a_r = 0.0025$ mm
ECT mm	Wheel speed V_{10} m/min	Removal Rate $SMRR_{10}$ mm^3/mm/ min	Work speed V_w m/min	
0.00050	1970	985	39	390
0.00033	2580	850	34	340
0.00017	2910	500	20	200

The grit size, however, is a major parameter. Fig. 6, shows that a high wheel speed, combined with a small grit size, say 320 Mesh, can achieve R_A values as small as 0.03 microns.

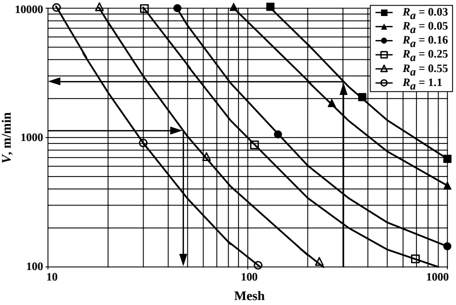


Fig. 6. Wheel speed vs. wheel mesh, *Ra* plotted

Spark-out Time.—Fig. 7 shows how spark-out time varies with system stiffness. As with surface finish, when wheel-life is short (high wear rate) the spark-out time decreases.

Equivalent Diameter (Work Conformity) Factor: The difference in curvature of the work and wheel in the contact region, determined by the equivalent diameter or work conformity formula, is an important factor for calculating spark-out time and forces, but has a

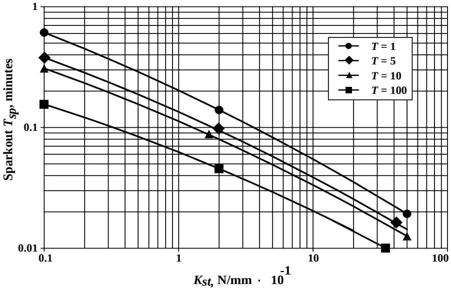


Fig. 7. Sparkout time vs. system stiffness, wheel-life *T* plotted

negligible influence on wheel-life. Therefore, an equivalent diameter, $D_e = D/(1 \pm (D/D_w))$, with the minus sign for internal grinding and the plus sign for external grinding operations, is used to consider the effect of conformity when using internal and external grinding with varying work and wheel diameters. D_e is equal to the wheel diameter in surface grinding (work flat); in internal grinding, the wheel conforms closely to the work and D_e is therefore larger than in external grinding.

Grinding Cutting Forces, Torque and Power.—Formulas to calculate the tangential cutting force, torque and required machining power are found in *Estimating Machining Power* on page 1046, but the values of K_c, specific cutting force or specific energy, are approximately 30 to 40 times higher in grinding than in turning, milling and drilling. This is primarily due to the fact that the *ECT* values in grinding are 1000 to 10000 times smaller, and also due to the negative rake angles of the grit. Average grinding rake angles are around −35 to −45 degrees. K_c for grinding unhardened steel is around 50000 to 70000 N/mm^2 and up to 150000 to 200000 N/mm^2 for hardened steels and heat resistant alloys. The grinding cutting forces are relatively small because the chip area is very small.

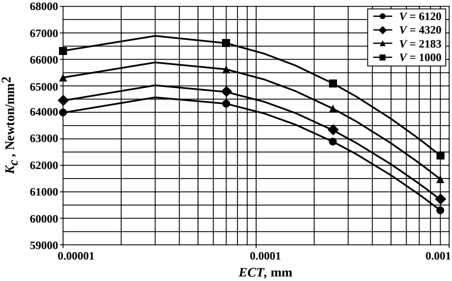


Fig. 8. Specific grinding force *Kc* vs. *ECT*; *V* plotted

As in the other metal cutting operations, the forces vary with *ECT* and to a smaller extent with the wheel speed *V*. An example is shown in Fig. 8, where K_c, specific cutting force, is plotted versus *ECT* at wheel speeds between 1000 and 6000 m/min. The material is medium unhardened carbon steel ground by an aluminum oxide wheel. The impact of wheel speed is relatively small (2 to 5% lower with increasing speed).

Example 5: Find the cutting force when ECT = 0.00017 mm, the cutting edge length (width of cut) CEL is 10 mm, and K_c=150000 N/mm^2.

The chip area is $ECT \times CEL$ = 0.0017 mm^2. For K_c=150000, the cutting force is 0.0017 × 150000 = 255 Newton.

Another difference compared to turning is the influence of the negative rake angles, illustrated by the ratio of F_H/F_C, where F_H is the normal force and F_C the tangential grinding force acting in the wheel speed direction. F_H is much larger than the grinding cutting force, generally F_H/F_C ratio is approximately 2 to 4. An example is shown in Fig. 9, where F_H/F_C, is plotted versus ECT at wheel speeds between 1000 and 6000 m/min, under the same conditions as in Fig. 8.

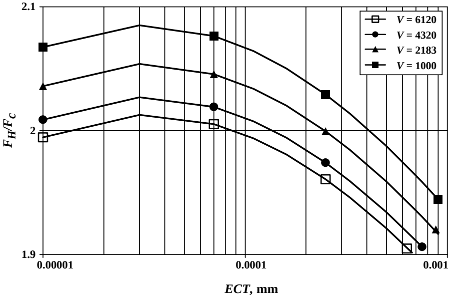


Fig. 9. F_H/F_C vs. ECT; cutting speed plotted

In both Fig. 8 and Fig. 9, it is apparent that both K_c and F_H/F_C attain maximum values for given small values of ECT, in this case approximately ECT = 0.00005 mm. This fact illustrates that forces and wheel-life are closely linked; for example, wheel speed has a maximum for constant wheel-life at approximately the same values of ECT shown in the two graphs (compare with the trends illustrated in Figs. 2a, 2b, 2c, and 3). As a matter of fact, force relationships obey the same type of relationships as those of wheel-life. Colding's force relationship uses the same 5 constants as the tool life equation, but requires values for the specific cutting force at ECT = 0.001 and an additional constant, obtained by a special data base generator. This requires more elaborate laboratory tests, or better, the design of a special test and follow-up program for parts running in the ordinary production.

Grinding Data Selection Including Wheel Life

The first estimate of machine settings is based on dividing work materials into 10 groups, based on grindability, as given in Table 1. Compositions of these work materials are found in the Handbook in the section *STANDARD STEELS* starting on page 403.

Grinding wheel nomenclature is described in *American National Standard Grinding Wheel Markings* starting on page 1141. The wheel compositions are selected according to the grade recommendations in the section *The Selection of Grinding Wheels* starting on page 1142. Grinding fluid recommendations are given in *Cutting Fluids for Machining* starting on page 1106.

Note: Maximum wheel speeds should always be checked using the safety standards in the section *Safe Operating Speeds* starting on page 1171, because the recommendations will sometimes lead to speeds above safety levels.

The material in this section is based on the use of a typical standard wheel composition such as 51-A-46-L-5-V-23, with wheel grade (wheel hardness) = L or above, and mesh (grit size) = 46 or above.

Table 1. Grindability Groups

Group	Examples
Group 1 Unhardened Steels	
Group 2 Stainless Steels	SAE 30201–30347, 51409–51501
Group 3 Cast Iron	
Group 4 Tool Steels	M1, M8, T1, H, O, L, F, 52100
Group 5 Tool Steels	M2, T2, T5, T6, D2, H41, H42, H43, M50
Group 6 Tool Steels	M3, M4, T3, D7
Group 7 Tool Steels	T15, M15
Group 8 Heat Resistant Steels	Inconel, Rene etc.
Group 9 Carbide Materials	P30 Diamond Wheel
Group 10 Ceramic Materials	

For each grindability group there is one table and 2 graphs (one with Taylor lines and the other with *SMRR* versus wheel speed *V*) that are used to get a first estimate of standardized machine settings, assuming a good system stiffness ($K_{ST} > 30$ N/μm). These data are then calibrated with the users own data in order to refine the estimate and optimize the grinding process, as discussed in *User Calibration of Recommendations*. The recommendations are valid for all grinding processes such as plunge grinding, cylindrical, and surface grinding with periphery or side of wheel, as well as for creep feed grinding.

The grinding data machinability system is based on the basic parameters equivalent chip thickness *ECT*, and wheel speed *V*, and is used to determine specific metal removal rates *SMRR* and wheel-life *T*, including the work speed V_w after the grinding depths for roughing and finishing are specified.

For each material group, the grinding data machinability system consists of *T–V* Taylor lines in log-log coordinates for 3 wheel speeds at wheel lives of 1, 10 and 100 minutes wheel-life with 4 different values of equivalent chip thickness *ECT*. The wheel speeds are designated V_1, V_{10}, and V_{100} respectively. In each table the corresponding specific metal removal rates *SMRR* are also tabulated and designated as $SMRR_1$, $SMRR_{10}$ and $SMRR_{100}$ respectively. The user can select any value of *ECT* and interpolate between the Taylor lines. These curves look the same in grinding as in the other metal cutting processes and the slope is set at n = 0.26, so each Taylor line is formulated by $V \times T^{0.26} = C$, where C is a constant tabulated at four *ECT* values, $ECT = 17, 33, 50$ and 75×10^{-5} mm, for each material group. Hence, for each value of *ECT*, $V_1 \times 1^{0.26} = V_{10} \times 10^{0.26} = V_{100} \times 100^{0.26} = C$.

Side Feed, Roughing and Finishing.—In cylindrical grinding, the side feed, $f_s = C \times$ Width, does not impact on the values in the tables, but on the feed rate F_R, where the fraction of the wheel width C is usually selected for roughing and in finishing operations, as shown in the following table.

Work Material	Roughing, C	Finishing, C
Unhardened Steel	2⁄3–3⁄4	1⁄3–3⁄8
Stainless Steel	1⁄2	1⁄4
Cast Iron	3⁄4	3⁄8
Hardened Steel	1⁄2	1⁄4

Finishing: The depth of cut in rough grinding is determined by the allowance and usually set at $a_r = 0.01$ to 0.025 mm. The depth of cut for finishing is usually set at $a_r = 0.0025$ mm and accompanied by higher wheel speeds in order to improve surface finish. However, the most important criterion for critical parts is to increase the work speed in order to avoid thermal damage and surface cracks. In cylindrical grinding, a reduction of side feed f_s

improves R_a as well. Small grit sizes are very important when very small finishes are required. See Figs. 4, 5, and 6 for reference.

Terms and Definitions

a_a = depth of cut

a_r = radial depth of cut, mm

C = fraction of grinding wheel width

CEL = cutting edge length, mm

C_U = Taylor constant

D = wheel diameter, mm

$DIST$ = grinding distance, mm

d_w = work diameter, mm

ECT = equivalent chip thickness = $f(a_r, V, V_w, f_s)$, mm

$$= 1 \div (V \div V_w \div a_r + 1 \div f_s) = \frac{V_w f_s (a_r + 1)}{V}$$

$= \text{approximately } V_w \times a_r \div V = SMRR \div V \div 1000$

$= z \times f_z \times a_r \times a_a \div CEL \div (\pi D)$ mm

F_R = feed rate, mm/min

$= f_s \times RPM_w$ for cylindrical grinding

$= f_i \times RPM_w$ for plunge (in-feed) grinding

f_i = in-feed in plunge grinding, mm/rev of work

f_s = side feed or engaged wheel width in cylindrical grinding = $C \times Width$ = a_a approximately equal to the cutting edge length CEL

Grindingratio = $MRR \div W^* = SMRR \times T \div W^* = 1000 \times ECT \times V \times T \div W^*$

MRR = metal removal rate = $SMRR \times T = 1000 \times f_s \times a_r \times V_w$ mm^3/min

$SMRR$ = specific metal removal rate obtained by dividing MRR by the engaged wheel width ($C \times Width$) = $1000 \times a_r \times V_w$ mm^3/mm width/min
Note: 100 mm^3/mm/min = 0.155 in^3/in/min, and 1 in^3/in/min = 645.16 mm^3/mm/min

T, T_U = wheel-life = $Grinding\ ratio \times W \div (1000 \times ECT \times V)$ minutes

t_c = grinding time per pass = $DIST \div F_R$ min

$= DIST \div F_R + t_{sp}$ (min) when spark-out time is included

= # Strokes $\times (DIST \div F_R + t_{sp})$ (min) when spark-out time and strokes are included

t_{sp} = spark-out time, minutes

V, V_U = wheel speed, m/min

V_w, V_{wU} = work speed = $SMRR \div 1000 \div a_r$ m/min

W^* = volume wheel wear, mm^3

$Width$ = wheel width (mm)

RPM = wheel speed = $1000 \times V \div D \div \pi$ rpm

RPM_w = work speed = $1000 \times V_w \div D_w \div \pi$ rpm

Relative Grindability.—An overview of grindability of the data base, which must be based on a constant wheel wear rate, or wheel-life, is demonstrated using 10 minutes wheel-life shown in Table 2.

Table 2. Grindability Overview

Material Group	$ECT \times 10^{-5}$	V_{10}	$SMRR_{10}$	V_w	
				Roughing Depth $a_r = 0.025$	*Finishing Depth* $a_r = 0.0025$
1 Unhardened	33	3827	1263	50	500
2 Stainless	33	1080	360	15	150
3 Cast Iron	33	4000	1320	53	530
4 Tool Steel	33	3190	1050	42	420
5 Tool Steel	33	2870	950	38	380
6 Tool Steel	33	2580	850	35	350
7Tool Steel	33	1080	360	15	150
8 Heat resistant	33	1045	345	14	140
9 Carbide with Diamond Wheel	5	$V_{600} = 1200$	$SMRR_{600} = 50$	2	20
10 Ceramics with Diamond Wheel	5	$V_{600} = 411$	$SMRR_{600} = 21$	0.84	84

Procedure to Determine Data.—The following wheel-life recommendations are designed for 4 values of *ECT* = 0.00017, 0.00033, 0.00050 and 0.00075 mm (shown as 17, 33, 50 and 75 in the tables). Lower values of *ECT* than 0.00010 mm (0.000004 in.) are not recommended as these may lie to the left of the *H*-curve.

The user selects any one of the *ECT* values, or interpolates between these, and selects the wheel speed for 10 or 100 minutes life, denoted by V_{10} and V_{100}, respectively. For other desired wheel lives the wheel speed can be calculated from the tabulated Taylor constants C and $n = 0.26$ as follows:

$(V \times T_{(desired)})^{0.26}$ = C, the value of which is tabulated for each *ECT* value. C is the value of cutting speed *V* at $T = 1$ minute, hence is the same as for the speed V_1 ($V_1 \times 1$^0.26 =C)

$$V_{10} = \mathrm{C} \div 10^{0.26} = \mathrm{C} \div 1.82$$

$$V_{100} = \mathrm{C} \div 100^{0.26} = \mathrm{C} \div 3.31.$$

Example 6: A tool steel in material group 6 with *ECT* = 0.00033, has constant C= 4690, $V_{10} = 2578$ m/min, and $V_{100} = 1417$ m/min. From this information, find the wheel speed for desired wheel-life of $T = 15$ minutes and $T = 45$ minutes

For $T = 15$ minutes we get $V_{15} = 4690 \div 15^{0.26} = 2319$ m/min (7730 fpm) and for $T = 45$ minutes $V_{45} = 4690 \div 45^{0.26} = 1743$ m/min (5810 fpm).

The Tables are arranged in 3 sections:

1. Speeds V_{10} and V_1 = Constant CST(standard) for 4 *ECT* values 0.00017, 0.00033, 0.00050 and 0.00075 mm. Values $\mathrm{C_U}$ and $\mathrm{V_{10U}}$ refer to user calibration of the standard values in each material group, explained in the following.

2. Speeds V_{100} (first row of 3), $\mathrm{V_{10}}$ and V_1 (last in row) corresponding to wheel lives 100, 10 and 1 minutes, for 4 *ECT* values 0.00017, 0.00033, 0.00050 and 0.00075 mm.

3. Specific metal removal rates $SMRR_{100}$, $SMRR_{10}$ and $SMRR_1$ corresponding to wheel lives 100, 10 and 1 minutes, for the 4 *ECT* values 0.00017, 0.00033, 0.00050, and 0.00075 mm

The 2 Graphs show: wheel life versus wheel speed in double logarithmic coordinates (Taylor lines); and, *SMRR* versus wheel speed in double logarithmic coordinates for 4 *ECT* values: 0.00017, 0.00033, 0.00050 and 0.00075 mm.

Table 1. Group 1—Unhardened Steels

Tool Life T (min)	ECT = 0.00017 mm		ECT = 0.00033 mm		ECT = 0.00050 mm		ECT = 0.00075 mm	
	Constant C = 8925		Constant C = 6965		Constant C = 5385		Constant C = 3885	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	2695	460	2105	695	1625	815	1175	880
10	**4905**	**835**	**3830**	**1265**	**2960**	**1480**	**2135**	**1600**
1	8925	1520	6965	2300	5385	2695	3885	2915

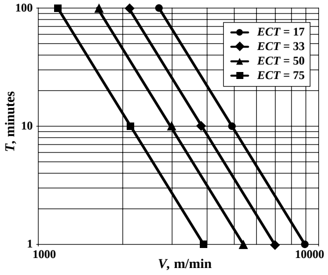


Fig. 1a. T–V

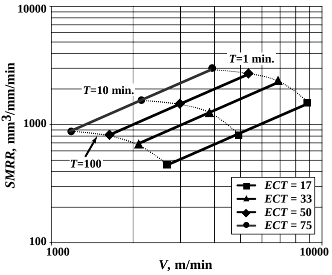


Fig. 1b. *SMRR* vs. V, T = 100, 10, 1 minutes

Table 2. Group 2—Stainless Steels SAE 30201 – 30347, SAE 51409 – 51501

Tool Life T (min)	ECT = 0.00017 mm		ECT = 0.00033 mm		ECT = 0.00050 mm		ECT = 0.00075 mm	
	Constant C = 2270		Constant C = 1970		Constant C = 1505		Constant C = 1010	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	685	115	595	195	455	225	305	230
10	**1250**	**210**	**1080**	**355**	**825**	**415**	**555**	**415**
1	2270	385	1970	650	1505	750	1010	760

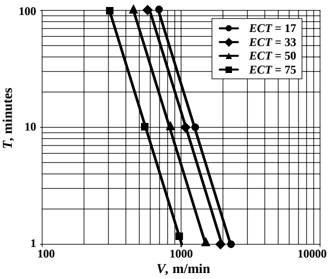


Fig. 2a. T–V

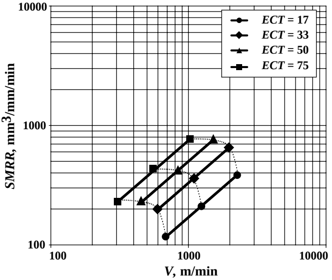


Fig. 2b. *SMRR* vs. V, T = 100, 10, 1 minutes

Table 3. Group 3—Cast Iron

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 10710		Constant C = 8360		Constant C = 6465		Constant C = 4665	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	3235	550	2525	835	1950	975	1410	1055
10	**5885**	**1000**	**4595**	**1515**	**3550**	**1775**	**2565**	**1920**
1	10710	1820	8360	2760	6465	3230	4665	3500

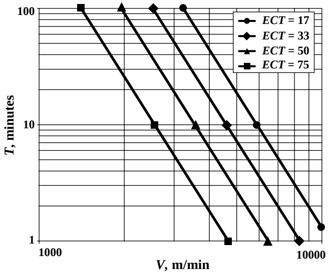


Fig. 3a. *T*–*V*

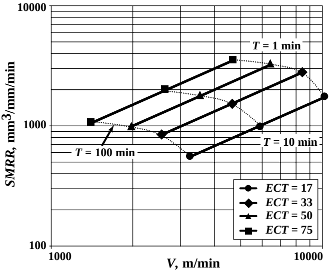


Fig. 3b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 4. Group 4—Tool Steels, M1, M8, T1, H, O, L, F, 52100

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 7440		Constant C = 5805		Constant C = 4490		Constant C = 3240	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	2245	380	1755	580	1355	680	980	735
10	**4090**	**695**	**3190**	**1055**	**2465**	**1235**	**1780**	**1335**
1	7440	1265	5805	1915	4490	2245	3240	2430

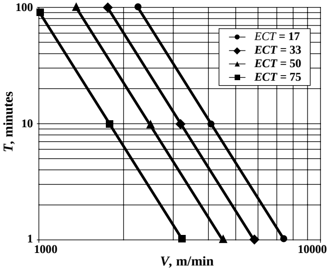


Fig. 4a. *T*–*V*

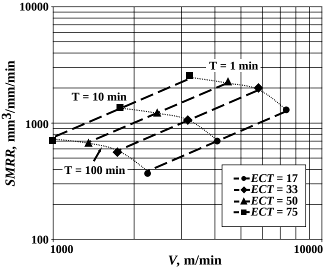


Fig. 4b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 5. Group 5—Tool Steels, M2, T2, T5, T6, D2, D5, H41, H42, H43, M50

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 6695		Constant C = 5224		Constant C = 4040		Constant C = 2915	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	2020	345	1580	520	1220	610	880	660
10	**3680**	**625**	**2870**	**945**	**2220**	**1110**	**1600**	**1200**
1	6695	1140	5225	1725	4040	2020	2915	2185

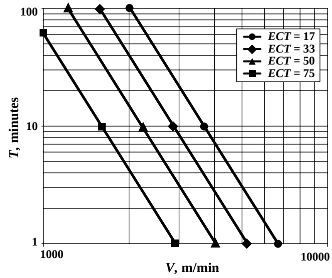


Fig. 5a. *T*–*V*

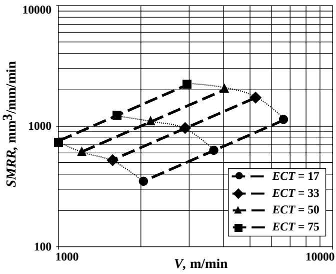


Fig. 5b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 6. Group 6—Tool Steels, M3, M4, T3, D7

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 5290		Constant C = 4690		Constant C = 3585		Constant C = 2395	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	1600	270	1415	465	1085	540	725	540
10	**2910**	**495**	**2580**	**850**	**1970**	**985**	**1315**	**985**
1	5290	900	4690	1550	3585	1795	2395	1795

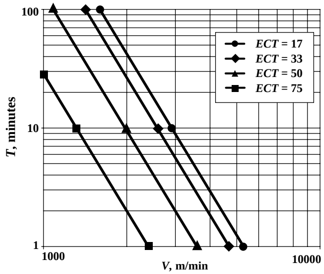


Fig. 6a. Group 6 Tool Steels *T*–*V*

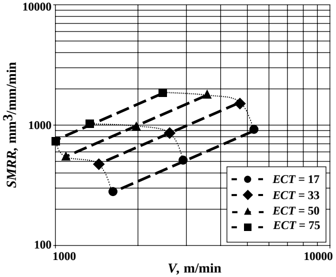


Fig. 6b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 7. Group 7—Tool Steels, T15, M15

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 2270		Constant C = 1970		Constant C = 1505		Constant C = 1010	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	685	115	595	195	455	225	305	230
10	**1250**	**210**	**1080**	**355**	**825**	**415**	**555**	**415**
1	2270	385	1970	650	1505	750	1010	760

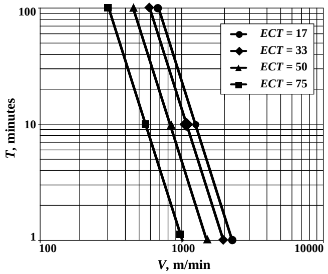


Fig. 7a. *T–V*

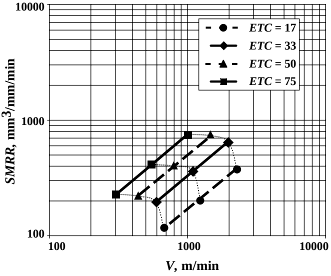


Fig. 7b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 8. Group 8—Heat Resistant Alloys, Inconel, Rene, etc.

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 2150		Constant C = 1900		Constant C = 1490		Constant C = 1035	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	650	110	575	190	450	225	315	235
10	**1185**	**200**	**1045**	**345**	**820**	**410**	**570**	**425**
1	2150	365	1900	625	1490	745	1035	780

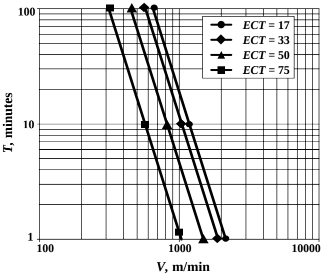


Fig. 8a. *T–V*

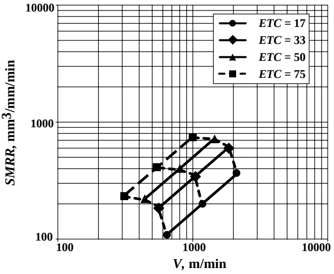


Fig. 8b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 9. Group 9—Carbide Materials, Diamond Wheel

Tool Life *T* (min)	*ECT* = 0.00002 mm		*ECT* = 0.00003 mm		*ECT* = 0.00005 mm		*ECT* = 0.00008 mm	
	Constant C = 9030		Constant C = 8030		Constant C = 5365		Constant C = 2880	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
4800	1395	30	1195	35	760	40	390	30
600	2140	45	1855	55	1200	60	625	50
10	4960	100	4415	130	2950	145	1580	125

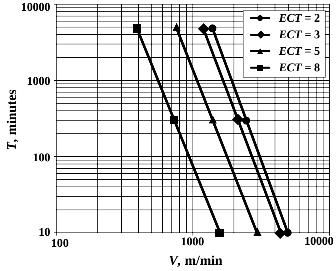


Fig. 9a. *T*–*V*

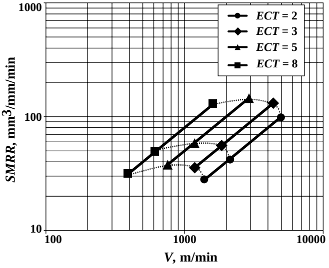


Fig. 9b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Table 10. Group 10 — Ceramic Materials Al_20_3, ZrO_2, SiC, Si_3N_4, Diamond Wheel

Tool Life *T* (min)	*ECT* = 0.00002 mm		*ECT* = 0.00003 mm		*ECT* = 0.00005 mm		*ECT* = 0.00008 mm	
	Constant C = 2460		Constant C = 2130		Constant C = 1740		Constant C = 1420	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
4800	395	8	335	10	265	13	210	17
600	595	12	510	15	410	20	330	25
10	1355	25	1170	35	955	50	780	60

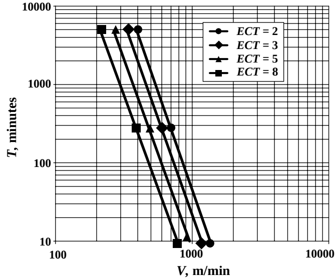


Fig. 10a. *T*–*V*

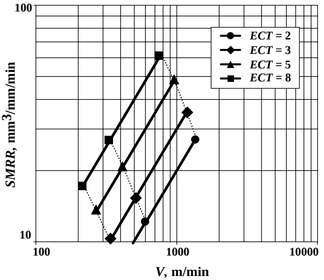


Fig. 10b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

User Calibration of Recommendations

It is recommended to copy or redraw the standard graph for any of the material groups before applying the data calibration method described below. The method is based on the user's own experience and data. The procedure is described in the following and illustrated in Table 11 and Fig. 12.

Only one shop data set is needed to adjust all four Taylor lines as shown below. The required shop data is the user's wheel-life T_U obtained at the user's wheel speed V_U, the user's work speed V_{wU}, and depth of cut a_r.

25) First the user finds out which wheel-life T_U was obtained in the shop, and the corresponding wheel speed V_U, depth of cut a_r and work speed V_{wU}.

26) Second, calculate:

a) $ECT = V_{wU} \times ar \div V_U$

b) the user Taylor constant $C_U = V_U \times T_U^{0.26}$

c) $V_{10U} = C_U \div 10^{0.26}$ d) $V_{100U} = C_U \div 100^{0.26}$

27) Thirdly, the user Taylor line is drawn in the pertinent graph. If the user wheel-life T_U is longer than that in the standard graph the speed values will be higher, or if the user wheel-life is shorter the speeds C_U, V_{10U}, V_{100U} will be lower than the standard values C, V_{10} and V_{100}.

The results are a series of lines moved to the right or to the left of the standard Taylor lines for $ECT = 17, 33, 50$ and 75×10^{-5} mm. Each standard table contains the values $C = V_1$, V_{10}, V_{100} and empty spaces for filling out the calculated user values: $C_U = V_U \times T_U^{0.26}$, $V_{10U} = C_U \div 10^{0.26}$ and $V_{100U} = C_U \div 100^{0.26}$.

Example 7: Assume the following test results on a Group 6 material: user speed is V_U = 1800 m/min, wheel-life T_U = 7 minutes, and ECT = 0.00017 mm. The Group 6 data is repeated below for convenience.

Standard Table Data, Group 6 Material

Tool Life *T* (min)	*ECT* = 0.00017 mm		*ECT* = 0.00033 mm		*ECT* = 0.00050 mm		*ECT* = 0.00075 mm	
	Constant C = 5290		Constant C = 4690		Constant C = 3585		Constant C = 2395	
	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*	V_T	*SMRR*
100	1600	270	1415	465	1085	540	725	540
10	**2910**	**495**	**2580**	**850**	**1970**	**985**	**1315**	**985**
1	5290	900	4690	1550	3585	1795	2395	1795

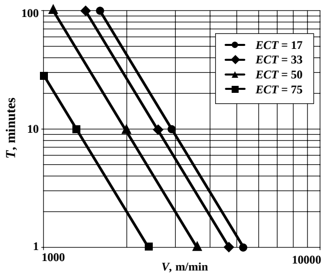


Fig. 11a. Group 6 Tool Steels, *T–V*

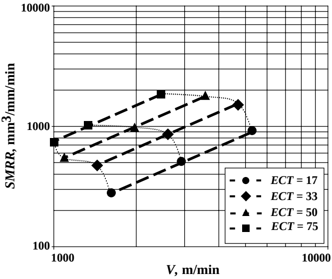


Fig. 11b. *SMRR* vs. *V*, *T* = 100, 10, 1 minutes

Calculation Procedure

1) Calculate V_{1U}, V_{10U}, V_{100U} and $SMRR_{1U}$, $SMRR_{10U}$, $SMRR_{100U}$ for $ECT = 0.00017$ mm

a) V_{1U} = the user Taylor constant $C_U = V_U \times T_U^{0.26} = 1800 \times 7^{0.26} = 2985$ m/min, and $SMRR_{1U} = 1000 \times 2985 \times 0.00017 = 507$ mm³/mm width/min

b) $V_{10U} = C_U \div 10^{0.26} = 2985 \div 10^{0.26} = 1640$ m/min, and $SMRR_{10U} = 1000 \times 1640 \times 0.00017 = 279$ mm³/mm width/min

c) $V_{100U} = C_U \div 100^{0.26} = 2985 \div 100^{0.26} = 900$ m/min, and $SMRR_{100U} = 1000 \times 900 \times 0.00017 = 153$ mm³/mm width/min

2) For $ECT = 0.00017$ mm, calculate the ratio of user Taylor constant to standard Taylor constant from the tables = $C_U \div C_{ST} = C_U \div V_1 = 2985 \div 5290 = 0.564$ (see Table 6 for the value of $C_{ST} = V_1$ at $ECT = 0.00017$ mm).

3) For ECT = 0.00033, 0.00050, and 0.00075 mm calculate the user Taylor constants from $C_U = C_{ST} \times$ (the ratio calculated in step 2) = $V_1 \times 0.564 = V_{1U}$. Then, calculate V_{10U} and V_{100U} and $SMRR_{1U}$, $SMRR_{10U}$, $SMRR_{100U}$ using the method in items 1b) and 1c) above.

a) For $ECT = 0.00033$ mm

$V_{1U} = C_U = 4690 \times 0.564 = 2645$ m/min

$V_{10U} = C_U \div 10^{0.26} = 2645 \div 10^{0.26} = 1455$ m/min

$V_{100U} = C_U \div 100^{0.26} = 2645 \div 100^{0.26} = 800$ m/min

$SMRR_{1U}$, $SMRR_{10U}$, and $SMRR_{100U}$ = 876, 480, and 264 mm³/mm width/min

b) For $ECT = 0.00050$ mm

$V_{1U} = C_U = 3590 \times 0.564 = 2025$ m/min

$V_{10U} = C_U \div 10^{0.26} = 2025 \div 10^{0.26} = 1110$ m/min

$V_{100U} = C_U \div 100^{0.26} = 2025 \div 100^{0.26} = 610$ m/min

$SMRR_{1U}$, $SMRR_{10U}$, and $SMRR_{100U}$ = 1013, 555, and 305 mm³/mm width/min

c) For $ECT = 0.00075$ mm

$V_{1U} = C_U = 2395 \times 0.564 = 1350$ m/min

$V_{10U} = C_U \div 10^{0.26} = 1350 \div 10^{0.26} = 740$ m/min

$V_{100U} = C_U \div 100^{0.26} = 1350 \div 100^{0.26} = 405$ m/min

$SMRR_{1U}$, $SMRR_{10U}$, and $SMRR_{100U}$ = 1013, 555, and 305 mm³/mm width/min

Thus, the wheel speed for any desired wheel-life at a given ECT can be calculated from $V = C_U \div T^{0.26}$. For example, at $ECT = 0.00050$ mm and desired tool-life $T = 9$, $V_9 = 2025 \div 9^{0.26} = 1144$ m/min. The corresponding specific metal removal rate is $SMRR = 1000 \times 1144 \times 0.0005 = 572$ mm³/mm width/min (0.886 in³/inch width/min).

Table 11. User Calculated Data, Group 6 Material

Tool Life T (min)	ECT = 0.00017 mm		ECT = 0.00033 mm		ECT = 0.00050 mm		ECT = 0.00075 mm	
	User Constant $C_U = 2985$		User Constant $C_U = 2645$		User Constant $C_U = 2025$		User Constant $C_U = 1350$	
	V_T	$SMRR$	V_T	$SMRR$	V_T	$SMRR$	V_T	$SMRR$
100	900	153	800	264	610	305	405	305
10	**1640**	**279**	**1455**	**480**	**1110**	**555**	**740**	**555**
1	2985	507	2645	876	2025	1013	1350	1013

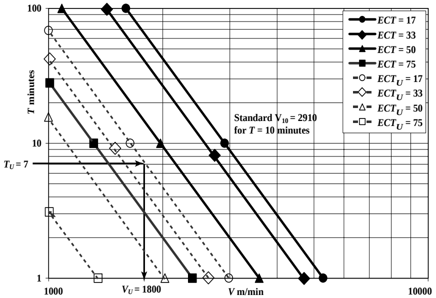


Fig. 12. Calibration of user grinding data to standard Taylor Lines
User Input: $V_U = 1800$ m/min, $T_U = 7$ minutes, $ECT = 0.00017$ mm

Optimization.— As shown, a global optimum occurs along the *G*-curve, in selected cases for values of *ECT* around 0.00075, i.e. at high metal removal rates as in other machining operations. It is recommended to use the simple formula for economic life: $T_E = 3 \times T_V$ minutes. $T_V = T_{RPL} + 60 \times C_E \div H_R$, minutes, where T_{RPL} is the time required to replace wheel, C_E = cost per wheel dressing = wheel cost + cost per dressing, and H_R is the hourly rate.

In grinding, values of T_V range between 2 and 5 minutes in conventional grinders, which means that the economic wheel lives range between 6 and 15 minutes indicating higher metal removal rates than are commonly used. When wheels are sharpened automatically after each stroke as in internal grinding, or when grits are continually replaced as in abrasive grinding (machining), T_V may be less than one minute. This translates into wheel lives around one minute in order to achieve minimum cost grinding.

Grinding Cost, Optimization and Process Planning: More accurate results are obtained when the firm collects and systemizes the information on wheel lives, wheel and work speeds, and depths of cut from production runs. A computer program can be used to plan the grinding process and apply the rules and formulas presented in this chapter. A complete grinding process planning program, such as that developed by Colding International Corporation, can be used to optimize machine settings for various feed-speed preferences corresponding wheel-life requirements, minimum cost or maximum production rate grinding, required surface finish and sparkout time; machine and fixture requirements based on the grinding forces, torque and power for sharp and worn grinding wheels; and, detailed time and cost analysis per part and per batch including wheel dressing and wheel changing schedules.

Table 12 summarizes the time and cost savings per batch as it relates to tool life. The sensitivity of how grinding parameters are selected is obvious. Minimum cost conditions yield a 51% reduction of time and 44% reduction of cost, while maximum production rate reduces total time by 65% but, at the expense of heavy wheel consumption (continuous dressing), cost by only 18%.

Table 12. Wheel Life vs. Cost

Preferences	Time per Batch, minutes	Cost per Batch, $		Reduction from Long Life,%	
		Tooling	Total Cost	Time	Cost
Long Life	2995	39	2412	—	—
Economic Life	2433	252	2211	19	8
Minimum Cost	1465	199	1344	51	44
Max Production Rate	1041	1244	1980	65	18

GRINDING AND OTHER ABRASIVE PROCESSES

Processes and equipment discussed under this heading use abrasive grains for shaping workpieces by means of machining or related methods. Abrasive grains are hard crystals either found in nature or manufactured. The most commonly used materials are aluminum oxide, silicon carbide, cubic boron nitride and diamond. Other materials such as garnet, zirconia, glass and even walnut shells are used for some applications. Abrasive products are used in three basic forms by industry:

A) *Bonded* to form a solid shaped tool such as disks (the basic shape of grinding wheels), cylinders, rings, cups, segments, or sticks to name a few.

B) *Coated* on backings made of paper or cloth, in the form of sheets, strips, or belts.

C) *Loose,* held in some liquid or solid carrier (for lapping, polishing, tumbling), or propelled by centrifugal force, air, or water pressure against the work surface (blast cleaning).

The applications for abrasive processes are multiple and varied. They include:

A) *Cleaning* of surfaces, also the coarse removal of excess material—such as rough off-hand grinding in foundries to remove gates and risers.

B) *Shaping,* such as in form grinding and tool sharpening.

C) *Sizing,* a general objective, but of primary importance in precision grinding.

D) *Surface finish improvement,* either primarily as in lapping, honing, and polishing or as a secondary objective in other types of abrasive processes.

E) *Separating,* as in cut-off or slicing operations.

The main field of application of abrasive processes is in metalworking, because of the capacity of abrasive grains to penetrate into even the hardest metals and alloys. However, the great hardness of the abrasive grains also makes the process preferred for working other hard materials, such as stones, glass, and certain types of plastics. Abrasive processes are also chosen for working relatively soft materials, such as wood, rubber, etc., for such reasons as high stock removal rates, long-lasting cutting ability, good form control, and fine finish of the worked surface.

Grinding Wheels

Abrasive Materials.—In earlier times, only natural abrasives were available. From about the beginning of this century, however, manufactured abrasives, primarily silicon carbide and aluminum oxide, have replaced the natural materials; even natural diamonds have been almost completely supplanted by synthetics. Superior and controllable properties, and dependable uniformity characterize the manufactured abrasives.

Both silicon carbide and aluminum oxide abrasives are very hard and brittle. This brittleness, called friability, is controllable for different applications. Friable abrasives break easily, thus forming sharp edges. This decreases the force needed to penetrate into the work material and the heat generated during cutting. Friable abrasives are most commonly used for precision and finish grinding. Tough abrasives resist fracture and last longer. They are used for rough grinding, snagging, and off-hand grinding.

As a general rule, although subject to variation:

1) Aluminum oxide abrasives are used for grinding plain and alloyed steel in a soft or hardened condition.

2) Silicon carbide abrasives are selected for cast iron, nonferrous metals, and nonmetallic materials.

3) Diamond is the best type of abrasive for grinding cemented carbides. It is also used for grinding glass, ceramics, and hardened tool steel.

4) Cubic Boron Nitride (CBN) is known by several trade names including Borazon (General Electric Co.), ABN (De Beers), Sho-bon (Showa-Denko), and Elbor (USSR). CBN is a synthetic superabrasive used for grinding hardened steels and wear-resistant superalloys. (See *Cubic Boron Nitride (CBN)* starting on page 982.) CBN grinding wheels have long lives and can maintain close tolerances with superior surface finishes.

Bond Properties and Grinding Wheel Grades.—The four main types of bonds used for grinding wheels are the vitrified, resinoid, rubber, and metal.

Vitrified bonds are used for more than half of all grinding wheels made, and are preferred because of their strength and other desirable qualities. Being inert, glass-like materials, vitrified bonds are not affected by water or by the chemical composition of different grinding fluids. Vitrified bonds also withstand the high temperatures generated during normal grinding operations. The structure of vitrified wheels can be controlled over a wide range of strength and porosity. Vitrified wheels, however, are more sensitive to impact than those made with organic bonds.

Resinoid bonds are selected for wheels subjected to impact, or sudden loads, or very high operating speeds. They are preferred for snagging, portable grinder uses, or roughing operations. The higher flexibility of this type of bond—essentially a filled thermosetting plastic—helps it withstand rough treatment.

Rubber bonds are even more flexible than the resinoid type, and for that reason are used for producing a high finish and for resisting sudden rises in load. Rubber bonded wheels are commonly used for wet cut-off wheels because of the nearly burr-free cuts they produce, and for centerless grinder regulating wheels to provide a stronger grip and more reliable workpiece control.

Metal bonds are used in CBN and diamond wheels. In metal bonds produced by electrodeposition, a single layer of superabrasive material (diamond or CBN) is bonded to a metal core by a matrix of metal, usually nickel. The process is so controlled that about 30–40 per cent of each abrasive particle projects above the deposited surface, giving the wheel a very aggressive and free-cutting action. With proper use, such wheels have remarkably long lives. When dulled, or worn down, the abrasive can be stripped off and the wheel renewed by a further deposit process. These wheels are also used in electrical discharge grinding and electrochemical grinding where an electrically conductive wheel is needed.

In addition to the basic properties of the various bond materials, each can also be applied in different proportions, thereby controlling the grade of the grinding wheel.

Grinding wheel grades commonly associated with hardness, express the amount of bond material in a grinding wheel, and hence the strength by which the bond retains the individual grains.

During grinding, the forces generated when cutting the work material tend to dislodge the abrasive grains. As the grains get dull and if they don't fracture to resharpen themselves, the cutting forces will eventually tear the grains from their supporting bond. For a “soft” wheel the cutting forces will dislodge the abrasive grains before they have an opportunity to fracture. When a “hard” wheel is used, the situation is reversed. Because of the extra bond in the wheel the grains are so firmly held that they never break loose and the wheel becomes glazed. During most grinding operations it is desirable to have an intermediate wheel where there is a continual slow wearing process composed of both grain fracture and dislodgement.

The grades of the grinding wheels are designated by capital letters used in alphabetical order to express increasing “hardness” from A to Z.

Grinding Wheel Structure.—The individual grains, which are encased and held together by the bond material, do not fill the entire volume of the grinding wheel; the intermediate open space is needed for several functional purposes such as heat dissipation, coolant application, and particularly, for the temporary storage of chips. It follows that the spacing of the grains must be greater for coarse grains which cut thicker chips and for large contact areas within which the chips have to be retained on the surface of the wheel before being disposed of. On the other hand, a wide spacing reduces the number of grains that contact the work surface within a given advance distance, thereby producing a coarser finish.

In general, denser structures are specified for grinding hard materials, for high-speed grinding operations, when the contact area is narrow, and for producing fine finishes and/or accurate forms. Wheels with open structure are used for tough materials, high stock removal rates, and extended contact areas, such as grinding with the face of the wheel. There are, however, several exceptions to these basic rules, an important one being the grinding of parts made by powder metallurgy, such as cemented carbides; although they represent one of the hardest industrial materials, grinding carbides requires wheels with an open structure.

Most kinds of general grinding operations, when carried out with the periphery of the wheel, call for medium spacing of the grains. The structure of the grinding wheels is expressed by numerals from 1 to 16, ranging from dense to open. Sometimes, “induced porosity” is used with open structure wheels. This term means that the grinding wheel manufacturer has placed filler material (which later burns out when the wheel is fired to vitrify the bond) in the grinding wheel mix. These fillers create large “pores” between grain clusters without changing the total volume of the “pores” in the grinding wheel. Thus, an A46-H12V wheel and an A46H12VP wheel will contain the same amounts of bond, abrasive, and air space. In the former, a large number of relatively small pores will be distributed throughout the wheel. The latter will have a smaller number of larger pores.

American National Standard Grinding Wheel Markings.—ANSI Standard B74.13-1990“ Markings for Identifying Grinding Wheels and Other Bonded Abrasives,” applies to grinding wheels and other bonded abrasives, segments, bricks, sticks, hones, rubs, and other shapes that are for removing material, or producing a desired surface or dimension. It does not apply to specialities such as sharpening stones and provides only a standard system of markings. Wheels having the same standard markings but made by different wheel manufacturers may not—and probably will not—produce exactly the same grinding action. This desirable result cannot be obtained because of the impossibility of closely correlating any measurable physical properties of bonded abrasive products in terms of their grinding action.

Symbols for designating diamond and cubic boron wheel compositions are given on page 1166.

Sequence of Markings.—The accompanying illustration taken from ANSI B74.13-1990 shows the makeup of a typical wheel or bonded abrasive marking.

Prefix	1 Abrasive Type	2 Grain Size	3 Grade	4 Structure	5 Bond Type	6 Manufacturer's Record
51 —	A —	36 —	L —	5 —	V —	23

The meaning of each letter and number in this or other markings is indicated by the following complete list.

1) *Abrasive Letters:* The letter (A) is used for aluminum oxide, (C) for silicon carbide, and (Z) for aluminum zirconium. The manufacturer may designate some particular type in any one of these broad classes, by using his own symbol as a prefix (example, 51).

2) *Grain Size:* The grain sizes commonly used and varying from coarse to very fine are indicated by the following numbers: 8, 10, 12, 14, 16, 20, 24, 30, 36, 46, 54, 60,70, 80, 90, 100, 120, 150, 180, and 220. The following additional sizes are used occasionally: 240, 280, 320, 400, 500, and 600. The wheel manufacturer may add to the regular grain number an additional symbol to indicate a special grain combination.

3) *Grade:* Grades are indicated by letters of the alphabet from A to Z in all bonds or processes. Wheel grades from A to Z range from soft to hard.

4) *Structure:* The use of a structure symbol is optional. The structure is indicated by Nos. 1 to 16 (or higher, if necessary) with progressively higher numbers indicating a progressively wider grain spacing (more open structure).

5) *Bond or Process:* Bonds are indicated by the following letters: V, vitrified; S, silicate; E, shellac or elastic; R, rubber; RF, rubber reinforced; B, resinoid (synthetic resins); BF, resinoid reinforced; O, oxychloride.

6) *Manufacturer's Record:* The sixth position may be used for manufacturer's private factory records; this is optional.

American National Standard Shapes and Sizes of Grinding Wheels.—The ANSI Standard B74.2-1982 which includes shapes and sizes of grinding wheels, gives a wide variety of grinding wheel shape and size combinations. These are suitable for the majority of applications. Although grinding wheels can be manufactured to shapes and dimensions different from those listed, it is advisable, for reasons of cost and inventory control, to avoid using special shapes and sizes, unless technically warranted.

Standard shapes and size ranges as given in this Standard together with typical applications are shown in Table for inch dimensions and in Table for metric dimensions.

The operating surface of the grinding wheel is often referred to as the wheel face. In the majority of cases it is the periphery of the grinding wheel which, when not specified otherwise, has a straight profile. However, other face shapes can also be supplied by the grinding wheel manufacturers, and also reproduced during usage by appropriate truing. ANSI B74.2-1982 standard offers 13 different shapes for grinding wheel faces, which are shown in Table 2.

The Selection of Grinding Wheels.—In selecting a grinding wheel, the determining factors are the composition of the work material, the type of grinding machine, the size range of the wheels used, and the expected grinding results, in this approximate order.

The Norton Company has developed, as the result of extensive test series, a method of grinding wheel recommendation that is more flexible and also better adapted to taking into consideration pertinent factors of the job, than are listings based solely on workpiece categories. This approach is the basis for Tables 3 through 6, inclusive. Tool steels and constructional steels are considered in the detailed recommendations presented in these tables.

Table 3 assigns most of the standardized tool steels to five different grindability groups. The AISI-SAE tool steel designations are used.

After having defined the grindability group of the tool steel to be ground, the operation to be carried out is found in the first column of Table . The second column in this table distinguishes between different grinding wheel size ranges, because wheel size is a factor in determining the contact area between wheel and work, thus affecting the apparent hardness of the grinding wheel. Distinction is also made between wet and dry grinding.

Finally, the last two columns define the essential characteristics of the recommended types of grinding wheels under the headings of first and second choice, respectively. Where letters are used *preceding* A, the standard designation for aluminum oxide, they indicate a degree of friability different from the regular, thus: SF = semi friable (Norton equivalent 16A) and F = friable (Norton equivalent 33A and 38A). The suffix P, where applied, expresses a degree of porosity that is more open than the regular.

Table 1a. Standard Shapes and Inch Size Ranges of Grinding Wheels

ANSI B74.2-1982

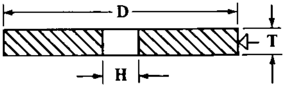
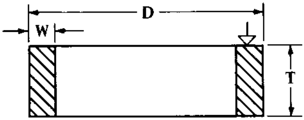
	Size Ranges of Principal Dimensions, Inches		
Applications	*D* = Dia.	*T* = Thick.	*H* = Hole
	**Type 1. Straight Wheel** For peripheral grinding.		
CUTTING OFF (Organic bonds only)	1 to 48	1⁄64 to ⅜	1⁄16 to 6
CYLINDRICAL GRINDING Between centers	12 to 48	½ to 6	5 to 20
CYLINDRICAL GRINDING Centerless grinding wheels	14 to 30	1 to 20	5 or 12
CYLINDRICAL GRINDING Centerless regulating wheels	8 to 14	1 to 12	3 to 6
INTERNAL GRINDING	¼ to 4	¼ to 2	3⁄32 to ⅞
OFFHAND GRINDING Grinding on the periphery			
General purpose	6 to 36	½ to 4	½ to 3
For wet tool grinding only	30 or 36	3 or 4	20
SAW GUMMING (F-type face)	6 to 12	¼ to 1½	½ to 1¼
SNAGGING Floor stand machines	12 to 24	1 to 3	1¼ to 2½
SNAGGING Floor stand machines (Organic bond, wheel speed over 6500 sfpm)	20 to 36	2 to 4	6 or 12
SNAGGING Mechanical grinders (Organic bond, wheel speed up to 16,500 sfpm)	24	2 to 3	12
SNAGGING Portable machines	3 to 8	¼ to 1	⅜ to ⅝
SNAGGING Portable machines (Reinforced organic bond, 17,000 sfpm)	6 or 8	¾ or 1	1
SNAGGING Swing frame machines	12 to 24	2 to 3	3½ to 12
SURFACE GRINDING Horizontal spindle machines	6 to 24	½ to 6	1¼ to 12
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc.	6 to 10	¼ to ½	⅝ to 5
	**Type 2. Cylindrical Wheel** Side grinding wheel — mounted on the diameter; may also be mounted in a chuck or on a plate.		
			W = Wall
SURFACE GRINDING Vertical spindle machines	8 to 20	4 or 5	1 to 4

Table 1a. *(Continued)* **Standard Shapes and Inch Size Ranges of Grinding Wheels** *ANSI B74.2-1982*

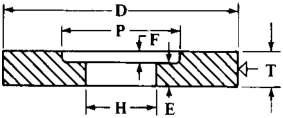
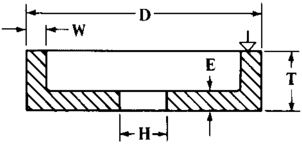
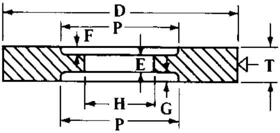
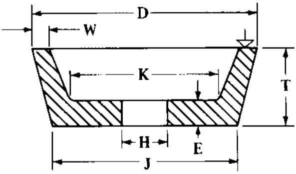
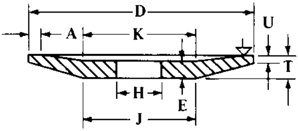
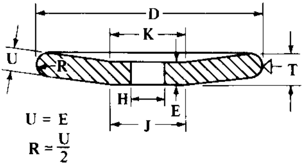
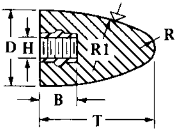
Applications	Size Ranges of Principal Dimensions, Inches		
	D = Dia.	*T* = Thick.	*H* = Hole
	**Type 5. Wheel,** recessed one side For peripheral grinding. Allows wider faced wheels than the available mounting thickness, also grinding clearance for the nut and flange.		
CYLINDRICAL GRINDING Between centers	12 to 36	1½ to 4	5 or 12
CYLINDRICAL GRINDING Centerless regulating wheel	8 to 14	3 to 6	3 or 5
INTERNAL GRINDING	⅜ to 4	⅜ to 2	⅛ to ⅞
SURFACE GRINDING Horizontal spindle machines	7 to 24	¾ to 6	1¼ to 12
	**Type 6. Straight-Cup Wheel** Side grinding wheel, in whose dimensioning the wall thickness (*W*) takes precedence over the diameter of the recess. Hole is ⅝-11UNC-2B threaded for the snagging wheels and ½ or 1¼″ for the tool grinding wheels.		
			W = Wall
SNAGGING Portable machines, organic bond only.	4 to 6	2	¾ to 1½
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc.	2 to 6	1 ¼ to 2	5⁄16 or ⅜
	**Type 7. Wheel,** recessed two sides Peripheral grinding. Recesses allow grinding clearance for both flanges and also narrower mounting thickness than overall thickness.		
CYLINDRICAL GRINDING Between centers	12 to 36	1½ to 4	5 or 12
CYLINDRICAL GRINDING Centerless regulating wheel	8 to 14	4 to 20	3 to 6
SURFACE GRINDING Horizontal spindle machines	12 to 24	2 to 6	5 to 12

Table 1a. *(Continued)* **Standard Shapes and Inch Size Ranges of Grinding Wheels** *ANSI B74.2-1982*

<table>
<tr><th rowspan="2">Applications</th><th colspan="3">Size Ranges of Principal Dimensions, Inches</th></tr>
<tr><th><i>D</i> = Dia.</th><th><i>T</i> = Thick.</th><th><i>H</i> = Hole</th></tr>
<tr><td>
</td><td colspan="3">Type 11. Flaring-Cup Wheel Side grinding wheel with wall tapered outward from the back; wall generally thicker in the back.</td></tr>
<tr><td>SNAGGING
Portable machines, organic bonds only, threaded hole</td><td>4 to 6</td><td>2</td><td>⅝-11 UNC-2B</td></tr>
<tr><td>TOOL GRINDING
Broaches, cutters, mills, reamers, taps, etc.</td><td>2 to 5</td><td>1 ¼ to 2</td><td>½ to 1 ¼</td></tr>
<tr><td>
</td><td colspan="3">Type 12. Dish Wheel Grinding on the side or on the U-face of the wheel, the U-face being always present in this type.</td></tr>
<tr><td>TOOL GRINDING
Broaches, cutters, mills, reamers, taps, etc.</td><td>3 to 8</td><td>½ or ¾</td><td>½ to 1 ¼</td></tr>
<tr><td>
</td><td colspan="3">Type 13. Saucer Wheel Peripheral grinding wheel, resembling the shape of a saucer, with cross section equal throughout.</td></tr>
<tr><td>SAW GUMMING
Saw tooth shaping and sharpening</td><td>8 to 12</td><td>½ to 1 ¾
<i>U</i> & <i>E</i>
¼ to 1½</td><td>¾ to 1 ¼</td></tr>
<tr><td>

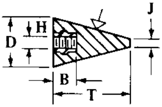

</td><td colspan="3">Type 16. Cone, Curved Side
Type 17. Cone, Straight Side, Square Tip
Type 17R. Cone, Straight Side, Round Tip (Tip Radius <i>R</i> = <i>J</i>/2)</td></tr>
<tr><td>SNAGGING
Portable machine, threaded holes</td><td>1¼ to 3</td><td>2 to 3½</td><td>⅜-24UNF-2B
to
⅝-11UNC-2B</td></tr>
</table>

Table 1a. *(Continued)* Standard Shapes and Inch Size Ranges of Grinding Wheels *ANSI B74.2-1982*

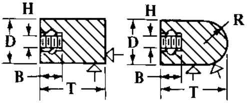
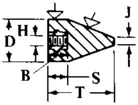
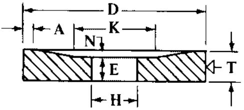
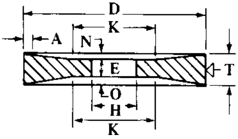
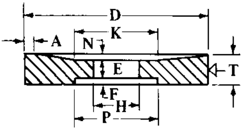
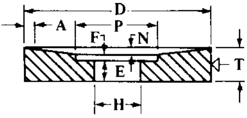
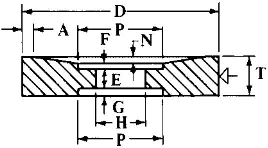
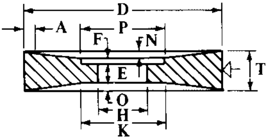
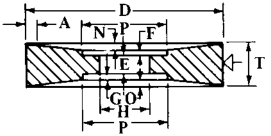
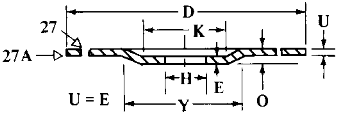
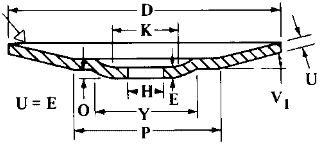
Applications	Size Ranges of Principal Dimensions, Inches		
	D = Dia.	*T* = Thick.	*H* = Hole
	**Type 18. Plug, Square End** **Type 18R. Plug, Round End** *R* = *D*/2		
	**Type 19. Plugs, Conical End, Square Tip** **Type 19R. Plugs, Conical End, Round Tip** (Tip Radius *R* = *J*/2)		
SNAGGING Portable machine, threaded holes	1¼ to 3	2 to 3½	⅜-24UNF-2B to ⅝-11UNC-2B
	**Type 20. Wheel, Relieved One Side** Peripheral grinding wheel, one side flat, the other side relieved to a flat.		
CYLINDRICAL GRINDING Between centers	12 to 36	¾ to 4	5 to 20
	**Type 21. Wheel, Relieved Two Sides** Both sides relieved to a flat.		
	**Type 22. Wheel, Relieved One Side, Recessed Other Side** One side relieved to a flat.		
	**Type 23. Wheel, Relieved and Recessed Same Side** The other side is straight.		
CYLINDRICAL GRINDING Between centers, with wheel periphery	20 to 36	2 to 4	12 or 20

Table 1a. *(Continued)* **Standard Shapes and Inch Size Ranges of Grinding Wheels**
ANSI B74.2-1982

Applications	Size Ranges of Principal Dimensions, Inches		
	D = Dia.	*T* = Thick.	*H* = Hole
	**Type 24. Wheel, Relieved and Recessed One Side, Recessed Other Side** One side recessed, the other side is relieved to a recess.		
	**Type 25. Wheel, Relieved and Recessed One Side, Relieved Other Side** One side relieved to a flat, the other side relieved to a recess.		
	**Type 26. Wheel, Relieved and Recessed Both Sides**		
CYLINDRICAL GRINDING Between centers, with the periphery of the wheel	20 to 36	2 to 4	12 or 20
	**TYPES 27 & 27A. Wheel, Depressed Center** 27. *Portable Grinding:* Grinding normally done by contact with work at approx. a 15° angle with face of the wheel. 27A. *Cutting-off:* Using the periphery as grinding face.		
CUTTING OFF Reinforced organic bonds only	16 to 30	$U = E = \frac{5}{32}$ to $\frac{1}{4}$	1 or 1 $\frac{1}{2}$
SNAGGING Portable machine	3 to 9	*U* = Uniform thick. $\frac{1}{8}$ to $\frac{3}{8}$	$\frac{3}{8}$ or $\frac{7}{8}$
	**Type 28. Wheel, Depressed Center (Saucer Shaped Grinding Face)** Grinding at approx. 15° angle with wheel face.		
SNAGGING Portable machine	7 or 9	*U* = Uniform thickness $\frac{1}{4}$	$\frac{7}{8}$

Throughout table large open-head arrows indicate grinding surfaces.

Table 1b. Standard Shapes and Metric Size Ranges of Grinding Wheels

ANSI B74.2-1982

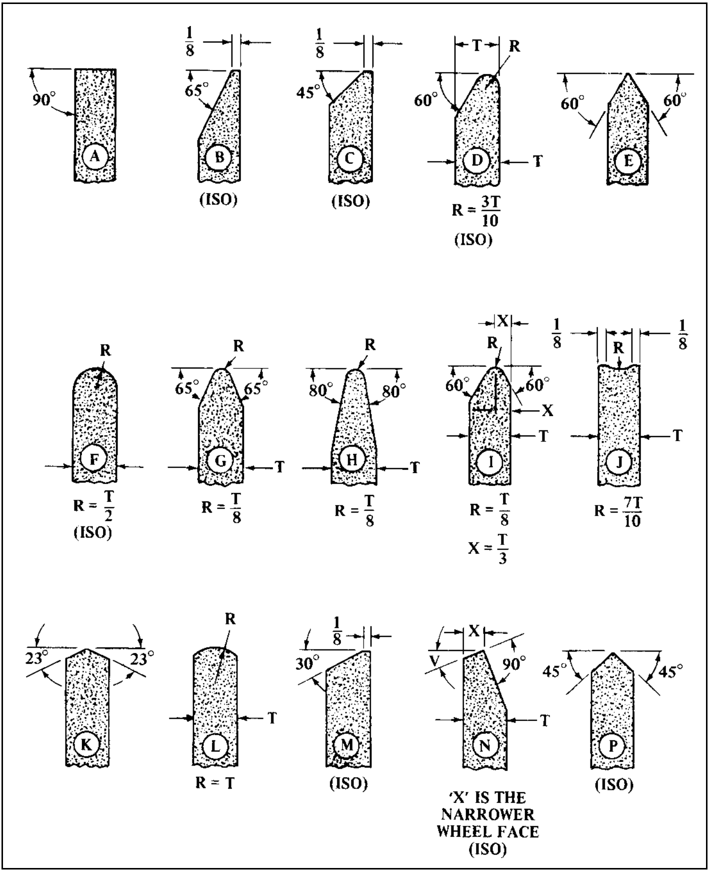
Applications	Size Ranges of Principal Dimensions, Millimeters		
	D = Diam.	*T* = Thick.	*H* = Hole
Type 1. Straight Wheel[a]			
CUTTING OFF (nonreinforced and reinforced organic bonds only)	150 to 1250	0.8 to 10	16 to 152.4
CYLINDRICAL GRINDING Between centers	300 to 1250	20 to 160	127 to 508
CYLINDRICAL GRINDING Centerless grinding wheels	350 to 750	25 to 500	127 or 304.8
CYLINDRICAL GRINDING Centerless regulating wheels	200 to 350	25 to 315	76.2 to 152.4
INTERNAL GRINDING	6 to 100	6 to 50	2.5 to 25
OFFHAND GRINDING Grinding on the periphery			
General purpose	150 to 900	13 to 100	20 to 76.2
For wet tool grinding only	750 or 900	80 or 100	508
SAW GUMMING (F-type face)	150 to 300	6 to 40	32
SNAGGING Floor stand machines	300 to 600	25 to 80	32 to 76.2
SNAGGING Floor stand machines(organic bond, wheel speed over 33 meters per second)	500 to 900	50 to 100	152.4 or 304.8
SNAGGING Mechanical grinders (organic bond, wheel speed up to 84 meters per second)	600	50 to 80	304.8
SNAGGING Portable machines	80 to 200	6 to 25	10 to 16
SNAGGING Swing frame machines (organic bond)	300 to 600	50 to 80	88.9 to 304.8
SURFACE GRINDING Horizontal spindle machines	150 to 600	13 to 160	32 to 304.8
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc.	150 to 250	6 to 20	32 to 127
Type 2. Cylindrical Wheel[a]			
			W = Wall
SURFACE GRINDING Vertical spindle machines	200 to 500	100 or 125	25 to 100

Table 1b. *(Continued)* Standard Shapes and Metric Size Ranges of Grinding Wheels *ANSI B74.2-1982*

Applications	Size Ranges of Principal Dimensions, Millimeters		
	D = Diam.	*T* = Thick.	*H* = Hole
Type 5. Wheel, recessed one side[a]			
CYLINDRICAL GRINDING Between centers	300 to 900	40 to 100	127 or 304.8
CYLINDRICAL GRINDING Centerless regulating wheels	200 to 350	80 to 160	76.2 or 127
INTERNAL GRINDING	10 to 100	10 to 50	3.18 to 25
Type 6. Straight-Cup Wheel[a]			
			W = Wall
SNAGGING Portable machines, organic bond only (hole is 5⁄8-11 UNC-2B)	100 to 150	50	20 to 40
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc. (Hole is 13 to 32 mm)	50 to 150	32 to 50	8 or 10
Type 7. Wheel, recessed two sides[a]			
CYLINDRICAL GRINDING Between centers	300 to 900	40 to 100	127 or 304.8
CYLINDRICAL GRINDING Centerless regulating wheels	200 to 350	100 to 500	76.2 to 152.4
Type 11. Flaring-Cup Wheel[a]			
SNAGGING Portable machines, organic bonds only, threaded hole	100 to 150	50	5⁄8-11 UNC-2B
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc.	50 to 125	32 to 50	13 to 32
Type 12. Dish Wheel[a]			
TOOL GRINDING Broaches, cutters, mills, reamers, taps, etc.	80 to 200	13 or 20	13 to 32
Type 27 and 27A. Wheel, depressed center[a]			
CUTTING OFF Reinforced organic bonds only	400 to 750	*U* = *E* = 6	25.4 or 38.1
SNAGGING Portable machines	80 to 230	*U* = *E* = 3.2 to 10	9.53 or 22.23

[a] See Table 1a for diagrams and descriptions of each wheel type.
All dimensions in millimeters.

Table 2. Standard Shapes of Grinding Wheel Faces *ANSI B74.2-1982*



Recommendations, similar in principle, yet somewhat less discriminating have been developed by the Norton Company for *constructional steels.* These materials can be ground either in their original state (soft) or in their after-hardened state (directly or following carburization). Constructional steels must be distinguished from structural steels which are used primarily by the building industry in mill shapes, without or with a minimum of machining.

Constructional steels are either plain carbon or alloy type steels assigned in the AISI-SAE specifications to different groups, according to the predominant types of alloying elements. In the following recommendations no distinction is made because of different compositions since that factor generally, has a minor effect on grinding wheel choice in constructional steels. However, separate recommendations are made for soft (Table 5) and hardened (Table 6) constructional steels. For the relatively rare instance where the use of a single type of wheel for both soft and hardened steel materials is considered more important than the selection of the best suited types for each condition of the work materials, Table 5 lists "All Around" wheels in its last column.

For applications where cool cutting properties of the wheel are particularly important, Table 6 lists, as a second alternative, porous-type wheels. The sequence of choices as presented in these tables does not necessarily represent a second, or third best; it can also apply to conditions where the first choice did not provide optimum results and by varying slightly the composition of the grinding wheel, as indicated in the subsequent choices, the performance experience of the first choice might be improved.

Table 3. Classification of Tool Steels by their Relative Grindability

Relative Grindability Group	AISI-SAE Designation of Tool Steels
GROUP 1—Any area of work surface	W1, W2, W5
	S1, S2, S4, S5, S6, S7
High grindability tool and die steels	O1, O2, O6, O7
(Grindability index greater than 12)	H10, H11, H12, H13, H14
	L2, L6
GROUP 2—Small area of work surface	H19, H20, H21, H22, H23, H24, H26
(*as found in tools*)	P6, P20, P21
	T1, T7, T8
Medium grindability tool and die steels	M1, M2, M8, M10, M33, M50
(Grindability index 3 to 12)	D1, D2, D3, D4, D5, D6
	A2, A4, A6, A8, A9, A10
GROUP 3—Small area of work surface	T4, T5, T6, T8
(*as found in tools*)	M3, M6, M7, M34, M36, M41, M42, M46, M48, M52, M62
Low grindability tool and die steels	D2, D5
(Grindability index between 1.0 and 3)	A11
GROUP 4—Large area of work surface	
(as found in dies)	All steels found in Groups 2 and 3
Medium and low grindability tool and die steels	
(Grindability index between 1.0 and 12)	
GROUP 5—Any area of work surface	D3, D4, D7
	M4
Very low grindability tool and die steels	A7
(Grindability index less than 1.0)	T15

Table 4. Grinding Wheel Recommendations for Hardened Tool Steels According to their Grindability

Operation	Wheel or Rim Diameter, Inches	First-Choice Specifications	Second-Choice Specifications
Group 1 Steels			
Surfacing			
Surfacing wheels	14 and smaller	Wet FA46-I8V	SFA46-G12VP
	14 and smaller	Dry FA46-H8V	FA46-F12VP
	Over 14	Wet FA36-I8V	SFA36-I8V
Segments or Cylinders	1½ rim or less	Wet FA30-H8V	FA30-F12VP
Cups	¾ rim or less	Wet FA36-H8V	FA46-F12VP
	(for rims wider than 1½ inches, go one grade softer in available specifications)		
Cutter sharpening			
Straight wheel	…	Wet FA46-K8V	FA60-K8V
	…	Dry FA46-J8V	FA46-H12VP
Dish shape	…	Dry FA60-J8V	FA60-H12VP
Cup shape	…	Dry FA46-L8V	FA60-H12VP
	…	Wet SFA46-L5V	SFA60-L5V
Form tool grinding	8 and smaller	Wet FA60-L8V to FA100-M7V	
	8 and smaller	Dry FA60-K8V to FA100-L8V	
	10 and larger	Wet FA60-L8V to FA80-M6V	
Cylindrical	14 and smaller	Wet SFA60-L5V	…
	16 and larger	Wet SFA60-M5V	…
Centerless	…	Wet SFA60-M5V	…
Internal			
Production grinding	Under ½	Wet SPA80-N6V	SFA80-N7V
	½ to 1	Wet SFA60-M5V	SFA60-M6V
	Over 1 to 3	Wet SFA54-L5V	SFA54-L6V
	Over 3	Wet SFA46-L5V	SFA46-K5V
Tool room grinding	Under ½	Dry FA80-L6V	SFA80-L7V
	½ to 1	Dry FA70-K7V	SFA70-K7V
	Over 1 to 3	Dry FA60-J8V	FA60-H12VP
	Over 3	Dry FA46-J8V	FA54-H12VP
Group 2 Steels			
Surfacing			
Straight wheels	14 and smaller	Wet FA46-I8V	FA46-G12VP
	14 and smaller	Dry FA46-H8V	FA46-F12VP
	Over 14	Wet FA46-H8V	SFA46-I8V
Segments or Cylinders	1½ rim or less	Wet FA30-G8V	FA36-E12VP
Cups	¾ rim or less	Wet FA36-H8V	FA46-F12VP
	(for rims wider than 1½ inches, go one grade softer in available specifications)		

Table 4. *(Continued)* **Grinding Wheel Recommendations for Hardened Tool Steels According to their Grindability**

Operation	Wheel or Rim Diameter, Inches	First-Choice Specifications	Second-Choice Specifications
Cutter sharpening			
Straight wheel	…	Wet FA46-L5V	FA60-K8V
	…	Dry FA46-J8V	FA60-H12VP
Dish shape	…	Dry FA60-J5V	FA60-G12VP
Cup shape	…	Dry FA46-K5V	FA60-G12VP
	…	Wet FA46-L5V	FA60-J8V
Form tool grinding	8 and smaller	Wet FA60-K8V to FA120-L8V	
	8 and smaller	Dry FA80-K8V to FA150-K8V	
	10 and larger	Wet FA60-K8V to FA120-L8V	
Cylindrical	14 and less	Wet FA60-L5V	SFA60-L5V
	16 and larger	Wet FA60-K5V	SFA60-K5V
Centerless	…	Wet FA60-M5V	SFA60-M5V
Internal			
Production grinding	Under ½	Wet FA80-L6V	SFA80-L6V
	½ to 1	Wet FA70-K5V	SFA70-K5V
	Over 1 to 3	Wet FA60-J8V	SFA60-J7V
	Over 3	Wet FA54-J8V	SFA54-J8V
Tool room grinding	Under ½	Dry FA80-I8V	SFA80-K7V
	½ to 1	Dry FA70-J8V	SFA70-J7V
	Over 1 to 3	Dry FA60-I8V	FA60-G12VP
	Over 3	Dry FA54-I8V	FA54-G12VP
Group 3 Steels			
Surfacing			
Straight wheels	14 and smaller	Wet FA60-I8V	FA60-G12VP
	14 and smaller	Dry FA60-H8V	FA60-F12VP
	Over 14	Wet FA60-H8V	SFA60-I8V
Segments or Cylinders	1½ rim or less	Wet FA46-G8V	FA46-E12VP
Cups	¾ rim or less	Wet FA46-G8V	FA46-E12VP
	(for rims wider than 1½ inches, go one grade softer in available specifications)		
Cutter grinding			
Straight wheel	…	Wet FA46-J8V	FA60-J8V
	…	Dry FA46-I8V	FA46-G12VP
Dish shape	…	Dry FA60-H8V	FA60-F12VP
Cup shape	…	Dry FA46-I8V	FA60-F12VP
	…	Wet FA46-J8V	FA60-J8V
Form tool grinding	8 and smaller	Wet FA80-K8V to FA150-L9V	
	8 and smaller	Dry FA100-J8V to FA150-K8V	
	10 and larger	Wet FA80-J8V to FA150-J8V	

Table 4. *(Continued)* **Grinding Wheel Recommendations for Hardened Tool Steels According to their Grindability**

Operation	Wheel or Rim Diameter, Inches	First-Choice Specifications	Second-Choice Specifications
Cylindrical	14 and less	Wet FA80-L5V	SFA80-L6V
	16 and larger	Wet FA60-L6V	SFA60-K5V
Centerless	…	Wet FA60-L5V	SFA60-L5V
Internal			
Production grinding	Under ½	Wet FA90-L6V	SFA90-L6V
	½ to 1	Wet FA80-L6V	SFA80-L6V
	Over 1 to 3	Wet FA70-K5V	SFA70-K5V
	Over 3	Wet FA60-J5V	SFA60-J5V
Tool room grinding	Under ½	Dry FA90-K8V	SFA90-K7V
	½ to 1	Dry FA80-J8V	SFA80-J7V
	Over 1 to 3	Dry FA70-I8V	SFA70-G12VP
	Over 3	Dry FA60-I8V	SFA60-G12VP
Group 4 Steels			
Surfacing			
Straight wheels	14 and smaller	Wet FA60-I8V	C60-JV
	14 and smaller	Wet FA60-H8V	C60-IV
	Over 14	Wet FA46-H8V	C60-HV
Segments	1 ½ rim or less	Wet FA46-G8V	C46-HV
Cylinders	1 ½ rim or less	Wet FA46-G8V	C60-HV
Cups	¾ rim or less	Wet FA46-G6V	C60-IV
	(for rims wider than 1 ½ inches, go one grade softer in available specifications)		
Form tool grinding	8 and smaller	Wet FA60-J8V to FA150-K8V	
	8 and smaller	Dry FA80-I8V to FA180-J8V	
	10 and larger	Wet FA60-J8V to FA150-K8V	
Cylindrical	14 and less	Wet FA80-K8V	C60-KV
	16 and larger	Wet FA60-J8V	C60-KV
Internal			
Production grinding	Under ½	Wet FA90-L8V	C90-LV
	½ to 1	Wet FA80-K5V	C80-KV
	Over 1 to 3	Wet FA70-J8V	C70-JV
	Over 3	Wet FA60-I8V	C60-IV
Tool room grinding	Under ½	Dry FA90-K8V	C90-KV
	½ to 1	Dry FA80-J8V	C80-JV
	Over 1 to 3	Dry FA70-I8V	C70-IV
	Over 3	Dry FA60-H8V	C60-HV

Table 4. *(Continued)* **Grinding Wheel Recommendations for Hardened Tool Steels According to their Grindability**

Operation	Wheel or Rim Diameter, Inches	First-Choice Specifications	Second-Choice Specifications	Third-Choice Specifications
Group 5 Steels				
Surfacing				
Straight wheels	14 and smaller	Wet SFA60-H8V	FA60-E12VP	C60-IV
	14 and smaller	Dry SFA80-H8V	FA80-E12VP	C80-HV
	Over 14	Wet SFA60-H8V	FA60-E12VP	C60-HV
Segments or Cylinders	1 ½ rim or less	Wet SFA46-G8V	FA46-E12VP	C46-GV
Cups	¾ rim or less	Wet SFA60-G8V	FA60-E12VP	C60-GV
	(for rims wider than 1 ½ inches, go one grade softer in available specifications)			
Cutter grinding				
Straight wheels	…	Wet SFA60-I8V	SFA60-G12VP	…
	…	Dry SFA60-H8V	SFA80-F12VP	…
Dish shape	…	Dry SFA80-H8V	SFA80-F12VP	…
Cup shape	…	Dry SFA60-I8V	SFA60-G12VP	…
	…	Wet SFA60-J8V	SFA60-H12VP	…
Form tool grinding	8 and smaller	Wet FA80-J8V to FA180-J9V		…
	8 and smaller	Dry FA100-I8V to FA220-J9V		…
	10 and larger	Wet FA80-J8V to FA180-J9V		…
Cylindrical	14 and less	Wet FA80-J8V	C80-KV	FA80-H12VP
	16 and larger	Wet FA80-I8V	C80-KV	FA80-G12VP
Centerless	…	Wet FA80-J5V	C80-LV	…
Internal				
Production grinding	Under ½	Wet FA100-L8V	C90-MV	…
	½ to 1	Wet FA90-K8V	C80-LV	…
	Over 1 to 3	Wet FA80-J8V	C70-KV	FA80-H12VP
	Over 3	Wet FA70-I8V	C60-JV	FA70-G12VP
Tool room grinding	Under ½	Dry FA100-K8V	C90-KV	…
	½ to 1	Dry FA90-J8V	C80-JV	…
	Over 1 to 3	Dry FA80-I8V	C70-IV	FA80-G12VP
	Over 3	Dry FA70-I8V	C60-IV	FA70-G12VP

Table 5. Grinding Wheel Recommendations for Constructional Steels (Soft)

Grinding Operation	Wheel or Rim Diameter, Inches	First Choice	Alternate Choice (Porous type)	All-Around Wheel
Surfacing				
Straight wheels	14 and smaller	Wet FA46-J8V	FA46-H12VP	FA46-J8V
	14 and smaller	Dry FA46-I8V	FA46-H12VP	FA46-I8V
	Over 14	Wet FA36-J8V	FA36-H12VP	FA36-J8V
Segments	1½ rim or less	Wet FA24-H8V	FA30-F12VP	FA24-H8V
Cylinders	1½ rim or less	Wet FA24-I8V	FA30-G12VP	FA24-H8V
Cups	¾ rim or less	Wet FA24-H8V	FA30-F12VP	FA30-H8V
	(for wider rims, go one grade softer)			
Cylindrical	14 and smaller	Wet SFA60-M5V	…	SFA60-L5V
	16 and larger	Wet SFA54-M5V	…	SFA54-L5V
Centerless	…	Wet SFA54-N5V	…	SFA60-M5V
Internal	Under ½	Wet SFA60-M5V	…	SFA80-L6V
	½ to 1	Wet SFA60-L5V	…	SFA60-K5V
	Over 1 to 3	Wet SFA54-K5V	…	SFA54-J5V
	Over 3	Wet SFA46-K5V	…	SFA46-J5V

Table 6. Grinding Wheel Recommendations for Constructional Steels (Hardened or Carburized)

Grinding Operation	Wheel or Rim Diameter, Inches	First Choice	Alternate Choice (Porous Type)
Surfacing			
Straight wheels	14 and smaller	Wet FA46-I8V	FA46-G12VP
	14 and smaller	Dry FA46-H8V	FA46-F12VP
	Over 14	Wet FA36-I8V	FA36-G12VP
Segments or Cylinders	1½ rim or less	Wet FA30-H8V	FA36-F12VP
Cups	¾ rim or less	Wet FA36-H8V	FA46-F12VP
	(for wider rims, go one grade softer)		
Forms and Radius Grinding	8 and smaller	Wet FA60-L7V to FA100-M8V	
	8 and smaller	Dry FA60-K8V to FA100-L8V	
	10 and larger	Wet FA60-L7V to FA80-M7V	
Cylindrical			
Work diameter			
1 inch and smaller	14 and smaller	Wet SFA80-L6V	…
Over 1 inch	14 and smaller	Wet SFA80-K5V	…
1 inch and smaller	16 and larger	Wet SFA60-L5V	…
Over 1 inch	16 and larger	Wet SFA60-L5V	…
Centerless	…	Wet SFA80-M6V	…
Internal	Under ½	Wet SFA80-N6V	…
	½ to 1	Wet SFA60-M5V	…
	Over 1 to 3	Wet SFA54-L5V	…
	Over 3	Wet SFA46-K5V	…
	Under ½	Dry FA80-L6V	…
	½ to 1	Dry FA70-K8V	…
	Over 1 to 3	Dry FA60-J8V	FA60-H12VP
	Over 3	Dry FA46-J8V	FA54-H12VP

Cubic Boron Nitride (CBN) Grinding Wheels.—Although CBN is not quite as hard, strong, and wear-resistant as a diamond, it is far harder, stronger, and more resistant to wear than aluminum oxide and silicon carbide. As with diamond, CBN materials are available in different types for grinding workpieces of 50 Rc and above, and for superalloys of 35 Rc and harder. Microcrystalline CBN grinding wheels are suitable for grinding mild steels, medium-hard alloy steels, stainless steels, cast irons, and forged steels. Wheels with larger mesh size grains (up to 20⁄30), now available, provide for higher rates of metal removal.

Special types of CBN are produced for resin, vitrified, and electrodeposited bonds. Wheel standards and nomenclature generally conform to those used for diamond wheels (page 1163), except that the letter **B** instead of **D** is used to denote the type of abrasive. Grinding machines for CBN wheels are generally designed to take full advantage of the ability of CBN to operate at high surface speeds of 9,000–25,000 sfm. CBM is very responsive to changes in grinding conditions, and an increase in wheel speed from 5,000 to 10,000 sfm can increase wheel life by a factor of 6 or more. A change from a water-based coolant to a coolant such as a sulfochlorinated or sulfurized straight grinding oil can increase wheel life by a factor of 10 or more.

Machines designed specifically for use with CBN grinding wheels generally use either electrodeposited wheels or have special trueing systems for other CBN bond wheels, and are totally enclosed so they can use oil as a coolant. Numerical control systems are used, often running fully automatically, including loading and unloading. Machines designed for CBN grinding with electrodeposited wheels are extensively used for form and gear grinding, special systems being used to ensure rapid mounting to exact concentricity and truth in running, no trueing or dressing being required. CBN wheels can produce workpieces having excellent accuracy and finish, with no trueing or dressing for the life of the wheel, even over many hours or days of production grinding of hardened steel components.

Resin-, metal-, and vitrified-bond wheels are used extensively in production grinding, in standard and special machines. Resin-bonded wheels are used widely for dry tool and cutter resharpening on conventional hand-operated tool and cutter grinders. A typical wheel for such work would be designated 11V9 cup type, 100⁄120 mesh, 75 concentration, with a $\frac{1}{16}$ or $\frac{1}{8}$ in. rim section. Special shapes of resin-bonded wheels are used on dedicated machines for cutting tool manufacture. These types of wheels are usually self-dressing, and allow full machine control of the operation without the need for an operator to see, hear, or feel the action.

Metal-bonded CBN wheels are usually somewhat cheaper than those using other types of bond because only a thin layer of abrasive is present. Metal bonding is also used in manufacture of CBN honing stones. Vitrified-bond CBN wheels are a recent innovation, and high-performance bonds are still being developed. These wheels are used for grinding cams, internal diameters, and bearing components, and can be easily redressed.

An important aspect of grinding with CBN and diamond wheels is reduced heating of the workpiece, thought to result from their superior thermal conductivity compared with aluminum oxide, for instance. CBN and diamond grains also are harder, which means that they stay sharp longer than aluminum oxide grains. The superior ability to absorb heat from the workpiece during the grinding process reduces formation of untempered martensite in the ground surface, caused by overheating followed by rapid quenching. At the same time, a higher compressive residual stress is induced in the surface, giving increased fatigue resistance, compared with the tensile stresses found in surfaces ground with aluminum oxide abrasives. Increased fatigue resistance is of particular importance for gear grinding, especially in the root area.

Variations from General Grinding Wheel Recommendations.—Recommendations for the selection of grinding wheels are usually based on average values with regard to both operational conditions and process objectives. With variations from such average values, the composition of the grinding wheels must be adjusted to obtain optimum results. Although it is impossible to list and to appraise all possible variations and to define their effects on the selection of the best suited grinding wheels, some guidance is obtained from experience. The following tabulation indicates the general directions in which the characteristics of the initially selected grinding wheel may have to be altered in order to approach optimum performance. Variations in a sense opposite to those shown will call for wheel characteristic changes in reverse.

Conditions or Objectives	Direction of Change
To increase cutting rate	Coarser grain, softer bond, higher porosity
To retain wheel size and/or form	Finer grain, harder bond
For small or narrow work surface	Finer grain, harder bond
For larger wheel diameter	Coarser grain
To improve finish on work	Finer grain, harder bond, or resilient bond
For increased work speed or feed rate	Harder bond
For increased wheel speed	Generally, softer bond, except for high-speed grinding, which requires a harder bond for added wheel strength
For interrupted or coarse work surface	Harder bond
For thin walled parts	Softer bond
To reduce load on the machine drive motor	Softer bond

Dressing and Truing Grinding Wheels.—The perfect grinding wheel operating under ideal conditions will be self sharpening, i.e., as the abrasive grains become dull, they will tend to fracture and be dislodged from the wheel by the grinding forces, thereby exposing new, sharp abrasive grains. Although in precision machine grinding this ideal sometimes may be partially attained, it is almost never attained completely. Usually, the grinding wheel must be dressed and trued after mounting on the precision grinding machine spindle and periodically thereafter.

Dressing may be defined as any operation performed on the face of a grinding wheel that improves its cutting action. Truing is a dressing operation but is more precise, i.e., the face of the wheel may be made parallel to the spindle or made into a radius or special shape. Regularly applied truing is also needed for accurate size control of the work, particularly in automatic grinding. The tools and processes generally used in grinding wheel dressing and truing are listed and described in Table .

Table 1. Tools and Methods for Grinding Wheel Dressing and Truing

Designation	Description	Application
Rotating Hand Dressers	Freely rotating discs, either star-shaped with protruding points or discs with corrugated or twisted perimeter, supported in a fork-type handle, the lugs of which can lean on the tool rest of the grinding machine.	Preferred for bench- or floor-type grinding machines; also for use on heavy portable grinders (snagging grinders) where free-cutting proper ties of the grinding wheel are primarily sought and the accuracy of the trued profile is not critical.
Abrasive Sticks	Made of silicon carbide grains with a hard bond. Applied directly or supported in a handle. Less frequently abrasive sticks are also made of boron carbide.	Usually hand held and use limited to smaller-size wheels. Because it also shears the grains of the grinding wheel, or preshaping, prior to final dressing with, e.g., a diamond.

Table 1. *(Continued)* **Tools and Methods for Grinding Wheel Dressing and Truing**

Designation	Description	Application
Abrasive Wheels (Rolls)	Silicon carbide grains in a hard vitrified bond are cemented on ball-bearing mounted spindles. Use either as hand tools with handles or rigidly held in a supporting member of the grinding machine. Generally freely rotating; also available with adjustable brake for diamond wheel dressing.	Preferred for large grinding wheels as a diamond saver, but also for improved control of the dressed surface characteristics. By skewing the abrasive dresser wheel by a few degrees out of parallel with the grinding wheel axis, the basic crushing action is supplemented with wiping and shearing, thus producing the desired degree of wheel surface smoothness.
Single-Point Diamonds	A diamond stone of selected size is mounted in a steel nib of cylindrical shape with or without head, dimensioned to fit the truing spindle of specific grinding machines. Proper orientation and retainment of the diamond point in the setting is an important requirement.	The most widely used tool for dressing and truing grinding wheels in precision grinding. Permits precisely controlled dressing action by regulating infeed and cross feed rate of the truing spindle when the latter is guided by cams or templates for accurate form truing.
Single-Point Form Truing Diamonds	Selected diamonds having symmetrically located natural edges with precisely lapped diamond points, controlled cone angles and vertex radius, and the axis coinciding with that of the nib.	Used for truing operations requiring very accurately controlled, and often steeply inclined wheel profiles, such as are needed for thread and gear grinding, where one or more diamond points participate in generating the resulting wheel periphery form. Dependent on specially designed and made truing diamonds and nibs.
Cluster-Type Diamond Dresser	Several, usually seven, smaller diamond stones are mounted in spaced relationship across the working surface of the nib. In some tools, more than a single layer of such clusters is set at parallel levels in the matrix, the deeper positioned layer becoming active after the preceding layer has worn away.	Intended for straight-face dressing and permits the utilization of smaller, less expensive diamond stones. In use, the holder is canted at a 3° to 10° angle, bringing two to five points into contact with the wheel. The multiple-point contact permits faster cross feed rates during truing than may be used with single-point diamonds for generating a specific degree of wheel-face finish.
Impregnated Matrix-Type Diamond Dressers	The operating surface consists of a layer of small, randomly distributed, yet rather uniformly spaced diamonds that are retained in a bond holding the points in an essentially common plane. Supplied either with straight or canted shaft, the latter being used to cancel the tilt of angular truing posts.	For the truing of wheel surfaces consisting of a single or several flat elements. The nib face should be held tangent to the grinding wheel periphery or parallel with a flat working surface. Offers economic advantages where technically applicable because of using less expensive diamond splinters presented in a manner permitting efficient utilization.
Form- Generating Truing Devices	Swiveling diamond holder post with adjustable pivot location, arm length, and swivel arc, mounted on angularly adjustable cross slides with controlled traverse movement, permits the generation of various straight and circular profile elements, kept in specific mutual locations.	Such devices are made in various degrees of complexity for the positionally controlled interrelation of several different profile elements. Limited to regular straight and circular sections, yet offers great flexibility of setup, very accurate adjustment, and unique versatility for handling a large variety of frequently changing profiles.

Table 1. *(Continued)* **Tools and Methods for Grinding Wheel Dressing and Truing**

Designation	Description	Application
Contour-Duplicating Truing Devices	The form of a master, called cam or template, shaped to match the profile to be produced on the wheel, or its magnified version, is translated into the path of the diamond point by means of mechanical linkage, a fluid actuator, or a pantograph device.	Preferred single-point truing method for profiles to be produced in quantities warranting the making of special profile bars or templates. Used also in small- and medium-volume production when the complexity of the profile to be produced excludes alternate methods of form generation.
Grinding Wheel Contouring by Crush Truing	A hardened steel or carbide roll, which is free to rotate and has the desired form of the workpiece, is fed gradually into the grinding wheel, which runs at slow speed. The roll will, by crushing action, produce its reverse form in the wheel. Crushing produces a free-cutting wheel face with sharp grains.	Requires grinding machines designed for crush truing, having stiff spindle bearings, rigid construction, slow wheel speed for truing, etc. Due to the cost of crush rolls and equipment, the process is used for repetitive work only. It is one of the most efficient methods for precisely duplicating complex wheel profiles that are capable of grinding in the 8-microinch AA range. Applicable for both surface and cylindrical grinding.
Rotating Diamond Roll-Type Grinding Wheel Truing	Special rolls made to agree with specific profile specifications have their periphery coated with a large number of uniformly distributed diamonds, held in a matrix into which the individual stones are set by hand (for larger diamonds) or bonded by a plating process (for smaller elements).	The diamond rolls must be rotated by an air, hydraulic, or electric motor at about one-fourth of the grinding wheel surface speed and in opposite direction to the wheel rotation. Whereas the initial costs are substantially higher than for single-point diamond truing the savings in truing time warrants the method's application in large-volume production of profile-ground components.
Diamond Dressing Blocks	Made as flat blocks for straight wheel surfaces, are also available for radius dressing and profile truing. The working surface consists of a layer of electroplated diamond grains, uniformly distributed and capable of truing even closely toleranced profiles.	For straight wheels, dressing blocks can reduce dressing time and offer easy installation on surface grinders, where the blocks mount on the magnetic plate. Recommended for small- and medium-volume production for truing intricate profiles on regular surface grinders, because the higher pressure developed in crush dressing is avoided.

Guidelines for Truing and Dressing with Single-Point Diamonds.—The diamond nib should be canted at an angle of 10 to 15 degrees in the direction of the wheel rotation and also, if possible, by the same amount in the direction of the cross feed traverse during the truing (see diagram). The dragging effect resulting from this “angling,” combined with the occasional rotation of the diamond nib in its holder, will prolong the diamond life by limiting the extent of wear facets and will also tend to produce a pyramid shape of the diamond tip. The diamond may also be set to contact the wheel at about $\frac{1}{8}$ to $\frac{1}{4}$ inch below its centerline.

Depth of Cut: This amount should not exceed 0.001 inch per pass for general work, and will have to be reduced to 0.0002 to 0.0004 inch per pass for wheels with fine grains used for precise finishing work.

Diamond crossfeed rate: This value may be varied to some extent depending on the required wheel surface: faster crossfeed for free cutting, and slower crossfeed for producing fine finishes. Such variations, however, must always stay within the limits set by the grain size of the wheel. Thus, the advance rate of the truing diamond per wheel revolution should not exceed the diameter of a grain or be less than half of that rate. Consequently, the diamond crossfeed must be slower for a large wheel than for a smaller wheel having the same grain size number.

Typical crossfeed values for frequently used grain sizes are given in Table 2.

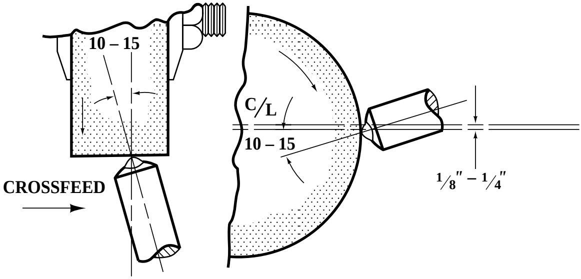


Table 2. Typical Diamond Truing and Crossfeeds

Grain Size	30	36	46	50
Crossfeed per Wheel Rev., in.	0.014–0.024	0.012–0.019	0.008–0.014	0.007–0.012
Grain Size	60	80	120	…
Crossfeed per Wheel Rev., in.	0.006–0.010	0.004–0.007	0.0025–0.004	…

These values can be easily converted into the more conveniently used inch-per-minute units, simply by multiplying them by the rpm of the grinding wheel.

Example: For a 20-inch diameter wheel, Grain No. 46, running at 1200 rpm: Crossfeed rate for roughing-cut truing—approximately 17 ipm, for finishing-cut truing—approximately 10 ipm

Coolant should be applied before the diamond comes into contact with the wheel and must be continued in generous supply while truing.

The speed of the grinding wheel should be at the regular grinding rate, or not much lower. For that reason, the feed wheels of centerless grinding machines usually have an additional speed rate higher than functionally needed, that speed being provided for wheel truing only.

The initial approach of the diamond to the wheel surface must be carried out carefully to prevent sudden contact with the diamond, resulting in penetration in excess of the selected depth of cut. It should be noted that the highest point of a worn wheel is often in its center portion and not at the edge from which the crossfeed of the diamond starts.

The general conditions of the truing device are important for best truing results and for assuring extended diamond life. A rigid truing spindle, well-seated diamond nib, and firmly set diamond point are mandatory. Sensitive infeed and smooth traverse movement at uniform speed also must be maintained.

Resetting of the diamond point.: Never let the diamond point wear to a degree where the grinding wheel is in contact with the steel nib. Such contact can damage the setting of the diamond point and result in its loss. Expert resetting of a worn diamond can repeatedly add to its useful life, even when applied to lighter work because of reduced size.

Size Selection Guide for Single-Point Truing Diamonds.—There are no rigid rules for determining the proper size of the diamond for any particular truing application because of the very large number of factors affecting that choice. Several of these factors are related to the condition, particularly the rigidity, of the grinding machine and truing device, as well as to such characteristics of the diamond itself as purity, crystalline structure, etc. Although these factors are difficult to evaluate in a generally applicable manner, the expected effects of several other conditions can be appraised and should be considered in the selection of the proper diamond size.

The recommended sizes in Table 3 must be considered as informative only and as representing minimum values for generally favorable conditions. Factors calling for larger diamond sizes than listed are the following:

Silicon carbide wheels (Table 3 refers to aluminum oxide wheels)

Dry truing

Grain sizes coarser than No. 46

Bonds harder than M

Wheel speed substantially higher than 6500 sfm.

It is advisable to consider any single or pair of these factors as justifying the selection of one size larger diamond. As an example: for truing an SiC wheel, with grain size No. 36 and hardness P, select a diamond that is two sizes larger than that shown in Table 3 for the wheel size in use.

Table 3. Recommended Minimum Sizes for Single-Point Truing Diamonds

Diamond Size in Carats[a]	Index Number (Wheel Dia. × Width in Inches)	Examples of Max. Grinding Wheel Dimensions	
		Diameter	Width
0.25	3	4	0.75
0.35	6	6	1
0.50	10	8	1.25
0.60	15	10	1.50
0.75	21	12	1.75
1.00	30	12	2.50
1.25	48	14	3.50
1.50	65	16	4.00
1.75	80	20	4.00
2.00	100	20	5.00
2.50	150	24	6.00
3.00	200	24	8.00
3.50	260	30	8.00
4.00	350	36	10.00

[a] One carat equals 0.2 gram.

Single-point diamonds are available as loose stones, but are preferably procured from specialized manufacturers supplying the diamonds set into steel nibs. Expert setting, comprising both the optimum orientation of the stone and its firm retainment, is mandatory for assuring adequate diamond life and satisfactory truing. Because the holding devices for truing diamonds are not yet standardized, the required nib dimensions vary depending on the make and type of different grinding machines. Some nibs are made with angular heads, usually hexagonal, to permit occasional rotation of the nib either manually, with a wrench, or automatically.

Diamond Wheels

Diamond Wheels.—A diamond wheel is a special type of grinding wheel in which the abrasive elements are diamond grains held in a bond and applied to form a layer on the operating face of a non-abrasive core. Diamond wheels are used for grinding very hard or highly abrasive materials. Primary applications are the grinding of cemented carbides, such as the sharpening of carbide cutting tools; the grinding of glass, ceramics, asbestos, and cement products; and the cutting and slicing of germanium and silicon.

Shapes of Diamond Wheels.—The industry-wide accepted Standard (ANSI B74.3-1974) specifies ten basic diamond wheel core shapes which are shown in Table 1 with the applicable designation symbols. The applied diamond abrasive layer may have different cross-sectional shapes. Those standardized are shown in Table 2. The third aspect which is standardized is the location of the diamond section on the wheel as shown by the diagrams in Table . Finally, modifications of the general core shape together with pertinent designation letters are given in Table 4.

The characteristics of the wheel shape listed in these four tables make up the components of the standard designation symbol for diamond wheel shapes. An example of that symbol with arbitrarily selected components is shown in Fig. 1.

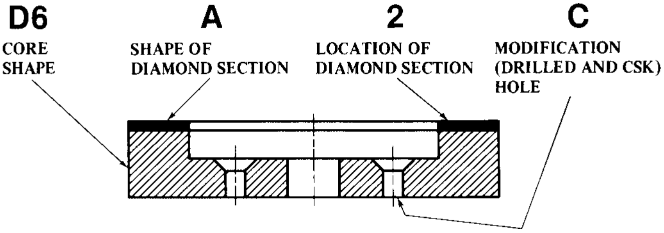


Fig. 1. A Typical Diamond Wheel Shape Designation Symbol

An explanation of these components is as follows:

Basic Core Shape: This portion of the symbol indicates the basic shape of the core on which the diamond abrasive section is mounted. The shape is actually designated by a number. The various core shapes and their designations are given in Table 1.

Diamond Cross-Section Shape: This, the second component, consisting of one or two letters, denotes the cross-sectional shape of the diamond abrasive section. The various shapes and their corresponding letter designations are given in Table 2.

Diamond Section Location: The third component of the symbol consists of a number which gives the location of the diamond section, i.e., periphery, side, corner, etc. An explanation of these numbers is shown in Table 3.

Modification: The fourth component of the symbol is a letter designating some modification, such as drilled and counterbored holes for mounting or special relieving of diamond section or core. This modification position of the symbol is used only when required. The modifications and their designations are given in Table 4.

Table 1. Diamond Wheel Core Shapes and Designations *ANSI B74.3-1974*

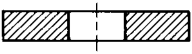
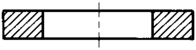
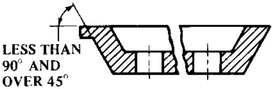
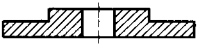
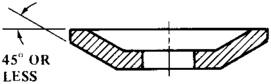
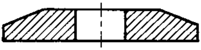
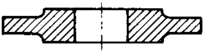
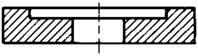
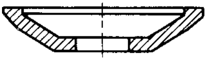
 1	 9
 2	 11
 3	 12
 4	 14
 6	 15

Table 2. Diamond Cross-sections and Designations
ANSI B74.3-1974

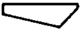
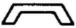
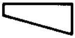
A 	**CH** 	**ER** 	**GN** 	**LL** 	**S** 
AH 	**D** 	**ET** 	**H** 	**M** 	**U** 
B 	**DD** 	**F** 	**J** 	**P** 	**V** 
BT 	**E** 	**FF** 	**K** 	**Q** 	**Y** 
C 	**EE** 	**G** 	**L** 	**QQ** 	

Table 3. Designations for Location of Diamond Section on Diamond Wheel *ANSI B74.3-1974*

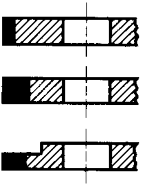
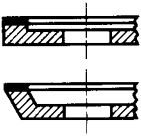
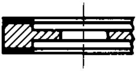
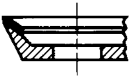
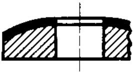
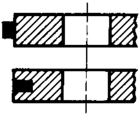
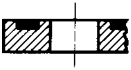
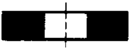
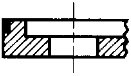
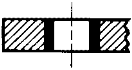
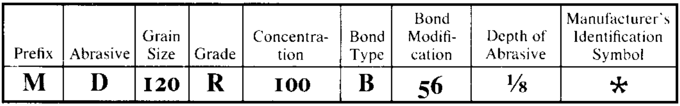
Designation No. and Location	Description	Illustration
1 — Periphery	The diamond section shall be placed on the periphery of the core and shall extend the full thickness of the wheel. The axial length of this section may be greater than, equal to, or less than the depth of diamond, measured radially. A hub or hubs shall not be considered as part of the wheel thickness for this definition.	
2 — Side	The diamond section shall be placed on the side of the wheel and the length of the diamond section shall extend from the periphery toward the center. It may or may not include the entire side and shall be greater than the diamond depth measured axially. It shall be on that side of the wheel which is commonly used for grinding purposes.	
3 — Both Sides	The diamond sections shall be placed on both sides of the wheel and shall extend from the periphery toward the center. They may or may not include the entire sides, and the radial length of the diamond section shall exceed the axial diamond depth.	
4 — Inside Bevel or Arc	This designation shall apply to the general wheel types 2, 6, 11, 12, and 15 and shall locate the diamond section on the side wall. This wall shall have an angle or arc extending from a higher point at the wheel periphery to a lower point toward the wheel center.	
5 — Outside Bevel or Arc	This designation shall apply to the general wheel types, 2, 6, 11, and 15 and shall locate the diamond section on the side wall. This wall shall have an angle or arc extending from a lower point at the wheel periphery to a higher point toward the wheel center.	
6 — Part of Periphery	The diamond section shall be placed on the periphery of the core but shall not extend the full thickness of the wheel and shall not reach to either side.	
7 — Part of Side	The diamond section shall be placed on the side of the core and shall not extend to the wheel periphery. It may or may not extend to the center.	

Table 3. *(Continued)* **Designations for Location of Diamond Section on Diamond Wheel** *ANSI B74.3-1974*

Designation No. and Location	Description	Illustration
8 — Throughout	Designates wheels of solid diamond abrasive section without cores.	
9 — Corner	Designates a location which would commonly be considered to be on the periphery except that the diamond section shall be on the corner but shall not extend to the other corner.	
10 — Annular	Designates a location of the diamond abrasive section on the inner annular surface of the wheel.	

Composition of Diamond and Cubic Boron Nitride Wheels.—According to American National Standard ANSI B74.13-1990, a series of symbols is used to designate the composition of these wheels. An example is shown below.



Prefix	Abrasive	Grain Size	Grade	Concentration	Bond Type	Bond Modification	Depth of Abrasive	Manufacturer's Identification Symbol
M	D	120	R	100	B	56	1/8	*

Fig. 2. Designation Symbols for Composition of Diamond and Cubic Boron Nitride Wheels

The meaning of each symbol is indicated by the following list:

1) *Prefix:* The prefix is a manufacturer's symbol indicating the exact kind of abrasive. Its use is optional.

2) *Abrasive Type:* The letter (B) is used for cubic boron nitride and (D) for diamond.

3) *Grain Size:* The grain sizes commonly used and varying from coarse to very fine are indicated by the following numbers: 8, 10, 12, 14, 16, 20, 24, 30, 36, 46, 54, 60, 70, 80, 90, 100, 120, 150, 180, and 220. The following additional sizes are used occasionally: 240, 280, 320, 400, 500, and 600. The wheel manufacturer may add to the regular grain number an additional symbol to indicate a special grain combination.

4) *Grade:* Grades are indicated by letters of the alphabet from A to Z in all bonds or processes. Wheel grades from A to Z range from soft to hard.

5) *Concentration:* The concentration symbol is a manufacturer's designation. It may be a number or a symbol.

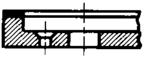
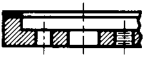
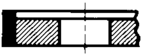
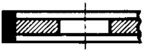
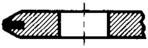
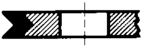
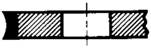
6) *Bond:* Bonds are indicated by the following letters: B, resinoid; V, vitrified; M, metal.

7) *Bond Modification:* Within each bond type a manufacturer may have modifications to tailor the bond to a specific application. These modifications may be identified by either letters or numbers.

8) *Abrasive Depth:* Abrasive section depth, in inches or millimeters (inches illustrated), is indicated by a number or letter which is the amount of total dimensional wear a user may expect from the abrasive portion of the product. Most diamond and CBN wheels are made with a depth of coating on the order of $\frac{1}{16}$ in., $\frac{1}{8}$ in., or more as specified. In some cases the diamond is applied in thinner layers, as thin as one thickness of diamond grains. The L is included in the marking system to identify a layered type product.

9) *Manufacturer's Identification Symbol:* The use of this symbol is optional.

Table 4. Designation Letters for Modifications of Diamond Wheels *ANSI B74.3-1974*

Designation Letter[a]	Description	Illustration
B — Drilled and Counterbored	Holes drilled and counterbored in core.	
C — Drilled and Countersunk	Holes drilled and countersunk in core.	
H — Plain Hole	Straight hole drilled in core.	
M — Holes Plain and Threaded	Mixed holes, some plain, some threaded, are in core.	
P — Relieved One Side	Core relieved on one side of wheel. Thickness of core is less than wheel thickness.	
R — Relieved Two Sides	Core relieved on both sides of wheel. Thickness of core is less than wheel thickness.	
S — Segmented-Diamond Section	Wheel has segmental diamond section mounted on core. (Clearance between segments has no bearing on definition.)	
SS — Segmental and Slotted	Wheel has separated segments mounted on a slotted core.	
T — Threaded Holes	Threaded holes are in core.	
Q — Diamond Inserted	Three surfaces of the diamond section are partially or completely enclosed by the core.	
V — Diamond Inverted	Any diamond cross section, which is mounted on the core so that the interior point of any angle, or the concave side of any arc, is exposed shall be considered inverted. *Exception:* Diamond cross section AH shall be placed on the core with the concave side of the arc exposed.	 

[a] Y — Diamond Inserted and Inverted. See definitions for Q and V.

The Selection of Diamond Wheels.—Two general aspects must be defined: (a) The shape of the wheel, also referred to as the basic wheel type and (b) The specification of the abrasive portion.

Table 5. General Diamond Wheel Recommendations for Wheel Type and Abrasive Specification

Typical Applications or Operation	Basic Wheel Type	Abrasive Specification
Single Point Tools (offhand grinding)	D6A2C	*Rough:* MD100-N100-B$\frac{1}{8}$ *Finish:* MD220-P75-B$\frac{1}{8}$
Single Point Tools (machine ground)	D6A2H	*Rough:* MD180-J100-B$\frac{1}{8}$ *Finish:* MD320-L75-B$\frac{1}{8}$
Chip Breakers	D1A1	MD150-R100-B$\frac{1}{8}$
Multitooth Tools and Cutters (face mills, end mills, reamers, broaches, etc.)		
Sharpening and Backing off	D11V9	Rough: MD100-R100-B$\frac{1}{8}$ Combination: MD150-R100-B$\frac{1}{8}$ *Finish:* MD220-R100-B$\frac{1}{8}$
Fluting	D12A2	MD180-N100-B$\frac{1}{8}$
Saw Sharpening	D12A2	MD180-R100-B$\frac{1}{8}$
Surface Grinding (horizontal spindle)	D1A1	*Rough:* MD120-N100-B$\frac{1}{8}$ *Finish:* MD240-P100-B$\frac{1}{8}$
Surface Grinding (vertical spindle)	D2A2T	MD80-R75-B$\frac{1}{8}$
Cylindrical or Centertype Grinding	D1A1	MD120-P100-B$\frac{1}{8}$
Internal Grinding	D1A1	MD150-N100-B$\frac{1}{8}$
Slotting and Cutoff	D1A1R	MD150-R100-B$\frac{1}{4}$
Lapping	Disc	MD400-L50-B$\frac{1}{16}$
Hand Honing	DH1, DH2	*Rough:* MD220-B$\frac{1}{16}$ *Finish:* MD320-B$\frac{1}{6}$

General recommendations for the dry grinding, with resin bond diamond wheels, of most grades of cemented carbides of average surface to ordinary finishes at normal rates of metal removal with average size wheels, as published by Cincinnati Milacron, are listed in Table 5.

A further set of variables are *the dimensions of the wheel,* which must be adapted to the available grinding machine and, in some cases, to the configuration of the work.

The general abrasive specifications in Table 5 may be modified to suit operating conditions by the following suggestions:

Use softer wheel grades for harder grades of carbides, for grinding larger areas or larger or wider wheel faces.

Use harder wheel grades for softer grades of carbides, for grinding smaller areas, for using smaller and narrower face wheels and for light cuts.

Use fine grit sizes for harder grades of carbides and to obtain better finishes.

Use coarser grit sizes for softer grades of carbides and for roughing cuts.

Use higher diamond concentration for harder grades of carbides, for larger diameter or wider face wheels, for heavier cuts, and for obtaining better finish.

Guidelines for the Handling and Operation of Diamond Wheels.—Grinding machines used for grinding with diamond wheels should be of the precision type, in good service condition, with true running spindles and smooth slide movements.

Mounting of Diamond Wheels: Wheel mounts should be used which permit the precise centering of the wheel, resulting in a runout of less than 0.001 inch axially and 0.0005 inch radially. These conditions should be checked with a 0.0001-inch type dial indicator. Once mounted and centered, the diamond wheel should be retained on its mount and stored in that condition when temporarily removed from the machine.

Truing and Dressing: Resinoid bonded diamond wheels seldom require dressing, but when necessary a soft silicon carbide stick may be hand-held against the wheel. Peripheral and cup type wheels may be sharpened by grinding the cutting face with a 60 to 80 grit silicon carbide wheel. This can be done with the diamond wheel mounted on the spindle of the machine, and with the silicon carbide wheel driven at a relatively slow speed by a specially designed table-mounted grinder or by a small table-mounted tool post grinder. The diamond wheel can be mounted on a special arbor and ground on a lathe with a tool post grinder; peripheral wheels can be ground on a cylindrical grinder or with a special brake-controlled truing device with the wheel mounted on the machine on which it is used. Cup and face type wheels are often lapped on a cast iron or glass plate using a 100 grit silicon carbide abrasive. Care must be used to lap the face parallel to the back, otherwise they must be ground to restore parallelism. Peripheral diamond wheels can be trued and dressed by grinding a silicon carbide block or a special diamond impregnated bronze block in a manner similar to surface grinding. Conventional diamonds must not be used for truing and dressing diamond wheels.

Speeds and Feeds in Diamond Grinding.—General recommendations are as follows:

Wheel Speeds: The generally recommended wheel speeds for diamond grinding are in the range of 5000 to 6000 surface feet per minute, with this upper limit as a maximum to avoid harmful “overspeeding.” Exceptions from that general rule are diamond wheels with coarse grains and high concentration (100 per cent) where the wheel wear in dry surface grinding can be reduced by lowering the speed to 2500–3000 sfpm. However, this lower speed range can cause rapid wheel breakdown in finer grit wheels or in those with reduced diamond concentration.

Work Speeds: In diamond grinding, work rotation and table traverse are usually established by experience, adjusting these values to the selected infeed so as to avoid excessive wheel wear.

Infeed per Pass: Often referred to as downfeed and usually a function of the grit size of the wheel. The following are general values which may be increased for raising the productivity, or lowered to improve finish or to reduce wheel wear.

Wheel Grit Size Range	Infeed per Pass
100 to 120	0.001 inch
150 to 220	0.0005 inch
250 and finer	0.00025 inch

Grinding Wheel Safety

Safety in Operating Grinding Wheels.—Grinding wheels, although capable of exceptional cutting performance due to hardness and wear resistance, are prone to damage caused by improper handling and operation. Vitrified wheels, comprising the major part of grinding wheels used in industry, are held together by an inorganic bond which is actually a type of pottery product and therefore brittle and breakable. Although most of the organic bond types are somewhat more resistant to shocks, it must be realized that all grinding wheels are conglomerates of individual grains joined by a bond material whose strength is limited by the need of releasing the dull, abrasive grains during use.

It must also be understood that during the grinding process very substantial forces act on the grinding wheel, including the centrifugal force due to rotation, the grinding forces resulting from the resistance of the work material, and shocks caused by sudden contact with the work. To be able to resist these forces, the grinding wheel must have a substantial minimum strength throughout that is well beyond that needed to hold the wheel together under static conditions.

Finally, a damaged grinding wheel can disintegrate during grinding, liberating dormant forces which normally are constrained by the resistance of the bond, thus presenting great hazards to both operator and equipment.

To avoid breakage of the operating wheel and, should such a mishap occur, to prevent damage or injury, specific precautions must be applied. These safeguards have been formulated into rules and regulations and are set forth in the American National Standard ANSI B7.1-1988, entitled the American National Standard Safety Requirements for the Use, Care, and Protection of Abrasive Wheels.

Handling, Storage and Inspection.—Grinding wheels should be hand carried, or transported, with proper support, by truck or conveyor. A grinding wheel must not be rolled around on its periphery.

The storage area, positioned not far from the location of the grinding machines, should be free from excessive temperature variations and humidity. Specially built racks are recommended on which the smaller or thin wheels are stacked lying on their sides and the larger wheels in an upright position on two-point cradle supports consisting of appropriately spaced wooden bars. Partitions should separate either the individual wheels, or a small group of identical wheels. Good accessibility to the stored wheels reduces the need of undesirable handling.

Inspection will primarily be directed at detecting visible damage, mostly originating from handling and shipping. Cracks which are not obvious can usually be detected by “ring testing,” which consists of suspending the wheel from its hole and tapping it with a nonmetallic implement. Heavy wheels may be allowed to rest vertically on a clean, hard floor while performing this test. A clear metallic tone, a “ring”, should be heard; a dead sound being indicative of a possible crack or cracks in the wheel.

Machine Conditions.—The general design of the grinding machines must ensure safe operation under normal conditions. The bearings and grinding wheel spindle must be dimensioned to withstand the expected forces and ample driving power should be provided to ensure maintenance of the rated spindle speed. For the protection of the operator, stationary machines used for dry grinding should have a provision made for connection to an exhaust system and when used for off-hand grinding, a work support must be available.

Wheel guards are particularly important protection elements and their material specifications, wall thicknesses and construction principles should agree with the Standard's specifications. The exposure of the wheel should be just enough to avoid interference with the grinding operation. The need for access of the work to the grinding wheel will define the boundary of guard opening, particularly in the direction of the operator.

Grinding Wheel Mounting.—The mass and speed of the operating grinding wheel makes it particularly sensitive to imbalance. Vibrations that result from such conditions are harmful to the machine, particularly the spindle bearings, and they also affect the ground surface, i.e., wheel imbalance causes chatter marks and interferes with size control. Grinding wheels are shipped from the manufacturer's plant in a balanced condition, but retaining the balanced state after mounting the wheel is quite uncertain. Balancing of the mounted wheel is thus required, and is particularly important for medium and large size wheels, as well as for producing acccurate and smooth surfaces. The most common way of balancing mounted wheels is by using balancing flanges with adjustable weights. The wheel and balancing flanges are mounted on a short balancing arbor, the two concentric and round stub ends of which are supported in a balancing stand.

Such stands are of two types: 1) the parallel straight-edged, which must be set up precisely level; and 2) the disk type having two pairs of ball bearing mounted overlapping disks, which form a V for containing the arbor ends without hindering the free rotation of the wheel mounted on that arbor.

The wheel will then rotate only when it is out of balance and its heavy spot is not in the lowest position. Rotating the wheel by hand to different positions will move the heavy spot, should such exist, from the bottom to a higher location where it can reveal its presence by causing the wheel to turn. Having detected the presence and location of the heavy spot, its effect can be cancelled by displacing the weights in the circular groove of the flange until a balanced condition is accomplished.

Flanges are commonly used means for holding grinding wheels on the machine spindle. For that purpose, the wheel can either be mounted directly through its hole or by means of a sleeve which slips over a tapered section of the machine spindle. Either way, the flanges must be of equal diameter, usually not less than one-third of the new wheel's diameter. The purpose is to securely hold the wheel between the flanges without interfering with the grinding operation even when the wheel becomes worn down to the point where it is ready to be discarded. Blotters or flange facings of compressible material should cover the entire contact area of the flanges.

One of the flanges is usually fixed while the other is loose and can be removed and adjusted along the machine spindle. The movable flange is held against the mounted grinding wheel by means of a nut engaging a threaded section of the machine spindle. The sense of that thread should be such that the nut will tend to tighten as the spindle revolves. In other words, to remove the nut, it must be turned in the direction that the spindle revolves when the wheel is in operation.

Safe Operating Speeds.—Safe grinding processes are predicated on the proper use of the previously discussed equipment and procedures, and are greatly dependent on the application of adequate operating speeds.

The Standard establishes maximum speeds at which grinding wheels can be operated, assigning the various types of wheels to several classification groups. Different values are listed according to bond type and to wheel strength, distinguishing between low, medium and high strength wheels.

For the purpose of general information, the accompanying table shows an abbreviated version of the Standard's specification. However, for the governing limits, the authoritative source is the manufacturer's tag on the wheel which, particularly for wheels of lower strength, might specify speeds below those of the table.

All grinding wheels of 6 inches or greater diameter must be test run in the wheel manufacturer's plant at a speed that for all wheels having operating speeds in excess of 5000 sfpm is 1.5 times the maximum speed marked on the tag of the wheel.

The table shows the permissible wheel speeds in surface feet per minute (sfpm) units, whereas the tags on the grinding wheels state, for the convenience of the user, the maximum operating speed in revolutions per minute (rpm). The sfpm unit has the advantage of remaining valid for worn wheels whose rotational speed may be increased to the applicable sfpm value. The conversion from either one to the other of these two kinds of units is a matter of simple calculation using the formulas:

$$\text{sfpm} = \text{rpm} \times \frac{D}{12} \times \pi$$

or

$$\text{rpm} = \frac{\text{sfpm} \times 12}{D \times \pi}$$

Where D = maximum diameter of the grinding wheel, in inches. Table 2, showing the conversion values from surface speed into rotational speed, can be used for the direct reading of the rpm values corresponding to several different wheel diameters and surface speeds.

Special Speeds: Continuing progress in grinding methods has led to the recognition of certain advantages that can result from operating grinding wheels above, sometimes even higher than twice, the speeds considered earlier as the safe limits of grinding wheel operations. Advantages from the application of high speed grinding are limited to specific processes, but the Standard admits, and offers code regulations for the use of wheels at special high speeds. These regulations define the structural requirements of the grinding machine and the responsibilities of the grinding wheel manufacturers, as well as of the users. High speed grinding should not be applied unless the machines, particularly guards, spindle assemblies, and drive motors, are suitable for such methods. Also, appropriate grinding wheels expressly made for special high speeds must be used and, of course, the maximum operating speeds indicated on the wheel's tag must never be exceeded.

Portable Grinders.—The above discussed rules and regulations, devised primarily for stationary grinding machines apply also to portable grinders. In addition, the details of various other regulations, specially applicable to different types of portable grinders are discussed in the Standard, which should be consulted, particularly for safe applications of portable grinding machines.

Table 1. Maximum Peripheral Speeds for Grinding Wheels Based on ANSI B7.1–1988

Classification No.	Types of Wheels[a]	Maximum Operating Speeds, sfpm, Depending on Strength of Bond	
		Inorganic Bonds	Organic Bonds
1	Straight wheels — Type 1, except classifications 6, 7, 9, 10, 11, and 12 below Type 4[b] — Taper Side Wheels Types 5, 7, 20, 21, 22, 23, 24, 25, 26 Dish wheels — Type 12 Saucer wheels — Type 13 Cones and plugs — Types 16, 17, 18, 19	5,500 to 6,500	6,500 to 9,500
2	Cylinder wheels — Type 2 Segments	5,000 to 6,000	5,000 to 7,000
3	Cup shape tool grinding wheels — Types 6 and 11 (for fixed base machines)	4,500 to 6,000	6,000 to 8,500
4	Cup shape snagging wheels — Types 6 and 11 (for portable machines)	4,500 to 6,500	6,000 to 9,500
5	Abrasive disks	5,500 to 6,500	5,500 to 8,500
6	Reinforced wheels — except cutting-off wheels (depending on diameter and thickness)	…	9,500 to 16,000
7	Type 1 wheels for bench and pedestal grinders, Types 1 and 5 also in certain sizes for surface grinders	5,500 to 7,550	6,500 to 9,500
8	Diamond and cubic boron nitride wheels Metal bond Steel centered cutting off	to 6,500 to 12,000 to 16,000	to 9,500 … to 16,000
9	Cutting-off wheels — Larger than 16-inch diameter (incl. reinforced organic)	…	9,500 to 14,200
10	Cutting-off wheels — 16-inch diameter and smaller (incl. reinforced organic)	…	9,500 to 16,000
11	Thread and flute grinding wheels	8,000 to 12,000	8,000 to 12,000
12	Crankshaft and camshaft grinding wheels	5,500 to 8,500	6,500 to 9,500

[a] See Tables and Tables starting on page 1148.

[b] Non-standard shape. For snagging wheels, 16 inches and larger — Type 1, internal wheels — Types 1 and 5, and mounted wheels, see ANSI B7.1–1988. Under no conditions should a wheel be operated faster than the maximum operating speed established by the manufacturer.

Values in this table are for general information only.

Table 2. Revolutions per Minute for Various Grinding Speeds and Wheel Diameters (Based on B7.1–1988)

Wheel Diameter, Inch	Peripheral (Surface) Speed, Feet per Minute																Wheel Diameter, Inch
	4,000	4,500	5,000	5,500	6,000	6,500	7,000	7,500	8,000	8,500	9,000	9,500	10,000	12,000	14,000	16,000	
	Revolutions per Minute																
1	15,279	17,189	19,099	21,008	22,918	24,828	26,738	28,648	30,558	32,468	34,377	36,287	38,197	45,837	53,476	61,115	1
2	7,639	8,594	9,549	10,504	11,459	12,414	13,369	14,324	15,279	16,234	17,189	18,144	19,099	22,918	26,738	30,558	2
3	5,093	5,730	6,366	7,003	7,639	8,276	8,913	9,549	10,186	10,823	11,459	12,096	12,732	15,279	17,825	20,372	3
4	3,820	4,297	4,775	5,252	5,730	6,207	6,685	7,162	7,639	8,117	8,594	9,072	9,549	11,459	13,369	15,279	4
5	3,056	3,438	3,820	4,202	4,584	4,966	5,348	5,730	6,112	6,494	6,875	7,257	7,639	9,167	10,695	12,223	5
6	2,546	2,865	3,183	3,501	3,820	4,138	4,456	4,775	5,093	5,411	5,730	6,048	6,366	7,639	8,913	10,186	6
7	2,183	2,456	2,728	3,001	3,274	3,547	3,820	4,093	4,365	4,638	4,911	5,184	5,457	6,548	7,639	8,731	7
8	1,910	2,149	2,387	2,626	2,865	3,104	3,342	3,581	3,820	4,058	4,297	4,536	4,775	5,730	6,685	7,639	8
9	1,698	1,910	2,122	2,334	2,546	2,759	2,971	3,183	3,395	3,608	3,820	4,032	4,244	5,093	5,942	6,791	9
10	1,528	1,719	1,910	2,101	2,292	2,483	2,674	2,865	3,056	3,247	3,438	3,629	3,820	4,584	5,348	6,112	10
12	1,273	1,432	1,592	1,751	1,910	2,069	2,228	2,387	2,546	2,706	2,865	3,024	3,183	3,820	4,456	5,093	12
14	1,091	1,228	1,364	1,501	1,637	1,773	1,910	2,046	2,183	2,319	2,456	2,592	2,728	3,274	3,820	4,365	14
16	955	1,074	1,194	1,313	1,432	1,552	1,671	1,790	1,910	2,029	2,149	2,268	2,387	2,865	3,342	3,820	16
18	849	955	1,061	1,167	1,273	1,379	1,485	1,592	1,698	1,804	1,910	2,016	2,122	2,546	2,971	3,395	18
20	764	859	955	1,050	1,146	1,241	1,337	1,432	1,528	1,623	1,719	1,814	1,910	2,292	2,674	3,056	20
22	694	781	868	955	1,042	1,129	1,215	1,302	1,389	1,476	1,563	1,649	1,736	2,083	2,431	2,778	22
24	637	716	796	875	955	1,035	1,114	1,194	1,273	1,353	1,432	1,512	1,592	1,910	2,228	2,546	24
26	588	661	735	808	881	955	1,028	1,102	1,175	1,249	1,322	1,396	1,469	1,763	2,057	2,351	26
28	546	614	682	750	819	887	955	1,023	1,091	1,160	1,228	1,296	1,364	1,637	1,910	2,183	28
30	509	573	637	700	764	828	891	955	1,019	1,082	1,146	1,210	1,273	1,528	1,783	2,037	30
32	477	537	597	657	716	776	836	895	955	1,015	1,074	1,134	1,194	1,432	1,671	1,910	32
34	449	506	562	618	674	730	786	843	899	955	1,011	1,067	1,123	1,348	1,573	1,798	34
36	424	477	531	584	637	690	743	796	849	902	955	1,008	1,061	1,273	1,485	1,698	36
38	402	452	503	553	603	653	704	754	804	854	905	955	1,005	1,206	1,407	1,608	38
40	382	430	477	525	573	621	668	716	764	812	859	907	955	1,146	1,337	1,528	40
42	364	409	455	500	546	591	637	682	728	773	819	864	909	1,091	1,273	1,455	42
44	347	391	434	477	521	564	608	651	694	738	781	825	868	1,042	1,215	1,389	44
46	332	374	415	457	498	540	581	623	664	706	747	789	830	996	1,163	1,329	46
48	318	358	398	438	477	517	557	597	637	676	716	756	796	955	1,114	1,273	48
53	288	324	360	396	432	468	504	541	577	613	649	685	721	865	1,009	1,153	53
60	255	286	318	350	382	414	446	477	509	541	573	605	637	764	891	1,019	60
72	212	239	265	292	318	345	371	398	424	451	477	504	531	637	743	849	72

Cylindrical Grinding

Cylindrical grinding designates a general category of various grinding methods that have the common characteristic of rotating the workpiece around a fixed axis while grinding outside surface sections in controlled relation to that axis of rotation.

The form of the part or section being ground in this process is frequently cylindrical, hence the designation of the general category. However, the shape of the part may be tapered or of curvilinear profile; the position of the ground surface may also be perpendicular to the axis; and it is possible to grind concurrently several surface sections, adjacent or separated, of equal or different diameters, located in parallel or mutually inclined planes, etc., as long as the condition of a common axis of rotation is satisfied.

Size Range of Workpieces and Machines: Cylindrical grinding is applied in the manufacture of miniature parts, such as instrument components and, at the opposite extreme, for grinding rolling mill rolls weighing several tons. Accordingly, there are cylindrical grinding machines of many different types, each adapted to a specific work-size range. Machine capacities are usually expressed by such factors as maximum work diameter, work length and weight, complemented, of course, by many other significant data.

Plain, Universal, and Limited-Purpose Cylindrical Grinding Machines.—The plain cylindrical grinding machine is considered the basic type of this general category, and is used for grinding parts with cylindrical or slightly tapered form.

The universal cylindrical grinder can be used, in addition to grinding the basic cylindrical forms, for the grinding of parts with steep tapers, of surfaces normal to the part axis, including the entire face of the workpiece, and for internal grinding independently or in conjunction with the grinding of the part's outer surfaces. Such variety of part configurations requiring grinding is typical of work in the tool room, which constitutes the major area of application for universal cylindrical grinding machines.

Limited-purpose cylindrical grinders are needed for special work configurations and for high-volume production, where productivity is more important than flexibility of adaptation. Examples of limited-purpose cylindrical grinding machines are crankshaft and camshaft grinders, polygonal grinding machines, roll grinders, etc.

Traverse or Plunge Grinding.—In traverse grinding, the machine table carrying the work performs a reciprocating movement of specific travel length for transporting the rotating workpiece along the face of the grinding wheel. At each or at alternate stroke ends, the wheel slide advances for the gradual feeding of the wheel into the work. The length of the surface that can be ground by this method is generally limited only by the stroke length of the machine table. In large roll grinders, the relative movement between work and wheel is accomplished by the traverse of the wheel slide along a stationary machine table.

In plunge grinding, the machine table, after having been set, is locked and, while the part is rotating, the wheel slide continually advances at a preset rate, until the finish size of the part is reached. The width of the grinding wheel is a limiting factor of the section length that can be ground in this process. Plunge grinding is required for profiled surfaces and for the simultaneous grinding of multiple surfaces of different diameters or located in different planes.

When the configuration of the part does not make use of either method mandatory, the choice may be made on the basis of the following general considerations: traverse grinding usually produces a better finish, and the productivity of plunge grinding is generally higher.

Work Holding on Cylindrical Grinding Machines.—The manner in which the work is located and held in the machine during the grinding process determines the configuration of the part that can be adapted for cylindrical grinding and affects the resulting accuracy of the ground surface. The method of work holding also affects the attainable production rate, because the mounting and dismounting of the part can represent a substantial portion of the total operating time.

Whatever method is used for holding the part on cylindrical types of grinding machines, two basic conditions must be satisfied: 1) the part should be located with respect to its correct axis of rotation; and 2) the work drive must cause the part to rotate, at a specific speed, around the established axis.

The lengthwise location of the part, although controlled, is not too critical in traverse grinding; however, in plunge grinding, particularly when shoulder sections are also involved, it must be assured with great accuracy.

Table 1 presents a listing, with brief discussions, of work-holding methods and devices that are most frequently used in cylindrical grinding.

Table 1. Work-Holding Methods and Devices for Cylindrical Grinding

Designation	Description	Discussion
Centers, nonrotating (“dead”), with drive plate	Headstock with nonrotating spindle holds the center. Around the spindle, an independently supported sleeve carries the drive plate for rotating the work. Tailstock for opposite center.	The simplest method of holding the work between two opposite centers is also the potentially most accurate, as long as correctly prepared and located center holes are used in the work.
Centers, driving type	Word held between two centers obtains its rotation from the concurrently applied drive by the live headstock spindle and live tailstock spindle.	Eliminates the drawback of the common center-type grinding with driver plate, which requires a dog attached to the workpiece. Driven spindles permit the grinding of the work up to both ends.
Chuck, geared, or cam-actuated	Two, three, or four jaws moved radially through mechanical elements, hand-, or power-operated, exert concentrically acting clamping force on the workpiece.	Adaptable to workpieces of different configurations and within a generally wide capacity of the chuck. Flexible in uses that, however, do not include high-precision work.
Chuck, diaphragm	Force applied by hand or power of a flexible diaphragm causes the attached jaws to deflect temporarily for accepting the work, which is held when force is released.	Rapid action and flexible adaptation to different work configurations by means of special jaws offer varied uses for the grinding of disk-shaped and similar parts.
Collets	Holding devices with externally or internally acting clamping force, easily adaptable to power actuation, assuring high centering accuracy.	Limited to parts with previously machined or ground holding surfaces, because of the small range of clamping movement of the collet jaws.
Face plate	Has four independently actuated jaws, any or several of which may be used, or entirely removed, using the base plate for supporting special clamps.	Used for holding bulky parts, or those of awkward shape, which are ground in small quantities not warranting special fixtures.
Magnetic plate	Flat plates, with pole distribution adapted to the work, are mounted on the spindle like chucks and may be used for work with the locating face normal to the axis.	Applicable for light cuts such as are frequent in tool making, where the rapid clamping action and easy access to both the O.D. and the exposed face are sometimes of advantage.
Steady rests	Two basic types are used: (a) the two-jaw type supporting the work from the back (back rest), leaving access by the wheel; (b) the three-jaw type (center rest).	A complementary work-holding device, used in conjunction with primary work holders, to provide additional support, particularly to long and/or slender parts.
Special fixtures	Single-purpose devices, designed for a particular workpiece, primarily for providing special locating elements.	Typical workpieces requiring special fixturing are, as examples, crankshafts where the holding is combined with balancing functions; or internal gears located on the pitch circle of the teeth for O.D. grinding.

Selection of Grinding Wheels for Cylindrical Grinding.—For cylindrical grinding, as for grinding in general, the primary factor to be considered in wheel selection is the work material. Other factors are the amount of excess stock and its rate of removal (speeds and feeds), the desired accuracy and surface finish, the ratio of wheel and work diameter, wet or dry grinding, etc. In view of these many variables, it is not practical to set up a complete list of grinding wheel recommendations with general validity. Instead, examples of recommendations embracing a wide range of typical applications and assuming common practices are presented in Table 2. This is intended as a guide for the starting selection of grinding-wheel specifications which, in case of a not entirely satisfactory performance, can be refined subsequently. The content of the table is a version of the grinding-wheel recommendations for cylindrical grinding by the Norton Company using, however, non-proprietary designations for the abrasive types and bonds.

Table 2. Wheel Recommendations for Cylindrical Grinding

Material	Wheel Marking	Material	Wheel Marking
Aluminum	SFA46-18V	Forgings	A46-M5V
Armatures (laminated)	SFA100-18V	Gages (plug)	SFA80-K8V
Axles (auto & railway)	A54-M5V	General-purpose grinding	SFA54-L5V
Brass	C36-KV	Glass	BFA220-011V
Bronze		Gun barrels	
Soft	C36-KV	Spotting and O.D.	BFA60-M5V
Hard	A46-M5V	Nitralloy	
Bushings (hardened steel)	BFA60-L5V	Before nitriding	A60-K5V
Bushings (cast iron)	C36-JV	After nitriding	
Cam lobes (cast alloy)		Commercial finish	SFA60-18V
Roughing	BFA54-N5V	High finish	C100-1V
Finishing	A70-P6B	Reflective finish	C500-19E
Cam lobes (hardened steel)		Pistons (aluminum)	SFA46-18V
Roughing	BFA54-L5V	(cast iron)	C36-KV
Finishing	BFA80-T8B	Plastics	C46-JV
Cast iron	C36-JV	Rubber	
Chromium plating		Soft	SFA20-K5B
Commercial finish	SFA60-J8V	Hard	C36-KB
High finish	A150-K5E	Spline shafts	SFA60-N5V
Reflective finish	C500-I9E	Sprayed metal	C60-JV
Commutators (copper)	C60-M4E	Steel	
Crankshafts (airplane)		Soft	
Pins	BFA46-K5V	1 in. dia. and smaller	SFA60-M5V
Bearings	A46-L5V	over 1 in dia.	SFA46-L5V
Crankshafts (automotive		Hardened	
pins and bearings)		1 in. dia. and smaller	SFA80-L8V
Finishing	A54-N5V	over 1 in. dia.	SFA60-K5V
Roughing & finishing	A54-O5V	300 series stainless	SFA46-K8V
Regrinding	A54-M5V	Stellite	BFA46-M5V
Regrinding, sprayed		Titanium	C60-JV
metal	C60-JV	Valve stems (automative)	BFA54-N5V
Drills	BFA54-N5V	Valve tappets	BFA54-M5V

Note: Prefixes to the standard designation "A" of aluminum oxide indicate modified abrasives as follows:

BFA = Blended friable (a blend of regular and friable).

SFA = Semifriable.

Operational Data for Cylindrical Grinding.—In cylindrical grinding, similarly to other metalcutting processes, the applied speed and feed rates must be adjusted to the operational conditions as well as to the objectives of the process. Grinding differs, however, from other types of metalcutting methods in regard to the cutting speed of the tool which, in grinding, is generally not a variable; it should be maintained at, or close to the optimum rate, commonly 6500 feet per minute peripheral speed.

In establishing the proper process values for grinding, of prime consideration are the work material, its condition (hardened or soft), and the type of operation (roughing or finishing). Other influencing factors are the characteristics of the grinding machine (stability, power), the specifications of the grinding wheel, the material allowance, the rigidity and balance of the workpiece, as well as several grinding process conditions, such as wet or dry grinding, the manner of wheel truing, etc.

Variables of the cylindrical grinding process, often referred to as *grinding data,* comprise the speed of work rotation (measured as the surface speed of the work); the infeed (in inches per pass for traverse grinding, or in inches per minute for plunge grinding); and, in the case of traverse grinding, the speed of the reciprocating table movement (expressed either in feet per minute, or as a fraction of the wheel width for each revolution of the work).

For the purpose of starting values in setting up a cylindrical grinding process, a brief listing of basic data for common cylindrical grinding conditions and involving frequently used materials, is presented in Table 3.

Table 3. Basic Process Data for Cylindrical Grinding

Traverse Grinding						
Work Material	Material Condition	Work Surface Speed, fpm	Infeed, Inch/Pass		Traverse for Each Work Revolution, In Fractions of the Wheel Width	
			Roughing	Finishing	Roughing	Finishing
Plain Carbon Steel	Annealed	100	0.002	0.0005	½	⅙
	Hardened	70	0.002	0.0003 to 0.0005	¼	⅛
Alloy Steel	Annealed	100	0.002	0.0005	½	⅙
	Hardened	70	0.002	0.0002 to 0.0005	¼	⅛
Tool Steel	Annealed	60	0.002	0.0005 max.	½	⅙
	Hardened	0.002	0.002	0.0001 to 0.0005	¼	⅛
Copper Alloys	Annealed or Cold Drawn	100	0.002	0.0005 max.	⅓	⅙
Aluminum Alloys	Cold Drawn or Solution Treated	150	0.002	0.0005 max.	⅓	⅙

Plunge Grinding		
Work Material	Infeed per Revolution of the Work, Inch	
	Roughing	Finishing
Steel, soft	0.0005	0.0002
Plain carbon steel, hardened	0.0002	0.000050
Alloy and tool steel, hardened	0.0001	0.000025

These data, which are, in general, considered conservative, are based on average operating conditions and may be modified subsequently,

reducing the values in case of unsatisfactory quality of the grinding or the occurrence of failures;

increasing the rates for raising the productivity of the process, particularly for rigid workpieces, substantial stock allowance, etc.

High-Speed Cylindrical Grinding.—The maximum peripheral speed of the wheels in regular cylindrical grinding is generally 6500 feet per minute; the commonly used grinding wheels and machines are designed to operate efficiently at this speed. Recently, efforts were made to raise the productivity of different grinding methods, including cylindrical grinding, by increasing the peripheral speed of the grinding wheel to a substantially higher than traditional level, such as 12,000 feet per minute or more. Such methods are designated by the distinguishing term of high-speed grinding.

For high-speed grinding, special grinding machines have been built with high dynamic stiffness and static rigidity, equipped with powerful drive motors, extra-strong spindles and bearings, reinforced wheel guards, etc., and using grinding wheels expressly made and tested for operating at high peripheral speeds. The higher stock-removal rate accomplished by high-speed grinding represents an advantage when the work configuration and material permit, and the removable stock allowance warrants its application.

CAUTION: High-speed grinding must *not* be applied on standard types of equipment, such as general types of grinding machines and regular grinding wheels. Operating grinding wheels, even temporarily, at higher than approved speed constitutes a grave safety hazard.

Areas and Degrees of Automation in Cylindrical Grinding.—Power drive for the work rotation and for the reciprocating table traverse are fundamental machine movements that, once set for a certain rate, will function without requiring additional attention. Loading and removing the work, starting and stopping the main movements, and applying infeed by hand wheel are carried out by the operator on cylindrical grinding machines in their basic degree of mechanization. Such equipment is still frequently used in tool room and jobbing-type work.

More advanced levels of automation have been developed for cylindrical grinders and are being applied in different degrees, particularly in the following principal respects:

A) *Infeed,* in which different rates are provided for rapid approach, roughing and finishing, followed by a spark-out period, with presetting of the advance rates, the cutoff points, and the duration of time-related functions.

B) *Automatic cycling* actuated by a single lever to start work rotation, table reciprocation, grinding-fluid supply, and infeed, followed at the end of the operation by wheel slide retraction, the successive stopping of the table movement, the work rotation, and the fluid supply.

C) *Table traverse dwells* (tarry) in the extreme positions of the travel, over preset periods, to assure uniform exposure to the wheel contact of the entire work section.

D) *Mechanized work loading,* clamping, and, after termination of the operation, unloading, combined with appropriate work-feeding devices such as indexing-type drums.

E) *Size control* by in-process or post-process measurements. Signals originated by the gage will control the advance movement or cause automatic compensation of size variations by adjusting the cutoff points of the infeed.

F) *Automatic wheel dressing* at preset frequency, combined with appropriate compensation in the infeed movement.

G) *Numerical control* obviates the time-consuming setups for repetitive work performed on small- or medium-size lots. As an application example: shafts with several sections of different lengths and diameters can be ground automatically in a single operation, grinding the sections in consecutive order to close dimensional limits, controlled by an in-process gage, which is also automatically set by means of the program.

The choice of the grinding machine functions to be automated and the extent of automation will generally be guided by economic considerations, after a thorough review of the available standard and optional equipment. Numerical control of partial or complete cycles is being applied to modern cylindrical and other grinding machines.

Cylindrical Grinding Troubles and Their Correction.—Troubles that may be encountered in cylindrical grinding may be classified as work defects (chatter, checking, burning, scratching, and inaccuracies), improperly operating machines (jumpy infeed or traverse), and wheel defects (too hard or soft action, loading, glazing, and breakage). The Landis Tool Company has listed some of these troubles, their causes, and corrections as follows:

Chatter.—Sources of chatter include: 1) faulty coolant; 2) wheel out of balance; 3) wheel out of round; 4) wheel too hard; 5) improper dressing; 6) faulty work support or rotation; 7) improper operation; 8) faulty traverse; 9) work vibration; 10) outside vibration transmitted to machine; 11) interference; 12) wheel base; and 13) headstock.

Suggested procedures for correction of these troubles are:

1) *Faulty coolant:* Clean tanks and lines. Replace dirty or heavy coolant with correct mixture.

2) *Wheel out of balance:* Rebalance on mounting before and after dressing. Run wheel without coolant to remove excess water. Store a removed wheel on its side to keep retained water from causing a false heavy side. Tighten wheel mounting flanges. Make sure wheel center fits spindle.

3) *Wheel out of round:* True before and after balancing. True sides to face.

4) *Wheel too hard:* Use coarser grit, softer grade, more open bond. See *Wheel Defects* on page 1183.

5) *Improper dressing:* Use sharp diamond and hold rigidly close to wheel. It must not overhang excessively. Check diamond in mounting.

6) *Faulty work support or rotation:* Use sufficient number of work rests and adjust them more carefully. Use proper angles in centers of work. Clean dirt from footstock spindle and be sure spindle is tight. Make certain that work centers fit properly in spindles.

7) *Improper operation:* Reduce rate of wheel feed.

8) *Faulty traverse:* See *Uneven Traverse or Infeed of Wheel Head* on page 1182.

9) *Work vibration:* Reduce work speed. Check workpiece for balance.

10) *Outside vibration transmitted to machine:* Check and make sure that machine is level and sitting solidly on foundation. Isolate machine or foundation.

11) *Interference:* Check all guards for clearance.

12) *Wheel base:* Check spindle bearing clearance. Use belts of equal lengths or uniform cross-section on motor drive. Check drive motor for unbalance. Check balance and fit of pulleys. Check wheel feed mechanism to see that all parts are tight.

13) *Headstock:* Put belts of same length and cross-section on motor drive; check for correct work speeds. Check drive motor for unbalance. Make certain that headstock spindle is not loose. Check work center fit in spindle. Check wear of face plate and jackshaft bearings.

Spirals on Work (traverse lines with same lead on work as rate of traverse).— Sources of spirals include: 1) machine parts out of line; and 2) truing.

Suggested procedures for correction of these troubles are:

1) *Machine parts out of line:* Check wheel base, headstock, and footstock for proper alignment.

2) *Truing:* Point truing tool down 3 degrees at the workwheel contact line. Round off wheel edges.

Check Marks on Work.—Sources of check marks include: 1) improper operation; 2) improper heat treatment; 3) improper size control; 4) improper wheel; and 5) improper dressing.

Suggested procedures for correction of these troubles are:

1) *Improper operation:* Make wheel act softer. See *Wheel Defects*. Do not force wheel into work. Use greater volume of coolant and a more even flow. Check the correct positioning of coolant nozzles to direct a copious flow of clean coolant at the proper location.

2) *Improper heat treatment:* Take corrective measures in heat-treating operations.

3) *Improper size control:* Make sure that engineering establishes reasonable size limits. See that they are maintained.

4) *Improper wheel:* Make wheel act softer. Use softer-grade wheel. Review the grain size and type of abrasive. A finer grit or more friable abrasive or both may be called for.

5) *Improper dressing:* Check that the diamond is sharp, of good quality, and well set. Increase speed of the dressing cycle. Make sure diamond is not cracked.

Burning and Discoloration of Work.—Sources of burning and discoloration are: 1) improper operation; and 2) improper wheel.

Suggested procedures for correction of these troubles are:

1) *Improper operation:* Decrease rate of infeed. Don't stop work while in contact with wheel.

2) *Improper wheel:* Use softer wheel or obtain softer effect. See *Wheel Defects*. Use greater volume of coolant.

Isolated Deep Marks on Work.—Source of trouble is an unsuitable wheel. Use a finer wheel and consider a change in abrasive type.

Fine Spiral or Thread on Work.—Sources of this trouble are: 1) improper operation; and 2) faulty wheel dressing.

Suggested procedures for corrections of these troubles are:

1) *Improper operation:* Reduce wheel pressure. Use more work rests. Reduce traverse with respect to work rotation. Use different traverse rates to break up pattern when making numerous passes. Prevent edge of wheel from penetrating by dressing wheel face parallel to work.

2) *Faulty wheel dressing:* Use slower or more even dressing traverse. Set dressing tool at least 3 degrees down and 30 degrees to the side from time to time. Tighten holder. Don't take too deep a cut. Round off wheel edges. Start dressing cut from wheel edge.

Narrow and Deep Regular Marks on Work.—Source of trouble is that the wheel is too coarse. Use finer grain size.

Wide, Irregular Marks of Varying Depth on Work.—Source of trouble is too soft a wheel. Use a harder grade wheel. See *Wheel Defects*.

Widely Spaced Spots on Work.—Sources of trouble are oil spots or glazed areas on wheel face. Balance and true wheel. Keep oil from wheel face.

Irregular "Fish-tail" Marks of Various Lengths and Widths on Work.—Source of trouble is dirty coolant. Clean tank frequently. Use filter for fine finish grinding. Flush wheel guards after dressing or when changing to finer wheel.

Wavy Traverse Lines on Work.—Source of trouble is wheel edges. Round off. Check for loose thrust on spindle and correct if necessary.

Irregular Marks on Work.—Cause is loose dirt. Keep machine clean.

Deep, Irregular Marks on Work.—Source of trouble is loose wheel flanges. Tighten and make sure blotters are used.

Isolated Deep Marks on Work.—Sources of trouble are: 1) grains pull out; coolant too strong; 2) coarse grains or foreign matter in wheel face; and 3) improper dressing.

Respective suggested procedures for corrections of these troubles are: 1) decrease soda content in coolant mixture; 2) dress wheel; and 3) use sharper dressing tool.

Brush wheel after dressing with stiff bristle brush.

Grain Marks on Work.—Sources of trouble are: 1) improper finishing cut; 2) grain sizes of roughing and finishing wheels differ too much; 3) dressing too coarse; and 4) wheel too coarse or too soft.

Respective suggested procedures for corrections of these troubles are: start with high work and traverse speeds; finish with high work speed and slow traverse, letting wheel "spark-out" completely; finish out better with roughing wheel or use finer roughing wheel; use shallower and slower cut; and use finer grain size or harder-grade wheel.

Inaccuracies in Work.—Work out-of-round, out-of-parallel, or tapered.

Sources of trouble are: 1) misalignment of machine parts; 2) work centers; 3) improper operation; 4) coolant; 5) wheel; 6) improper dressing; 7) spindle bearings; and 8) work.

Suggested procedures for corrections of these troubles are:

1) *Misalignment of machine parts:* Check headstock and tailstock for alignment and proper clamping.

2) *Work centers:* Centers in work must be deep enough to clear center point. Keep work centers clean and lubricated. Check play of footstock spindle and see that footstock spindle is clean and tightly seated. Regrind work centers if worn. Work centers must fit taper of work-center holes. Footstock must be checked for proper tension.

3) *Improper operation:* Don't let wheel traverse beyond end of work. Decrease wheel pressure so work won't spring. Use harder wheel or change feeds and speeds to make wheel act harder. Allow work to "spark-out." Decrease feed rate. Use proper number of work rests. Allow proper amount of tarry. Workpiece must be balanced if it is an odd shape.

4) *Coolant:* Use greater volume of coolant.

5) *Wheel:* Rebalance wheel on mounting before and after truing.

6) *Improper dressing:* Use same positions and machine conditions for dressing as in grinding.

7) *Spindle bearings:* Check clearance.

8) *Work:* Work must come to machine in reasonably accurate form.

Inaccurate Work Sizing (when wheel is fed to same position, it grinds one piece to correct size, another oversize, and still another undersize).—Sources of trouble are:

1) improper work support or rotation; 2) wheel out of balance; 3) loaded wheel;

4) improper infeed; 5) improper traverse; 6) coolant; 7) misalignment; and 8) work.

Suggested procedures for corrections of these troubles are:

1) *Improper work support or rotation:* Keep work centers clean and lubricated. Regrind work-center tips to proper angle. Be sure footstock spindle is tight. Use sufficient work rests, properly spaced.

2) *Wheel out of balance:* Balance wheel on mounting before and after truing.

3) *Loaded wheel:* See *Wheel Defects*.

4) *Improper infeed:* Check forward stops of rapid feed and slow feed. When readjusting position of wheel base by means of the fine feed, move the wheel base back after making the adjustment and then bring it forward again to take up backlash and relieve strain in feed-up parts. Check wheel spindle bearings. Don't let excessive lubrication of wheel base slide cause "floating." Check and tighten wheel feed mechanism. Check parts for wear. Check pressure in hydraulic system. Set infeed cushion properly. Check to see that pistons are not sticking.

5) *Improper traverse:* Check traverse hydraulic system and the operating pressure. Prevent excessive lubrication of carriage ways with resultant "floating" condition. Check to see if carriage traverse piston rods are binding. Carriage rack and driving gear must not bind. Change length of tarry period.

6) *Coolant:* Use greater volume of clean coolant.

7) *Misalignment:* Check level and alignment of machine.

8) *Work:* Workpieces may vary too much in length, permitting uneven center pressure.

Uneven Traverse or Infeed of Wheel Head.—Sources of uneven traverse or infeed of wheel head are: carriage and wheel head, hydraulic system, interference, unbalanced conditions, and wheel out of balance. Suggested procedures for correction of these troubles are:

1) *Carriage and wheel head:* Ways may be scored. Be sure to use recommended oil for both lubrication and hydraulic system. Make sure ways are not so smooth that they press out oil film. Check lubrication of ways. Check wheel feed mechanism, traverse gear, and carriage rack clearance. Prevent binding of carriage traverse cylinder rods.

2) *Hydraulic systems:* Remove air and check pressure of hydraulic oil. Check pistons and valves for oil leakage and for gumminess caused by incorrect oil. Check worn valves or pistons that permit leakage.

3) *Interference:* Make sure guard strips do not interfere.

4) *Unbalanced conditions:* Eliminate loose pulleys, unbalanced wheel drive motor, uneven belts, or high spindle keys.

5) *Wheel out of balance:* Balance wheel on mounting before and after truing.

Wheel Defects.—When *wheel is acting too hard*, such defects as glazing, some loading, lack of cut, chatter, and burning of work result.

Suggested procedures for correction of these faults are: 1) Increase work and traverse speeds as well as rate of in-feed; 2) decrease wheel speed, diameter, or width; 3) dress more sharply; 4) use thinner coolant; 5) don't tarry at end of traverse; 6) select softer wheel grade and coarser grain size; 7) avoid gummy coolant; and 8) on hardened work select finer grit, more fragile abrasive or both to get penetration. Use softer grade.

When *wheel is acting too soft,* such defects as wheel marks, tapered work, short wheel life, and not-holding-cut result.

Suggested procedures for correction of these faults are: 1) Decrease work and traverse speeds as well as rate of in-feed; 2) increase wheel speed, diameter, or width; 3) dress with little in-feed and slow traverse; 4) use heavier coolants; 5) don't let wheel run off work at end of traverse; and 6) select harder wheel or less fragile grain or both.

Wheel Loading and Glazing.—Sources of the trouble of wheel loading or glazing are:

1) Incorrect wheel; 2) improper dress; 3) faulty operation; 4) faulty coolant; and 5) gummy coolant.

Suggested procedures for correction of these faults are:

1) *Incorrect wheel:* Use coarser grain size, more open bond, or softer grade.

2) *Improper dressing:* Keep wheel sharp with sharp dresser, clean wheel after dressing, use faster dressing traverse, and deeper dressing cut.

3) *Faulty operation:* Control speeds and feeds to soften action of wheel. Use less in-feed to prevent loading; more in-feed to stop glazing.

4) *Faulty coolant:* Use more, cleaner and thinner coolant, and less oily coolant.

5) *Gummy coolant:* To stop wheel glazing, increase soda content and avoid the use of soluble oils if water is hard. In using soluble oil coolant with hard water a suitable conditioner or "softener" should be added.

Wheel Breakage.—Suggested procedures for the correction of a radial break with three or more pieces are: 1) Reduce wheel speed to or below rated speed; 2) mount wheel properly, use blotters, tight arbors, even flange pressure and be sure to keep out dirt between flange and wheel; 3) use plenty of coolant to prevent over-heating; 4) use less in-feed; and 5) don't allow wheel to become jammed on work.

A radial break with two pieces may be caused by excessive side strain. To prevent an irregular wheel break, don't let wheel become jammed on work; don't allow striking of wheel; and never use wheels that have been damaged in handling. In general, do not use a wheel that is too tight on the arbor since the wheel is apt to break when started. Prevent excessive hammering action of wheel. Follow rules of the American National Standard Safety Requirements for the Use, Care, and Protection of Abrasive Wheels (ANSI B7.1-1978).

Centerless Grinding

In centerless grinding the work is supported on a work rest blade and is between the grinding wheel and a regulating wheel. The regulating wheel generally is a rubber bonded abrasive wheel. In the normal grinding position the grinding wheel forces the work downward against the work rest blade and also against the regulating wheel. The latter imparts a uniform rotation to the work giving it its same peripheral speed which is adjustable.

The higher the work center is placed above the line joining the centers of the grinding and regulating wheels the quicker the rounding action. Rounding action is also increased by a high work speed and a slow rate of traverse (if a through-feed operation). It is possible to have a higher work center when using softer wheels, as their use gives decreased contact pressures and the tendency of the workpiece to lift off the work rest blade is lessened.

Long rods or bars are sometimes ground with their centers below the line-of-centers of the wheels to eliminate the whipping and chattering due to slight bends or kinks in the rods or bars, as they are held more firmly down on the blade by the wheels.

There are three general methods of centerless grinding which may be described as through-feed, in-feed, and end-feed methods.

Through-feed Method of Grinding.—The through-feed method is applied to straight cylindrical parts. The work is given an axial movement by the regulating wheel and passes between the grinding and regulating wheels from one side to the other. The rate of feed depends upon the diameter and speed of the regulating wheel and its inclination which is adjustable. It may be necessary to pass the work between the wheels more than once, the number of passes depending upon such factors as the amount of stock to be removed, the roundness and straightness of the unground work, and the limits of accuracy required.

The work rest fixture also contains adjustable guides on either side of the wheels that directs the work to and from the wheels in a straight line.

In-feed Method of Centerless Grinding.—When parts have shoulders, heads or some part larger than the ground diameter, the in-feed method usually is employed. This method is similar to “plungecut” form grinding on a center type of grinder. The length or sections to be ground in any one operation are limited by the width of the wheel. As there is no axial feeding movement, the regulating wheel is set with its axis approximately parallel to that of the grinding wheel, there being a slight inclination to keep the work tight against the end stop.

End-feed Method of Grinding.—The end-feed method is applied only to taper work. The grinding wheel, regulating wheel, and the work rest blade are set in a fixed relation to each other and the work is fed in from the front mechanically or manually to a fixed end stop. Either the grinding or regulating wheel, or both, are dressed to the proper taper.

Automatic Centerless Grinding.—The grinding of relatively small parts may be done automatically by equipping the machine with a magazine, gravity chute, or hopper feed, provided the shape of the part will permit using these feed mechanisms.

Internal Centerless Grinding.—Internal grinding machines based upon the centerless principle utilize the outside diameter of the work as a guide for grinding the bore which is concentric with the outer surface. In addition to straight and tapered bores, interrupted and “blind” holes can be ground by the centerless method. When two or more grinding operations such as roughing and finishing must be performed on the same part, the work can be rechucked in the same location as often as required.

Centerless Grinding Troubles.—A number of troubles and some corrective measures compiled by a manufacturer are listed here for the through-feed and in-feed methods of centerless grinding.

Chattermarks: are caused by having the work center too high above the line joining the centers of the grinding and regulating wheels; using too hard or too fine a grinding wheel; using too steep an angle on the work support blade; using too thin a work support blade; “play” in the set-up due to loosely clamped members; having the grinding wheel fit loosely on the spindle; having vibration either transmitted to the machine or caused by a defective drive in the machine; having the grinding wheel out-of-balance; using too heavy a stock removal; and having the grinding wheel or the regulating wheel spindles not properly adjusted.

Feed lines or spiral marks: in through-feed grinding are caused by too sharp a corner on the exit side of the grinding wheel which may be alleviated by dressing the grinding wheel to a slight taper about $\frac{1}{2}$ inch from the edge, dressing the edge to a slight radius, or swiveling the regulating wheel a bit.

Scored work: is caused by burrs, abrasive grains, or removed material being imbedded in or fused to the work support blade. This condition may be alleviated by using a coolant with increased lubricating properties and if this does not help a softer grade wheel should be used.

Work not ground round: may be due to the work center not being high enough above the line joining the centers of the grinding and regulating wheels. Placing the work center higher and using a softer grade wheel should help to alleviate this condition.

Work not ground straight: in through-feed grinding may be due to an incorrect setting of the guides used in introducing and removing the work from the wheels, and the existence of convex or concave faces on the regulating wheel. For example, if the work is tapered on the front end, the work guide on the entering side is deflected toward the regulating wheel. If tapered on the back end, then the work guide on the exit side is deflected toward the regulating wheel. If both ends are tapered, then both work guides are deflected toward the regulating wheel. The same barrel-shaped pieces are also obtained if the face of the regulating wheel is convex at the line of contact with the work. Conversely, the work would be ground with hollow shapes if the work guides were deflected toward the grinding wheel or if the face of the regulating wheel were concave at the line of contact with the work. The use of a warped work rest blade may also result in the work not being ground straight and the blade should be removed and checked with a straight edge.

In in-feed grinding, in order to keep the wheel faces straight which will insure straightness of the cylindrical pieces being ground, the first item to be checked is the straightness and the angle of inclination of the work rest blade. If this is satisfactory then one of three corrective measures may be taken: the first might be to swivel the regulating wheel to compensate for the taper, the second might be to true the grinding wheel to that angle that will give a perfectly straight workpiece, and the third might be to change the inclination of the regulating wheel (this is true only for correcting very slight tapers up to 0.0005 inch).

Difficulties in sizing: the work in in-feed grinding are generally due to a worn in-feed mechanism and may be overcome by adjusting the in-feed nut.

Flat spots: on the workpiece in in-feed grinding usually occur when grinding heavy work and generally when the stock removal is light. This condition is due to insufficient driving power between the work and the regulating wheel which may be alleviated by equipping the work rest with a roller that exerts a force against the workpiece; and by feeding the workpiece to the end stop using the upper slide.

Surface Grinding

The term surface grinding implies, in current technical usage, the grinding of surfaces which are essentially flat. Several methods of surface grinding, however, are adapted and used to produce surfaces characterized by parallel straight line elements in one direction, while normal to that direction the contour of the surface may consist of several straight line sections at different angles to each other (e.g., the guideways of a lathe bed); in other cases the contour may be curved or profiled (e.g., a thread cutting chaser).

Advantages of Surface Grinding.—Alternate methods for machining work surfaces similar to those produced by surface grinding are milling and, to a much more limited degree, planing. Surface grinding, however, has several advantages over alternate methods that are carried out with metal-cutting tools. Examples of such potential advantages are as follows:

1) Grinding is applicable to very hard and/or abrasive work materials, without significant effect on the efficiency of the stock removal.

2) The desired form and dimensional accuracy of the work surface can be obtained to a much higher degree and in a more consistent manner.

3) Surface textures of very high finish and—when the appropriate system is utilized—with the required lay, are generally produced.

4) Tooling for surface grinding as a rule is substantially less expensive, particularly for producing profiled surfaces, the shapes of which may be dressed into the wheel, often with simple devices, in processes that are much more economical than the making and the maintenance of form cutters.

5) Fixturing for work holding is generally very simple in surface grinding, particularly when magnetic chucks are applicable, although the mechanical holding fixture can also be simpler, because of the smaller clamping force required than in milling or planing.

6) Parallel surfaces on opposite sides of the work are produced accurately, either in consecutive operations using the first ground surface as a dependable reference plane or, simultaneously, in double face grinding, which usually operates without the need for holding the parts by clamping.

7) Surface grinding is well adapted to process automation, particularly for size control, but also for mechanized work handling in the large volume production of a wide range of component parts.

Principal Systems of Surface Grinding.—Flat surfaces can be ground with different surface portions of the wheel, by different arrangements of the work and wheel, as well as by different interrelated movements. The various systems of surface grinding, with their respective capabilities, can best be reviewed by considering two major distinguishing characteristics:

1) *The operating surface of the grinding wheel,* which may be the periphery or the face (the side);

2) *The movement of the work during the process,* which may be traverse (generally reciprocating) or rotary (continuous), depending on the design of a particular category of surface grinders.

The accompanying table provides a concise review of the principal surface grinding systems, defined by the preceding characteristics. It should be noted that many surface grinders are built for specific applications, and do not fit exactly into any one of these major categories.

Selection of Grinding Wheels for Surface Grinding.—The most practical way to select a grinding wheel for surface grinding is to base the selection on the work material. Table gives the grinding wheel recommendations for Types 1, 5, and 7 straight wheels used on reciprocating and rotary table surface grinders with horizontal spindles. Table 1b gives the grinding wheel recommendations for Type 2 cylinder wheels, Type 6 cup wheels, and wheel segments used on vertical spindle surface grinders.

The last letters (two or three) that may follow the bond designation V (vitrified) or B (resinoid) refer to: 1) bond modification, "BE" being especially suitable for surface grinding; 2) special structure, "P" type being distinctively porous; and 3) for segments made of 23A type abrasives, the term 12VSM implies porous structure, and the letter "P" is not needed.

Table 1a. Grinding Wheel Recommendations for Surface Grinding—Using Straight Wheel Types 1, 5, and 7

Horizontal-spindle, reciprocating-table surface grinders		
Material	Wheels less than 16 inches in diameter	Wheels 16 inches in diameter and over
Cast iron	37C36-K8V or 23A46-I8VBE	23A36-I8VBE
Nonferrous metal	37C36-K8V	37C36-K8V
Soft steel	23A46-J8VBE	23A36-J8VBE

Table 1a. *(Continued)* Grinding Wheel Recommendations for Surface Grinding—Using Straight Wheel Types 1, 5, and 7

Horizontal-spindle, reciprocating-table surface grinders		
Material	Wheels less than 16 inches in diameter	Wheels 16 inches in diameter and over
Hardened steel—broad contact	32A46-H8VBE or 32A60-F12VBEP	32A36-H8VBE or 32A36-F12VBEP
Hardened steel—narrow contact or interrupted cut	32A46-I8VBE	32A36-J8VBE
General-purpose wheel	23A46-H8VBE	23A36-I8VBE
Cemented carbides	Diamond wheels[a]	Diamond wheels[a]

[a] General diamond wheel recommendations are listed in Table 5 on page 1168.

Horizontal-spindle, rotary-table surface grinders	
Material	Wheels of any diameter
Cast iron	37C36-K8V or 23A46-I8VBE
Nonferrous metals	37C36-K8V
Soft steel	23A46-J8VBE
Hardened steel—broad contact	32A46-I8VBE
Hardened steel—narrow contact or interrupted cut	32A46-J8VBE
General-purpose wheel	23A46-I8VBE
Cemented carbides—roughing	Diamond wheels[a]

Courtesy of Norton Company

Table 1b. Grinding Wheel Recommendations for Surface Grinding—Using Type 2 Cylinder Wheels, Type 6 Cup Wheels, and Wheel Segments

Material	Type 2 Cylinder Wheels	Type 6 Cup Wheels	Wheel Segments
High tensile cast iron and nonferrous metals	37C24-HKV	37C24-HVK	37C24-HVK
Soft steel, malleable cast iron, steel castings, boiler plate	23A24-I8VBE or 23A30-G12VBEP	23A24-I8VBE	23A24-I8VSM or 23A30-H12VSM
Hardened steel—broad contact	32A46-G8VBE or 32A36-E12VBEP	32A46-G8VBE or 32A60-E12VBEP	32A36-G8VBE or 32A46-E12VBEP
Hardened steel—narrow contact or interrupt cut	32A46-H8VBE	32A60-H8VBE	32A46-G8VBE or 32A60-G12VBEP
General-purpose use	23A30-H8VBE or 23A30-E12VBEP	…	23A30-H8VSM or 23A30-G12VSM

The wheel markings in the tables are those used by the Norton Co., complementing the basic standard markings with Norton symbols. The complementary symbols used in these tables, that is, those preceding the letter designating A (aluminum oxide) or C (silicon carbide), indicate the special type of basic abrasive that has the friability best suited for particular work materials. Those preceding A (aluminum oxide) are

57—a versatile abrasive suitable for grinding steel in either a hard or soft state.

38—the most friable abrasive.

32—the abrasive suited for tool steel grinding.

23—an abrasive with intermediate grinding action, and

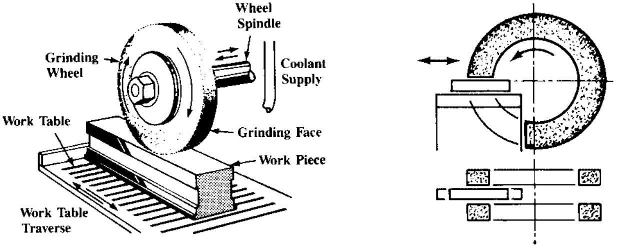
19—the abrasive produced for less heat-sensitive steels.

Those preceding C (silicon carbide) are

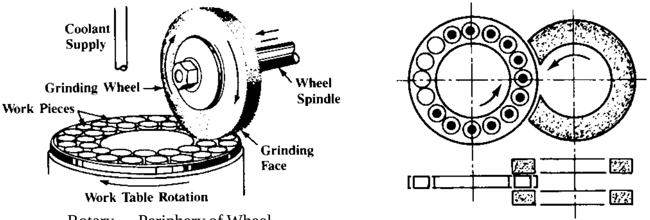
37—a general application abrasive, and

39—an abrasive for grinding hard cemented carbide.

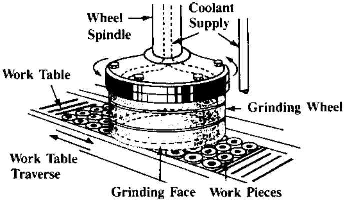
Principal Systems of Surface Grinding — Diagrams



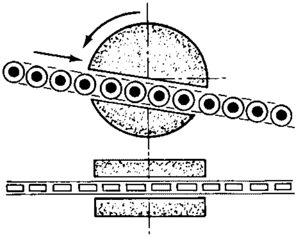
Reciprocating — Periphery of Wheel



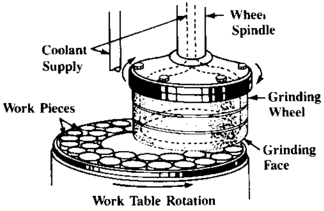
Rotary — Periphery of Wheel



Reciprocating — Face (Side) of Wheel



Traverse Along Straight Line or Arcuate Path — Face (Side) of Wheel



Rotary — Face (Side) of Wheel

Principal Systems of Surface Grinding—Principles of Operation

Effective Grinding Surface—Periphery of Wheel Movement of Work—Reciprocating
Work is mounted on the horizontal machine table that is traversed in a reciprocating movement at a speed generally selected from a steplessly variable range. The transverse movement, called cross feed of the table or of the wheel slide, operates at the end of the reciprocating stroke and assures the gradual exposure of the entire work surface, which commonly exceeds the width of the wheel. The depth of the cut is controlled by the downfeed of the wheel, applied in increments at the reversal of the transverse movement.
Effective Grinding Surface—Periphery of Wheel Movement of Work—Rotary
Work is mounted, usually on the full-diameter magnetic chuck of the circular machine table that rotates at a preset constant or automatically varying speed, the latter maintaining an approximately equal peripheral speed of the work surface area being ground. The wheelhead, installed on a cross slide, traverses over the table along a radial path, moving in alternating directions, toward and away from the center of the table. Infeed is by vertical movement of the saddle along the guideways of the vertical column, at the end of the radial wheelhead stroke. The saddle contains the guideways along which the wheelhead slide reciprocates.
Effective Grinding Surface—Face (Side) of Wheel Movement of Work—Reciprocating
Operation is similar to the reciprocating table-type peripheral surface grinder, but grinding is with the face, usually with the rim of a cup-shaped wheel, or a segmental wheel for large machines. Capable of covering a much wider area of the work surface than the peripheral grinder, thus frequently no need for cross feed. Provides efficient stock removal, but is less adaptable than the reciprocating table-type peripheral grinder.
Effective Grinding Surface—Face (Side) of Wheel Movement of Work—Rotary
The grinding wheel, usually of segmental type, is set in a position to cover either an annular area near the periphery of the table or, more commonly, to reach beyond the table center. A large circular magnetic chuck generally covers the entire table surface and facilitates the mounting of workpieces, even of fixtures, when needed. The uninterrupted passage of the work in contact with the large wheel face permits a very high rate of stock removal and the machine, with single or double wheelhead, can be adapted also to automatic operation with continuous part feed by mechanized work handling.
Effective Grinding Surface—Face (Side) of Wheel Movement of Work—Traverse Along Straight or Arcuate Path
Operates with practically the entire face of the wheel, which is designated as an abrasive disc (hence "disc grinding") because of its narrow width in relation to the large diameter. Built either for one or, more frequently, for two discs operating with opposed faces for the simultaneous grinding of both sides of the workpiece. The parts pass between the operating faces of the wheel (a) pushed-in and retracted by the drawerlike movement of a feed slide; (b) in an arcuate movement carried in the nests of a rotating feed wheel; (c) nearly diagonally advancing along a rail. Very well adapted to fully mechanized work handling.

Process Data for Surface Grinding.—In surface grinding, similarly to other metal-cutting processes, the speed and feed rates that are applied must be adjusted to the operational conditions as well as to the objectives of the process. Grinding differs, however, from other types of metal cutting methods in regard to the cutting speed of the tool; the peripheral speed of the grinding wheel is maintained within a narrow range, generally 5500 to 6500 surface feet per minute. Speed ranges different from the common one are used in particular processes which require special wheels and equipment.

Table 2. Basic Process Data for Peripheral Surface Grinding on Reciprocating Table Surface Grinders

Work Material	Hardness	Material Condition	Wheel Speed, fpm	Table Speed, fpm	Downfeed, in. per pass		Crossfeed per pass, fraction of wheel width
					Rough	Finish	
Plain carbon steel	52 Rc max.	Annealed, Cold drawn	5500 to 6500	50 to 100	0.003	0.0005 max.	¼
	52 to 65 Rc	Carburized and/or quenched and tempered	5500 to 6500	50 to 100	0.003	0.0005 max.	⅒
Alloy steels	52 Rc max.	Annealed or quenched and tempered	5500 to 6500	50 to 100	0.003	0.001 max.	¼
	52 to 65 Rc	Carburized and/or quenched and tempered	5500 to 6500	50 to 100	0.003	0.0005 max.	⅒
Tool steels	150 to 275 Bhn	Annealed	5500 to 6500	50 to 100	0.002	0.0005 max.	⅕
	56 to 65 Rc	Quenched and tempered	5500 to 6500	50 to 100	0.002	0.0005 max.	⅒
Nitriding steels	200 to 350 Bhn	Normalized, annealed	5500 to 6500	50 to 100	0.003	0.001 max.	¼
	60 to 65 Rc	Nitrided	5500 to 6500	50 to 100	0.003	0.0005 max.	⅒
Cast steels	52 Rc max.	Normalized, annealed	5500 to 6500	50 to 100	0.003	0.001 max.	¼
	Over 52 Rc	Carburized and/or quenched and tempered	5500 to 6500	50 to 100	0.003	0.0005 max.	⅒
Gray irons	52 Rc max.	As cast, annealed, and/or quenched and tempered	5000 to 6500	50 to 100	0.003	0.001 max.	⅓
Ductile irons	52 Rc max.	As cast, annealed or quenched and tempered	5500 to 6500	50 to 100	0.003	0.001 max.	⅕
Stainless steels, martensitic	135 to 235 Bhn	Annealed or cold drawn	5500 to 6500	50 to 100	0.002	0.0005 max.	¼
	Over 275 Bhn	Quenched and tempered	5500 to 6500	50 to 100	0.001	0.0005 max.	⅛
Aluminum alloys	30 to 150 Bhn	As cast, cold drawn or treated	5500 to 6500	50 to 100	0.003	0.001 max.	⅓

In establishing the proper process values for grinding, of prime consideration are the work material, its condition, and the type of operation (roughing or finishing). Table 2 gives basic process data for peripheral surface grinding on reciprocating table surface grinders. For different work materials and hardness ranges data are given regarding table speeds, downfeed (infeed) rates and cross feed, the latter as a function of the wheel width.

Common Faults and Possible Causes in Surface Grinding.—Approaching the ideal performance with regard to both the quality of the ground surface and the efficiency of surface grinding, requires the monitoring of the process and the correction of conditions adverse to the attainment of that goal.

Table 3. Common Faults and Possible Causes in Surface Grinding

CAUSES	FAULTS	WORK DIMENSION			METALLURGICAL DEFECTS		SURFACE QUALITY				WHEEL CONDITION			WORK RETAINMENT	
		Work not flat	Work not parallel	Poor size holding	Burnishing of work	Burning or checking	Feed lines	Chatter marks	Scratches on surface	Poor finish	Wheel loading	Wheel glazing	Rapid wheel wear	Not firmly seated	Work sliding on chuck
WORK CONDITION	Heat treat stresses	X	..	..	..	..	..	..	..	..	..	..	..	..	..
	Work too thin	X	X	..	..	..	..	..	..	..	..	..	..	..	..
	Work warped	X	..	..	..	..	..	..	..	..	..	..	..	X	..
	Abrupt section changes	X	X	..	..	..	..	..	..	..	..	..	..	..	..
GRINDING WHEEL	Grit too fine	..	..	..	X	X	..	..	..	..	X	X	..	..	..
	Grit too coarse	..	..	..	..	..	..	..	..	X	..	..	..	..	..
	Grade too hard	X	..	..	X	X	..	X	..	..	X	X	..	..	..
	Grade too soft	..	..	X	..	..	..	X	X	..	..	..	X	..	..
	Wheel not balanced	..	..	..	..	..	..	X	..	..	..	..	..	..	..
	Dense structure	..	..	..	..	..	..	..	..	..	X	X	..	..	..
TOOLING AND COOLANT	Improper coolant	..	..	..	..	..	..	..	..	..	X	..	..	..	..
	Insufficient coolant	X	X	..	X	X	..	..	..	..	..	X	..	..	..
	Dirty coolant	..	..	..	..	..	..	..	X	..	X	..	..	..	..
	Diamond loose or chipped	X	X	..	..	..	..	..	X	..	..	..	..	..	..
	Diamond dull	..	..	X	..	..	..	..	..	X	X	X	..	..	..
	No or poor magnetic force	..	..	X	..	..	..	..	X	..	..	..	..	X	X
	Chuck surface worn or burred	X	X	..	..	..	..	..	X	..	..	..	..	X	..
MACHINE AND SETUP	Chuck not aligned	X	X	..	..	..	..	..	..	..	..	..	..	..	..
	Vibrations in machine	..	..	..	..	..	..	X	..	..	..	..	..	..	..
	Plane of movement out of parallel	X	X	..	..	..	..	..	..	..	..	..	..	..	..
OPERATIONAL CONDITIONS	Too low work speed	..	..	..	..	..	..	..	..	..	X	..	..	..	..
	Too light feed	..	..	..	..	..	..	..	..	..	..	X	..	..	..
	Too heavy cut	X	..	..	X	..	..	..	..	X	..	..	..	..	..
	Chuck retained swarf	X	X	..	..	..	..	..	..	..	..	..	..	..	X
	Chuck loading improper	X	X	..	..	..	..	..	..	..	..	..	..	..	X
	Insufficient blocking of parts	..	..	..	..	..	..	..	X	..	..	..	..	..	..
	Wheel runs off the work	..	X	X	..	X	..	..	..	..	..	..	X	..	..
	Wheel dressing too fine	X	..	..	..	..	..	..	..	..	..	..	..	..	..
	Wheel edge not chamfered	..	..	..	..	..	X	..	..	..	..	..	..	..	..
	Loose dirt under guard	..	..	..	..	..	..	..	X	..	..	..	..	..	..

Defective, or just not entirely satisfactory surface grinding may have any one or more of several causes. Exploring and determining the cause for eliminating its harmful effects is facilitated by knowing the possible sources of the experienced undesirable performance. Table 3, associating the common faults with their possible causes, is intended to aid in determining the actual cause, the correction of which should restore the desired performance level.

While the table lists the more common faults in surface grinding, and points out their frequent causes, other types of improper performance and/or other causes, in addition to those indicated, are not excluded.

Offhand Grinding

Offhand grinding consists of holding the wheel to the work or the work to the wheel and grinding to broad tolerances and includes such operations as certain types of tool sharpening, weld grinding, snagging castings and other rough grinding. Types of machines that are used for rough grinding in foundries are floor- and bench-stand machines. Wheels for these machines vary from 6 to 30 inches in diameter. Portable grinding machines (electric, flexible shaft, or air-driven) are used for cleaning and smoothing castings.

Many rough grinding operations on castings can be best done with shaped wheels, such as cup wheels (including plate mounted) or cone wheels, and it is advisable to have a good assortment of such wheels on hand to do the odd jobs the best way.

Floor- and Bench-Stand Grinding.—The most common method of rough grinding is on double-end floor and bench stands. In machine shops, welding shops, and automotive repair shops, these grinders are usually provided with a fairly coarse grit wheel on one end for miscellaneous rough grinding and a finer grit wheel on the other end for sharpening tools. The pressure exerted is a very important factor in selecting the proper grinding wheel. If grinding is to be done mostly on hard sharp fins, then durable, coarse and hard wheels are required, but if grinding is mostly on large gate and riser pads, then finer and softer wheels should be used for best cutting action.

Portable Grinding.—Portable grinding machines are usually classified as air grinders, flexible shaft grinders, and electric grinders. The electric grinders are of two types; namely, those driven by standard 60 cycle current and so-called high-cycle grinders. Portable grinders are used for grinding down and smoothing weld seams; cleaning metal before welding; grinding out imperfections, fins and parting lines in castings and smoothing castings; grinding punch press dies and patterns to proper size and shape; and grinding manganese steel castings.

Wheels used on portable grinders are of three bond types; namely, resinoid, rubber, and vitrified. By far the largest percentage is resinoid. Rubber bond is used for relatively thin wheels and where a good finish is required. Some of the smaller wheels such as cone and plug wheels are vitrified bonded.

Grit sizes most generally used in wheels from 4 to 8 inches in diameter are 16, 20, and 24. In the still smaller diameters, finer sizes are used, such as 30, 36, and 46.

The particular grit size to use depends chiefly on the kind of grinding to be done. If the work consists of sharp fins and the machine has ample power, a coarse grain size combined with a fairly hard grade should be used. If the job is more in the nature of smoothing or surfacing and a fairly good finish is required, then finer and softer wheels are called for.

Swing-Frame Grinding.—This type of grinding is employed where a considerable amount of material is to be removed as on snagging large castings. It may be possible to remove 10 times as much material from steel castings using swing-frame grinders as with portable grinders; and 3 times as much material as with high-speed floor-stand grinders.

The largest field of application for swing-frame machines is on castings which are too heavy to handle on a floor stand; but often it is found that comparatively large gates and risers on smaller castings can be ground more quickly with swing-frame grinders, even if fins and parting lines have to be ground on floor stands as a second operation.

In foundries, the swing-frame machines are usually suspended from a trolley on a jib that can be swung out of the way when placing the work on the floor with the help of an overhead crane. In steel mills when grinding billets, a number of swing-frame machines are usually suspended from trolleys on a line of beams which facilitate their use as required.

The grinding wheels used on swing-frame machines are made with coarser grit sizes and harder grades than wheels used on floor stands for the same work. The reason is that greater grinding pressures can be obtained on the swing-frame machines.

Mounted Wheels and Mounted Points.—These wheels and points are used in hard-to-get-at places and are available with a vitrified bond. The wheels are available with aluminum oxide or silicon carbide abrasive grains. The aluminum oxide wheels are used to grind tough and tempered die steels and the silicon carbide wheels, cast iron, chilled iron, bronze, and other non-ferrous metals.

The illustrations on pages 1205 and 1206 give the standard shapes of mounted wheels and points as published by the Grinding Wheel Institute. A note about the maximum operating speed for these wheels is given at the bottom of the first page of illustrations. Metric sizes are given on page 1204.

Abrasive Belt Grinding

Abrasive belts are used in the metalworking industry for removing stock, light cleaning up of metal surfaces, grinding welds, deburring, breaking and polishing hole edges, and finish grinding of sheet steel. The types of belts that are used may be coated with aluminum oxide (the most common coating) for stock removal and finishing of all alloy steels, high-carbon steel, and tough bronzes; and silicon carbide for use on hard, brittle, and low-tensile strength metals which would include aluminum and cast irons.

Table 1 is a guide to the selection of the proper abrasive belt, lubricant, and contact wheel. This table is entered on the basis of the material used and type of operation to be done and gives the abrasive belt specifications (type of bonding andabrasive grain size and material), the range of speeds at which the belt may best be operated, the type of lubricant to use, and the type and hardness of the contact wheel to use. Table 2 serves as a guide in the selection of contact wheels. This table is entered on the basis of the type of contact wheel surface and the contact wheel material. The table gives the hardness and/or density, the type of abrasive belt grinding for which the contact wheel is intended, the character of the wheel action and such comments as the uses, and hints for best use. Both tables are intended only as guides for general shop practice; selections may be altered to suit individual requirements.

There are three types of abrasive belt grinding machines. One type employs a contact wheel behind the belt at the point of contact of the workpiece to the belt and facilitates a high rate of stock removal. Another type uses an accurate parallel ground platen over which the abrasive belt passes and facilitates the finishing of precision parts. A third type which has no platens or contact wheel is used for finishing parts having uneven surfaces or contours. In this type there is no support behind the belt at the point of contact of the belt with the workpiece. Some machines are so constructed that besides grinding against a platen or a contact wheel the workpiece may be moved and ground against an unsupported portion of the belt, thereby in effect making it a dual machine.

Although abrasive belts at the time of their introduction were used dry, since the advent of the improved waterproof abrasive belts, they have been used with coolants, oil-mists, and greases to aid the cutting action. The application of a coolant to the area of contact retards loading, resulting in a cool, free cutting action, a good finish and a long belt life.

Table 1. Guide to the Selection and Application of Abrasive Belts

						Contact Wheel	
Material	Type of Operation	Abrasive Belt[a]	Grit	Belt Speed, fpm	Type of Grease Lubricant	Type	Durometer Hardness
Hot-and Cold-Rolled Steel	Roughing	R/R Al_2O_3	24–60	4000–65000	Light-body or none	Cog-tooth, serrated rubber	70–90
	Polishing	R/G or R/R Al_2O_3	80–150	4500–7000	Light-body or none	Plain or serrated rubber, sectional or finger-type cloth wheel, free belt	20–60
	Fine Polishing	R/G or electro-coated Al_2O_3 cloth	180–500	4500–7000	Heavy or with abrasive compound	Smooth-faced rubber or cloth	20–40
Stainless Steel	Roughing	R/R Al_2O_3	50–80	3500–5000	Light-body or none	Cog-tooth, serrated rubber	70–90
	Polishing	R/G or R/R Al_2O_3	80–120	4000–5500	Light-body or none	Plain or serrated rubber, sectional or finger-type cloth wheel, free belt	30–60
	Fine Pol.	Closed-coat SiC	150–280	4500–5500	Heavy or oil mist	Smooth-faced rubber or cloth	20–40
Aluminum, Cast or Fabricated	Roughing	R/R SiC or Al_2O_3	24–80	5000–6500	Light	Cog-tooth, serrated rubber	70–90
	Polishing	R/G SiC or Al_2O_3	100–180	4500–6500	Light	Plain or serrated rubber, sectional or finger-type cloth wheel, free belt	30–50
	Fine Polishing	Closed-coat SiC or electro-coated Al_2O_3	220–320	4500–6500	Heavy or with abrasive compound	Plain faced rubber, finger-type cloth or free belt	20–50
Copper Alloys or Brass	Roughing	R/R SiC or Al_2O_3	36–80	2200–4500	Light-body	Cog-tooth, serrated rubber	70–90
	Polishing	Closed-coat SiC or electro-coated Al_2O_3 or R/G SiC or Al_2O_3	100–150	4000–6500	Light-body	Plain or serrated rubber, sectional or finger-type cloth wheel, free belt	30–50
	Fine Polishing	Closed-coat SiC or electro-coated Al_2O_3	180–320	4000–6500	Light or with abrasive compound	Same as for polishing	20–30
Non-ferrous Die-castings	Roughing	R/R SiC or Al_2O_3	24–80	4500–6500	Light-body	Hard wheel depending on application	50–70
	Polishing	R/G SiC or Al_2O_3	100–180	4500–6500	Light-body	Plain rubber, cloth or free belt	30–50
	Fine Polishing	Electro-coated Al_2O_3 or closed-coat SiC	220–320	4500–6500	Heavy or with abrasive compound	Plain or finger-type cloth wheel, or free belt	20–30
Cast Iron	Roughing	R/R Al_2O_3	24–60	2000–4000	None	Cog-tooth, serrated rubber	70–90
	Polishing	R/R Al_2O_3	80–150	4000–5500	None	Serrated rubber	30–70
	Fine Polishing	R/R Al_2O_3	120–240	4000–5500	Light-body	Smooth-faced rubber	30–40
Titanium	Roughing	R/R SiC or Al_2O_3	36–50	700–1500	Sulfur-chlorinated	Small-diameter, cog-tooth serrated rubber	70–80
	Polishing	R/R SiC	60–120	1200–2000	Light-body	Standard serrated rubber	50
	Fine Pol.	R/R SiC	120–240	1200–2000	Light-body	Smooth-faced rubber or cloth	20–40

[a] R/R indicates that both the making and sizing bond coats are resin. R/G indicates that the making coat is glue and the sizing coat is resin. The abbreviations Al_2O_3 for aluminum oxide and SiC for silicon carbide are used. Almost all R/R and R/G Al_2O_3 and SiC belts have a heavy-drill weight cloth backing. Most electro-coated Al_2O_3 and closed-coat SiC belts have a jeans weight cloth backing.

Table 2. Guide to the Selection and Application of Contact Wheels

Surface	Material	Hardness and Density	Purposes	Wheel Action	Comments
Cog-tooth	Rubber	70 to 90 durometer	Roughing	Fast cutting, allows long belt life.	For cutting down projections on castings and weld beads.
Standard serrated	Rubber	40 to 50 durometer, medium density	Roughing	Leaves rough- to medium-ground surface.	For smoothing projections and face defects.
X-shaped serrations	Rubber	20 to 50 durometer	Roughing and polishing	Flexibility of rubber allows entry into contours. Medium polishing, light removal.	Same as for standard serrated wheels but preferred for soft non-ferrous metals.
Plain face	Rubber	20 to 70 durometer	Roughing and polishing	Plain wheel face allows controlled penetration of abrasive grain. Softer wheels give better finishes.	For large or small flat faces.
Flat flexible	Compressed canvas	About nine densities from very hard to very soft	Roughing and polishing	Hard wheels can remove metal, but not as quickly as cog-tooth rubber wheels. Softer wheels polish well.	Good for medium-range grinding and polishing.
Flat flexible	Solid sectional canvas	Soft, medium, and hard	Polishing	Uniform polishing. Avoids abrasive pattern on work. Adjusts to contours. Can be performed for contours.	A low-cost wheel with uniform density at the face. Handles all types of polishing.
Flat flexible	Buff section canvas	Soft	Contour polishing	For fine polishing and finishing.	Can be widened or narrowed by adding or removing sections. Low cost.
Flat flexible	Sponge rubber inserts	5 to 10 durometer, soft	Polishing	Uniform polishing and finishing. Polishes and blends contours.	Has replaceable segments. Polishes and blends contours. Segments allow density changes.
Flexible	Fingers of canvas attached to hub	Soft	Polishing	Uniform polishing and finishing.	For polishing and finishing.
Flat flexible	Rubber segments	Varies in hardness	Roughing and polishing	Grinds or polishes depending on density and hardness of inserts.	For portable machines. Uses replaceable segments that save on wheel costs and allow density changes.
Flat flexible	Inflated rubber	Air pressure controls hardness	Roughing and polishing	Uniform finishing.	Adjusts to contours.

Abrasive Cutting

Abrasive cut-off wheels are used for cutting steel, brass and aluminum bars and tubes of all shapes and hardnesses, ceramics, plastics, insulating materials, glass and cemented carbides. Originally a tool or stock room procedure, this method has developed into a high-speed production operation. While the abrasive cut-off machine and cut-off wheel can be said to have revolutionized the practice of cutting-off materials, the metal saw continues to be the more economical method for cutting-off large cross-sections of certain materials. However, there are innumerable materials and shapes that can be cut with much greater speed and economy by the abrasive wheel method. On conventional chop-stroke abrasive cutting machines using 16-inch diameter wheels, 2-inch diameter bar stock is the maximum size that can be cut with satisfactory wheel efficiency, but bar stock up to 6 inches in diameter can be cut efficiently on oscillating-stroke machines. Tubing up to 3½ inches in diameter can also be cut efficiently.

Abrasive wheels are commonly available in four types of bonds: Resinoid, rubber, shellac and fiber or fabric reinforced. In general, resinoid bonded cut-off wheels are used for dry cutting where burrs and some burn are not objectionable and rubber bonded wheels are used for wet cutting where cuts are to be smooth, clean and free from burrs. Shellac bonded wheels have a soft, free cutting quality which makes them particularly useful in the tool room where tool steels are to be cut without discoloration. Fiber reinforced bonded wheels are able to withstand severe flexing and side pressures and fabric reinforced bonded wheels which are highly resistant to breakage caused by extreme side pressures, are fast cutting and have a low rate of wear.

The types of abrasives available in cut-off wheels are: Aluminum oxide, for cutting steel and most other metals; silicon carbide, for cutting non-metallic materials such as carbon, tile, slate, ceramics, etc.; and diamond, for cutting cemented carbides. The method of denoting abrasive type, grain size, grade, structure and bond type by using a system of markings is the same as for grinding wheels (see page 1141). Maximum wheel speeds given in the American National Standard Safety Requirements for The Use, Care, and Protection of Abrasive Wheels (ANSI B7.1-1988) range from 9500 to 14,200 surface feet per minute for organic bonded cut-off wheels larger than 16 inches in diameter and from 9500 to 16,000 surface feet per minute for organic bonded cut-off wheels 16 inches in diameter and smaller. Maximum wheel speeds specified by the manufacturer should never be exceeded even though they may be lower than those given in the B7.1.

There are four basic types of abrasive cutting machines: Chop-stroke, oscillating stroke, horizontal stroke and work rotating. Each of these four types may be designed for dry cutting or for wet cutting (includes submerged cutting).

The accompanying table based upon information made available by The Carborundum Co. gives some of the probable causes of cutting off difficulties that might be experienced when using abrasive cut-off wheels.

Probable Causes of Cutting-Off Difficulties

Difficulty	Probable Cause
Angular Cuts and Wheel Breakage	(1) Inadequate clamping which allows movement of work while the wheel is in the cut. The work should be clamped on both sides of the cut. (2) Work vise higher on one side than the other causing wheel to be pinched. (3) Wheel vibration resulting from worn spindle bearings. (4) Too fast feeding into the cut when cutting wet.
Burning of Stock	(1) Insufficient power or drive allowing wheel to stall. (2) Cuts too heavy for grade of wheel being used. (3) Wheel fed through the work too slowly. This causes a heating up of the material being cut. This difficulty encountered chiefly in dry cutting.
Excessive Wheel Wear	(1) Too rapid cutting when cutting wet. (2) Grade of wheel too hard for work, resulting in excessive heating and burning out of bond. (3) Inadequate coolant supply in wet cutting. (4) Grade of wheel too soft for work. (5) Worn spindle bearings allowing wheel vibration.
Excessive Burring	(1) Feeding too slowly when cutting dry. (2) Grit size in wheel too coarse. (3) Grade of wheel too hard. (4) Wheel too thick for job.

Honing Process

The hone-abrading process for obtaining cylindrical forms with precise dimensions and surfaces can be applied to internal cylindrical surfaces with a wide range of diameters such as engine cylinders, bearing bores, pin holes, etc. and also to some external cylindrical surfaces.

The process is used to: 1) eliminate inaccuracies resulting from previous operations by generating a true cylindrical form with respect to roundness and straightness within minimum dimensional limits; 2) generate final dimensional size accuracy within low tolerances, as may be required for interchangeability of parts; 3) provide rapid and economical stock removal consistent with accomplishment of the other results; and 4) generate surface finishes of a specified degree of surface smoothness with high surface quality.

Amount and Rate of Stock Removal.—Honing may be employed to increase bore diameters by as much as 0.100 inch or as little as 0.001 inch. The amount of stock removed by the honing process is entirely a question of processing economy. If other operations are performed before honing then the bulk of the stock should be taken off by the operation that can do it most economically. In large diameter bores that have been distorted in heat treating, it may be necessary to remove as much as 0.030 to 0.040 inch from the diameter to make the bore round and straight. For out-of-round or tapered bores, a good "rule of thumb" is to leave twice as much stock (on the diameter) for honing as there is error in the bore. Another general rule is: For bores over one inch in diameter, leave 0.001 to 0.0015 inch stock per inch of diameter. For example, 0.002 to 0.003 inch of stock is left in two-inch bores and 0.010 to 0.015 inch in ten-inch bores. Where parts are to be honed for finish only, the amount of metal to be left for removing tool marks may be as little as 0.0002 to 0.015 inch on the diameter.

In general, the honing process can be employed to remove stock from bore diameters at the rate of 0.009 to 0.012 inch per minute on cast-iron parts and from 0.005 to 0.008 inch per minute on steel parts having a hardness of 60 to 65 Rockwell C. These rates are based on parts having a length equal to three or four times the diameter. Stock has been removed from long parts such as gun barrels, at the rate of 65 cubic inches per hour. Recommended honing speeds for cast iron range from 110 to 200 surface feet per minute of rotation and from 50 to 110 lineal feet per minute of reciprocation. For steel, rotating surface speeds range from 50 to 110 feet per minute and reciprocation speeds from 20 to 90 lineal feet per minute. The exact rotation and reciprocation speeds to be used depend upon the size of the work, the amount and characteristics of the material to be removed and the quality of the finish desired. In general, the harder the material to be honed, the lower the speed. Interrupted bores are usually honed at faster speeds than plain bores.

Formula for Rotative Speeds.—Empirical formulas for determining rotative speeds for honing have been developed by the Micromatic Hone Corp. These formulas take into consideration the type of material being honed, its hardness and its surface characteristics; the abrasive area; and the type of surface pattern and degree of surface roughness desired. Because of the wide variations in material characteristics, abrasives available, and types of finishes specified, these formulas should be considered as a guide only in determining which of the available speeds (pulley or gear combinations) should be used for any particular application.

The formula for rotative speed, S, in surface feet per minute is:

$$S = \frac{K \times D}{W \times N}$$

The formula for rotative speed in revolutions per minute is:

$$\text{R.P.M} = \frac{R}{W \times N}$$

where, K and R are factors taken from the table on the following page, D is the diameter of the bore in inches, W is the width of the abrasive stone or stock in inches, and N is the number of stones.

Although the actual speed of the abrasive is the resultant of both the rotative speed and the reciprocation speed, this latter quantity is seldom solved for or used. The reciprocation speed is not determined empirically but by testing under operating conditions. Changing the reciprocation speed affects the dressing action of the abrasive stones, therefore, the reciprocation speed is adjusted to provide for a desired surface finish which is usually a well lubricated bearing surface that will not scuff.

Table of Factors for Use in Rotative Speed Formulas

Character of Surface[a]	Material	Hardness[b]					
		Soft		Medium		Hard	
		Factors					
		K	*R*	*K*	*R*	*K*	*R*
Base Metal	Cast Iron	110	420	80	300	60	230
	Steel	80	300	60	230	50	190
Dressing Surface	Cast Iron	150	570	110	420	80	300
	Steel	110	420	80	300	60	230
Severe Dressing	Cast Iron	200	760	150	570	110	420
	Steel	150	570	110	420	80	300

[a] The character of the surface is classified according to its effect on the abrasive; *Base Metal* being a honed, ground or fine bored section that has little dressing action on the grit; *Dressing Surface* being a rough bored, reamed or broached surface or any surface broken by cross holes or ports; *Severe Dressing* being a surface interrupted by keyways, undercuts or burrs that dress the stones severely. If over half of the stock is to be removed after the surface is cleaned up, the speed should be computed using the *Base Metal* factors for *K* and *R*.

[b] Hardness designations of soft, medium and hard cover the following ranges on the Rockwell " C" hardness scale, respectively: 15 to 45, 45 to 60 and 60 to 70.

Possible Adjustments for Eliminating Undesirable Honing Conditions

Undesirable Condition	Adjustment Required to Correct Condition[a]								
	Abrasive[b]				Other				
	Friability	Grain Size	Hardness	Structure	Feed Pressure	Reciprocation	R.P.M.	Runout Time	Stroke Length
Abrasive Glazing	+	– –	– –	+	+ +	+ +	– –	–	0
Abrasive Loading	0	– –	–	–	+ +	+	– –	0	0
Too Rough Surface Finish	0	+ +	+ +	–	–	–	+ +	+	0
Too Smooth Surface Finish	0	– –	– –	+	+	+	– –	–	0
Poor Stone Life	–	+	+ +	–	–	–	+	0	0
Slow Stock Removal	+	– –	–	+	+ +	+ +	– –	0	0
Taper — Large at Ends	0	0	0	0	0	0	0	0	–
Taper — Small at Ends	0	0	0	0	0	0	0	0	+

[a] The + and + + symbols generally indicate that there should be an increase or addition while the – and – – symbols indicate that there should be a reduction or elimination. In each case, the double symbol indicates that the contemplated change would have the greatest effect. The 0 symbol means that a change would have no effect.

[b] For the abrasive adjustments the + and + + symbols indicate a more friable grain, a finer grain, a harder grade or a more open structure and the – and – – symbols just the reverse.

Compiled by Micromatic Hone Corp.

Abrasive Stones for Honing.—Honing stones consist of aluminum oxide, silicon carbide, CBN or diamond abrasive grits, held together in stick form by a vitrified clay, resinoid or metal bond. CBN metal-bond stones are particularly suitable and widely used for honing. The grain and grade of abrasive to be used in any particular honing operation depend upon the quality of finish desired, the amount of stock to be removed, the material being honed and other factors.

The following general rules may be followed in the application of abrasive for honing:

1) Silicon-carbide abrasive is commonly used for honing cast iron, while aluminum-oxide abrasive is generally used on steel; 2) The harder the material being honed, the softer the abrasive stick used; 3) A rapid reciprocating speed will tend to make the abrasive cut fast because the dressing action on the grits will be severe; and 4) To improve the finish, use a finer abrasive grit, incorporate more multi-direction action, allow more "run-out" time after honing to size, or increase the speed of rotation.

Surface roughnesses ranging from less than 1 micro-inch r.m.s. to a relatively coarse roughness can be obtained by judicious choice of abrasive and honing time but the most common range is from 3 to 50 micro-inches r.m.s.

Adjustments for Eliminating Undesirable Honing Conditions.—The accompanying table indicates adjustments that may be made to correct certain undesirable conditions encountered in honing. Only one change should be made at a time and its effect noted before making other adjustments.

Tolerances.—For bore diameters above 4 inches the tolerance of honed surfaces with respect to roundness and straightness ranges from 0.0005 to 0.001 inch; for bore diameters from 1 to 4 inches, 0.0003 to 0.0005 inch; and for bore diameters below 1 inch, 0.00005 to 0.0003 inch.

Laps and Lapping

Material for Laps.—Laps are usually made of soft cast iron, copper, brass or lead. In general, the best material for laps to be used on very accurate work is soft, close-grained cast iron. If the grinding, prior to lapping, is of inferior quality, or an excessive allowance has been left for lapping, copper laps may be preferable. They can be charged more easily and cut more rapidly than cast iron, but do not produce as good a finish. Whatever material is used, the lap should be softer than the work, as, otherwise, the latter will become charged with the abrasive and cut the lap, the order of the operation being reversed. A common and inexpensive form of lap for holes is made of lead which is cast around a tapering steel arbor. The arbor usually has a groove or keyway extending lengthwise, into which the lead flows, thus forming a key that prevents the lap from turning. When the lap has worn slightly smaller than the hole and ceases to cut, the lead is expanded or stretched a little by the driving in of the arbor. When this expanding operation has been repeated two or three times, the lap usually must be trued or replaced with a new one, owing to distortion.

The tendency of lead laps to lose their form is an objectionable feature. They are, however, easily molded, inexpensive, and quickly charged with the cutting abrasive. A more elaborate form for holes is composed of a steel arbor and a split cast-iron or copper shell which is sometimes prevented from turning by a small dowel pin. The lap is split so that it can be expanded to accurately fit the hole being operated upon. For hardened work, some toolmakers prefer copper to either cast iron or lead. For holes varying from $\frac{1}{4}$ to $\frac{1}{2}$ inch in diameter, copper or brass is sometimes used; cast iron is used for holes larger than $\frac{1}{2}$ inch in diameter. The arbors for these laps should have a taper of about $\frac{1}{4}$ or $\frac{3}{8}$ inch per foot. The length of the lap should be somewhat greater than the length of the hole, and the thickness of the shell or lap proper should be from $\frac{1}{8}$ to $\frac{1}{6}$ its diameter.

External laps are commonly made in the form of a ring, there being an outer ring or holder and an inner shell which forms the lap proper. This inner shell is made of cast iron, copper, brass or lead. Ordinarily the lap is split and screws are provided in the holder for adjustment. The length of an external lap should at least equal the diameter of the work, and might well be longer. Large ring laps usually have a handle for moving them across the work.

Laps for Flat Surfaces.—Laps for producing plane surfaces are made of cast iron. In order to secure accurate results, the lapping surface must be a true plane. A flat lap that is used for roughing or “blocking down” will cut better if the surface is scored by narrow grooves. These are usually located about $\frac{1}{2}$ inch apart and extend both lengthwise and crosswise, thus forming a series of squares similar to those on a checker-board. An abrasive of No. 100 or 120 emery and lard oil can be used for charging the roughing lap. For finer work, a lap having an unscored surface is used, and the lap is charged with a finer abrasive. After a lap is charged, all loose abrasive should be washed off with gasoline, for fine work, and when lapping, the surface should be kept moist, preferably with kerosene. Gasoline will cause the lap to cut a little faster, but it evaporates so rapidly that the lap soon becomes dry and the surface caked and glossy in spots. Loose emery should not be applied while lapping, for if the lap is well charged with abrasive in the beginning, is kept well moistened and not crowded too hard, it will cut for a considerable time. The pressure upon the work should be just enough to insure constant contact. The lap can be made to cut only so fast, and if excessive pressure is applied it will become "stripped" in places. The causes of scratches are: Loose abrasive on the lap; too much pressure on the work, and poorly graded abrasive. To produce a perfectly smooth surface free from scratches, the lap should be charged with a very fine abrasive.

Grading Abrasives for Lapping.—For high-grade lapping, abrasives can be evenly graded as follows: A quantity of flour-emery or other abrasive is placed in a heavy cloth bag, which is gently tapped, causing very fine particles to be sifted through. When a sufficient quantity has been obtained in this way, it is placed in a dish of lard or sperm oil. The largest particles will then sink to the bottom and in about one hour the oil should be poured into another dish, care being taken not to disturb the sediment at the bottom. The oil is then allowed to stand for several hours, after which it is poured again, and so on, until the desired grade is obtained.

Charging Laps.—To charge a flat cast-iron lap, spread a very thin coating of the prepared abrasive over the surface and press the small cutting particles into the lap with a hard steel block. There should be as little rubbing as possible. When the entire surface is apparently charged, clean and examine for bright spots; if any are visible, continue charging until the entire surface has a uniform gray appearance. When the lap is once charged, it should be used without applying more abrasive until it ceases to cut. If a lap is over-charged and an excessive amount of abrasive is used, there is a rolling action between the work and lap which results in inaccuracy. The surface of a flat lap is usually finished true, prior to charging, by scraping and testing with a standard surface-plate, or by the well-known method of scraping-in three plates together, in order to secure a plane surface. In any case, the bearing marks or spots should be uniform and close together. These spots can be blended by covering the plates evenly with a fine abrasive and rubbing them together. While the plates are being ground in, they should be carefully tested and any high spots which may form should be reduced by rubbing them down with a smaller block.

To charge cylindrical laps for internal work, spread a thin coating of prepared abrasive over the surface of a hard steel block, preferably by rubbing lightly with a cast-iron or copper block; then insert an arbor through the lap and roll the latter over the steel block, pressing it down firmly to embed the abrasive into the surface of the lap. For external cylindrical laps, the inner surface can be charged by rolling-in the abrasive with a hard steel roller that is somewhat smaller in diameter than the lap. The taper cast-iron blocks which are sometimes used for lapping taper holes can also be charged by rolling-in the abrasive, as previously described; there is usually one roughing and one finishing lap, and when charging the former, it may be necessary to vary the charge in accordance with any error which might exist in the taper.

Rotary Diamond Lap.—This style of lap is used for accurately finishing very small holes, which, because of their size, cannot be ground. While the operation is referred to as lapping, it is, in reality, a grinding process, the lap being used the same as a grinding wheel. Laps employed for this work are made of mild steel, soft material being desirable because it can be charged readily. Charging is usually done by rolling the lap between two hardened steel plates. The diamond dust and a little oil is placed on the lower plate, and as the lap revolves, the diamond is forced into its surface. After charging, the lap should be washed in benzine. The rolling plates should also be cleaned before charging with dust of a finer grade. It is very important not to force the lap when in use, especially if it is a small size. The lap should just make contact with the high spots and gradually grind them off. If a diamond lap is lubricated with kerosene, it will cut freer and faster. These small laps are run at very high speeds, the rate depending upon the lap diameter. Soft work should never be ground with diamond dust because the dust will leave the lap and charge the work.

When using a diamond lap, it should be remembered that such a lap will not produce sparks like a regular grinding wheel; hence, it is easy to crowd the lap and "strip" some of the diamond dust. To prevent this, a sound intensifier or "harker" should be used. This is placed against some stationary part of the grinder spindle, and indicates when the lap touches the work, the sound produced by the slightest contact being intensified.

Grading Diamond Dust.—The grades of diamond dust used for charging laps are designated by numbers, the fineness of the dust increasing as the numbers increase. The diamond, after being crushed to powder in a mortar, is thoroughly mixed with high-grade olive oil. This mixture is allowed to stand five minutes and then the oil is poured into another receptacle. The coarse sediment which is left is removed and labeled No. 0, according to one system. The oil poured from No. 0 is again stirred and allowed to stand ten minutes, after which it is poured into another receptacle and the sediment remaining is labeled No. 1. This operation is repeated until practically all of the dust has been recovered from the oil, the time that the oil is allowed to stand being increased as shown by the following table. This is done in order to obtain the smaller particles that require a longer time for precipitation:

To obtain No. 1 — 10 minutes

To obtain No. 2 — 30 minutes

To obtain No. 3 — 1 hour

To obtain No. 4 — 2 hours

To obtain No. 5 — 10 hours

To obtain No. 6 — until oil is clear

The No. 0 or coarse diamond which is obtained from the first settling is usually washed in benzine, and re-crushed unless very coarse dust is required. This No. 0 grade is sometimes known as "ungraded" dust. In some places the time for settling, in order to obtain the various numbers, is greater than that given in the table.

Cutting Properties of Laps and Abrasives.—In order to determine the cutting properties of abrasives when used with different lapping materials and lubricants, a series of tests was conducted, the results of which were given in a paper by W. A. Knight and A. A. Case, presented before the American Society of Mechanical Engineers. In connection with these tests, a special machine was used, the construction being such that quantitative results could be obtained with various combinations of abrasive, lubricant, and lap material. These tests were confined to surface lapping.

It was not the intention to test a large variety of abrasives, three being selected as representative; namely, Naxos emery, carborundum, and alundum. Abrasive No. 150 was used in each case, and seven different lubricants, five different pressures, and three different lap materials were employed. The lubricants were lard oil, machine oil, kerosene, gasoline, turpentine, alcohol, and soda water.

These tests indicated throughout that there is, for each different combination of lap and lubricant, a definite size of grain that will give the maximum amount of cutting. With all the tests, except when using the two heavier lubricants, some reduction in the size of the grain below that used in the tests (No. 150) seemed necessary before the maximum rate of cutting was reached. This reduction, however, was continuous and soon passed below that which gave the maximum cutting rate.

Cutting Qualities with Different Laps.—The surfaces of the steel and cast-iron laps were finished by grinding. The hardness of the different laps, as determined by the scleroscope was, for cast-iron, 28; steel, 18; copper, 5. The total amount ground from the test-pieces with each of the three laps showed that, taking the whole number of tests as a standard, there is scarcely any difference between the steel and cast iron, but that copper has somewhat better cutting qualities, although, when comparing the laps on the basis of the highest and lowest values obtained with each lap, steel and cast iron are as good for all practical purposes as copper, when the proper abrasive and lubricant are used.

Wear of Laps.—The wear of laps depends upon the material from which they are made and the abrasive used. The wear on all laps was about twice as fast with carborundum as with emery, while with alundum the wear was about one and one-fourth times that with emery. On an average, the wear of the copper lap was about three times that of the cast-iron lap. This is not absolute wear, but wear in proportion to the amount ground from the test-pieces.

Lapping Abrasives.—As to the qualities of the three abrasives tested, it was found that carborundum usually began at a lower rate than the other abrasives, but, when once started, its rate was better maintained. The performance gave a curve that was more nearly a straight line. The charge or residue as the grinding proceeded remained cleaner and sharper and did not tend to become pasty or mucklike, as is so frequently the case with emery. When using a copper lap, carborundum shows but little gain over the cast-iron and steel laps, whereas, with emery and alundum, the gain is considerable.

Effect of Different Lapping Lubricants.—The action of the different lubricants, when tested, was found to depend upon the kind of abrasive and the lap material.

Lard and Machine Oil The test showed that lard oil, without exception, gave the higher rate of cutting, and that, in general, the initial rate of cutting is higher with the lighter lubricants, but falls off more rapidly as the test continues. The lowest results were obtained with machine oil, when using an emery-charged, cast-iron lap. When using lard oil and a carborundum-charged steel lap, the highest results were obtained.

Gasoline and Kerosene On the cast-iron lap, gasoline was superior to any of the lubricants tested. Considering all three abrasives, the relative value of gasoline, when applied to the different laps, is as follows: Cast iron, 127; copper, 115; steel, 106. Kerosene, like gasoline, gives the best results on cast iron and the poorest on steel. The values obtained by carborundum were invariably higher than those obtained with emery, except when using gasoline and kerosene on a copper lap.

Turpentine and Alcohol Turpentine was found to do good work with carborundum on any lap. With emery, turpentine did fair work on the copper lap, but, with the emery on cast-iron and steel laps, it was distinctly inferior. Alcohol gives the lowest results with emery on the cast-iron and steel laps.

Soda Water Soda water gives medium results with almost any combination of lap and abrasives, the best work being on the copper lap and the poorest on the steel lap. On the cast-iron lap, soda water is better than machine or lard oil, but not so good as gasoline or kerosene. Soda water when used with alundum on the copper lap, gave the highest results of any of the lubricants used with that particular combination.

Lapping Pressures.—Within the limits of the pressures used, that is, up to 25 pounds per square inch, the rate of cutting was found to be practically proportional to the pressure. The higher pressures of 20 and 25 pounds per square inch are not so effective on the copper lap as on the other materials.

Wet and Dry Lapping.—With the “wet method” of using a surface lap, there is a surplus of oil and abrasive on the surface of the lap. As the specimen being lapped is moved over it, there is more or less movement or shifting of the abrasive particles. With the “dry method,” the lap is first charged by rubbing or rolling the abrasive into its surface. All surplus oil and abrasive are then washed off, leaving a clean surface, but one that has embedded uniformly over it small particles of the abrasive. It is then like the surface of a very fine oilstone and will cut away hardened steel that is rubbed over it. While this has been termed the dry method, in practice, the lap surface is kept moistened with kerosene or gasoline.

Experiments on dry lapping were carried out on the cast-iron, steel, and copper laps used in the previous tests, and also on one of tin made expressly for the purpose. Carborundum alone was used as the abrasive and a uniform pressure of 15 pounds per square inch was applied to the specimen throughout the tests. In dry lapping, much depends upon the manner of charging the lap. The rate of cutting decreased much more rapidly after the first 100 revolutions than with the wet method. Considering the amounts ground off during the first 100 revolutions, and the best result obtained with each lap taken as the basis of comparison, it was found that with a tin lap, charged by rolling No. 150 carborundum into the surface, the rate of cutting, when dry, approached that obtained with the wet method. With the other lap materials, the rate with the dry method was about one-half that of the wet method.

Summary of Lapping Tests.—The initial rate of cutting does not greatly differ for different abrasives. There is no advantage in using an abrasive coarser than No. 150. The rate of cutting is practically proportional to the pressure. The wear of the laps is in the following proportions: cast iron, 1.00; steel, 1.27; copper, 2.62. In general, copper and steel cut faster than cast iron, but, where permanence of form is a consideration, cast iron is the superior metal. Gasoline and kerosene are the best lubricants to use with a cast-iron lap. Machine and lard oil are the best lubricants to use with copper or steel laps. They are, however, least effective on a cast-iron lap. In general, wet lapping is from 1.2 to 6 times as fast as dry lapping, depending upon the material of the lap and the manner of charging.

Portable Grinding Tools

Circular Saw Arbors.—ANSI Standard B107.4-1982 "Driving and Spindle Ends for Portable Hand, Air, and Air Electric Tools" calls for a round arbor of ⅝-inch diameter for nominal saw blade diameters of 6 to 8.5 inches, inclusive, and a ¾-inch diameter round arbor for saw blade diameters of 9 to 12 inches, inclusive.

Spindles for Geared Chucks.—Recommended threaded and tapered spindles for portable tool geared chucks of various sizes are as given in the following table:

Recommended Spindle Sizes

Chuck Sizes, Inch	Recommended Spindles	
	Threaded	Taper[a]
3/16 and ¼ Light	⅜–24	1
¼ and 5/16 Medium	⅜–24 or ½–20	2 Short
⅜ Light	⅜–24 or ½–20	2
⅜ Medium	½–20 or ⅝–16	2
½ Light	½–20 or ⅝–16	33
½ Medium	⅝–16 or ¾–16	6
⅝ and ¾ Medium	⅝–16 or ¾–16	3

[a] Jacobs number.

Vertical and Angle Portable Tool Grinder Spindles.—The ⅝–11 spindle with a length of 1⅛ inches shown on page 1209 is designed to permit the use of a jam nut with threaded cup wheels. When a revolving guard is used, the length of the spindle is measured from the wheel bearing surface of the guard. For unthreaded wheels with a ⅞-inch hole, a safety sleeve nut is recommended. The unthreaded wheel with ⅝-inch hole is not recommended because a jam nut alone may not resist the inertia effect when motor power is cut off.

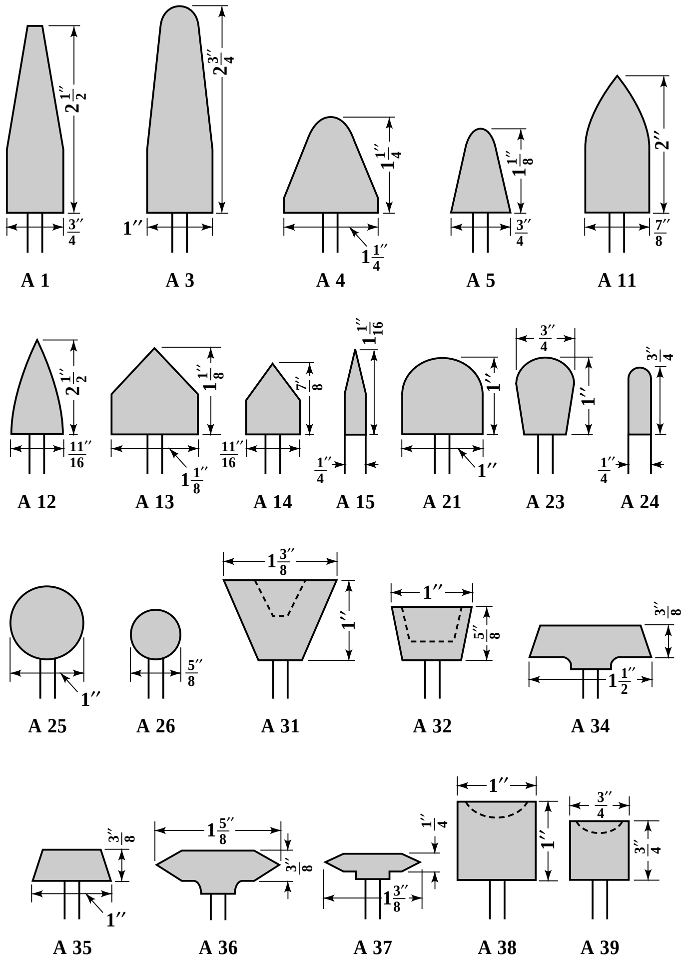
Standard Shapes and Metric Sizes of Mounted Wheels and Points
ANSI B74.2-1982

Abrasive Shape No.[a]	Abrasive Shape Size		Abrasive Shape No.[a]	Abrasive Shape Size	
	Diameter	Thickness		Diameter	Thickness
A 1	20	65	A 24	6	20
A 3	22	70	A 25	25	…
A 4	30	30	A 26	16	…
A 5	20	28	A 31	35	26
A 11	21	45	A 32	25	20
A 12	18	30	A 34	38	10
A 13	25	25	A 35	25	10
A 14	18	22	A 36	40	10
A 15	6	25	A 37	30	6
A 21	25	25	A 38	25	25
A 23	20	25	A 39	20	20
B 41	16	16	B 97	3	10
B 42	13	20	B 101	16	18
B 43	6	8	B 103	16	5
B 44	5.6	10	B 104	8	10
B 51	11	20	B 111	11	18
B 52	10	20	B 112	10	13
B 53	8	16	B 121	13	…
B 61	20	8	B 122	10	…
B 62	13	10	B 123	5	…
B 71	16	3	B 124	3	…
B 81	20	5	B 131	13	13
B 91	13	16	B 132	10	13
B 92	6	6	B 133	10	10
B 96	3	6	B 135	6	13
W 144	3	6	W 196	16	26
W 145	3	10	W 197	16	50
W 146	3	13	W 200	20	3
W 152	5	6	W 201	20	6
W 153	5	10	W 202	20	10
W 154	5	13	W 203	20	13
W 158	6	3	W 204	20	20
W 160	6	6	W 205	20	25
W 162	6	10	W 207	20	40
W 163	6	13	W 208	20	50
W 164	6	20	W 215	25	3
W 174	10	6	W 216	25	6
W 175	10	10	W 217	25	10
W 176	10	13	W 218	25	13
W 177	10	20	W 220	25	25
W 178	10	25	W 221	25	40
W 179	10	30	W 222	25	50
W 181	13	1.5	W 225	30	6
W 182	13	3	W 226	30	10
W 183	13	6	W 228	30	20
W 184	13	10	W 230	30	30
W 185	13	13	W 232	30	50
W 186	13	20	W 235	40	6
W 187	13	25	W 236	40	13
W 188	13	40	W 237	40	25
W 189	13	50	W 238	40	40
W 195	16	20	W 242	50	25

[a] See shape diagrams on pages 1205 and 1206.

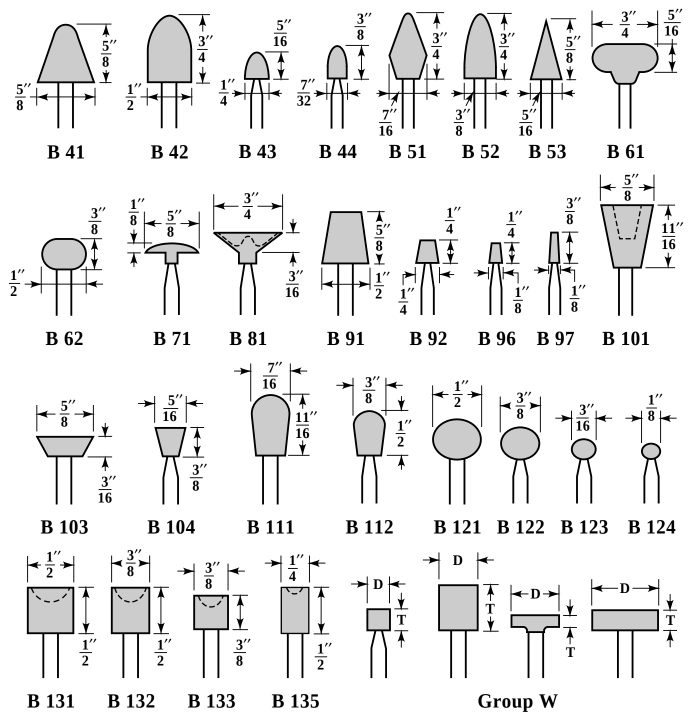
All dimensions are in millimeters.

Standard Shapes and Inch Sizes of Mounted Wheels and Points
ANSI B74.2-1982 — 1



The maximum speeds of mounted vitrified wheels and points of average grade range from about 38,000 to 152,000 rpm for diameters of 1 inch down to ¼ inch. However, the safe operating speed usually is limited by the critical speed (speed at which vibration or whip tends to become excessive) which varies according to wheel or point dimensions, spindle diameter, and overhang.

Standard Shapes and Inch Sizes of Mounted Wheels and Points
ANSI B74.2-1982 — 2



Abrasive Shape No.	Abrasive Shape Size		Abrasive Shape No.	Abrasive Shape Size		Abrasive Shape No.	Abrasive Shape Size	
	D	*T*		*D*	*T*		*D*	*T*
W 144	⅛	¼	W 182	½	⅛	W 208	¾	2
W 145	⅛	⅜	W 183	½	¼	W 215	1	⅛
W 146	⅛	½	W 184	½	⅜	W 216	1	¼
W 152	3⁄16	¼	W 185	½	½	W 217	1	⅜
W 153	3⁄16	⅜	W 186	½	¾	W 218	1	½
W 154	3⁄16	½	W 187	½	1	W 220	1	1
W 158	¼	⅛	W 188	½	1½	W 221	1	1½
W 160	¼	¼	W 189	½	2	W 222	1	2
W 162	¼	⅜	W 195	⅝	¾	W 225	1¼	¼
W 163	¼	½	W 196	⅝	1	W 226	1¼	⅜
W 164	¼	¾	W 197	⅝	2	W 228	1¼	¾
W 174	⅜	¼	W 200	¾	⅛	W 230	1¼	1¼
W 175	⅜	⅜	W 201	¾	¼	W 232	1¼	2
W 176	⅜	½	W 202	¾	⅜	W 235	1½	¼
W 177	⅜	¾	W 203	¾	½	W 236	1½	½
W 178	⅜	1	W 204	¾	¾	W 237	1½	1
W 179	⅜	1¼	W 205	¾	1	W 238	1½	1½
W 181	½	1⁄16	W 207	¾	1½	W 242	2	1

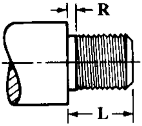
Straight Grinding Wheel Spindles for Portable Tools.—Portable grinders with pneumatic or induction electric motors should be designed for the use of organic bond wheels rated 9500 feet per minute. Light-duty electric grinders may be designed for vitrified wheels rated 6500 feet per minute. Recommended maximum sizes of wheels of both types are as given in the following table:

Recommended Maximum Grinding Wheel Sizes for Portable Tools

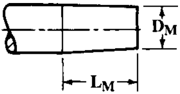
Spindle Size	Maximum Wheel Dimensions			
	9500 fpm		6500 fpm	
	Diameter *D*	Thickness *T*	Diameter *D*	Thickness *T*
⅜-24 × 1⅛	2½	½	4	½
½–13 × 1¾	4	¾	5	¾
⅝–11 × 2⅛	8	1	8	1
⅝–11 × 3⅛	6	2	…	…
⅝–11 × 3⅛	8	1½	…	…
¾–10 × 3¼	8	2	…	…

Minimum *T* with the first three spindles is about ⅛ inch to accommodate cutting off wheels. Flanges are assumed to be according to ANSI B7.1 and threads to ANSI B1.1.

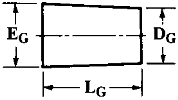
American Standard Threaded and Tapered Spindles for Portable Air and Electric Tools *ASA B5.38-1958*



Threaded Spindle



Taper Spindle (Jacobs)



Master Plug Gage

Nom. Dia. and Thd.	Pitch Dia.		*R*	*L*	No.[a]	D_M	L_M	E_G	D_G	L_G	Taper per Foot[b]
	Max.	Min.									
⅜–24	0.3479	0.3455	1/16	9/16[c]	1	0.335-0.333	0.656	0.38400	0.33341	0.65625	0.92508
½–20	0.4675	0.4649	1/16	9/16	2S[d]	0.490-0.488	0.750	0.54880	0.48764	0.7500	0.97861
					2	0.490-0.488	0.875	0.55900	0.48764	0.87500	0.97861
⅝–16	0.5844	0.5812	3/32	11/16	33	0.563-0.561	1.000	0.62401	0.56051	1.000	0.76194
					6	0.626-0.624	1.000	0.67600	0.62409	1.000	0.62292
¾–16	0.7094	0.7062	3/32	11/16	3	0.748-0.746	1.219	0.81100	0.74610	1.21875	0.63898

[a] Jacobs taper number.

[b] Calculated from E_G, D_G, L_G for the master plug gage.

[c] Also 7/16 inch.

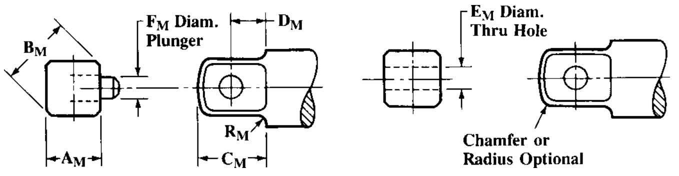
[d] 2S stands for 2 Short.

All dimensions in inches.

Threads are per inch and right-hand.

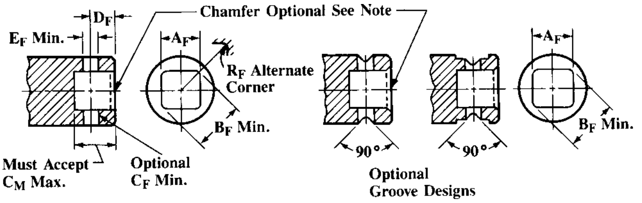
Tolerances: On *R*, plus or minus 1/64 inch; on *L*, plus 0.000, minus 0.030 inch.

American Standard Square Drives for Portable Air and Electric Tools
ASA B5.38-1958



DESIGN A **DESIGN B**

Drive Size	Male End										
	Design.	A_M		B_M	C_M		D_M		E_M	F_M	R_M
		Max.	Min.	Max.	Max.	Min.	Max.	Min.	Min.	Max.	Max.
¼	A	0.252	0.247	0.330	0.312	0.265	0.165	0.153	…	0.078	0.015
⅜	A	0.377	0.372	0.500	0.438	0.406	0.227	0.215	…	0.156	0.031
½	A	0.502	0.497	0.665	0.625	0.531	0.321	0.309	…	0.187	0.031
⅝	A	0.627	0.622	0.834	0.656	0.594	0.321	0.309	…	0.187	0.047
¾	B	0.752	0.747	1.000	0.938	0.750	0.415	0.403	0.216	…	0.047
1	B	1.002	0.997	1.340	1.125	1.000	0.602	0.590	0.234	…	0.063
1½	B	1.503	1.498	1.968	1.625	1.562	0.653	0.641	0.310	…	0.094



DESIGN A **DESIGN B**

Drive Size	Female End							
	Design	A_F		B_F	D_F		E_F	R_F
		Max.	Min.	Min.	Max.	Min.	Min.	Max.
¼	A	0.258	0.253	0.335	0.159	0.147	0.090	…
⅜	A	0.383	0.378	0.505	0.221	0.209	0.170	…
½	A	0.508	0.503	0.670	0.315	0.303	0.201	…
⅝	A	0.633	0.628	0.839	0.315	0.303	0.201	…
¾	B	0.758	0.753	1.005	0.409	0.397	0.216	0.047
1	B	1.009	1.004	1.350	0.596	0.584	0.234	0.062
1½	B	1.510	1.505	1.983	0.647	0.635	0.310	0.125

All dimensions in inches.

Incorporating fillet radius (R_M) at shoulder of male tang precludes use of minimum diameter cross-hole in socket (E_F), unless female drive end is chamfered (shown as optional).

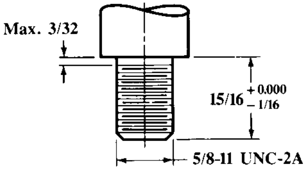
If female drive end is not chamfered, socket cross-hole diameter (E_F) is increased to compensate for fillet radius R_M, max.

Minimum clearance across flats male to female is 0.001 inch through ¾-inch size; 0.002 inch in 1- and 1½-inch sizes. For impact wrenches A_M should be held as close to maximum as practical.

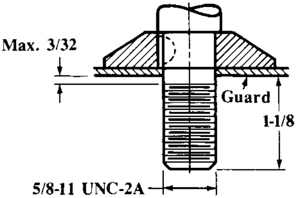
C_F, min. for both designs A and B should be equal to C_M, max.

American Standard Abrasion Tool Spindles for Portable Air and Electric Tools
ASA B5.38-1958

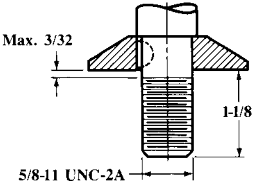
Sanders and Polishers



Vertical and Angle Grinders

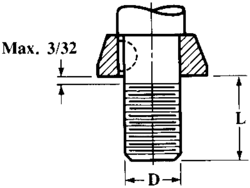


WITH REVOLVING CUP GUARD



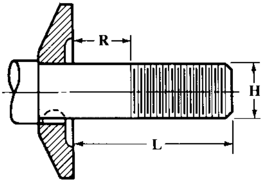
STATIONARY GURAD

Cone Wheel Grinders



D	*L*
3⁄8–24 UNF-2A	9⁄16
1⁄2–13 UNC-2A	11⁄16
5⁄8–11 UNC-2A	15⁄16

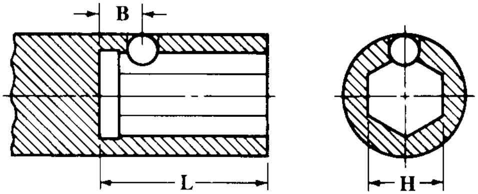
Straight Wheel Grinders



H	*R*	*L*
3⁄8–24 UNF-2A	1⁄4	1 1⁄8
1⁄2–13 UNC-2A	3⁄8	1 3⁄4
5⁄8–11 UNC-2A	1⁄2	2 1⁄8
5⁄8–11 UNC-2A	1	3 1⁄8
3⁄4–10 UNC-2A	1	3 1⁄4

All dimensions in inches. Threads are right-hand.

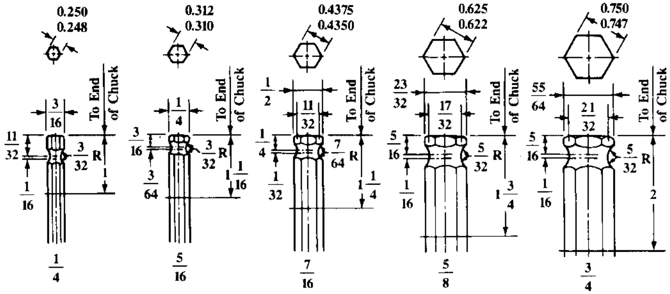
American Standard Hexagonal Chucks for Portable Air and Electric Tools
ASA B5.38-1958



Nominal Hexagon	*H* Min.	*H* Max.	*B*	*L* Max.	Nominal Hexagon	*H* Min.	*H* Max.	*B*	*L* Max.
¼	0.253	0.255	3⁄8	15⁄16	5⁄8	0.630	0.632	11⁄32	1 5⁄8
5⁄16	0.314	0.316	13⁄64	1	¾	0.755	0.758	11⁄32	1 7⁄8
7⁄16	0.442	0.444	17⁄64	1 1⁄8	…	…	…	…	…

All dimensions in inches.
Tolerances on *B* is plus or minus 0.005 inch.

American Standard Hexagon Shanks for Portable Air and Electric Tools
ASA B5.38-1958



KNURLS AND KNURLING

ANSI Standard Knurls and Knurling.—The ANSI/ASME Standard B94.6-1984 covers knurling tools with standardized diametral pitches and their dimensional relations with respect to the work in the production of straight, diagonal, and diamond knurling on cylindrical surfaces having teeth of uniform pitch parallel to the cylinder axis or at a helix angle not exceeding 45 degrees with the work axis.

These knurling tools and the recommendations for their use are equally applicable to general purpose and precision knurling. The advantage of this ANSI Standard system is the provision by which good tracking (the ability of teeth to mesh as the tool penetrates the work blank in successive revolutions) is obtained by tools designed on the basis of diametral pitch instead of TPI (teeth per inch) when used with work blank diameters that are multiples of $\frac{1}{64}$ inch for 64 and 128 diametral pitch or $\frac{1}{32}$ inch for 96 and 160 diametral pitch. The use of knurls and work blank diameters which will permit good tracking should improve the uniformity and appearance of knurling, eliminate the costly trial and error methods, reduce the failure of knurling tools and production of defective work, and decrease the number of tools required. Preferred sizes for cylindrical knurls are given in Table 1 and detailed specifications appear in Table 2.

Table 1. ANSI Standard Preferred Sizes for Cylindrical Type Knurls
ANSI/ASME B94.6-1984

Nominal Outside Diameter D_{nt}	Width of Face F	Diameter of Hole A	Standard Diametral Pitches, P			
			64	96	128	160
			Number of Teeth, N_t, for Standard Pitches			
$\frac{1}{2}$	$\frac{3}{16}$	$\frac{3}{16}$	32	48	64	80
$\frac{5}{8}$	$\frac{1}{4}$	$\frac{1}{4}$	40	60	80	100
$\frac{3}{4}$	$\frac{3}{8}$	$\frac{1}{4}$	48	72	96	120
$\frac{7}{8}$	$\frac{3}{8}$	$\frac{1}{4}$	56	84	112	140
Additional Sizes for Bench and Engine Lathe Tool Holders						
$\frac{5}{8}$	$\frac{5}{16}$	$\frac{7}{32}$	40	60	80	100
$\frac{3}{4}$	$\frac{5}{8}$	$\frac{1}{4}$	48	72	96	120
1	$\frac{3}{8}$	$\frac{5}{16}$	64	96	128	160

The 96 diametral pitch knurl should be given preference in the interest of tool simplification. Dimensions D_{nt}, F, and A are in inches.

Table 2. ANSI Standard Specifications for Cylindrical Knurls with Straight or Diagonal Teeth ***ANSI/ASME B94.6-1984***

Diametral Pitch P	Nominal Diameter, D_{nt}					Tracking Correction Factor Q	Tooth Depth, h, $+0.0015$, -0.0000		Radius at Root R
	$\frac{1}{2}$	$\frac{5}{8}$	$\frac{3}{4}$	$\frac{7}{8}$	1		Straight	Diagonal	
	Major Diameter of Knurl, D_{ot}, $+0.0000$, -0.0015								
64	0.4932	0.6165	0.7398	0.8631	0.9864	0.0006676	0.024	0.021	0.0070 0.0050
96	0.4960	0.6200	0.7440	0.8680	0.9920	0.0002618	0.016	0.014	0.0060 0.0040
128	0.4972	0.6215	0.7458	0.8701	0.9944	0.0001374	0.012	0.010	0.0045 0.0030
160	0.4976	0.6220	0.7464	0.8708	0.9952	0.00009425	0.009	0.008	0.0040 0.0025

All dimensions except diametral pitch are in inches.

Approximate angle of space between sides of adjacent teeth for both straight and diagonal teeth is 80 degrees. The permissible eccentricity of teeth for all knurls is 0.002 inch maximum (total indicator reading).

Number of teeth in a knurl equals diametral pitch multiplied by nominal diameter.

Diagonal teeth have 30-degree helix angle, ψ.

The term *Diametral Pitch* applies to the quotient obtained by dividing the total number of teeth in the circumference of the work by the basic blank diameter; in the case of the knurling tool it would be the total number of teeth in the circumference divided by the *nominal* diameter. In the Standard the diametral pitch and number of teeth are always measured in a transverse plane which is perpendicular to the axis of rotation for diagonal as well as straight knurls and knurling.

Cylindrical Knurling Tools.—The cylindrical type of knurling tool comprises a tool holder and one or more knurls. The knurl has a centrally located mounting hole and is provided with straight or diagonal teeth on its periphery. The knurl is used to reproduce this tooth pattern on the work blank as the knurl and work blank rotate together.

[*]*Formulas for Cylindrical Knurls*

$$P = \text{diametral pitch of knurl} = N_t \div D_{nt} \qquad (1)$$

$$D_{nt} = \text{nominal diameter of knurl} = N_t \div P \qquad (2)$$

$$N_t = \text{no. of teeth on knurl} = P \times D_{nt} \qquad (3)$$

$$^*P_{nt} = \text{circular pitch on nominal diameter} = \pi \div P \qquad (4)$$

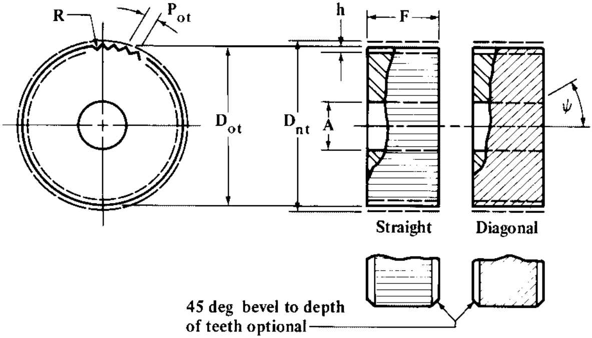
$$^*P_{ot} = \text{circular pitch on major diameter} = \pi D_{ot} \div N_t \qquad (5)$$

$$D_{ot} = \text{major diameter of knurl} = D_{nt} - (N_t Q \div \pi) \qquad (6)$$

$$Q = P_{nt} - P_{ot} = \text{tracking correction factor in Formula} \qquad (7)$$

Tracking Correction Factor Q: Use of the preferred pitches for cylindrical knurls, Table 2, results in good tracking on all fractional work-blank diameters which are multiples of $\frac{1}{64}$ inch for 64 and 128 diametral pitch, and $\frac{1}{32}$ inch for 96 and 160 diametral pitch; an indication of good tracking is evenness of marking on the work surface during the first revolution of the work.

The many variables involved in knurling practice require that an empirical correction method be used to determine what actual circular pitch is needed at the major diameter of the knurl to produce good tracking and the required circular pitch on the workpiece. The empirical tracking correcton factor, *Q,* in Table 2 is used in the calculation of the major diameter of the knurl, Formula (6).



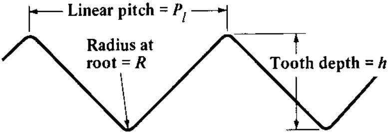
Cylindrical Knurl

---

[*] *Note:* For diagonal knurls, P_{nt} and P_{ot} are the transverse circular pitches which are measured in the plane perpendicular to the axis of rotation.

Flat Knurling Tools.—The flat type of tool is a knurling die, commonly used in reciprocating types of rolling machines. Dies may be made with either single or duplex faces having either straight or diagonal teeth. No preferred sizes are established for flat dies.

Flat Knurling Die with Straight Teeth:



$R =$ radius at root

$P =$ diametral pitch $= N_w \div D_w$ (8)

$D_w =$ work blank (pitch) diameter $= N_w \div P$ (9)

$N_w =$ number of teeth on work $= P \times D_w$ (10)

$h =$ tooth depth

$Q =$ tracking correction factor (see Table 2)

$P_l =$ linear pitch on die

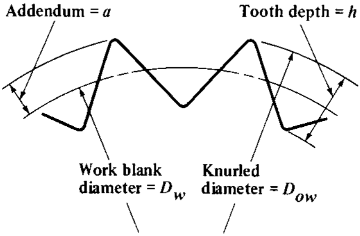
$=$ circular pitch on work pitch diameter $= P - Q$ (11)

Table 3. ANSI Standard Specifications for Flat Knurling Dies
ANSI/ASME B94.6-1984

Diametral Pitch, P	Linear Pitch,[a] P_l	Tooth Depth, h		Radius at Root, R	Diametral Pitch, P	Linear Pitch,[a] P_l	Tooth Depth, h		Radius at Root, R
		Straight	Diagonal				Straight	Diagonal	
64	0.0484	0.024	0.021	0.0070 0.0050	128	0.0244	0.012	0.010	0.0045 0.0030
96	0.0325	0.016	0.014	0.0060 0.0040	160	0.0195	0.009	0.008	0.0040 0.0025

[a] The linear pitches are theoretical. The exact linear pitch produced by a flat knurling die may vary slightly from those shown depending upon the rolling condition and the material being rolled.

All dimensions except diametral pitch are in inches.



Teeth on Knurled Work

Formulas Applicable to Knurled Work.—The following formulas are applicable to knurled work with straight, diagonal, and diamond knurling.

Formulas for Straight or Diagonal Knurling with Straight or Diagonal Tooth Cylindrical Knurling Tools Set with Knurl Axis Parallel with Work Axis:

$$P = \text{diametral pitch} = N_w \div D_w \quad (12)$$

$$D_w = \text{work blank diameter} = N_w \div P \quad (13)$$

$$N_w = \text{no. of teeth on work} = P \times D_w \quad (14)$$

$$a = \text{“addendum” of tooth on work} = (D_{ow} - D_w) \div 2 \quad (15)$$

$$h = \text{tooth depth (see Table 2)}$$

$$D_{ow} = \text{knurled diameter (outside diameter after knurling)} = D_w + 2a \quad (16)$$

Formulas for Diagonal and Diamond Knurling with Straight Tooth Knurling Tools Set at an Angle to the Work Axis:

If, ψ = angle between tool axis and work axis
P = diametral pitch on tool
P_ψ = diametral pitch produced on work blank (as measured in the transverse plane) by setting tool axis at an angle ψ with respect to work blank axis
D_w = diameter of work blank; and
N_w = number of teeth produced on work blank (as measured in the transverse plane)

then, $P_\psi = P \cos\psi$ (17)

and, $N = D_w P \cos\psi$ (18)

For example, if 30 degree diagonal knurling were to be produced on 1-inch diameter stock with a 160 pitch straight knurl:

$$N_w = D_w P \cos 30^\circ = 1.000 \times 160 \times 0.86603 = 138.56 \text{ teeth}$$

Good tracking is theoretically possible by changing the helix angle as follows to correspond to a whole number of teeth (138):

$$\cos\psi = N_w \div D_w P = 138 \div (1 \times 160) = 0.8625$$

$$\psi = 30\tfrac{1}{2} \text{ degrees, approximately}$$

Whenever it is more practical to machine the stock, good tracking can be obtained by reducing the work blank diameter as follows to correspond to a whole number of teeth (138):

$$D_w = \frac{N_w}{P\cos\psi} = \frac{138}{160 \times 0.866} = 0.996 \text{ inch}$$

Table 4. ANSI Standard Recommended Tolerances on Knurled Diameters
ANSI/ASME B94.6-1984

Tolerance Class	Diametral Pitch							
	64	96	128	160	64	96	128	160
	Tolerance on Knurled Outside Diameter				Tolerance on Work-Blank Diameter Before Knurling			
I	+ 0.005 − 0.012	+ 0.004 − 0.010	+ 0.003 − 0.008	+ 0.002 − 0.006	± 0.0015	± 0.0010	± 0.0007	± 0.0005
II	+ 0.000 − 0.010	+ 0.000 − 0.009	+ 0.000 − 0.008	+ 0.000 − 0.006	± 0.0015	± 0.0010	± 0.0007	± 0.0005
III	+ 0.000 − 0.006	+ 0.000 − 0.005	+ 0.000 − 0.004	+ 0.000 − 0.003	+ 0.000 − 0.0015	+ 0.0000 − 0.0010	+ 0.000 − 0.0007	+ 0.0000 − 0.0005

Recommended Tolerances on Knurled Outside Diameters.—The recommended applications of the tolerance classes shown in Table 4 are as follows:

Class I: Tolerances in this classification may be applied to straight, diagonal and raised diamond knurling where the knurled outside diameter of the work need not be held to close dimensional tolerances. Such applications include knurling for decorative effect, grip on thumb screws, and inserts for moldings and castings.

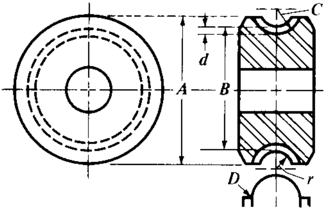
Class II: Tolerances in this classification may be applied to straight knurling only and are recommended for applications requiring closer dimensional control of the knurled outside diameter than provided for by Class I tolerances.

Class III: Tolerances in this classification may be applied to straight knurling only and are recommended for applications requiring closest possible dimensional control of the knurled outside diameter. Such applications include knurling for close fits.

Note: The width of the knurling should not exceed the diameter of the blank, and knurling wider than the knurling tool cannot be produced unless the knurl starts at the end of the work.

Marking on Knurls and Dies.—Each knurl and die should be marked as follows: *a.* when straight to indicate its diametral pitch; *b.* when diagonal, to indicate its diametral pitch, helix angle, and hand of angle.

Concave Knurls.—The radius of a concave knurl should not be the same as the radius of the piece to be knurled. If the knurl and the work are of the same radius, the material compressed by the knurl will be forced down on the shoulder *D* and spoil the appearance of the work. A design of concave knurl is shown in the accompanying illustration, and all the important dimensions are designated by letters. To find these dimensions, the pitch of the knurl required must be known, and also, approximately, the throat diameter *B*. This diameter must suit the knurl holder used, and be such that the circumference contains an even number of teeth with the required pitch. When these dimensions have been decided upon, all the other unknown factors can be found by the following formulas: Let R = radius of piece to be knurled; r = radius of concave part of knurl; C = radius of cutter or hob for cutting the teeth in the knurl; B = diameter over concave part of knurl (throat diameter); A = outside diameter of knurl; d = depth of tooth in knurl; P = pitch of knurl (number of teeth per inch circumference); p = circular pitch of knurl; then $r = R + \frac{1}{2}d$; $C = r + d$; $A = B + 2r - (3d + 0.010 \text{ inch})$; and $d = 0.5 \times p \times \cot \alpha/2$, where α is the included angle of the teeth.



As the depth of the tooth is usually very slight, the throat diameter *B* will be accurate enough for all practical purposes for calculating the pitch, and it is not necessary to take into consideration the pitch circle. For example, assume that the pitch of a knurl is 32, that the throat diameter *B* is 0.5561 inch, that the radius *R* of the piece to be knurled is $\frac{1}{16}$ inch, and that the angle of the teeth is 90 degrees; find the dimensions of the knurl. Using the notation given:

$$p = \frac{1}{P} = \frac{1}{32} = 0.03125 \text{ inch} \qquad d = 0.5 \times 0.03125 \times \cot 45^\circ = 0.0156 \text{ inch}$$

$$r = \frac{1}{16} + \frac{0.0156}{2} = 0.0703 \text{ inch} \qquad C = 0.0703 + 0.0156 = 0.0859 \text{ inch}$$

$$A = 0.5561 + 0.1406 - (0.0468 + 0.010) = 0.6399 \text{ inch}$$

MACHINE TOOL ACCURACY

Accuracy, Repeatability, and Resolution: In machine tools, accuracy is the maximum spread in measurements made of slide movements during successive runs at a number of target points, as discussed below. Repeatability is the spread of the normal curve at the target point that has the largest spread. A rule of thumb says that repeatability is approximately half the accuracy value, or twice as good as the accuracy, but this rule is somewhat nullified due to the introduction of error-compensation features on NC machines. Resolution refers to the smallest units of measurement that the system (controller plus servo) can recognize. Resolution is an electronic/electrical term and the unit is usually smaller than either the accuracy or the repeatability. Low values for resolution are usually, though not necessarily, applied to machines of high accuracy. In addition to high cost, a low-resolution-value design usually has a low maximum feed rate and the use of such designs is usually restricted to applications requiring high accuracy.

*Positioning Accuracy:*The positioning accuracy of a numerically controlled machine tool refers to the ability of an NC machine to place the tip of a tool at a preprogrammed target. Although no metal cutting is involved, this test is very significant for a machine tool and the cost of an NC machine will rise almost geometrically with respect to its positioning accuracy. Care, therefore, should be taken when deciding on the purchase of such a machine, to avoid paying the premium for unneeded accuracy but instead to obtain a machine that will meet the tolerance requirements for the parts to be produced.

Accuracy can be measured in many ways. A tool tip on an NC machine could be moved, for example, to a target point whose *X*-coordinate is 10.0000 inches. If the move is along the *X*-axis, and the tool tip arrives at a point that measures 10.0001 inches, does this mean that the machine has an accuracy of 0.0001 inch? What if a repetition of this move brought the tool tip to a point measuring 10.0003 inches, and another repetition moved the tool to a point that measured 9.9998 inches? In practice, it is expected that there would be a scattering or distribution of measurements and some kind of averaging is normally used.

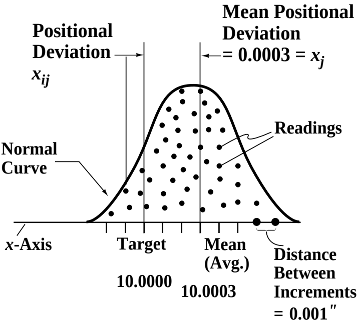


Fig. 1. In a Normal Distribution, Plotted Points Cluster Around the Mean.

Although averaging the results of several runs is an improvement over a single run, the main problem with averaging is that it does not consider the extent or width of the spread of readings. For example, if one measurement to the 10.0000-inch target is 9.9000 inches and another is 10.1000 inches, the difference of the two readings is 0.2000 inch, and the accuracy is poor. However, the readings average a perfect 10 inches. Therefore, the average and the spread of several readings must both be considered in determining the accuracy.

Plotting the results of a large number of runs generates a *normal distribution curve,* as shown in Fig. 1. In this example, the readings are plotted along the *X*-axis in increments of

0.0001 inch (0.0025 mm). Usually, five to ten such readings are sufficient. The distance of any one reading from the target is called the *positional deviation* of the point. The distance of the mean, or average, for the normal distribution from the target is called the *mean positional deviation.*

The spread for the normal curve is determined by a mathematical formula that calculates the distance from the mean that a certain percentage of the readings fall into. The mathematical formula used calculates one *standard deviation,* which represents approximately 32 per cent of the points that will fall within the normal curve, as shown in Fig. 2. One standard deviation is also called one sigma, or 1σ. Plus or minus one sigma ($\pm 1\sigma$) represents 64 per cent of all the points under the normal curve. A wider range on the curve, $\pm 2\sigma$, means that 95.44 per cent of the points are within the normal curve, and $\pm 3\sigma$ means that 99.74 per cent of the points are within the normal curve. If an infinite number of runs were made, almost all the measurements would fall within the $\pm 3\sigma$ range.

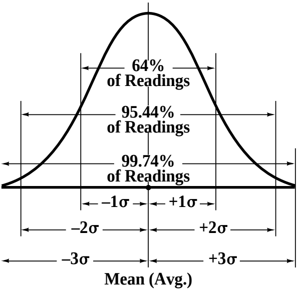


Fig. 2. Percentages of Points Falling in the ±1σ (64%), ±2σ (95.44%), and ±3σ (99.74%) Ranges

The formula for calculating one standard deviation is

$$1\sigma = \sqrt{\frac{1}{n-1}\sum_{i=1}^{n}(X_{ij}-\bar{X}_j)^2}$$

where n = number of runs to the target; i = identification for any one run; X_{ij} = positional deviation for any one run (see Fig. 1); and, $\bar{X}_j$ = mean positional deviation (see Fig. 1).

The bar over the $\bar{X}$ in the formula indicates that the value is the mean or average for the normal distribution.

Example: From Fig. 3, five runs were made at a target point that is 10.0000 inches along the X-axis and the positional deviations for each run were:
$x_{1j} = -0.0002$, $x_{2j} = +0.0002$, $x_{3j} = +0.0005$, $x_{4j} = +0.0007$, and $x_{5j} = +0.0008$ inch. The algebraic total of these five runs is +0.0020, and the mean positional deviation = $\bar{X}_j = 0.0020/5 = 0.0004$.

The calculations for one standard deviation are:

$$1\sigma = \sqrt{\frac{1}{n-1}[(X_{1j}-\bar{X}_j)^2 + (X_{2j}-\bar{X}_j)^2 + (X_{3j}-\bar{X}_j)^2 + (X_{4j}-\bar{X}_j)^2 + (X_{5j}-\bar{X}_j)^2]}$$

$$1\sigma = \sqrt{\frac{1}{5-1}\begin{matrix}[(-0.0002-0.0004)^2 + (0.0002-0.0004)^2 \\ (0.0005-0.0004)^2 + (0.0007-0.0004)^2 + (0.0008-0.0004)^2]\end{matrix}}$$

$$= \sqrt{\frac{1}{4}(0.00000066)} = \sqrt{0.17\times10^{-6}} = 0.0004$$

Three sigma variations or 3σ, is 3 times sigma, equal to 0.0012 for the example.

If an infinite number of trials were made to the target position of 10.0000 inches for the ongoing example, 99.74 per cent of the points would fall between 9.9992 and 10.0016 inches, giving a spread of $\pm 3\sigma$, or 0.0024 inch. This spread alone is not considered as the *accuracy* but rather the repeatability for the target point 10.0000.

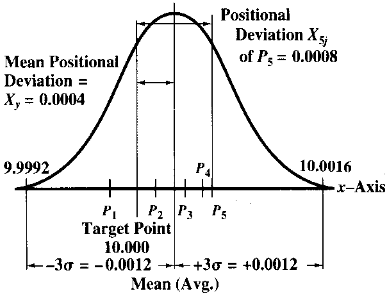


Fig. 3. Readings for Five Runs to Target Points P_1, P_2, P_3, P_4, and P_5 Result in a Mean Positional Deviation of 0.0004

To calculate the accuracy, it is not sufficient to make a number of runs to one target point along a particular axis, but rather to a number of points along the axis, the number depending on the length of axis travel provided. For example, a travel of about 3 ft requires 5, and a travel of 6 ft requires 10 target points. The standard deviation and spread for the normal curve must be determined at each target point, as shown in Fig. 4. The *accuracy* for the axis would then be the spread between the normal curve with the most negative position and the normal curve with the most positive position. Technically, the accuracy is a spread rather than a ± figure, but it is often referred to as a ± figure and it may be assumed that a ±0.003, for expediency, is equal to a spread of 0.006.

The above description for measuring accuracy considers unidirectional approaches to target points. Bidirectional movements (additional movements to the same target point from either direction) will give different results, mostly due to backlash in the lead-screw, though backlash is small with ballnut leadscrews. Measurements made with bidirectional movements will show greater spreads and somewhat less accuracy than will unidirectional movements.

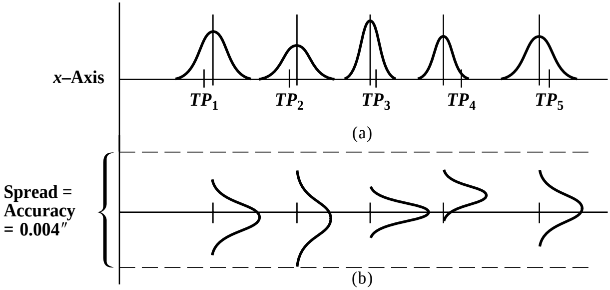


Fig. 4. Two Ways of Plotting Five Target Point Spreads

Rules for determining accuracy were standardized in guidelines last revised by the Association for Manufacturing Technology (AMT) in 1972. Some European machine tool builders use the VDI/DGQ 3441 (German) guidelines, which are similar to those of the

AMT in that normal distributions are used and a number of target points are selected along an axis. Japanese standards JIS-B-6201, JIS-B-6336, and JIS-B-6338 are somewhat simpler and consider only the spread of the readings, so that the final accuracy figure may be almost double that given by the AMT or VDI methods. The International Standards Organization (ISO), in 1988, issued ISO 230-2, which follows the procedures discussed above, but is somewhat less strict than the AMT recommendations. Table 1 lists some types of NC machines and the degree of accuracy that they normally provide.

Table 1. Degrees of Accuracy Expected with NC Machine Tools

Type of NC Machine	Accuracy	
	inches	mm
Large boring machines or boring mills	0.0010–0.0020	0.025–0.050
Small milling machines	0.0006–0.0010	0.015–0.025
Large machining centers	0.0005–0.0008	0.012–0.020
Small and medium-sized machining centers	0.0003–0.0006	0.008–0.015
Lathes, slant bed, small and medium sizes	0.0002–0.0005	0.006–0.012
Lathes, small precision	0.0002–0.0003	0.004–0.008
Horizontal jigmill	0.0002–0.0004	0.004–0.010
Vertical jig boring machines	0.0001–0.0002	0.002–0.005
Vertical jig grinding machines	0.0001–0.0002	0.002–0.005
Cylindrical grinding machines, small to medium sizes	0.00004–0.0003	0.001–0.007
Diamond turning lathes	0.00002–0.0001	0.0005–0.003

*Significance of Accuracy:*Numerically controlled machines are generally considered to be more accurate and more consistent in their movements than their conventional counterparts. CNC controllers have improved the accuracy by providing the ability to compensate for mechanical inaccuracies. Thus, compensation for errors in the lead-screw, parallelism and squareness of the machine ways, and for the effects of heating can be made automatically on NC machines. Some machine tool types are expected to be more accurate than others; for instance, grinding machines are more accurate than milling machines, and lathes for diamond turning are more accurate than normal slant-bed lathes.

Accuracy of machine tools depends on temperature, air pressure, local vibrations, and humidity. ISO standard 230-2 requires that, where possible, the ambient temperature for conducting such tests be held between 67.1 and 68.9 degrees F (19.5 and 20.5 degrees C).

*Autocollimation:*Checks on movements of slides and spindles, and alignment and other characteristics of machine tools are performed with great accuracy by means of an autocollimator, which is an optical, noncontact, angle-measuring instrument. Flatness, straightness, perpendicularity, and runout can also be checked by autocollimation. The instrument is designed to project a beam of light from a laser or an incandescent bulb onto an optically flat mirror. When the light beam is reflected back to the instrument, the distance traveled by the beam, also deviations from a straight line, can be detected by the projector and calculated electronically or measured by the scale.

Autocollimators have a small angular measuring range and are usually calibrated in arc-seconds. One arc-second is an angle of 4.85 millionths of an inch (0.00000485 in.) per inch of distance from the vertex, and is often rounded to 5 millionths of an inch per inch. Angles can also be described in terms of radians and 1 arc-second is equal to 4.85 microradians, or 0.0000573 deg.

In practice, the interferometer or autocollimator is fixed to a rigid structure and the optical mirror, which should have a flatness of one-quarter wavelength of the light used (see page 696), is fixed to the workpiece to be measured. The initial reading is taken, and then the workpiece is moved to another position. Readings of movement can be made to within a few millionths of an inch. Angular displacements, corresponding to successive positions, of about 1 arc-second can be taken from most autocollimators, in azimuth or elevation or a combination of the two. Generally, the line width of the reticle limits the accuracy of reading such instruments.

Laser interferometers are designed to allow autocollimation readings to be taken by a photodetector instead of the eye, and some designs can measure angles to 0.001 arc-second, closer than is required for most machine shop applications. Output from an electronic autocollimator is usually transferred to a computer for recording or analysis if required. The computer calculates, lists, and plots the readings for the target points automatically, under control of the inspection program.

A typical plot from such a setup is seen in Fig. 5, where the central line connects the averages for the normal distributions at each target point. The upper line connects the positive outer limits and the lower line the negative outer limits for the normal distributions. The normal spread, indicating the accuracy of positioning, is 0.00065 inch (0.016 mm), for the *Y*-axis along which the measurements were taken.

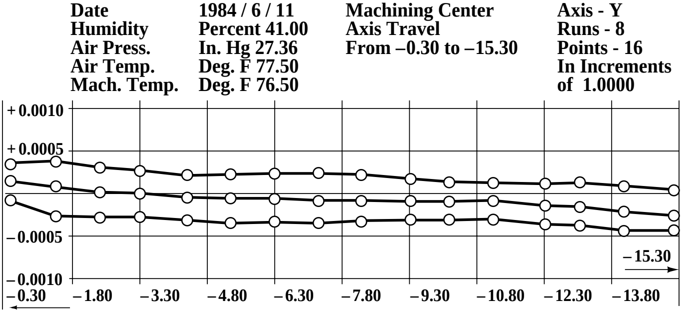


Fig. 5. Laser Interferometer Plots of Movements of Slides on a Large Horizontal Machining Center Showing an Accuracy of 0.00065 inch (0.016 mm) for the *y* Axis

Effect of Machine Accuracy on Part Tolerances

Part tolerances are usually shown on prints, usually in a control block to ANSI Standard 14.5M-1994 (see *Geometric Dimensioning and Tolerancing* starting on page 606.) Table 2 shows some part tolerance symbols that relate to machine tool positioning accuracy. The accuracy of a part is affected by machine and cutting tool dynamics, alignment, fixture accuracy, operator settings, and accuracies of the cutting tools, holders, and collets, but the positioning accuracy of the machine probably has the greatest influence. Spindle rotation accuracy, or runout, also has a large influence on part accuracy.

The ratio of the attainable part accuracy to the no-load positioning accuracy can vary from 1.7:1 to 8.31:1, depending on the type of cutting operation. For instance, making a hole by drilling, followed by a light boring or reaming operation, produces a quite accurate result in about the 1.7:1 range, whereas contour milling on hard material could be at the higher end of the range. A good average for part accuracy versus machine positioning accuracy is 3.3:1, which means that the part accuracy is 3.3 times the positioning accuracy.

Table 2. Symbols and Feature Control Frames *ANSI 14.5M-1994*

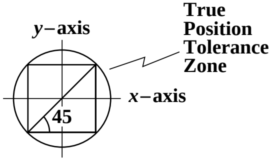
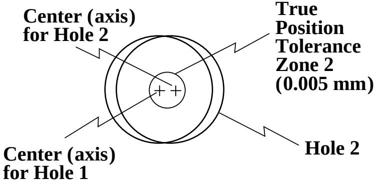
Symbol	Characteristic	Meaning of Characteristic	Relationship to the Machine Tool
⌖	Position	The allowable true position tolerance of a feature from a datum (assume feature to be a drilled hole). Feature control block might appear as: ⌖ ⌀ **0.005** **A** A is the datum, which can be another surface, another hole, or other feature	Assume tolerance is 0.005 mm. Machine positioning accuracy would be at least 0.005 × 0.707 = 0.0035 mm even if it is assumed that the hole accuracy is the same as the positioning accuracy. Machine could be milling, drilling, or machining center. 
	Position	Assume feature to be a turned circumference, the axis of which has to be within a tolerance to another feature. Feature control block would appear as follows if feature A were the axis of hole 1: ⌖ ⌀ **0.005** **A**	

Table 2. Symbols and Feature Control Frames *ANSI 14.5M-1994*

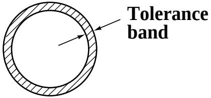
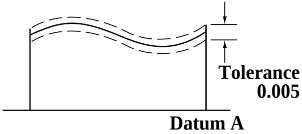
Symbol	Characteristic	Meaning of Characteristic	Relationship to the Machine Tool
○	Roundness	The roundness tolerance establishes a band.  	This tolerance would apply to turning and would be the result of radial spindle runout.
⌀	Diameter	Usually expressed as a ± tolerance attached to the dimension.	Diametral accuracy of the part would depend on the positioning accuracy of the cross-slide of lathe or grinder. Positioning accuracy would be from ½ to ¼ of part accuracy, depending chiefly on the rigidity of the tool, depth of cut, and material being cut.
⌓	Profile of a surface	Specifies a uniform boundary, along a true profile.  Feature control block might appear as: ⌓ ⌀ **0.005** **A**	Affected by positioning accuracy of machine. There would be side and/or end forces on the tool so expect part to machine positioning accuracy to be high, say, 5:1

Table 2. Symbols and Feature Control Frames *ANSI 14.5M-1994*

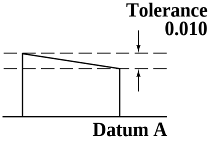
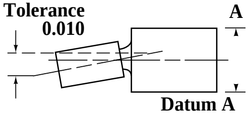
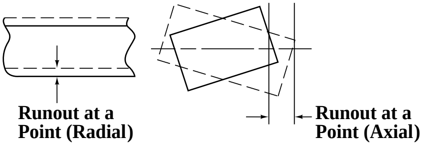
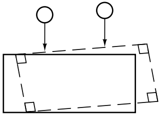
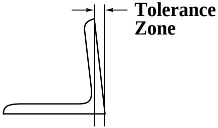
Symbol	Characteristic	Meaning of Characteristic	Relationship to the Machine Tool
//	Parallelism	A feature (surface) parallel to a datum plane or datum axis.  Feature control block might appear as: // │ ∅ **0.010** │ **A**	Affected by positioning accuracy, machine alignment, and fixturing.
◎	Concentricity	Applies to turning. The axis of the feature must lie within the tolerance zone of another axis.  Feature control block might appear as follows: ◎ │ ∅ **0.005** │ **A**	Affected by positioning accuracy, most likely along Z axis.

Table 2. Symbols and Feature Control Frames *ANSI 14.5M-1994*

Symbol	Characteristic	Meaning of Characteristic	Relationship to the Machine Tool
↗	Runout	Applies to the runout (both radial and axial) of a circular feature at any one position around the circumference or flat, perpendicular to the axis.  Runout at a Point (Radial)	Radial runout on part is not affected by spindle radial runout unless whole machine is untrue. Axial runout on part is affected by axial runout on machine. Feature would normally be perpendicular to datum. Feature control block might appear as: ↗ | ∅ **0.005** | **A**
⌰	Total runout	Similar to runout but applies to total surface and therefore consider both radial and axial runout. 	Would be affected by either radial or axial runout, or both, machine misalignment, or setup.
⊥	Perpendicularity	A feature is perpendicular to a datum plane or axis.  	Affected principally by misalignment of machine or fixturing.

NUMERICAL CONTROL

Introduction.—The Electronic Industries Association (EIA) defines numerical control as "a system in which actions are controlled by the direct insertion of numerical data at some point." More specifically, numerical control, or NC as it will be called here, involves machines controlled by electronic systems designed to accept numerical data and other instructions, usually in a coded form. These instructions may come directly from some source such as a punched tape, a floppy disk, directly from a computer, or from an operator.

The key to the success of numerical control lies in its flexibility. To machine a different part, it is only necessary to "play" a different tape. NC machines are more productive than conventional equipment and consequently produce parts at less cost even when the higher investment is considered. NC machines also are more accurate and produce far less scrap than their conventional counterparts. By 1985, over 110,000 NC machine tools were operating in the United States. Over 80 per cent of the dollars being spent on the most common types of machine tools, namely, drilling, milling, boring, and turning machines, are going into NC equipment.

NC is a generic term for the whole field of numerical control and encompasses a complete field of endeavor. Sometimes CNC, which stands for Computer Numerical Control and applies only to the control system, is used erroneously as a replacement term for NC. Albeit a monumental development, use of the term CNC should be confined to installations where the older hardware control systems have been replaced.

Metal cutting is the most popular application, but NC is being applied successfully to other equipment, including punch presses, EDM wire cutting machines, inspection machines, laser and other cutting and torching machines, tube bending machines, and sheet metal cutting and forming machines.

State of the CNC Technology Today.—Early numerical control machines were ordinary machines retrofitted with controls and motors to drive tools and tables. The operations performed were the same as the operations were on the machines replaced. Over the years, NC machines began to combine additional operations such as automatically changing tools and workpieces. The structure of the machines has been strengthened to provide more rigid platforms. These changes have resulted in a class of machine that can outperform its predecessors in both speed and accuracy. Typical capabilities of a modern machining center are accuracy better than ±0.00035 inch; spindle speeds in the range up to 25,000 rpm or more, and increasing; feed rates up to 400 inches per minute and increasing; tool change times hovering between 2 and 4 seconds and decreasing. Specialized machines have been built that can achieve accuracy better than one millionth (0.000001) of an inch.

Computer numerical control of machines has undergone a great deal of change in the last decade, largely as a result of rapid increases in computer capability. Development of new and improved materials for tooling and bearings, improvements in tool geometry, and the added structural stiffness of the new machines have made it possible to perform cutting operations at speeds and feeds that were formerly impossible to attain.

Numerical Control vs. Manual Operations.—The initial cost of a CNC machine is generally much higher than a manual machine of the same nominal capacity, and the higher initial cost leads to a higher overall cost of the machine per hour of its useful life. However, the additional cost of a CNC machine has to be considered against potential savings that the machine may make possible. Some of the individual factors that make NC and CNC machining attractive are considered below.

Labor is usually one of the highest costs in the production of a part, but the labor rate paid to a CNC machine operator may be lower than the rate paid to the operator of conventional machines. This statement is particularly true when there is a shortage of operators with specialized skills necessary for setting up and operating a manual machine. However, it should not be assumed that skilled CNC machine operators are not needed because most CNCs have manual overrides that allow the operator to adjust feeds and speeds and to manually edit or enter programs as necessary. Also, skilled setup personnel and operators are likely to promote better production rates and higher efficiency in the shop. In addition, the labor rate for setting up and operating a CNC machine can sometimes be divided between two or more machines, further reducing the labor costs and cost per part produced.

The quantity and quality requirements for an order of parts often determines what manufacturing process will be used to produce them. CNC machines are probably most effective when the jobs call for a small to medium number of components that require a wide range of operations to be performed. For example, if a large number of parts are to be machined and the allowable tolerances are large, then manual or automatic fixed-cycle machines may be the most viable process. But, if a large quantity of high quality parts with strict tolerances are required, then a CNC machine will probably be able to produce the parts for the lowest cost per piece because of the speed and accuracy of CNC machines. Moreover, if the production run requires designing and making a lot of specialized form tools, cams, fixtures, or jigs, then the economics of CNC machining improves even more because much of the preproduction work is not required by the nature of the CNC process.

CNC machines can be effective for producing one-of-a-kind jobs if the part is complicated and requires a lot of different operations that, if done manually, would require specialized setups, jigs, fixtures, etc. On the other hand, a single component requiring only one or two setups might be more practical to produce on a manual machine, depending on the tolerances required. When a job calls for a small to medium number of components that require a wide range of operations, CNC is usually preferable. CNC machines are also especially well suited for batch jobs where small numbers of components are produced from an existing part program, as inventory is needed. Once the part program has been tested, a batch of the parts can be run whenever necessary. Design changes can be incorporated by changing the part program as required. The ability to process batches also has an additional benefit of eliminating large inventories of finished components.

CNC machining can help reduce machine idle time. Surveys have indicated that when machining on manual machines, the average time spent on material removal is only about 40 per cent of the time required to complete a part. On particularly complicated pieces, this ratio can drop to as low as 10 per cent or even less. The balance of the time is spent on positioning the tool or work, changing tools, and similar activities. On numerically controlled machines, the metal removal time frequently has been found to be in excess of 70 per cent of the total time spent on the part. CNC nonmachining time is lower because CNC machines perform quicker tool changes and tool or work positioning than manual machines. CNC part programs require a skilled programmer and cost additional preproduction time, but specialized jigs and fixtures that are frequently required with manual machines are not usually required with CNC machines, thereby reducing setup time and cost considerably.

Additional advantages of CNC machining are reduced lead time; improved cutting efficiency and longer tool life, as a result of better control over the feeds and speeds; improved quality and consistently accurate parts, reduced scrap, and less rework; lower inspection costs after the first part is produced and proven correct; reduced handling of parts because more operations can be performed per setup; and faster response to design changes because most part changes can be made by editing the CNC program.

Numerical Control Standards.—Standards for NC hardware and software have been developed by many organizations, and copies of the latest standards may be obtained from the following: Electronic Industries Association (EIA), 2001 Pennsylvania Avenue NW, Washington, DC 20006 (EIA and ANSI/EIA); American Society of Mechanical Engineers (ASME), 345 East 47th Street, New York, NY 10017 (ANSI/ASME); American National Standards Institute (ANSI), II West 42nd Street, New York, NY 10017 (ANSI, ANSI/EIA, ANSI/ASME, and ISO); National Standards Association, Inc. (NSA), 1200 Quince Orchard Boulevard, Gaithersburg, MD 20878; NMTBA The Association for Manufacturing Technology, 7901 Westpark Drive, McLean, VA 22102. Some of the standards and their contents are listed briefly in the accompanying table.

Numerical Control Standards

Standard Title	Description
ANSI/CAM-I 101-1990	Dimensional Measuring Interface Specification
ANSI/ASME B5.50	V-Flange Tool Shanks for Machining Centers with Automatic Tool Changers
ANSI/ASME B5.54-1992	Methods for Performance Evaluation of Computer Numerically Controlled Machining Centers
ANSI/ASME B89.1.12M	Methods for Performance Evaluation of Coordinate Measuring Machines
ANSI/EIA 227-A	1-inch Perforated Tape
ANSI/EIA 232-D	Interface Between Data Terminal Equipment and Data Circuit-Terminating Equipment Employing Serial Binary Data Interchange
ANSI/EIA 267-B	Axis and Motion Nomenclature for Numerically Controlled Machines
ANSI/EIA 274-D	Interchangeable Variable Block Data Format for Positioning, Contouring and Contouring/Positioning Numerically Controlled Machines
ANSI/EIA 358-B	Subset of American National Standarde Code for Information Interchange for Numerical Machine Control Perforated Tape
ANSI/EIA 408	Interface Between NC Equipment and Data Terminal Equipment Employing Parallel Binary Data Interchange
ANSI/EIA 423-A	Electrical Characteristics of Unbalanced Voltage Digital Interface Circuits
ANSI/EIA 431	Electrical Interface Between Numerical Control and Machine Tools
ANSI/EIA 441	Operator Interface Function of Numerical Controls
ANSI/EIA 449	General Purpose 37-position and 9-position Interface for Data Terminal Equipment and Data Circuit-Terminating Equipment Employing Serial Binary Data Interchange
ANSI/EIA 484	Electrical and Mechanical Interface Characteristics and Line Control Protocol Using Communication Control Characters for Serial Data Link between a Direct Numerical Control System and Numerical Control Equipment Employing Asynchronous Full Duplex Transmission
ANSI/EIA 491-A -1990	Interface between a Numerical Control Unit and Peripheral Equipment Employing Asynchronous Binary Data Interchange over Circuits having EIA-423-A Electrical Characteristics
ANSI/EIA 494	32-bit Binary CL Interchange (BCL) Input Format for Numerically Controlled Machines
EIA AB_3-D	Glossary of Terms for Numerically Controlled Machines
EIA Bulletin 12	Application Notes on Interconnection between Interface Circuits Using RS-449 and RS-232-C
ANSI X 3.94	Programming Aid for Numerically Controlled Manufacturing
ANSI X 3.37	Programming Language APT
ANSI X 3.20	1-inch Perforated Tape Take-up Reels for Information Interchange
ANSI X 3.82	One-sided Single Density Unformatted 5.25 inch Flexible Disc Cartridges

Numerical Control Standards *(Continued)*

Standard Title	Description
ISO 841	Numerical Control of Machines—Axis and Motion Nomenclature
ISO 2806	Numerical Control of Machines—Bilingual Vocabulary
ISO 2972	Numerical Control of Machines—Symbols
ISO 3592	Numerical Control of Machines—Numerical Control Processor Output, Logical Structure and Major Words
ISO 4336	Numerical Control of Machines—Specification of Interface Signals between the Numerical Control Unit and the Electrical Equipment of a Numerically Controlled Machine
ISO 4343	Numerical Control of Machines—NC Processor Output— Minor Elements of 2000-type Records (Post Processor Commands)
ISO TR 6132	Numerical Control of Machines—Program Format and Definition of Address Words—Part 1: Data Format for Positioning, Line Motion and Contouring Control Systems
ISO 230-1	Geometric Accuracy of Machines Operating Under No-Load or Finishing Conditions
ISO 230-2	Determination of Accuracy and Repeatability of Positioning of Numerically Controlled Machine Tools
NAS 911	Numerically Controlled Skin/Profile Milling Machines
NAS 912	Numerically Controlled Spar Milling Machines
NAS 913	Numerically Controlled Profiling and Contouring Milling Machines
NAS 914	Numerically Controlled Horizontal Boring, Drilling and Milling Machines
NAS 960	Numerically Controlled Drilling Machines
NAS 963	Computer Numerically Controlled Vertical and Horizontal Jig Boring Machines
NAS 970	Basic Tool Holders for Numerically Controlled Machine Tools
NAS 971	Precision Numerically Controlled Measuring/Inspection Machines
NAS 978	Numerically Controlled Machining Centers
NAS 990	Numerically Controlled Composite Filament Tape Laying Machines
NAS 993	Direct Numerical Control System
NAS 994	Adaptive Control System for Numerically Controlled Milling Machines
NAS 995	Specification for Computerized Numerical Control (CNC)
NMTBA	Common Words as They Relate to Numerical Control Software
NMTBA	Definition and Evaluation of Accuracy and Repeatability of Numerically Controlled Machine Tools
NMTBA	Numerical Control Character Code Cross Reference Chart
NMTBA	Selecting an Appropriate Numerical Control Programming Method
NEMA 1A1	Industrial Cell Controller Classification Concepts and Selection Guide

Programmable Controller.—Frequently referred to as a PC or PLC (the latter term meaning Programmable Logic Controller), a programmable controller is an electronic unit or small computer. PLCs are used to control machinery, equipment, and complete processes, and to assist CNC systems in the control of complex NC machine tools and flexible manufacturing modules and cells. In effect, PLCs are the technological replacements for electrical relay systems.

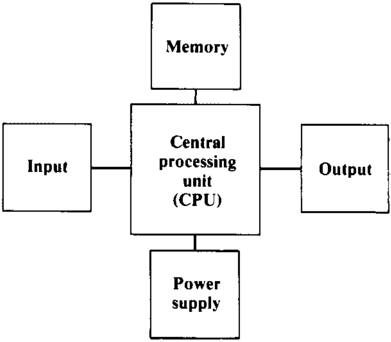


Fig. 1. Programmable Controllers' Four Basic Elements

As shown in Fig. 1, a PLC is composed of four basic elements: the equipment for handling input and output (I/O) signals, the central processing unit (CPU), the power supply, and the memory. Generally, the CPU is a microprocessor and the brain of the PLC. Early PLCs used hardwired special-purpose electronic logic circuits, but most PLCs now being offered are based on microprocessors and have far more logic and control capabilities than was possible with hardwired systems. The CPU scans the status of the input devices continuously, correlates these inputs with the control logic in the memory, and produces the appropriate output responses needed to control the machine or equipment.

Input to a PLC is either discrete or continuous. Discrete inputs may come from push buttons, micro switches, limit switches, photocells, proximity switches or pressure switches, for instance. Continuous inputs may come from sources such as thermocouples, potentiometers, or voltmeters. Outputs from a PLC normally are directed to actuating hardware such as solenoids, solenoid valves, and motor starters. The function of a PLC is to examine the status of an input or set of inputs and, based on this status, actuate or regulate an output or set of outputs.

Digital control logic and sensor input signals are stored in the memory as a series of binary numbers (zeros and ones). Each memory location holds only one “bit” (either 0 or 1) of binary information; however, most of the data in a PLC are used in groups of 8 bits, or bytes. A word is a group of bytes that is operated on at one time by the PLC. The word size in modern PLCs ranges from 8 to 32 bits (1 to 4 bytes), depending on the design of the PLC. In general, the larger the word size that a system is able to operate on (that is, to work on at one time), the faster the system is going to perform. New systems are now beginning to appear that can operate on 64 bits of information at a time.

There are two basic categories of memory: volatile and nonvolatile. Volatile memory loses the stored information when the power is turned off, but nonvolatile memory retains its logic even when power is cut off. A backup battery must be used if the information stored in volatile memory is to be retained. There are six commonly used types of memory. Of these six, random-access memory (RAM) is the most common type because it is the easiest to program and edit. RAM is also the only one of the six common types that is volatile memory. The five nonvolatile memory types are: core memory, read-only memory (ROM), programmable read-only memory (PROM), electronically alterable programmable read-only memory (EAPROM), and electronically erasable programmable read-only memory (EEPROM). EEPROMs are becoming more popular due to their relative ease of programming and their nonvolatile characteristic. ROM is often used as a generic term to refer to the general class of read-only memory types and to indicate that this type of memory is not usually reprogrammed.

More than 90 per cent of the microprocessor PLCs now in the field use RAM memory. RAM is primarily used to store data, which are collected or generated by a process, and to store programs that are likely to change frequently. For example, a part program for machining a workpiece on a CNC machining center is loaded into and stored in RAM. When a different part is to be made, a different program can be loaded in its place. The nonvolatile memory types are usually used to store programs and data that are not expected to be changed. Programs that directly control a specific piece of equipment and contain specific instructions that allow other programs (such as a part program stored in RAM) to access and operate the hardware are usually stored in nonvolatile memory or ROM. The benefit of ROM is that stored programs and data do not have to be reloaded into the memory after the power has been turned off.

PLCs are used primarily with handling systems such as conveyors, automatic retrieval and storage systems, robots, and automatic guided vehicles (AGV), such as are used in flexible manufacturing cells, modules, and systems (see *Flexible Manufacturing Systems (FMS)*, *Flexible Manufacturing Cell*, and *Flexible Manufacturing Module*). PLCs are also to be found in applications as diverse as combustion chamber control, chemical process control, and printed-circuit-board manufacturing.

Types of Programmable Controllers

Type	No. of I/Os	General Applications	Math Capability
Mini	32	Replaces relays, timers, and counters.	Yes
Micro	32–64	Replaces relays, timers, and counters.	Yes
Small	64–128	Replaces relays, timers, and counters. Used for materials handling, and some process control.	Yes
Medium	128–512	Replaces relays, timers, and counters. Used for materials handling, process control, and data collection.	Yes
Large	512+	Replaces relays, timers, and counters. Master control for other PLCs and cells and for generation of reports. High-level network capability	Yes

Types of PLCs may be divided into five groups consisting of micro, mini, small, medium, and large according to the number of I/Os, functional capabilities, and memory capacity. The smaller the number of I/Os and memory capacity, and the fewer the functions, the simpler the PLC. Micro and mini PLCs are usually little more than replacements for relay systems, but larger units may have the functional capabilities of a small computer and be able to handle mathematical functions, generate reports, and maintain high-level communications.

The preceding guidelines have some gray areas because mini, micro, and small PLCs are now available with large memory sizes and functional capacities normally reserved for medium and large PLCs. The accompanying table compares the various types of PLCs and their applications.

Instructions that are input to a PLC are called programs. Four major programming languages are used with PLCs, comprising ladder diagrams, Boolean mnemonics, functional blocks, and English statements. Some PLC systems even support high-level programming languages such as BASIC and PASCAL. Ladder diagrams and Boolean mnemonics are the basic control-level languages. Functional blocks and English statements are considered high-level languages. Ladder diagrams were used with electrical relay systems before these systems were replaced by PLCs and are still the most popular programming method, so they will be discussed further.

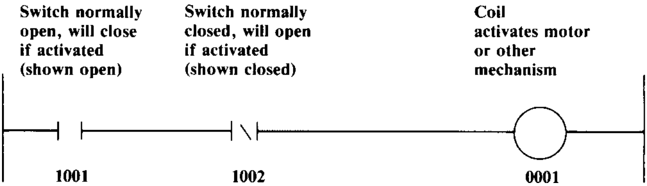


Fig. 2. One Rung on a Ladder Diagram

A ladder diagram consists of symbols, or ladder logic elements, that represent relay contacts or switches and other elements in the control system. One of the more basic symbols represents a normally open switch and is described by the symbol `1/`. Another symbol is the normally closed switch, described by the symbol `1\/`. When the normally open switch is activated, it will close, and when the normally closed switch is activated, it will open. Fig. 2 shows one rung (line) on a ladder diagram. Switch 1001 is normally open and switch 1002 is closed. A symbol for a coil (0001) is shown at the right. If switch 1001 is actuated, it will close. If switch 1002 is not activated, it will stay closed. With the two switches closed, current will flow through the line and energize coil 0001. The coil will activate some mechanism such as an electric motor, a robot, or an NC machine tool, for instance.

As an example, Fig. 3 shows a flexible manufacturing module (FMM), consisting of a turning center (NC lathe), an infeed conveyor, an outfeed conveyor, a robot that moves workpieces between the infeed conveyor, the turning center, and the outfeed conveyor, and a PLC. The arrowed lines show the signals going to and coming from the PLC.

Fig. 4 shows a ladder diagram for a PLC that would control the operations of the FMM by:

1) Activating the infeed conveyor to move the workpiece to a position where the robot can pick it up

2) Activating the robot to pick up the workpiece and load it into the chuck on the NC lathe

3) Activating the robot to remove the finished workpiece and place it on the outfeed conveyor

4) Activating the outfeed conveyor to move the workpiece to the next operation

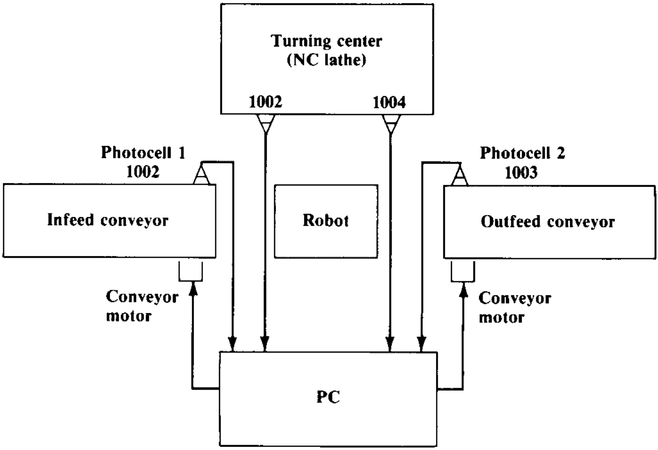


Fig. 3. Layout of a Flexible Manufacturing Module

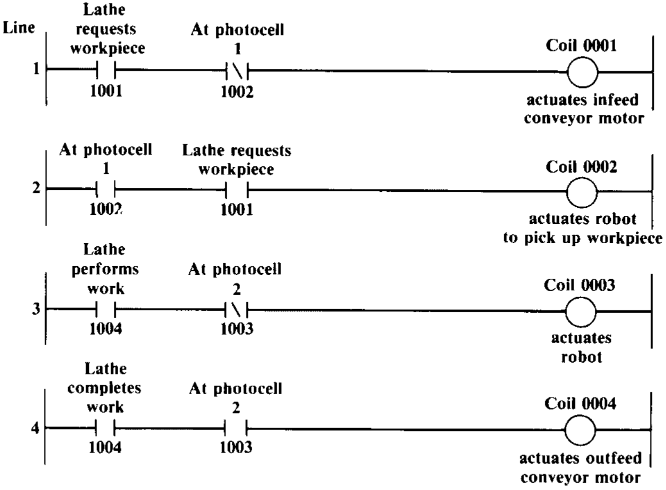


Fig. 4. Portion of a Typical Ladder Diagram for Control of a Flexible Manufacturing Module Including a Turning Center, Conveyors, a Robot, and a Programmable Controller

In Rung 1 of Fig. 4, a request signal for a workpiece from the NC lathe closes the normally open switch 1001. Switch 1002 will remain closed if photocell 1 is not activated, i.e., if it does not detect a workpiece. The signal therefore closes the circuit, energizes the coil, and starts the conveyor motor to bring the next workpiece into position for the robot to grasp.

In Rung 2, switch 1002 (which has been changed in the program of the PLC from a normally closed to a normally open switch) closes when it is activated as photocell 1 detects the workpiece. The signal thus produced, together with the closing of the now normally open switch 1001, energizes the coil, causing the robot to pick up the workpiece from the infeed conveyor.

In Rung 3, switch 1004 on the lathe closes when processing of the part is completed and it is ready to be removed by the robot. Photocell 2 checks to see if there is a space on the conveyor to accept the completed part. If no part is seen by photocell 2, switch 1003 will remain closed, and with switch 1004 closed, the coil will be energized, activating the robot to transfer the completed part to the outfeed conveyor.

Rung 4 shows activation of the output conveyor when a part is to be transferred. Normally open switch 1004 was closed when processing of the part was completed. Switch 1003 (which also was changed from a normally closed to a normally open switch by the program) closes if photocell 2 detects a workpiece. The circuit is then closed and the coil is energized, starting the conveyor motor to move the workpiece clear to make way for the succeeding workpiece.

Closed-Loop System.—Also referred to as a servo or feedback system, a closed-loop system is a control system that issues commands to the drive motors of an NC machine. The system then compares the results of these commands as measured by the movement or location of the machine component, such as the table or spindlehead. The feedback devices normally used for measuring movement or location of the component are called resolvers, encoders, Inductosyns, or optical scales. The resolver, which is a rotary analog mechanism, is the least expensive, and has been the most popular since the first NC machines were developed. Resolvers are normally connected to the lead-screws of NC machines. Linear measurement is derived from monitoring the angle of rotation of the leadscrew and is quite accurate.

Encoders also are normally connected to the leadscrew of the NC machine, and measurements are in digital form. Pulses, or a binary code in digital form, are generated by rotation of the encoder, and represent turns or partial turns of the leadscrew. These pulses are well suited to the digital NC system, and encoders have therefore become very popular with such systems. Encoders generally are somewhat more expensive than resolvers.

The Inductosyn (a trade name of Farrand Controls, Inc.) also produces analog signals, but is attached to the slide or fixed part of a machine to measure the position of the table, spindlehead, or other component. The Inductosyn provides almost twice the measurement accuracy of the resolver, but is considerably more expensive, depending on the length of travel to be measured.

Optical scales generally produce information in digital form and, like the Inductosyn, are attached to the slide or fixed part of the machine. Optical scale measurements are more accurate than either resolvers or encoders and, because of their digital nature, are well suited to the digital computer in a CNC system. Like the Inductosyn, optical scales are more costly than either resolvers or encoders.

Open-Loop System.—A control system that issues commands to the drive motors of an NC machine and has no means of assessing the results of these commands is known as an open-loop system. In such a system, no provision is made for feedback of information concerning movement of the slide(s), or rotation of the leadscrew(s). Stepping motors are popular as drives for open-loop systems.

Adaptive Control.—Measuring performance of a process and then adjusting the process to obtain optimum performance is called adaptive control. In the machine tool field, adaptive control is a means of adjusting the feed and/or speed of the cutting tool, based on sensor feedback information, to maintain optimum cutting conditions. A typical arrangement is seen in Fig. 5. Adaptive control is used primarily for cutting higher-strength materials such as titanium, although the concept is applicable to the cutting of any material. The costs of the sensors and software have restricted wider use of the feature.

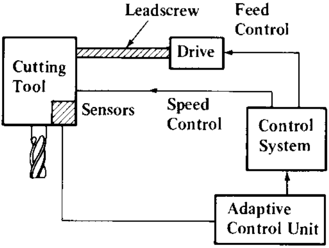


Fig. 5.

The sensors used for adaptive control are generally mounted on the machine drive shafts, tools, or even built into the drive motor. Typically, sensors are used to provide information such as the temperature at the tip of the cutting tool and the cutting force exerted by the tool. The information measured by the sensors is used by the control system computer to analyze the cutting process and adjust the feeds and speeds of the machine to maximize the material removal rate or to optimize another process variable such as surface finish. For the computer to effectively evaluate the process in real time (i.e., while cutting is in progress), details such as maximum allowable tool temperature, maximum allowable cutting force, and information about the drive system need to be integrated into the computer program monitoring the cutting process.

Adaptive control can be used to detect worn, broken, or dull tooling. Ordinarily, the adaptive control system monitors the cutting process to keep the process variables (cutting speed and feed rate, for example) within the proper range. Because the force required to machine a workpiece is lowest when the tool is new or recently resharpened, a steady increase in cutting force during a machining operation, assuming that the feed remains the same, is an indication that the tool is becoming dull (temperature may increase as well). Upon detecting cutting forces that are greater than a predetermined maximum allowable force, the control system causes the feed rate, the cutting speed, or both to be adjusted to maintain the cutting force within allowable limits. If the cutting force cannot be maintained without causing the speed and/or feed rate to be adjusted outside its allowable limits, the machine will be stopped, indicating that the tool is too dull and must be resharpened or replaced.

On some systems, the process monitoring equipment can interface directly with the machine control system, as discussed above. On other systems, the adaptive control is implemented by a separate monitoring system that is independent of the machine control system. These systems include instrumentation to monitor the operations of the machine tool, but do not have the capability to directly change operating parameters, such as feeds and speeds. In addition, this type of control does not require any modification of the existing part programs for control of the machine.

Flexible Manufacturing Systems (FMS).—A flexible manufacturing system (FMS) is a computer-controlled machining arrangement that can perform a variety of continuous metal-cutting operations on a range of components without manual intervention. The objective of such a system is to produce components at the lowest possible cost, especially components of which only small quantities are required. Flexibility, or the ability to switch from manufacture of one type of component to another, or from one type of machining to another, without interrupting production, is the prime requirement of such a system. In general, FMS are used for production of numbers of similar parts between 200 and 2000, although larger quantities are not uncommon. An FMS involves almost all the departments in a company, including engineering, methods, tooling and part programming, planning and scheduling, purchasing, sales and customer service, accounting, maintenance, and quality control. Initial costs of an FMS are estimated as being borne (percentages in parentheses) by machine tools (46.2), materials handling systems (7.7), tooling and fixtures (5.9), pallets (1.9), computer hardware (3.7), computer software (2.2), wash stations (2.8), automatic storage and retrieval systems (6.8), coolant and chip systems (2.4), spares (2), and others (18.4).

FMS are claimed to bring reductions in direct labor (80–90), production planning and control (65), and inspection (70). Materials handling and shop supervision are reduced, and individual productivity is raised. In the materials field, savings are made in tooling (35), scrap and rework (65), and floor space (50). Inventory is reduced and many other costs are avoided. Intangible savings claimed to result from FMS include reduced tooling changeover time, ability to produce complex parts, to incorporate engineering changes more quickly and efficiently than with other approaches, and to make special designs, so that a company can adapt quickly to changing market conditions. Requirements for spare parts with good fit are easily met, and the lower costs combine with higher quality to improve market share. FMS also are claimed to improve morale among workers, leading to higher productivity, with less paper work and more orderly shop operations. Better control of costs and improved cost data help to produce more accurate forecasts of sales and manpower requirements. Response to surges in demand and more economical materials ordering are other advantages claimed with FMS.

Completion of an FMS project is said to average 57 months, including 20 months from the time of starting investigations to the placing of the purchase order. A further 13 months are needed for delivery and a similar period for installation. Debugging and building of production takes about another 11 months before production is running smoothly. FMS are expensive, requiring large capital outlays and investments in management time, software, engineering, and shop support. Efficient operation of FMS also require constant workflow because gaps in the production cycle are very costly.

Flexible Manufacturing Cell.—A flexible manufacturing cell usually consists of two or three NC machines with some form of pallet-changing equipment or an industrial robot. Prismatic-type parts, such as would be processed on a machining center, are usually handled on pallets. Cylindrical parts, such as would be machined on an NC lathe, usually are handled with an overhead type of robot. The cell may be controlled by a computer, but is often run by programmable controllers. The systems can be operated without attendants, but the mixture of parts usually must be less than with a flexible manufacturing system (FMS).

Flexible Manufacturing Module.—A flexible manufacturing module is defined as a single machining center (or turning center) with some type of automatic materials handling equipment such as multiple pallets for machining centers, or robots for manipulating cylindrical parts and chucks for turning centers. The entire module is usually controlled by one or more programmable logic controllers.

Axis Nomenclature.—To distinguish among the different motions, or axes, of a machine tool, a system of letter addresses has been developed. A letter is assigned, for example, to the table of the machine, another to the saddle, and still another to the spindle head. These letter addresses, or axis designations, are necessary for the electronic control system to assign movement instructions to the proper machine element. The assignment of these letter addresses has been standardized on a worldwide basis and is contained in three standards, all of which are in agreement. These standards are EIA RS-267-B, issued by the Electronics Industries Association; AIA NAS-938, issued by the Aerospace Industries Association; and ISO/R 841, issued by the International Organization for Standardization.

The standards are based on a “right-hand rule,” which describes the orientation of the motions as well as whether the motions are positive or negative. If a right hand is laid palm up on the table of a vertical milling machine, as shown in Fig. 1, for example, the thumb will point in the positive *X*-direction, the forefinger in the positive *Y*-direction, and the erect middle finger in the positive *Z*-direction, or up. The direction signs are based on the motion of the cutter relative to the workpiece. The movement of the table shown in Fig. 2 is therefore positive, even though the table is moving to the left, because the motion of the cutter relative to the workpiece is to the right, or in the positive direction. The motions are considered from the part programmer's viewpoint, which assumes that the cutter always moves around the part, regardless of whether the cutter or the part moves. The right-hand rule also holds with a horizontal-spindle machine and a vertical table, or angle plate, as shown in Fig. 3. Here, spindle movement back and away from the angle plate, or workpiece, is a positive *Z*-motion, and movement toward the angle plate is a negative *Z*-motion.

Rotary motions also are governed by a right-hand rule, but the fingers are joined and the thumb is pointed in the positive direction of the axis. Fig. 4 shows the designations of the rotary motions about the three linear axes, *X*, *Y*, and *Z*. Rotary motion about the *X*-axis is designated as *A*; rotary motion about the *Y*-axis is *B*; and rotary motion about the *Z*-axis is *C*. The fingers point in the positive rotary directions. Movement of the rotary table around the *Y*-axis shown in Fig. 4 is a *B* motion and is common with horizontal machining centers. Here, the view is from the spindle face looking toward the rotary table. Referring, again, to linear motions, if the spindle is withdrawn axially from the work, the motion is a positive *Z*. A move toward the work is a negative *Z*.

When a second linear motion is parallel to another linear motion, as with the horizontal boring mill seen in Fig. 5, the horizontal motion of the spindle, or quill, is designated as *Z* and a parallel motion of the angle plate is *W*. A movement parallel to the *X*-axis is *U* and a movement parallel to the *Y*-axis is *V*. Corresponding motions are summarized as follows:

Linear	Rotary	Linear and Parallel
X	*A*	*U*
Y	*B*	*V*
Z	*C*	*W*

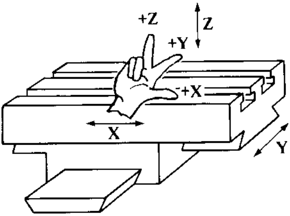


Fig. 1.

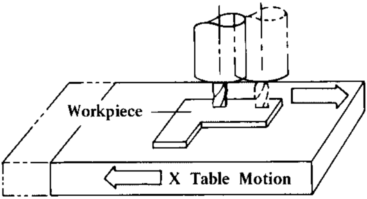


Table movement to the left in effect moves the cutter to the right, relative to the workpiece, or in the + direction

Fig. 2.

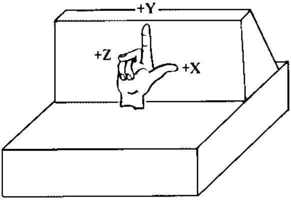


Fig. 3.

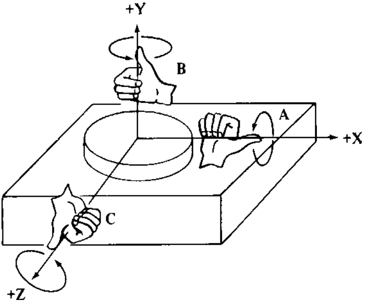


Fig. 4.

Axis designations for a lathe are shown in Fig. 6. Movement of the cross-slide away from the workpiece, or the centerline of the spindle, is noted as a plus *X*. Movement toward the workpiece is a minus *X*. The middle finger points in the positive *Z*-direction; therefore, movement away from the headstock is positive and movement toward the headstock is negative. Generally, there is no *Y*-movement.

The machine shown in Fig. 6 is of conventional design, but most NC lathes look more like that shown in Fig. 7. The same right-hand rule applies to this four-axis lathe, on which each turret moves along its own two independent axes. Movement of the outside-diameter or upper turret, up and away from the workpiece, or spindle centerline, is a positive *X*-motion, and movement toward the workpiece is a negative *X*-motion. The same rules apply to the *U*-movement of the inside-diameter, or boring, turret. Movement of the lower turret parallel to the *Z*-motion of the outside-diameter turret is called the *W*-motion. A popular lathe configuration is to have both turrets on one slide, giving a two-axis system rather than the four-axis system shown. *X*-and *Z*-motions may be addressed for either of the two heads. Upward movement of the boring head therefore is a positive *X*-motion.

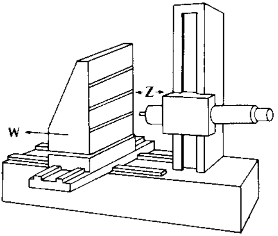


Fig. 5.

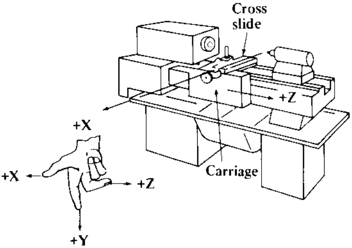


Fig. 6.

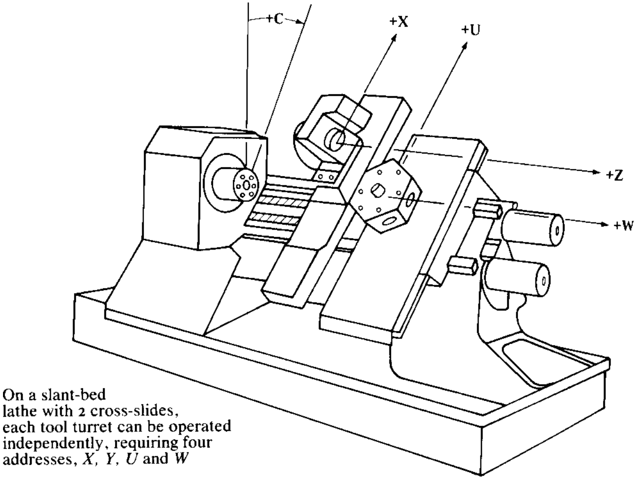


Fig. 7.

Axis nomenclature for other machine configurations is shown in Fig. 9. The letters with the prime notation (e.g., X', Y', Z', W', A', and B') mean that the motion shown is positive, because the movement of the cutter with respect to the work is in a positive direction. In these instances, the workpiece is moving rather than the cutter.

Total Indicator Reading (TIR).—Total indicator reading is used as a measure of the range of machine tool error. TIR is particularly useful for describing the error in a machine tool spindle, referred to as runout. As shown in Fig. 8, there are two types of runout: axial and radial, which can be measured with a dial indicator. Axial runout refers to the wobble of a spindle and is measured at the spindle face. Radial runout is the range of movement of the spindle centerline and is measured on the side of the spindle or quill.

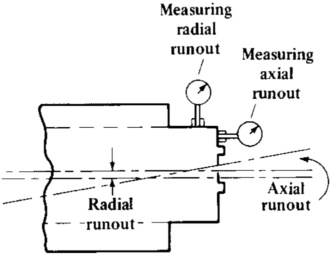
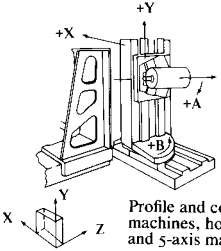
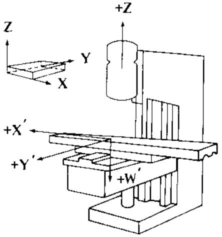


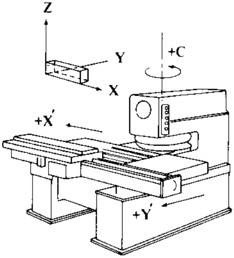
Fig. 8.



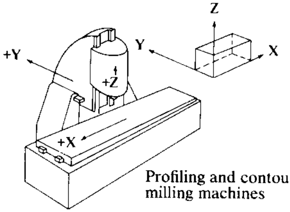
Profile and contour milling machines, horizontal spindle and 5-axis machines



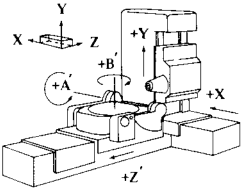
Vertical knee-type milling, drilling, or jig-boring machines



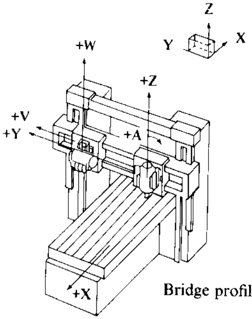
Turret type punch presses



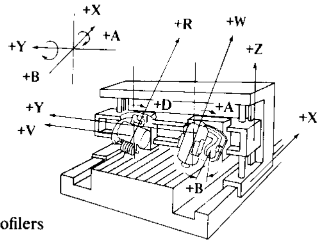
Profiling and contour milling machines



Tilting table, profile and contour milling and 5-axis machines



Bridge profilers



Gantry profilers

Fig. 9.

NUMERICAL CONTROL PROGRAMMING

Programming.—A numerical control (NC) program is a list of instructions (commands) that completely describes, in sequence, every operation to be carried out by a machine. When a program is run, each instruction is interpreted by the machine controller, which causes an action such as starting or stopping of a spindle or coolant, changing of spindle speed or rotation, or moving a table or slide a specified direction, distance, or speed. The form that program instructions can take, and how programs are stored and/or loaded into the machine, depends on the individual machine/control system. However, program instructions must be in a form (language) that the machine controller can understand.

A programming language is a system of symbols, codes, and rules that describes the manner in which program instructions can be written. One of the earliest and most widely recognized numerical control programming languages is based on the Standard ANSI/EIA RS-274-D-1980. The standard defines a recommended data format and codes for sending instructions to machine controllers. Although adherence to the standard is not mandatory, most controller manufacturers support it and most NC machine controllers (especially controllers on older NC machines using tape input) can accept data in a format that conforms, at least in part, with the recommended codes described in the RS-274-D standard. Most newer controllers also accept instructions written in proprietary formats offered (specified) by the controller's manufacturer.

One of the primary benefits of a standardized programming format is easy transfer of programs from one machine to another, but even standardized code formats such as RS-274-D are implemented differently on different machines. Consequently, a program written for one machine may not operate correctly on another machine without some modification of the program. On the other hand, proprietary formats are attractive because of features that are not available using the standardized code formats. For example, a proprietary format may make available certain codes that allow a programmer, with only a few lines of code, to program complex motions that would be difficult or even impossible to do in the standard language. The disadvantage of proprietary formats is that transferring programs to another machine may require a great deal of program modification or even complete rewriting. Generally, with programs written in a standardized format, the modifications required to get a program written for one machine to work on another machine are not extensive.

In programming, before describing the movement of any machine part, it is necessary to establish a coordinate system(s) as a reference frame for identifying the type and direction of the motion. A description of accepted terminology used worldwide to indicate the types of motion and the orientation of machine axes is contained in a separate section (Axis Nomenclature). Part geometry is programmed with reference to the same axes as are used to describe motion.

Manual data input (MDI) permits the machine operator to insert machining instructions directly into the NC machine control system via push buttons, pressure pads, knobs, or other arrangements. MDI has been available since the earliest NC machines were designed, but the method was less efficient than tape for machining operations and was used primarily for setting up the NC machine. Computer numerical control (CNC) systems, with their canned cycles and other computing capabilities, have now made the MDI concept more feasible and for some work MDI may be more practical than preparing a program. The choice depends very much on the complexity of the machining work to be done and, to a lesser degree, on the skill of the person who prepares the program.

Conversational part programming is a form of MDI that requires the operator or programmer to answer a series of questions displayed on the control panel of the CNC. The operator replies to questions that describe the part, material, tool and machine settings, and machining operations by entering numbers that identify the material, blank size and thickness or diameter, tool definitions, and other required data. Depending on capability, some controls can select the required spindle speed and feed rate automatically by using a materials look-up table; other systems request the appropriate feed and speed data. Tool motions needed to machine a part are described by selecting a linear or circular motion programming mode and entering endpoint and intersection coordinates of lines and radius, diameter, tangent points, and directions of arcs and circles (with some controllers, intersection and tangent points are calculated automatically). Machined elements such as holes, slots, and bolt circles are entered by selecting the appropriate tool and describing its action, or with "canned routines" built into the CNC to perform specific machining operations. On some systems, if a feature is once described, it can be copied and/or moved by: translation (copy and/or move), rotation about a point, mirror image (copy and rotate about an axis), and scaling (copy and change size). On many systems, as each command is entered, a graphic image of the part or operation gives a visual check that the program is producing the intended results. When all the necessary data have been entered, the program is constructed and can be run immediately or saved on tape, floppy disk, or other storage media for later use.

Conversational programming gives complete control of machine operations to the shop personnel, taking advantage of the experience and practical skills of the machine operator/programmer. Control systems that provide conversational programming usually include many built-in routines (fixed or canned cycles) for commonly used machining operations and may also have routines for specialized operations. Built-in routines speed programming because one command may replace many lines of program code that would take considerable time to write. Some built-in cycles allow complex machining operations to be programmed simply by specifying the final component profile and the starting stock size, handling such details as developing tool paths, depth of cut, number of roughing passes, and cutter speed automatically. On turning machines, built-in cycles for reducing diameters, chamfer and radius turning, and cutting threads automatically are common. Although many CNC machines have a conversational programming mode, the programming methods used and the features available are not standardized. Some control systems cannot be programmed from the control panel while another program is running (i.e., while a part is being machined), but those systems that can be thus programmed are more productive because programming does not require the machine to be idle. Conversational programming is especially beneficial In reducing programming time in shops that do most of their part programming from the control panel of the machine.

Manual part programming describes the preparation of a part program by manually writing the part program in word addressed format. In the past, this method implied programming without using a computer to determine tool paths, speeds and feeds, or any of the calculations normally required to describe the geometry of a part. Today, however, computers are frequently used for writing and storing the program on disk, as well as for calculations required to program the part. Manual part programming consists of writing codes, in a format appropriate to the machine controller, that instruct the controller to perform a specific action. The most widely accepted form of coding the instructions for numerically controlled machines uses the codes and formats suggested in the ANSI/EIA RS-274-D-1980, standard. This type of programming is sometimes called G-code programming, referring to a commonly used word address used in the RS-274-D standard. Basic details of programming in this format, using the various codes available, are discussed in the next section (G-Code Programming).

Computer-assisted part programming (CAPP) uses a computer to help in the preparation of the detailed instructions for operating an NC machine. In the past, defining a curve or complicated surface profile required a series of complex calculationsto describe the features in intimate detail. However, with the introduction of the microprocessor as an integral part of the CNC machine, the process of defining many complex shapes has been reduced to the simple task of calling up a canned cycle to calculate the path of the cutter. Most new CNC systems have some graphic programming capability, and many use graphic images of the part "drawn" on a computer screen. The part programmer moves a cutter about the part to generate the part program or the detailed block format instructions required by the control system. Machining instructions, such as the speed and feed rate, are entered via the keyboard. Using the computer as an assistant is faster and far more accurate than the manual part programming method.

Computer-assisted part programming methods generally can be characterized as either language-based or graphics-based, the distinction between the two methods being primarily in the manner by which the tool paths are developed. Some modern-language-based programming systems, such as Compact II, use interactive alphanumeric input so that programming errors are detected as soon as they are entered. Many of these programming systems are completely integrated with computer graphics and display an image of the part or operation as soon as an instruction is entered. The language-based programming systems are usually based on, or are a variation of, the APT programming language, which is discussed separately within this section (APT Programming).

The choice between computer-assisted part programming and manual part programming depends on the complexity of the part (particularly its geometry) and how many parts need to be programmed. The more complicated the part, the more benefit to be gained by CAPP, and if many parts are to be programmed, even if they are simple ones, the benefits of a computer-aided system are substantial. If the parts are not difficult to program but involve much repetition, computer-assisted part programming may also be preferred. If parts are to be programmed for several different control systems, a high-level part programming language such as APT will make writing the part programs easier. Because almost all machines have some deviations from standard practices, and few control systems use exactly the same programming format, a higher-level language allows the programmer to concentrate primarily on part geometry and machining considerations. The postprocessors (see *Postprocessors* below) for the individual control systems accommodate most of the variations in the programming required. The programmer only needs to write the program; the postprocessor deals with the machine specifics.

Graphical programming involves building a two- or three-dimensional model of a part on a computer screen by graphically defining the geometric shapes and surfaces of the part using the facilities of a CAD program. In many cases, depending on features of the CAD software package, the same computer drawing used in the design and drafting stage of a project can also be used to generate the program to produce the part. The graphical entities, such as holes, slots, and surfaces, are linked with additional information required for the specific machining operations needed. Most of the cutter movements (path of the cutter), such as those needed for the generation of pockets and lathe roughing cuts, are handled automatically by the computer. The program may then sort the various machining operations into an efficient sequence so that all operations that can be performed with a particular tool are done together, if possible. The output of graphical part programming is generally an alphanumeric part programming language output file, in a format such as an APT or Compact II file.

The part programming language file can be manually checked, and modified, as necessary before being run, and to help detect errors, many graphics programming systems also include some form of part verification software that simulates machining the part on the computer screen. Nongraphic data, such as feed rates, spindle speeds and coolant on/off, must be typed in by the part programmer or entered from acomputer data base at the appropriate points in the program, although some programs prompt for this information when needed. When the part program language file is run or compiled, the result is a center line data (CL data) file describing the part. With most computer-aided part programming output files, the CL data file needs to be processed through a postprocessor (see *Postprocessors* below) to tailor the final code produced to the actual machine being used. Postprocessor output is in a form that can be sent directly to the control system, or can be saved on tape or magnetic media and transferred to the machine tool when necessary. The graphic image of the part and the alphanumeric output files are saved in separate files so that either can be edited in the future if changes in the part become necessary. Revised files must be run and processed again for the part modifications to be included in the part program. Software for producing part programs is discussed further in the CAD/CAM section.

Postprocessors.—A postprocessor is computer software that contains a set of computer instructions designed to tailor the cutter center line location data (CL data), developed by a computerized part programming language, to meet the requirements of a particular machine tool/system combination. Generally, when a machine tool is programmed in a graphical programming environment or any high-level language such as APT, a file is created that describes all movements required of a cutting tool to make the part. The file thus created is run, or compiled, and the result is a list of coordinates (CL data) that describes the successive positions of the cutter relative to the origin of the machine's coordinate system. The output of the program must be customized to fit the input requirements of the machine controller that will receive the instructions. Cutter location data must be converted into a format recognized by the control system, such as G codes and M codes, or into another language or proprietary format recognized by the controller. Generally, some instructions are also added or changed by the programmer at this point.

The lack of standardization among machine tool control systems means that almost all computerized part programming languages require a postprocessor to translate the computer-generated language instructions into a form that the machine controller recognizes. Postprocessors are software and are generally prepared for a fee by the machine tool builder, the control system builder, a third party vendor, or by the user.

G-Code Programming

Programs written to operate numerical control (NC) machines with control systems that comply with the ANSI/EIA RS-274-D-1980, Standard consist of a series of data blocks, each of which is treated as a unit by the controller and contains enough information for a complete command to be carried out by the machine. Each block is made up of one or more words that indicate to the control system how its corresponding action is to be performed. A word is an ordered set of characters, consisting of a letter plus some numerical digits, that triggers a specific action of a machine tool. The first letter of the word is called the letter address of the word, and is used to identify the word to the control system. For example, X is the letter address of a dimension word that requires a move in the direction of the X-axis, Y is the letter address of another dimension word; and F is the letter address of the feed rate. The assigned letter addresses and their meanings, as listed in ANSI/EIA RS-274-D, are shown in Table 1.

Format Classification.—The *format classification sheet* completely describes the format requirements of a control system and gives other important information required to program a particular control including: the type of machine, the format classification shorthand and format detail, a listing of specific letter address codes recognized by the system (for example, G-codes: G01, G02, G17, etc.) and the range of values the available codes may take (S range: 10 to 1800 rpm, for example), an explanation of any codes not specifically assigned by the Standard, and any other unique features of the system.

The format classification shorthand is a nine- or ten-digit code that gives the type of system, the number of motion and other words available, the type and format of dimensional data required by the system, the number of motion control channels, and the number of numerically controlled axes of the system. The *format detail* verysuccinctly summarizes details of the machine and control system. This NC shorthand gives the letter address words and word lengths that can be used to make up a block. The format detail defines the basic features of the control system and the type of machine tool to which it refers. For example, the format detail

Table 1. Letter Addresses Used in Numerical Control

Letter Address	Description	Refers to
A	Angular dimension about the *X*-axis. Measured in decimal parts of a degree	Axis nomenclature
B	Angular dimension about the *Y*-axis. Measured in decimal parts of a degree	Axis nomenclature
C	Angular dimension about the *Z*-axis. Measured in decimal parts of a degree	Axis nomenclature
D	Angular dimension about a special axis, or third feed function, or tool function for selection of tool compensation	Axis nomenclature
E	Angular dimension about a special axis or second feed function	Axis nomenclature
F	Feed word (code)	Feed words
G	Preparatory word (code)	Preparatory words
H	Unassigned	
I	Interpolation parameter or thread lead parallel to the *X*-axis	Circular interpolation and threading
J	Interpolation parameter or thread lead parallel to the *Y*-axis	Circular interpolation and threading
K	Interpolation parameter or thread lead parallel to the *Z*-axis	Circular interpolation and threading
L	Unassigned	
M	Miscellaneous or auxilliary function	Miscellaneous functions
N	Sequence number	Sequence number
O	Sequence number for secondary head only	Sequence number
P	Third rapid-traverse dimension or tertiary-motion dimension parallel to *X*	Axis nomenclature
Q	Second rapid-traverse dimension or tertiary-motion dimension parallel to *Y*	Axis nomenclature
R	First rapid-traverse dimension or tertiary-motion dimension parallel to *Z* or radius for constant surface-speed calculation	Axis nomenclature
S	Spindle-speed function	Spindle speed
T	Tool function	Tool function
U	Secondary-motion dimension parallel to *X*	Axis nomenclature
V	Secondary-motion dimension parallel to *Y*	Axis nomenclature
W	Secondary-motion dimension parallel to *Z*	Axis nomenclature
X	Primary *X*-motion dimension	Axis nomenclature
Y	Primary *Y*-motion dimension	Axis nomenclature
Z	Primary *Z*-motion dimension	Axis nomenclature

N4G2X + 24Y + 24Z + 24B24I24J24F31T4M2

specifies that the NC machine is a machining center (has *X*-, *Y*-, and *Z*-axes) and a tool changer with a four-digit tool selection code (T4); the three linear axes are programmed with two digits before the decimal point and four after the decimal point (X + 24Y + 24Z + 24) and can be positive or negative; probably has a horizontal spindle and rotary table (B24

= rotary motion about the *Y*-axis); has circular interpolation (I24J24); has a feed rate range in which there are three digits before and one after the decimal point (F31); and can handle a four-digit sequence number (N4), two-digit G-words (G2), and two-digit miscellaneous words (M2). The sequence of letter addresses in the format detail is also the sequence in which words with those addresses should appear when used in a block.

The information given in the format shorthand and format detail is especially useful when programs written for one machine are to be used on different machines. Programs that use the variable block data format described in RS-274-D can be used interchangeably on systems that have the same format classification, but for complete program compatibility between machines, other features of the machine and control system must also be compatible, such as the relationships of the axes and the availability of features and control functions.

Control systems differ in the way that the numbers may be written. Most newer CNC machines accept numbers written in a decimal-point format, however, some systems require numbers to be in a fixed-length format that does not use an explicit decimal point. In the latter case, the control system evaluates a number based on the number of digits it has, including zeros. *Zero suppression* in a control system is an arrangement that allows zeros before the first significant figure to be dropped (leading zero suppression) or allows zeros after the last significant figure to be dropped (trailing zero suppression). An *X*-axis movement of 05.3400, for example, could be expressed as 053400 if represented in the full field format, 53400 (leading zero suppression), or 0534 (trailing zero suppression). With decimal-point programming, the above number is expressed simply as 5.34. To ensure program compatibility between machines, all leading and trailing zeros should be included in numbers unless decimal-point programming is used.

Sequence Number (N-Word).—A block normally starts with a sequence number that identifies the block within the part program. Most control systems use a four-digit sequence number allowing step numbers up to N9999. The numbers are usually advanced by fives or tens in order to leave spaces for additional blocks to be inserted later if required. For example, the first block in a program would be N0000, the next block N0005; the next N0010; and so on. The slash character, /, placed in a block, before the sequence number, is called an *optional stop* and causes the block to be skipped over when actuated by the operator. The block that is being worked on by the machine is often displayed on a digital readout so that the operator may know the precise operation being performed.

Preparatory Word (G-Word).—A preparatory word (also referred to as a preparatory function or G-code) consists of the letter address G and usually two digits. The preparatory word is placed at the beginning of a block, normally following the sequence number. Most newer CNC machines allow more than one G-code to be used in a single block, although many of the older systems do not. To ensure compatability with older machines and with the RS-274-D Standard, only one G-code per block should be used.

The G-word indicates to the control system how to interpret the remainder of theblock. For example, G01 refers to linear interpolation and indicates that the words following in the block will move the cutter in a straight line. The G02 code indicates that the words following in the block will move the cutter in a clockwise circular path. A G-word can completely change the normal meaning of other words in a block. For example, X is normally a dimension word that describes a distance or position in the *X*-direction. However, if a block contains the G04 word, which is the code for a dwell, the X word represents the time, in seconds, that the machine is to dwell.

The majority of G-codes are designated as modal, which means that once used, the code remains in effect for succeeding blocks unless it is specifically changed or canceled. Therefore, it is not necessary to include modal G-codes in succeeding blocks except to change or cancel them. Unless a G-code is modal, it is only effective within its designated block for the operation it defines. Table , G-Code Addresses, lists standardized G-code addresses and modality.

Table 2. G-Code Addresses

Code	Description	Code	Description
G00 [ab*]	Rapid traverse, point to point (M,L)	G34 [ab*]	Thread cutting, increasing lead (L)
G01 [abc]	Linear interpolation (M,L)	G35 [abc]	Thread cutting, decreasing lead (L)
G02 [abc]	Circular interpolation — clockwise movement (M,L)	G36-G39 [ab]	Permanently unassigned
G03 [abc]	Circular interpolation—counter-clockwise movement (M,L)	G36 [c]	Used for automatic acceleration and deceleration when the blocks are short (M,L)
G04 [ab]	Dwell—a programmed time delay (M,L)	G37, G37.1, G37.2, G37.3	Used for tool gaging (M,L)
G05 [ab]	Unassigned	G37.4	
G06 [abc]	Parabolic interpolation (M,L)	G38	Used for probing to measure the diameter and center of a hole (M)
G07 [c]	Used for programming with cylindrical diameter values (L)	G38.1	Used with a probe to measure the parallelness of a part with respect to an axis (M)
G08 [ab]	Programmed acceleration (M,L). [d] Also for lathe programming with cylindrical diameter values	G39, G39.1	Generates a nonprogrammed block to improve cycle time and corner cutting quality when used with cutter compensation (M)
G09 [ab]	Programmed deceleration (M,L). [d] Used to stop the axis movement at a precise location (M,L)	G39	Tool tip radius compensation used with linear generated block (L)
G10–G12 [ab]	Unassigned. [d]Sometimes used for machine lock and unlock devices	G39.1	Tool tip radius compensation used used with circular generated block (L)
G13–G16 [ac]	Axis selection (M,L)	G40 [abc]	Cancel cutter compensation/ offset (M)
G13–G16 [b]	Unassigned	G41 [abc]	Cutter compensation, left (M)
G13	Used for computing lines and circle intersections (M,L)	G42 [abc]	Cutter compensation, right (M)
G14, G14.1 [c]	Used for scaling (M,L)	G43 [abc]	Cutter offset, inside corner (M,L)
G15–G16 [c]	Polar coordinate programming (M)	G44 [abc]	Cutter offset, outside corner (M,L)
G15, G16.1 [c]	Cylindrical interpolation—C axis (L)	G45–G49 [ab]	Unassigned
G16.2 [c]	End face milling—C axis (L)	G50–G59 [a]	Reserved for adaptive control (M,L)
G17–G19 [abc]	*X-Y*, *X-Z, Y-Z* plane selection, respectively (M,L)	G50 [bb]	Unassigned
G20	Unassigned	G50.1 [c]	Cancel mirror image (M,L)
G22–G32 [ab]	Unassigned	G51.1 [c]	Program mirror image (M,L)
G22–G23 [c]	Defines safety zones in which the machine axis may not enter (M,L)	G52 [b]	Unassigned
G22.1, G233.1 [c]	Defines safety zones in which the cutting tool may not exit (M,L)	G52	Used to offset the axes with respect to the coordinate zero point (see G92) (M,L)
G24 [c]	Single-pass rough-facing cycle (L)	G53 [bc]	Datum shift cancel
G27–G29	Used for automatically moving to and returning from home position (M,L)	G53 [c]	Call for motion in the machine coordinate system (M,L)
		G54–G59 [bc]	Datum shifts (M,L)
G30	Return to an alternate home position (M,L)	G54–G59.3 [c]	Allows for presetting of work coordinate systems (M,L)
G31, G31.1, G31.2, G31.3, G31.4	External skip function, moves an axis on a linear path until an external signal aborts the move (M,L)	G60–G62 [abc]	Unassigned
G33 [abc]	Thread cutting, constant lead (L)		

Table 2. *(Continued)* **G-Code Addresses**

Code	Description	Code	Description
G61[c]	Modal equivalent of G09 except that rapid moves are not taken to a complete stop before the next motion block is executed (M,L)	G80[abc]	Cancel fixed cycles
		G81[abc]	Drill cycle, no dwell and rapid out (M,L)
G62[c]	Automatic corner override, reduces the feed rate on an inside corner cut (M,L)	G82[abc]	Drill cycle, dwell and rapid out (M,L)
G63[a]	Unassigned	G83[abc]	Deep hole peck drilling cycle (M,L)
G63[bc]	Tapping mode (M,L)	G84[abc]	Right-hand tapping cycle (M,L)
G64–G69[abc]	Unassigned	G84.1[c]	Left-hand tapping cycle (M,L)
G64[c]	Cutting mode, usually set by the system installer (M,L)	G85[abc]	Boring cycle, no dwell, feed out (M,L)
G65[c]	Calls for a parametric macro (M,L)	G86[abc]	Boring cycle, spindle stop, rapid out (M,L)
G66[c]	Calls for a parametric macro. Applies to motion blocks only (M,L)	G87[abc]	Boring cycle, manual retraction (M,L)
		G88[abc]	Boring cycle, spindle stop, manual retraction (M,L)
G66.1[c]	Same as G66 but applies to all blocks (M,L)	G88.1	Pocket milling (rectangular and circular), roughing cycle (M)
G67[c]	Stop the modal parametric macro (see G65, G66, G66.1) (M,L)	G88.2	Pocket milling (rectangular and circular), finish cycle (M)
G68[c]	Rotates the coordinate system (i.e., the axes) (M)	G88.3	Post milling, roughs out material around a specified area (M)
G69[c]	Cancel axes rotation (M)	G88.4	Post milling, finish cuts material around a post (M)
G70[abc]	Inch programming (M,L)	G88.5	Hemisphere milling, roughing cycle (M)
G71[abc]	Metric programming (M,L)		
G72[ac]	Circular interpolation CW (three-dimensional) (M)	G88.6	Hemisphere milling, finishing cycle (M)
G72[b]	Unassigned		
G72[c]	Used to perform the finish cut on a turned part along the Z-axis after the roughing cuts initiated under G73, G74, or G75 codes (L)	G89[abc]	Boring cycle, dwell and feed out (M,L)
		G89.1	Irregular pocket milling, roughing cycle (M)
G73[b]	Unassigned		
G73[c]	Deep hole peck drilling cycle (M); OD and ID roughing cycle, running parallel to the Z-axis (L)	G89.2	Irregular pocket milling, finishing cycle (M)
G74[ac]	Cancel multiquadrant circular interpolation (M,L)	G90[abc]	Absolute dimension input (M,L)
G74[bc]	Move to home position (M,L)	G91[abc]	Incremental dimension input (M,L)
G74[c]	Left-hand tapping cycle (M)	G92[abc]	Preload registers, used to shift the coordinate axes relative to the current tool position (M,L)
G74	Rough facing cycle (L)	G93[abc]	Inverse time feed rate (velocity/distance) (M,L)
G75[ac]	Multiquadrant circular interpolation (M,L)	G94[c]	Feed rate in inches or millimeters per minute (ipm or mpm) (M,L)
G75[b]	Unassigned	G95[abc]	Feed rate given directly in inches or millimeters per revolution (ipr or mpr) (M,L)
G75	Roughing routine for castings or forgings (L)		
G76–G79[ab]	Unassigned	G96[abc]	Maintains a constant surface speed, feet (meters) per minute (L)
		G97[abc]	Spindle speed programmed in rpm (M,L)

Table 2. *(Continued)* **G-Code Addresses**

Code	Description	Code	Description
		G98–99 [ab]	Unassigned

[a] Adheres to ANSI/EIA RS-274-D;

[b] Adheres to ISO 6983/1,2,3 Standards; where both symbols appear together, the ANSI/EIA and ISO standard codes are comparable;

[c] This code is modal. All codes that are not identified as modal are nonmodal, when used according to the corresponding definition.

[d] Indicates a use of the code that does not conform with the Standard.

Symbols following a description: (M) indicates that the code applies to a mill or machining center; (L) indicates that the code applies to turning machines; (M,L) indicates that the code applies to both milling and turning machines.

Codes that appear more than once in the table are codes that are in common use, but are not defined by the Standard or are used in a manner that is different than that designated by the Standard (e.g., see G61).

Most systems that support the RS-274-D Standard codes do not use all the codes available in the Standard. Unassigned G-words in the Standard are often used by builders of machine tool control systems for a variety of special purposes, sometimes leading to confusion as to the meanings of unassigned codes. Even more confusing, some builders of systems and machine tools use the less popular standardized codes for other than the meaning listed in the Standard. For these reasons, machine code written specifically for one machine/controller will not necessarily work correctly on another machine controller without modification.

Dimension words contain numerical data that indicate either a distance or a position. The dimension units are selected by using G70 (inch programming) or G71 (metric programming) code. G71 is canceled by a G70 command, by miscellaneous functions M02 (end of program), or by M30 (end of data). The dimension words immediately follow the G-word in a block and on multiaxis machines should be placed in the following order: X, Y, Z, U, V, W, P, Q, R, A, B, C, D, and E.

Absolute programming (G90) is a method of defining the coordinate locations of points to which the cutter (or workpiece) is to move based on the fixed machine zero point. In Fig. 1, the $X - Y$ coordinates of P1 are $X = 1.0$, $Y = 0.5$ and the coordinates of P2 are $X = 2.0$, $Y = 1.1$. To indicate the movement of the cutter from one point to another when using the absolute coordinate system, only the coordinates of the destination point P2 are needed.

Incremental programming (G91) is a method of identifying the coordinates of a particular location in terms of the distance of the new location from the current location. In the example shown in Fig. 2, a move from P1 to P2 is written as X + 1.0, Y + 0.6. If there is no movement along the Z-axis, Z is zero and normally is not noted. An $X - Y$ incremental move from P2 to P3 in Fig. 2 is written as X + 1.0, Y − 0.7.

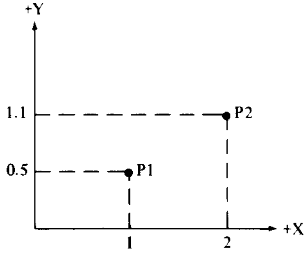


Fig. 1.

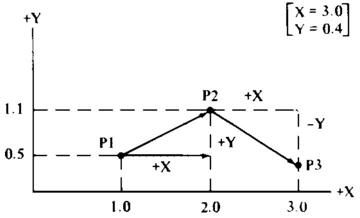


Fig. 2.

Most CNC systems offer both absolute and incremental part programming. The choice is handled by G-code G90 for absolute programming and G91 for incremental programming. G90 and G91 are both modal, so they remain in effect until canceled.

The G92 word is used to preload the registers in the control system with desired values. A common example is the loading of the axis-position registers in the control system for a lathe. Fig. 3 shows a typical home position of the tool tip with respect to the zero point on the machine. The tool tip here is registered as being 15.0000 inches in the *Z*-direction and 4.5000 inches in the *X*-direction from machine zero. No movement of the tool is required. Although it will vary with different control system manufacturers, the block to accomplish the registration shown in Fig. 3 will be approximately:

N0050 G92 X4.5 Z15.0

Miscellaneous Functions (M-Words).—Miscellaneous functions, or M-codes, also referred to as auxiliary functions, constitute on-off type commands. M functions are used to control actions such as starting and stopping of motors, turning coolant on and off, changing tools, and clamping and unclamping parts. M functions are made up of the letter M followed by a two-digit code. Table lists the standardized M-codes, however, the functions available will vary from one control system to another. Most systems provide fewer M functions than the complete list and may use some of the unassigned codes to provide additional functions that are not covered by the Standard. If an M-code is used in a block, it follows the T-word and is normally the last word in the block.

Table 3. Miscellaneous Function Words from *ANSI/EIA RS-274-D*

Code	Description
M00	Automatically *stops* the machine. The operator must push a button to continue with the remainder of the program.
M01	An *optional stop* acted upon only when the operator has previously signaled for this command by pushing a button. The machine will automatically stop when the control system senses the M01 code.
M02	This *end-of-program* code stops the machine when all commands in the block are completed. May include rewinding of tape.
M03	Start *spindle rotation* in a *clockwise* direction—looking out from the spindle face.
M04	Start *spindle rotation* in a *counterclockwise* direction—looking out from the spindle face.
M05	*Stop* the spindle in a normal and efficient manner.
M06	Command to *change a tool* (or tools) manually or automatically. Does not cover tool selection, as is possible with the T-words.
M07 to M08	M07 (coolant 2) and M08 (coolant 1) are codes to *turn on coolant.* M07 may control *flood* coolant and M08 *mist* coolant.
M09	Shuts off the coolant.
M10 to M11	M10 applies to automatic *clamping* of the machine slides, workpiece, fixture spindle, etc. M11 is an unclamping code.
M12	An inhibiting code used to synchronize multiple sets of axes, such as a four-axis lathe having two independently operated heads (turrets).
M13	Starts *CW spindle* motion and *coolant on* in the same command.
M14	Starts *CCW spindle* motion and *coolant on* in the same command.
M15 to M16	Rapid traverse of feed motion in either the +(M15) or –(M16) direction.
M17 to M18	Unassigned.
M19	Oriented spindle stop. Causes the spindle to stop at a predetermined angular position.
M20 to M29	Permanently unassigned.

Table 3. *(Continued)* **Miscellaneous Function Words from** *ANSI/EIA RS-274-D*

Code	Description
M30	An *end-of-tape* code similar to M02, but M30 will also rewind the tape; also may switch automatically to a second tape reader.
M31	A command known as *interlock bypass* for temporarily circumventing a normally provided interlock.
M32 to M35	Unassigned.
M36 to M39	Permanently unassigned.
M40 to M46	Used to signal gear changes if required at the machine; otherwise, unassigned.
M47	Continues program execution from the start of the program unless inhibited by an interlock signal.
M48 to M49	M49 deactivates a manual spindle or feed override and returns the parameter to the programmed value; M48 cancels M49.
M50 to M57	Unassigned.
M58 to M59	Holds the rpm constant at the value in use when M59 is initiated; M58 cancels M59.
M60 to M89	Unassigned.
M90 to M99	Reserved for use by the machine user.

Feed Function (F-Word).—F-word stands for feed-rate word or feed rate. The meaning of the feed word depends on the system of units in use and the feed mode. For example, F15 could indicate a feed rate of 0.15 inch (or millimeter) per revolution or 15 inches (or millimeters) per minute, depending on whether G70 or G71 is used to indicate inch or metric programming and whether G94 or G95 is used to specify feed rate expressed as inches (or mm) per minute or revolution. The G94 word is used to indicate inches/minute (ipm) or millimeters/minute (mmpm) and G95 is used for inches/revolution (ipr) or millimeters/revolution (mmpr). The default system of units is selected by G70 (inch programming) or G71 (metric programming) prior to using the feed function. The feed function is modal, so it stays in effect until it is changed by setting a new feed rate. In a block, the feed function is placed immediately following the dimension word of the axis to which it applies or immediately following the last dimension word to which it applies if it is used for more than one axis.

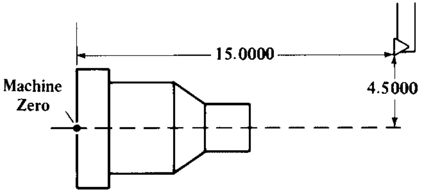


Fig. 3.

In turning operations, when G95 is used to set a constant feed rate per revolution, the spindle speed is varied to compensate for the changing diameter of the work — the spindle speed increases as the working diameter decreases. To prevent the spindle speed from increasing beyond a maximum value, the S-word, see *Spindle Function (S-Word)*, is used to specify the maximum allowable spindle speed before issuing the G95 command. If the spindle speed is changed after the G95 is used, the feed rate is also changed accordingly. If G94 is used to set a constant feed per unit of time (inches or millimeters per minute), changes in the spindle speed do not affect the feed rate.

Feed rates expressed in inches or millimeters per revolution can be converted to feed rates in inches or millimeters per minute by multiplying the feed rate by the spindle speed in revolutions per minute: feed/minute = feed/revolution × spindle speed in rpm. Feed rates for milling cutters are sometimes given in inches or millimeters per tooth. To convert feed per tooth to feed per revolution, multiply the feed rate per tooth by the number of cutter teeth: feed/revolution = feed/tooth × number of teeth.

For certain types of cuts, some systems require an inverse-time *feed command* that is the reciprocal of the time in minutes required to complete the block of instructions. The feed command is indicated by a G93 code followed by an F-word value found by dividing the feed rate, in inches (millimeters) or degrees per minute, by the distance moved in the block: feed command = feed rate/distance = (distance/time)/distance = 1/time.

Feed-rate override refers to a control, usually a rotary dial on the control system panel, that allows the programmer or operator to override the programmed feed rate. Feed-rate override does not change the program; permanent changes can only be made by modifying the program. The range of override typically extends from 0 to 150 per cent of the programmed feed rate on CNC machines; older hardwired systems are more restrictive and most cannot be set to exceed 100 per cent of the preset rate.

Spindle Function (S-Word).—An S-word specifies the speed of rotation of the spindle. The spindle function is programmed by the address S followed by the number of digits specified in the format detail (usually a four-digit number). Two G-codes control the selection of spindle speed input: G96 selects a constant cutting speed in surface feet per minute (sfm) or meters per minute (mpm) and G97 selects a constant spindle speed in revolutions per minute (rpm).

In turning, a constant spindle speed (G97) is applied for threading cycles and for machining parts in which the diameter remains constant. Feed rate can be programmed with either G94 (inches or millimeters per minute) or G95 (inches or millimeters per revolution) because each will result in a constant cutting speed to feed relationship.

G96 is used to select a constant cutting speed (i.e., a constant surface speed) for facing and other cutting operations in which the diameter of the workpiece changes. The spindle speed is set to an initial value specified by the S-word and then automatically adjusted as the diameter changes so that a constant surface speed is maintained. The control system adjusts spindle speed automatically, as the working diameter of the cutting tool changes, decreasing spindle speed as the working diameter increasesor increasing spindle speed as the working diameter decreases. When G96 is used for a constant cutting speed, G95 in a succeeding block maintains a constant feed rate per revolution.

Speeds given in surface feet or meters per minute can be converted to speeds in revolutions per minute (rpm) by the formulas:

$$\text{rpm} = \frac{\text{sfm} \times 12}{\pi \times d} \qquad \text{rpm} = \frac{\text{mpm} \times 1000}{\pi \times d}$$

where d is the diameter, in inches or millimeters, of the part on a lathe or of the cutter on a milling machine; and π is equal to 3.14159.

Tool Function (T-Word).—The T-word calls out the tool that is to be selected on a machining center or lathe having an automatic tool changer or indexing turret. On machines without a tool changer, this word causes the machine to stop and request a tool change. This word also specifies the proper turret face on a lathe. The word usually is accompanied by several numbers, as in T0101, where the first pair of numbers refers to the tool number (and carrier or turret if more than one) and the second pair of numbers refers to the tool offset number. Therefore, T0101 refers to tool 1, offset 1.

Information about the tools and the tool setups is input to the CNC system in the form of a *tool data table*. Details of specific tools are transferred from the table to the part program via the T-word. The tool nose radius of a lathe tool, for example, is recorded in the tool data table so that the necessary tool path calculations can be made by the CNC system. The miscellaneous code M06 can also be used to signal a tool change, either manually or automatically.

Compensation for variations in the tool nose radius, particularly on turning machines, allows the programmer to program the part geometry from the drawing and have the tool follow the correct path in spite of variations in the tool nose shape. Typical of the data required, as shown in Fig. 4, are the nose radius of the cutter, the *X* and *Z* distances from the gage point to some fixed reference point on the turret, and the orientation of the cutter (tool tip orientation code), as shown in Fig. 5. Details of nose radius compensation for numerical control is given in a separate section (Indexable Insert Holders for NC).

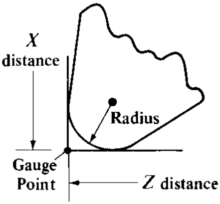


Fig. 4.

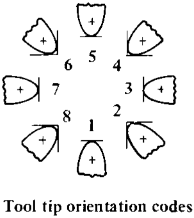


Fig. 5.

Tool offset, also called cutter offset, is the amount of cutter adjustment in a direction parallel to the axis of a tool. Tool offset allows the programmer to accommodate the varying dimensions of different tooling by assuming (for the sake of the programming) that all the tools are identical. The actual size of the tool is totally ignored by the programmer who programs the movement of the tools to exactly follow the profile of theworkpiece shape. Once tool geometry is loaded into the tool data table and the cutter compensation controls of the machine activated, the machine automatically compensates for the size of the tools in the programmed movements of the slide. In gage length programming, the tool length and tool radius or diameter are included in the program calculations. Compensation is then used only to account for minor variations in the setup dimensions and tool size.

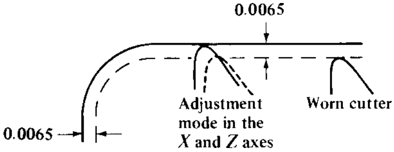


Fig. 6.

Customarily, the tool offset is used in the beginning of a program to initialize each individual tool. Tool offset also allows the machinist to correct for conditions, such as tool wear, that would cause the location of the cutting edge to be different from the programmed location. For example, owing to wear, the tool tip in Fig. 6 is positioned a distance of 0.0065 inch from the location required for the work to be done. To compensate for this wear, the operator (or part programmer), by means of the CNC control panel, adjusts the tool tip with reference to the *X*- and *Z*-axes, moving the tool closer to the work by

0.0065 inch throughout its traverse. The tool offset number causes the position of the cutter to be displaced by the value assigned to that offset number.

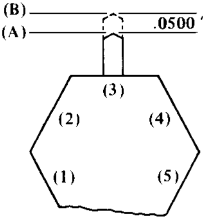


Fig. 7.

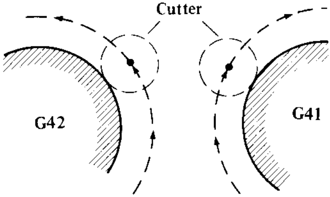


Fig. 8.

Changes to the programmed positions of cutting tool tip(s) can be made by *tool length offset* programs included in the control system. A dial or other means is generally provided on milling, drilling, and boring machines, and machining centers, allowing the operator or part programmer to override the programmed axial, or *Z*-axis, position. This feature is particularly helpful when setting the lengths of tools in their holders or setting a tool in a turret, as shown in Fig. 7, because an exact setting is not necessary. The tool can be set to an approximate length and the discrepancy eliminated by the control system.

The amount of offset may be determined by noting the amount by which the cutter is moved manually to a fixed point on the fixture or on the part, from the programmed *Z*-axis location. For example, in Fig. 7, the programmed *Z*-axis motion results in the cutter being moved to position A, whereas the required location for the tool is at B. Rather than resetting the tool or changing the part program, the tool length offset amount of 0.0500 inch is keyed into the control system. The 0.0500-inch amount is measured by moving the cutter tip manually to position B and reading the distance moved on the readout panel. Thereafter, every time that cutter is brought into the machining position, the programmed *Z*-axis location will be overridden by 0.0500 inch.

Manual adjustment of the cutter center path to correct for any variance between nominal and actual cutter radius is called *cutter compensation.* The net effect is to move the path of the center of the cutter closer to, or away from, the edge of the workpiece, as shown in Fig. 8. The compensation may also be handled via a tool data table.

When cutter compensation is used, it is necessary to include in the program a G41 code if the cutter is to be to the left of the part and a G42 code if to the right of the part, as shown in Fig. 8. A G40 code cancels cutter compensation. Cutter compensation with earlier hardwire systems was expensive, very limited, and usually held to ±0.0999 inch. The range for cutter compensation with CNC control systems can go as high as ±999.9999 inches, although adjustments of this magnitude are unlikely to be required.

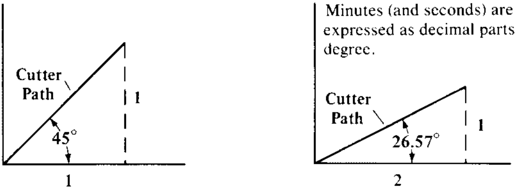


Fig. 9.

Linear Interpolation.—The ability of the control system to guide the workpiece along a straight-line path at an angle to the slide movements is called linear interpolation. Movements of the slides are controlled through simultaneous monitoring of pulses by the control system. For example, if monitoring of the pulses for the *X*-axis of a milling machine is at the same rate as for the *Y*-axis, the cutting tool will move at a 45-degree angle relative to the *X*-axis. However, if the pulses are monitored at twice the rate for the *X*-axis as for the *Y*-axis, the angle that the line of travel will make with the *X*-axis will be 26.57 degrees (tangent of 26.57 degrees = ½), as shown in Fig. 9. The data required are the distances traveled in the *X*- and *Y*-directions, and from these data, the control system will generate the straight line automatically. This monitoring concept also holds for linear motions along three axes. The required G-code for linear interpolation blocks is G01. The code is modal, which means that it will hold for succeeding blocks until it is changed.

Circular Interpolation.—A simplified means of programming circular arcs in one plane, using one block of data, is called circular interpolation. This procedure eliminates the need to break the arc into straight-line segments. Circular interpolation is usually handled in one plane, or two dimensions, although three-dimensional circular interpolation is described in the Standards. The plane to be used is selected by a G or preparatory code. In Fig. 10, G17 is used if the circle is to be formed in the *X*–*Y* plane, G18 if in the *X*–*Z* plane, and G19 if in the *Y*–*Z* plane. Often the control system is preset for the circular interpolation feature to operate in only one plane (e.g., the *X*–*Y* plane for milling machines or machining centers or the *X*–*Z* plane for lathes), and for these machines, the G-codes are not necessary.

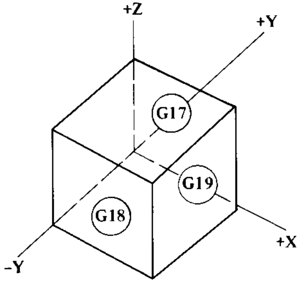


Fig. 10.

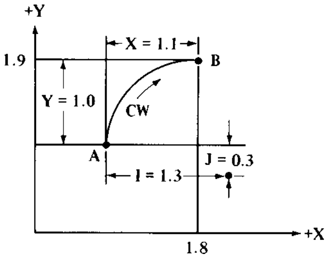


Fig. 11.

A circular arc may be described in several ways. Originally, the RS-274 Standard specified that, with incremental programming, the block should contain:

1) A G-code describing the direction of the arc, G02 for clockwise (CW), and G03 for counterclockwise (CCW).

2) Directions for the component movements around the arc parallel to the axes. In the example shown in Fig. 11, the directions are *X* = +1.1 inches and *Y* = +1.0 inch. The signs are determined by the direction in which the arc is being generated. Here, both *X* and *Y* are positive.

3) The I dimension, which is parallel to the *X*-axis with a value of 1.3 inches, and the J dimension, which is parallel to the *Y*-axis with a value of 0.3 inch. These values, which locate point A with reference to the center of the arc, are called offset dimensions. The block for this work would appear as follows:

```
N0025 G02 X011000 Y010000 I013000 J003000
```

(The sequence number, N0025, is arbitrary.)

The block would also contain the plane selection (i.e., G17, G18, or G19), if this selection is not preset in the system. Most of the newer control systems allow duplicate words in the same block, but most of the older systems do not. In these older systems, it is necessary to insert the plane selection code in a separate and prior block, for example, N0020 G17.

Another stipulation in the Standard is that the arc is limited to one quadrant. Therefore, four blocks would be required to complete a circle. Four blocks would also be required to complete the arc shown in Fig. 12, which extends into all four quadrants.

When utilizing absolute programming, the coordinates of the end point are described. Again from Fig. 11, the block, expressed in absolute coordinates, appears as:

N0055 G02 X01800 Y019000 I013000 J003000

where the arc is continued from a previous block; the starting point for the arc in this block would be the end point of the previous block.

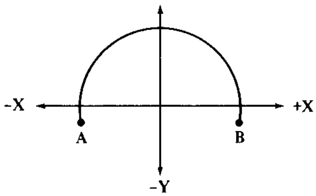


Fig. 12.

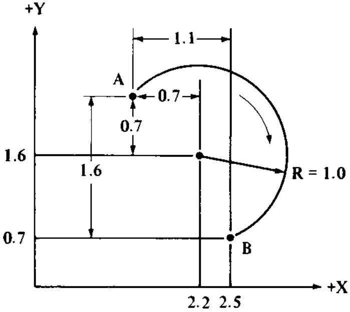


Fig. 13.

The Standard still contains the format discussed, but simpler alternatives have been developed. The latest version of the Standard (RS-274-D) allows *multiple quadrant programming* in one block, by inclusion of a G75 word. In the absolute-dimension mode (G90), the coordinates of the arc center are specified. In the incremental-dimension mode (G91), the signed (plus or minus) incremental distances from the beginning point of the arc to the arc center are given. Most system builders have introduced some variations on this format. One system builder utilizes the center and the end point of the arc when in an absolute mode, and might describe the block for going from A to B in Fig. 13 as:

N0065 G75 G02 X2.5 Y0.7 I2.2 J1.6

The I and the J words are used to describe the coordinates of the arc center. Decimal-point programming is also used here. A block for the same motion when programmed incrementally might appear as:

N0075 G75 G02 X1.1 Y – 1.6 I0.7 J0.7

This approach is more in conformance with the RS-274-D Standard in that the *X* and *Y* values describe the displacement between the starting and ending points (points A and B), and the I and J indicate the offsets of the starting point from the center. Another and even more convenient way of formulating a circular motion block is to note the coordinates of the ending point and the radius of the arc. Using absolute programming, the block for the motion in Fig. 13 might appear as:

N0085 G75 G02 X2.5 Y0.7 R10.0

The starting point is derived from the previous motion block. Multiquadrant circular interpolation is canceled by a G74 code.

Helical and Parabolic Interpolation.—Helical interpolation is used primarily for milling large threads and lubrication grooves, as shown in Fig. 14. Generally, helical interpolation involves motion in all three axes (*X*, *Y*, *Z*) and is accomplished by using circular interpolation (G02 or G03) while changing the third dimension. Parabolic interpolation (G06) is simultaneous and coordinated control of motion-such that the resulting cutter path describes part of a parabola. The RS-274-D Standard provides further details.

Subroutine.— A subroutine is a set of instructions or blocks that can be inserted into a program and repeated whenever required. Parametric subroutines permit letters or symbols to be inserted into the program in place of numerical values (see *Parametric Expressions and Macros*). Parametric subroutines can be called during part programming and values assigned to the letters or symbols. This facility is particularly helpful when dealing with families of parts.

A subprogram is similar to a subroutine except that a subprogram is not wholly contained within another program, as is a subroutine. Subprograms are used when it is necessary to perform the same task frequently, in different programs. The advantage of subprograms over subroutines is that subprograms may be called by any other program, whereas the subroutine can only be called by the program that contains the subroutine.

There is no standard subroutine format; however, the example below is typical of a program that might be used for milling the three pockets shown in Fig. 15. In the example, the beginning and end of the subroutine are indicated by the codes M92 and M93, respectively, and M94 is the code that is used to call the subroutine. The codes M92, M93, and M94 are not standardized (M-codes M90 through M99 are reserved for the user) and may be different from control system to control system. The subroutine functions may use different codes or may not be available at all on other systems.

N0010 G00 X.6 Y.85	Cutter is moved at a rapid traverse rate to a position over the corner of the first pocket to be cut.
N0020 M92	Tells the system that the subroutine is to start in the next block.
N0030 G01 Z–.25 F2.0	Cutter is moved axially into the workpiece 0.25 inch at 2.0 ipm.
N0040 X.8	Cutter is moved to the right 0.8 inch.
N0050 Y.2	Cutter is moved laterally up 0.2 inch.
N0060 X–.8	Cutter is moved to the left 0.8 inch.
N0070 Y.2	Cutter is moved laterally up 0.2 inch.

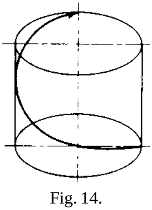


Fig. 14.

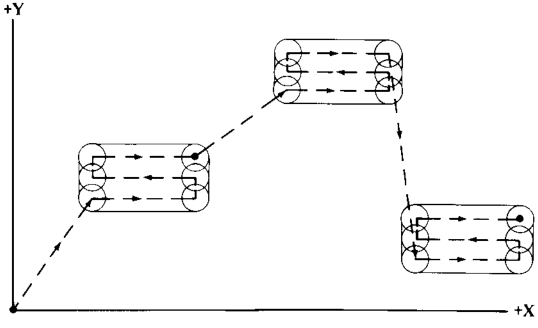


Fig. 15.

N0080 X.8	Cutter is moved to the right 0.8 inch.
N0090 G00 Z.25 M93	Cutter is moved axially out of pocket at rapid traverse rate. Last block of subroutine is signaled by word M93.
N0100 X.75 Y.5	Cutter is moved to bottom left-hand corner of second pocket at rapid traverse rate.
N0110 M94 N0030	Word M94 calls for repetition of the subroutine that starts at sequence number N0030 and ends at sequence number N0090.
N0120 G00 X.2 Y–I.3	After the second pocket is cut by repetition of sequence numbers N0030 through N0090, the cutter is moved to start the third pocket.
N0130 M94 N0030	Repetition of subroutine is called for by word M94 and the third pocket is cut.

Parametric Expressions and Macros.—Parametric programming is a method whereby a variable or replaceable parameter representing a value is placed in the machining code instead of using the actual value. In this manner, a section of code can be used several or many times with different numerical values, thereby simplifying the programming and reducing the size of the program. For example, if the values of X and Y in lines N0040 to N0080 of the previous example are replaced as follows:

```
N0040 X#1

N0050 Y#2

N0060 X#3

N0070 Y#4
```

then the subroutine starting at line N0030 is a parametric subroutine. That is, the numbers following the # signs are the variables or parameters that will be replaced with actual values when the program is run. In this example, the effect of the program changes is to allow the same group of code to be used for milling pockets of different sizes. If on the other hand, lines N0010, N0100, and N0120 of the original example were changed in a similar manner, the effect would be to move the starting location of each of the slots to the location specified by the replaceable parameters.

Before the program is run, the values that are to be assigned to each of the parameters or variables are entered as a list at the start of the part program in this manner:

```
#1 = .8
#2 = .2
#3 = .8
#4 = .2
```

All that is required to repeat the same milling process again, but this time creating a different size pocket, is to change the values assigned to each of the parameters #1, #2, #3, and #4 as necessary. Techniques for using parametric programming are not standardized and are not recognized by all control systems. For this reason, consult the programming manual of the particular system for specific details.

As with a parametric subroutine, macro describes a type of program that can be recalled to allow insertion of finite values for letter variables. The difference between a macro and a parametric subroutine is minor. The term macro normally applies toa source program that is used with computer-assisted part programming; the parametric subroutine is a feature of the CNC system and can be input directly into that system.

Conditional Expressions.—It is often useful for a program to make a choice between two or more options, depending on whether or not a certain condition exists. A program can contain one or more blocks of code that are not needed every time the program is run, but are needed some of the time. For example, refer to the previous program for milling three slots. An occasion arises that requires that the first and third slots be milled, but not the second one. If the program contained the following block of code, the machine could be easily instructed to skip the milling of the second slot:

```
N0095 IF [#5 EQ 0] GO TO N0120
```

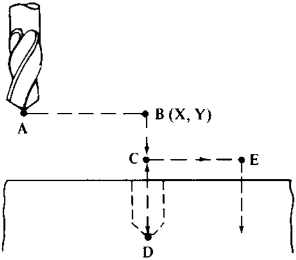
In this block, #5 is the name of a variable; EQ is a conditional expression meaning *equals;* and GO TO is a branch statement meaning resume execution of the program at the following line number. The block causes steps N0100 and N0110 of the program to be skipped if the value of #5 (a dummy variable) is set equal to zero. If the value assigned to #5 is any number other than zero, the expression (#5 EQ 0) is not true and the remaining instructions in block N0095 are not executed. Program execution continues with the next step, N0100, and the second pocket is milled. For the second pocket to be milled, parameter #5 is initialized at the beginning of the program with a statement such as #5 = 1 or #5 = 2. Initializing #5 = 0 guarantees that the pocket is not machined. On control systems that automatically initialize all variables to zero whenever the system is reset or a program is loaded, the second slot will not be machined unless the #5 is assigned a nonzero value each time the program is run.

Other conditional expressions are: NE = not equal to; GT = greater than; LT = less than; GE = greater than or equal to; and LE = less than or equal to. As with parametric expressions, conditional expressions may not be featured on all machines and techniques and implementation will vary. Therefore, consult the control system programming manual for the specific command syntax.

Fixed (Canned) Cycles.—Fixed (canned) cycles comprise sets of instructions providing for a preset sequence of events initiated by a single command or a block of data. Fixed cycles generally are offered by the builder of the control system or machine tool as part of the software package that accompanies the CNC system. Limited numbers of canned cycles began to appear on hardwire control systems shortly before their demise. The canned cycles offered generally consist of the standard G-codes covering driling, boring, and tapping operations, plus options that have been developed by the system builder such as thread cutting and turning cycles. (See *Thread Cutting* and *Turning Cycles*.) Some standard canned cycles included in RS-274-D are shown herewith. A block of data that might be used to generate the cycle functions is also shown above each illustration. Although the G-codes for the functions are standardized, the other words in the block and the block format are not, and different control system builders have different arrangements. The blocks shown are reasonable examples of fixed cycles and do not represent those of any particular system builder.

The G81 block for a simple drilling cycle is:
N_____ G81 X_____Y_____C_____D_____F_____EOB

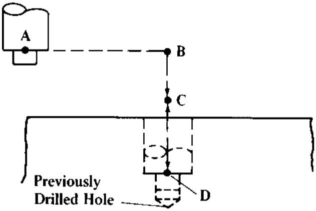
N_____X_____Y_____EOB



This G81 drilling cycle will move the drill point from position A to position B and then down to C at a rapid traverse rate; the drill point will next be fed from C to D at the programmed feed rate, then returned to C at the rapid traverse rate. If the cycle is to be repeated at a subsequent point, such as point E in the illustration, it is necessary Only to give the required X and Y coordinates. This repetition capability is typical of canned cycles.

The G82 block for a spotfacing or drilling cycle with a dwell is:

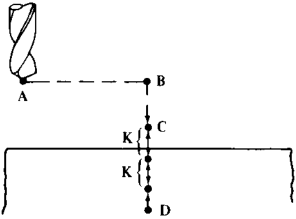
N_____G82 X_____Y_____C_____D_____T_____F_____EOB



This G82 code produces a cycle that is very similar to the cycle of the G81 code except for the dwell period at point D. The dwell period allows the tool to smooth out the bottom of the counterbore or spotface. The time for the dwell, in seconds, is noted as a T-word.

The G83 block for a peck-drilling cyle is:

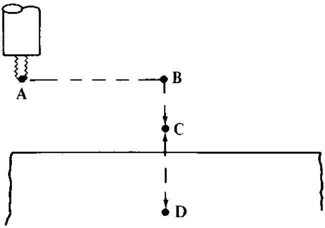
N_____G83 X_____Y_____C_____D_____K_____F_____EOB



In the G83 peck-drilling cycle, the drill is moved from point A to point B and then to point C at the rapid traverse rate; the drill is then fed the incremental distance K, followed by rapid return to C. Down feed again at the rapid traverse rate through the distance K is next, after which the drill is fed another distance K. The drill is thenrapid traversed back to C, followed by rapid traverse for a distance of K + K; down feed to D follows before the drill is rapid traversed back to C, to end the cycle.

The G84 block for a tapping cycle is:

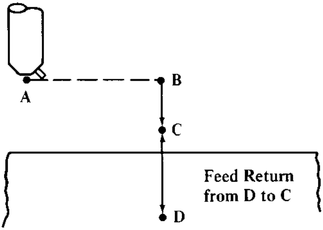
N_____G84 X_____Y_____C_____D_____F_____EOB



The G84 canned tapping cycle starts with the end of the tap being moved from point A to point B and then to point C at the rapid traverse rate. The tap is then fed to point D, reversed, and moved back to point C.

The G85 block for a boring cycle with tool retraction at the feed rate is:

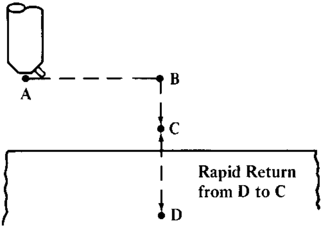
N_____G85 X_____Y_____C_____D_____F_____EOB



In the G85 boring cycle, the tool is moved from point A to point B and then to point C at the rapid traverse rate. The tool is next fed to point D and then, while still rotating, is moved back to point C at the same feed rate.

The G86 block for a boring cycle with rapid traverse retraction is:

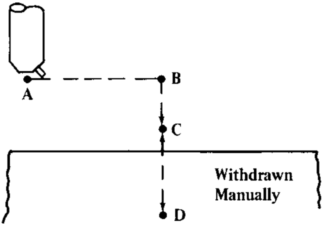
N_____G86 X_____Y_____C_____D_____F_____EOB



The G86 boring cycle is similar to the G85 cycle except that the tool is withdrawn at the rapid traverse rate.

The G87 block for a boring cycle with manual withdrawal of the tool is:

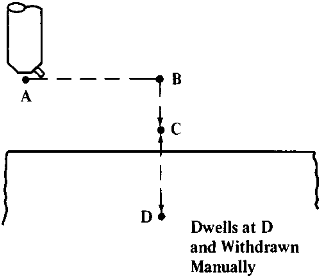
N_____G87 X_____Y_____C_____D_____F_____EOB



In the G87 canned boring cycle, the cutting tool is moved from A to B and then to C at the rapid traverse rate. The tool is then fed to D. The cycle is identical to the other boring cycles except that the tool is withdrawn manually.

The G88 block for a boring cycle with dwell and manual withdrawal is:

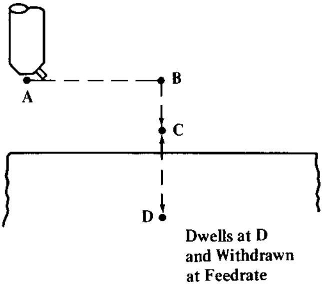
N_____G88 X_____Y_____C_____D_____T_____F_____EOB



In the G88 dwell cycle, the tool is moved from A to B to C at the rapid traverse rate and then fed at the prescribed feed rate to D. The tool dwells at D, then stops rotating and is withdrawn manually.

The G89 block for a boring cycle with dwell and withdrawal at the feed rate is:

N_____G89 X_____Y_____C_____D_____T_____F_____EOB



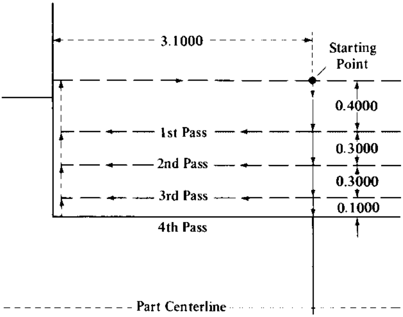


Fig. 16.

Turning Cycles.—Canned turning cycles are available from most system builders and are designed to allow the programmer to describe a complete turning operation in one or a few blocks. There is no standard for this type of operation, so a wide variety of programs have developed. Fig. 16 shows a hypothetical sequence in which the cutter is moved from the start point to depth for the first pass. If incremental programming is in effect, this distance is specified as D1. The depths of the other cuts will also be programmed as D2, D3, and so on. The length of the cut will be set by the W-word, and will remain the same with each pass. The preparatory word that calls for the roughing cycle is G77. The roughing feed rate is 0.03 ipr (inch per revolution), and the finishing feed rate (last pass) is 0.005 ipr. The block appears as follows:

```
N0054    G77    W = 3.1    D1 = .4    D2 = .3    D3 = .3    D4 = .1    F1 = .03    F2 = .005
```

Thread Cutting.—Most NC lathes can produce a variety of thread types including constant-lead threads, variable-lead threads (increasing), variable-lead threads (decreasing), multiple threads, taper threads, threads running parallel to the spindle axis, threads (spiral groove) perpendicular to the spindle axis, and threads containing a combination of the preceding. Instead of the feed rate, the lead is specified in the threading instruction block, so that the feed rate is made consistent with, and dependent upon, the selected speed (rpm) of the spindle.

The thread lead is generally noted by either an I- or a K-word. The I-word is used if the thread is parallel to the *X*-axis and the K-word if the thread is parallel to the *Z*-axis, the latter being by far the most common. The G-word for a constant-lead thread is G33, for an increasing variable-lead thread is G34, and for a decreasing variable-lead thread is G35. Taper threads are obtained by noting the *X*- and *Z*-coordinates of the beginning and end points of the thread if the G90 code is in effect (absolute programming), or the incremental movement from the beginning point to the end point of the thread if the G91 code (incremental programming) is in effect.

```
N0001 G91                     (Incremental programming)
N0002 G00 X–.1000             (Rapid traverse to depth)
N0003 G33 Z–1.0000 K.0625     (Produce a thread with a constant lead of
                                0.625 inch)
N0004 G00 X.1000              (Withdraw at rapid traverse)
N0005 Z1.0000                 (Move back to start point)
```

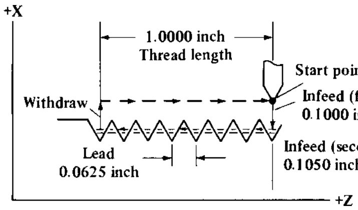


Fig. 17.

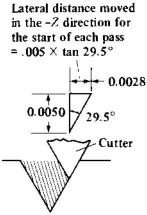


Fig. 18.

Multiple threads are specified by a code in the block that spaces the start of the threads equally around the cylinder being threaded. For example, if a triple thread is to be cut, the threads will start 120 degrees apart. Typical single-block thread cutting utilizing a plunge cut is illustrated in Fig. 17 and shows two passes. The passes areidentical except for the distance of the plunge cut. Builders of control systems and machine tools use different codewords for threading, but those shown below can be considered typical. For clarity, both zeros and decimal points are shown.

The only changes in the second pass are the depth of the plunge cut and the withdrawal. The blocks will appear as follows:

```
N0006 X – .1050
N0007 G33 Z – 1.0000 K.0625
N0008 G00 X.1050
N0009 Z1.000
```

Compound thread cutting, rather than straight plunge thread cutting, is possible also, and is usually used on harder materials. As illustrated in Fig. 18, the starting point for the thread is moved laterally in the -Z direction by an amount equal to the depth of the cut times the tangent of an angle that is slightly less than 30 degrees. The program for the second pass of the example shown in Fig. 18 is as follows:

```
N0006 X – .1050 Z – .0028
N0007 G33 Z – 1.0000 K.0625
N0008 G00 X.1050
N0009 Z1.0000
```

Fixed (canned), one-block cycles also have been developed for CNC systems to produce the passes needed to complete a thread. These cycles may be offered by the builder of the control system or machine tool as standard or optional features. Subroutines also can generally be prepared by the user to accomplish the same purpose (see Subroutine). A one-block fixed threading cycle might look something like:

```
N0048 G98 X – .2000 Z – 1.0000 D.0050 F.0010
```

where G98 = preparatory code for the threading cycle

X–.2000 = total distance from the starting point to the bottom of the thread

Z–1.0000 = length of the thread

D.0050 = depths of successive cuts

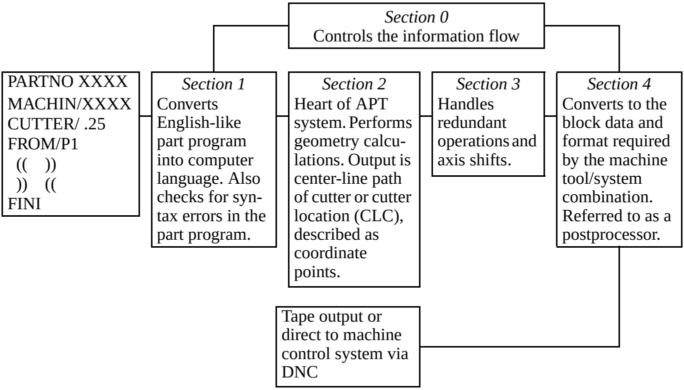
F.0010 = depth(s) of the finish cut(s)

APT Programming

APT.—APT stands for Automatically Programmed Tool and is one of many computer languages designed for use with NC machine tools. The selection of a computer-assisted part-programming language depends on the type and complexity of the parts being machined more than on any other factor. Although some of the other languages may be easier to use, APT has been chosen to be covered in this book because it is a nonproprietary language in the public domain, has the broadest range of capability, and is one of the most advanced and universally accepted NC programming languages available. APT (or a variation thereof) is also one of the languages that is output by many computer programs that produce CNC part programs directly from drawings produced with CAD systems.

APT is suitable for use in programming part geometry from simple to exceptionally complex shapes. APT was originally designed and used on mainframe computers, however, it is now available, in many forms, on mini- and microcomputers as well. APT has also been adopted as ANSI Standard X3.37and by the International Organization for Standardization (ISO) as a standardized language for NC programming. APT is a very dynamic program and is continually being updated. APT is being used as a processor for part-programming graphic systems, some of which have the capability of producing an APT program from a graphic screen display or CAD drawing and of producing a graphic display on the CAD system from an APT program.

APT is a high-level programming language. One difference between APT and the ANSI/EIA RS-274-D (G-codes) programming format discussed in the last section is that APT uses English like words and expressions to describe the motion of the tool or workpiece. APT has the capability of programming the machining of parts in up to five axes, and also allows computations and variables to be included in the programming statements so that a whole family of similar parts can be programmed easily. This section describes the general capabilities of the APT language and includes a ready reference guide to the basic geometry and motion statements of APT, which is suitable for use in programming the machining of the majority of cubic type parts involving two-dimensional movements. Some of the three-dimensional geometry capability of APT and a description of its five-dimensional capability are also included.



As shown above, the APT system can be thought of comprising the input program, the five sections 0 through IV, and the output program. The input program shown on the left progresses through the first four sections and all four are controlled by the fifth, section 0. Section IV, the postprocessor, is the software package that is added to sections II and III to customize the output and produce the necessary program format (including the G-words, M-words, etc.) so that the coded instructions will be recognizable by the control system. The postprocessor is software that is separate from the main body of the APT program, but for purposes of discussion, it may be easier to consider it as a unit within the APT program.

APT Computational Statements.—Algebraic and trigonometric functions and computations can be performed with the APT system as follows:

Arithmetic Form	APT Form	Arithmetic Form	APT Form	Arithmetic Form	APT Form
25×25	25*25	25^2	25**2	$\cos \theta$	COSF(θ)
$25 \div 25$	25/25	25^n	25**n	$\tan \theta$	TANF(θ)
$25 + 25$	25 + 25	$\sqrt{25}$	SQRTF (25)	arctan .5000	ATANF(.5)
$25 - 25$	25 – 25	$\sin \theta$	SINF(θ)		

Computations may be used in the APT system in two ways. One way is to let a factor equal the computation and then substitute the factor in a statement; the other is to put the computation directly into the statement. The following is a series of APT statements illustrating the first approach.

```
P1 = POINT/0,0,1
 T = (25*2/3 + (3**2 – 1))
P2 = POINT/T,0,0
```

The second way would be as follows;

```
P1 = POINT/0,0,1
P2 = POINT/(25*2/3 + (3**2 – 1)),0,0
```

Note: The parentheses have been used as they would be in an algebraic formula so that the calculations will be carried out in proper sequence. The operations within the inner parentheses would be carried out first. It is important for the total number of left-hand parentheses to equal the total number of right-hand parentheses; otherwise, the program will fail.

APT Geometry Statements.—Before movements around the geometry of a part can be described, the geometry must be defined. For example, in the statement GOTO/P1, the computer must know where P1 is located before the statement can be effective. P1 therefore must be described in a geometry statement, prior to its use in the motion statement GOTO/P1. The simplest and most direct geometry statement for a point is

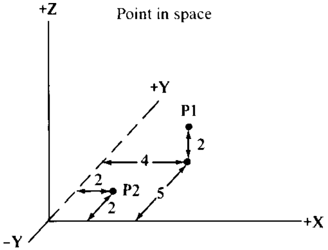
```
P1 = POINT/X ordinate, Y ordinate, Z ordinate
```

If the Z ordinate is zero and the point lies on the X–Y plane, the Z location need not be noted. There are other ways of defining the position of a point, such as at the intersection of two lines or where a line is tangent to a circular arc. These alternatives are described below, together with ways to define lines and circles. Referring to the preceding statement, P1 is known as a symbol. Any combination of letters and numbers may be used as a symbol providing the total does not exceed six characters and at least one of them is a letter. MOUSE2 would be an acceptable symbol, as would CAT3 or FRISBE. However, it is sensible to use symbols that help define the geometry. For example, C1 or CIR3 would be good symbols for a circle. A good symbol for a vertical line would be VL5.

Next, and after the equal sign, the particular geometry is noted. Here, it is a POINT. This word is a vocabulary word and must be spelled exactly as prescribed. Throughout, the designers of APT have tried to use words that are as close to English as possible. A slash follows the vocabulary word and is followed by a specific description of the particular geometry, such as the coordinates of the point P1. A usable statement for P1 might appear as P1 = POINT/1,5,4. The 1 would be the X ordinate; the 5, the Y ordinate; and the 4, the Z ordinate.

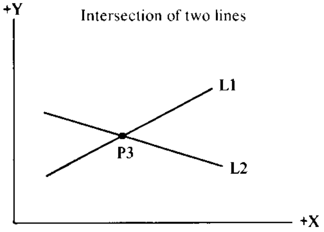
Lines as calculated by the computer are infinitely long, and circles consist of 360 degrees. As the cutter is moved about the geometry under control of the motion statements, the lengths of the lines and the amounts of the arcs are “cut” to their proper size. (Some of the geometry statements shown in the accompanying illustrations for defining POINTS, LINES, CIRCLES, TABULATED CYLINDERS, CYLINDERS, CONES, and SPHERES, in the APT language, may not be included in some two-dimensional [ADAPT] systems.)

Points



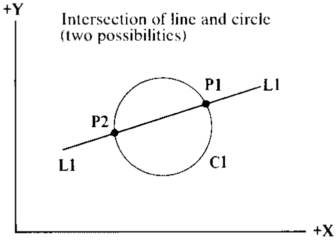
Point in space

P1 = POINT/4, 5, 2
P2 = POINT/2, 2



Intersection of two lines

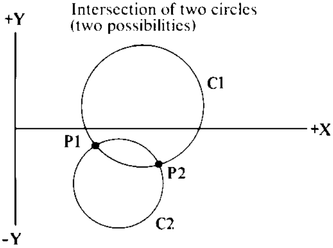
P3 = POINT/INTOF, L1, L2



Intersection of line and circle (two possibilities)

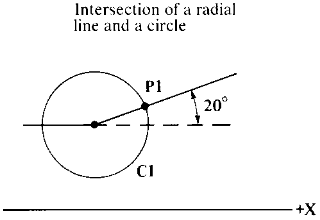
P1 = POINT/XLARGE, INTOF, L1, C1
or
P1 = POINT/YLARGE, INTOF, L1, C1
P2 = POINT/XSMALL, INTOF, L1, C1
or
P2 = POINT/YSMALL, INTOF, L1, C1

The *X* and *Y* ordinates of P1 are larger than the *X* and *Y* ordinates of P2



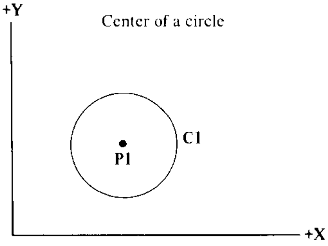
Intersection of two circles (two possibilities)

P1 = POINT/XSMALL, INTOF, C1, C2
or
P1 = POINT/YLARGE, INTOF, C1, C2
P2 = POINT/XLARGE, INTOF, C1, C2
or
P2 = POINT/YSMALL, INTOF, C1, C2



Intersection of a radial line and a circle

P1 = POINT/C1, ATANGL, 20

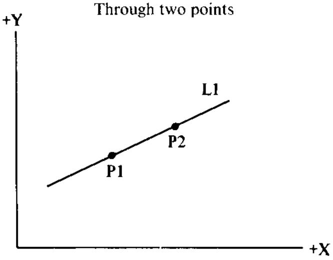


Center of a circle

P1 = POINT/CENTER, C1

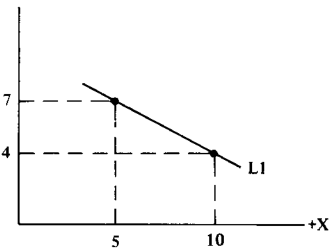
Lines

Through two points



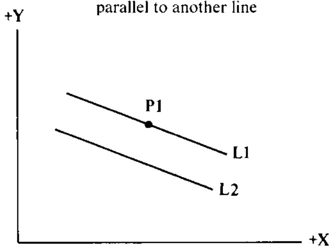
L1 = LINE/P1, P2

The coordinates of two points



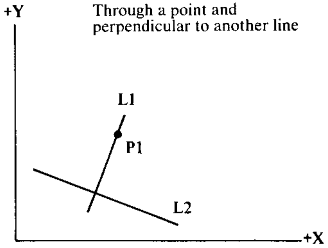
L1 = LINE/5, 7, 10, 4

Through a point and parallel to another line



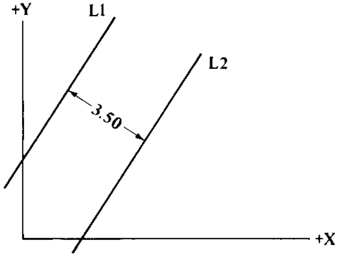
L1 = LINE/P1, PARLEL, L2

Through a point and perpendicular to another line



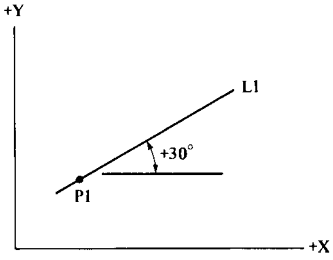
L1 = LINE/P1, PERPTO, L2

A parallel line and a perpendicular distance



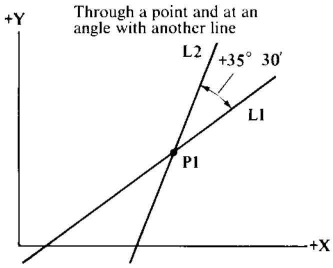
L1 = LINE/PARLEL, L2, XSMALL, 3.50
or
L1 = LINE/PARLEL, L2, YLARGE, 3.50
L2 = LINE/PARLEL, L1, XLARGE, 3.50
or
L2 = LINE/PARLEL, L1, YSMALL, 3.50

Through a point and at an angle with X-axis

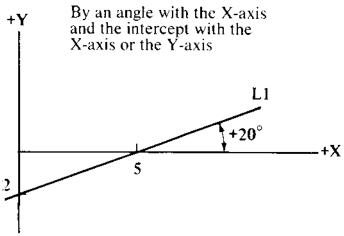


L1 = LINE/P1, ATANGL, 30

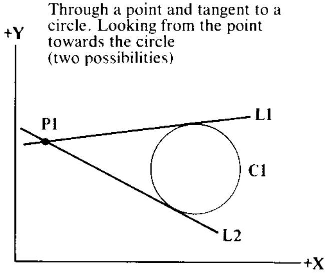
Lines *(Continued)*


Through a point and at an angle with another line

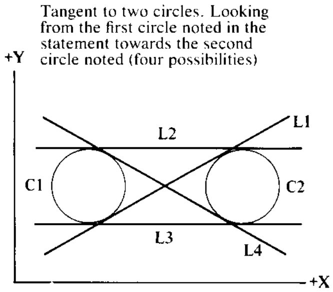
L2 = LINE/P1, ATANGL, 35.5, L1


By an angle with the X-axis and the intercept with the X-axis or the Y-axis

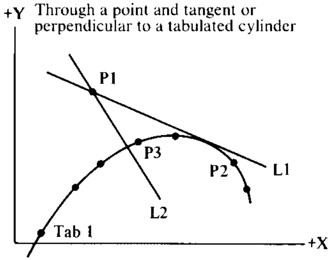
L1 = LINE/ATANGL, 20, INTERC, XAXIS, 5
L1 = LINE/ATANGL, 20, INTERC, YAXIS, 2


Through a point and tangent to a circle. Looking from the point towards the circle (two possibilities)

L1 = LINE/P1, LEFT, TANTO, C1
L2 = LINE/P1, RIGHT, TANTO, C1


Tangent to two circles. Looking from the first circle noted in the statement towards the second circle noted (four possibilities)

L1 = LINE/RIGHT, TANTO, C2, LEFT, TANTO, C1
or
L1 = LINE/RIGHT, TANTO, C1, LEFT, TANTO, C2
L2 = LINE/LEFT, TANTO, C1, LEFT, TANTO, C2
or
L2 = LINE/RIGHT, TANTO, C2, RIGHT, TANTO, C1
L3 = LINE/RIGHT, TANTO, C1, RIGHT, TANTO, C2
or
L3 = LINE/LEFT, TANTO, C2, LEFT, TANTO, C1
L4 = LINE/LEFT, TANTO, C2, RIGHT, TANTO, C1
or
L4 = LINE/LEFT, TANTO, C1, RIGHT, TANTO, C2

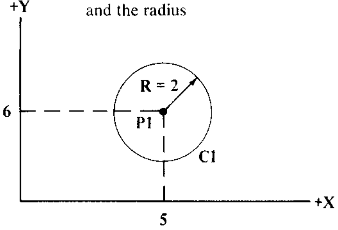

Through a point and tangent or perpendicular to a tabulated cylinder

L1 = LINE/P1, TANTO, TAB1, P3
L1 = LINE/P1, PERPTO, TAB1, P3

P2 and P3 are points close to the tangent points of L1 and the intersection point of L2, therefore cannot be end points of the tabulated cylinder

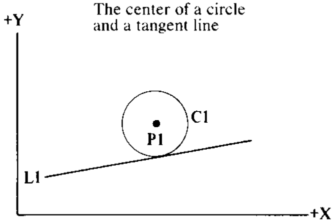
Circles

The center of the circle and the radius



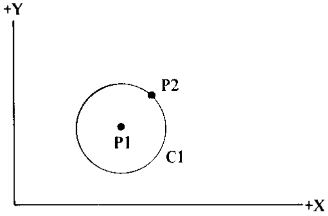
```
C1 = CIRCLE/5, 6, 2
       or
C1 = CIRCLE/5, 6, 0, 2
     (where 0 = Z ordinate)
       or
C1 = CIRCLE/CENTER, P1, RADIUS, 2
```

The center of a circle and a tangent line



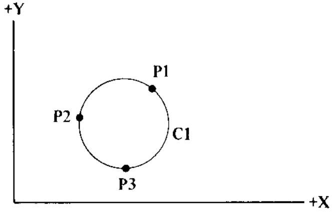
```
C1 = CIRCLE/CENTER, P1, TANTO, L1
```

The center of a circle and a point on the circumference



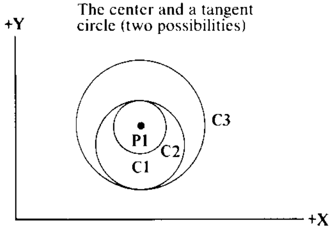
```
C1 = CIRCLE/CENTER, P1, P2
```

Three points on a circle



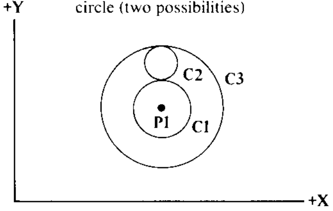
```
C1 = CIRCLE/P1, P2, P3
```

The center and a tangent circle (two possibilities)



```
C1 = CIRCLE/CENTER, P1, SMALL,
     TANTO, C2
                 or
C3 = CIRCLE/CENTER, P1, LARGE,
     TANTO, C2
```

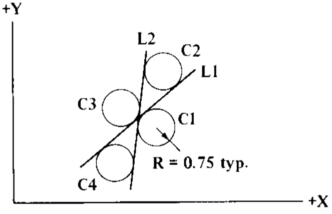
The center and a tangent circle (two possibilities)



```
C1 = CIRCLE/CENTER, P1, SMALL,
     TANTO, C2
                 or
C3 = CIRCLE/CENTER, LARGE,
     TANTO, C2
```

Circles

Tangent to two intersecting lines, given the radius (four possibilities)



C1 = CIRCLE/XLARGE, L2, YSMALL, L1, RADIUS, .75

or

C2 = CIRCLE/XLARGE, L2, YLARGE, L1, RADIUS, .75

or

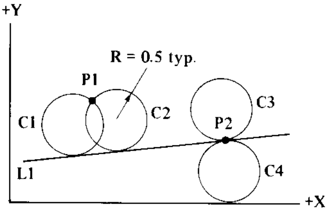
C3 = CIRCLE/XSMALL, L2, YLARGE, L1, RADIUS, .75

or

C4 = CIRCLE/XSMALL, L2, YSMALL, L1, RADIUS, .75

The modifiers (XLARGE, etc.) are used to indicate which of the four circles is wanted.

Tangent to a line and a point on the circumference, given the radius (two possibilities)



C1 = CIRCLE/TANTO, L1, XSMALL, P1, RADIUS, .5

or

C2 = CIRCLE/TANTO, L1, XLARGE, P1, RADIUS, .5

if the point lies on the line, then:

C3 = CIRCLE/TANTO, L1, YLARGE, P2, RADIUS, .5

or

C4 = CIRCLE/TANTO, L1, YSMALL, P2, RADIUS, .5

APT Motion Statements.—APT is based on the concept that a milling cutter is guided by two surfaces when in a contouring mode. Examples of these surfaces are shown in Fig. 1, and they are called the "part" and the "drive" surfaces. Usually, the partsurface guides the bottom of the cutter and the drive surface guides the side of the cutter. These surfaces may or may not be actual surfaces on the part, and although they may be imaginary to the part programmer, they are very real to the computer. The cutter is either stopped or redirected by a third surface called a check surface. If one were to look directly down on these surfaces, they would appear as lines, as shown in Figs. 2a through 2c.

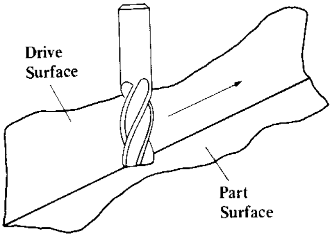


Fig. 1. Contouring Mode Surfaces

When the cutter is moving toward the check surface, it may move to it, onto it, or past it, as illustrated in Fig. 2a. When the cutter meets the check surface, it may go right, denoted by the APT command GORGT, or go left, denoted by the command GOLFT, in Fig. 2b.

Alternatively, the cutter may go forward, instructed by the command GOFWD, as in Fig. 2c. The command GOFWD is used when the cutter is moving either onto or off a tangent circular arc. These code instructions are part of what are called motion commands. Fig. 3 shows a cutter moving along a drive surface, L1, toward a check surface, L2. When it arrives at L2, the cutter will make a right turn and move along L2 and past the new check surface L3. Note that L2 changes from a check surface to a drive surface the moment the cutter begins to move along it. The APT motion statement for this move is:

GORGT/L2,PAST,L3

Contouring Cutter Movements

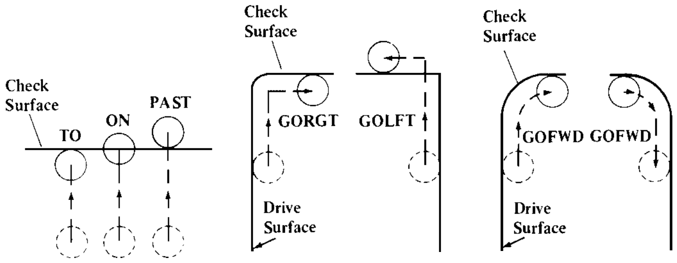


Fig. 2a. Fig. 2b. Fig. 2c.

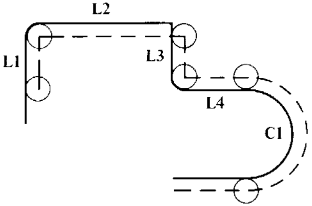


Fig. 3. Motion Statements for Movements Around a Workpiece

Still referring to Fig. 3, the cutter moves along L3 until it comes to L4. L3 now becomes the drive surface and L4 the check surface. The APT statement is:

GORGT/L3,TO,L4

The next statement is:

GOLFT/L4,TANTO,C1

Even though the cutter is moving to the right, it makes a left turn if one is looking in the direction of travel of the cutter. In writing the motion statements, the part programmers must imagine they are steering the cutter. The drive surface now becomes L4 and the check surface, C1. The next statement will therefore be:

GOFWD/C1,TANTO,L5

This movement could continue indefinitely, with the cutter being guided by the drive, part, and check surfaces.

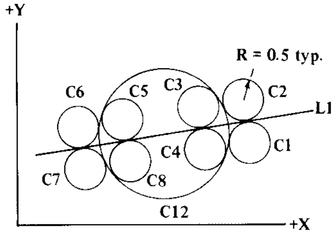
Start-Up Statements: For the cutter to move along them, it must first be brought into contact with the three guiding surfaces by means of a start-up statement. There are three different start-up statements, depending on how many surfaces are involved.

A three-surface start-up statement is one in which the cutter is moved to the drive, part, and check surfaces, as seen in Fig. 4a. A two-surface start-up is one in which the cutter is moved to the drive and part surfaces, as in Fig. 4b. A one-surface start-up is one in which the cutter is moved to the drive surface and the *X–Y* plane, where *Z* = 0, as in Fig. 4c. With the two- and one-surface start-up statements, the cutter moves in the most direct path, or perpendicular to the surfaces. Referring to Fig. 4a(three-surface start-up), the move is initiated from a point P1. The two statements that will move the cutter from P1 to the three surfaces are:

FROM/P1
GO/TO,DS,TO,PS,TO,CS

Circles

Tangent to a line and a circle, given the radius (eight possibilities)

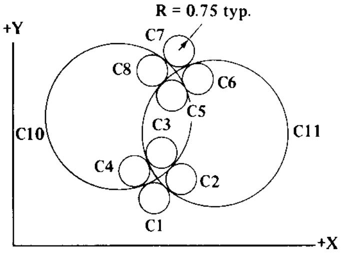


C1 = CIRCLE/YSMALL, L1, XLARGE, OUT, C12, RADIUS, .5
C2 = CIRCLE/YLARGE, L1, XLARGE, OUT, C12, RADIUS, .5
C3 = CIRCLE/YLARGE, L1, XLARGE, IN, C12, RADIUS, .5
C4 = CIRCLE/YSMALL, L1, XLARGE, IN, C12, RADIUS, .5
C5 = CIRCLE/YLARGE, L1, XSMALL, OUT, C12, RADIUS, .5
C6 = CIRCLE/XLARGE, L1, XSMALL, OUT, C12, RADIUS, .5
C7 = CIRCLE/YSMALL, L1, XSMALL, OUT, C12, RADIUS, .5
C8 = CIRCLE/YSMALL, L1, XSMALL, IN, C12, RADIUS, .5

Recommendations:

1. Note which side of line circle is on (e.g., YSMALL, L1).
2. Note whether the circle being defined is inside (IN), or outside (OUT), the known circle.
3. Of the two remaining circles, note whether the circle to be defined is XLARGE, XSMALL, or YLARGE or YSMALL, to arrive at the second modifier in the statement.

Tangent to two circles, given the radius (eight possibilities)



C1 = CIRCLE/YSMALL, OUT, C10, OUT, C11, RADIUS, .75
C2 = CIRCLE/YSMALL, OUT, C10, IN, C11, RADIUS, .75
C3 = CIRCLE/YSMALL, IN, C10, IN, C11, RADIUS, .75
C4 = CIRCLE/YSMALL, IN, C10, OUT, C11, RADIUS, .75
C5 = CIRCLE/YLARGE, IN, C10, IN, C11, RADIUS, .75
C6 = CIRCLE/YLARGE, OUT, C10, IN, C11, RADIUS, .75
C7 = CIRCLE/YLARGE, OUT, C10, OUT, C11, RADIUS, .75
C8 = CIRCLE/YLARGE, IN, C10, OUT, C11, RADIUS, .75

Recommendations

1. Apply IN, OUT modifiers.
2. Apply XLARGE, etc., modifiers.

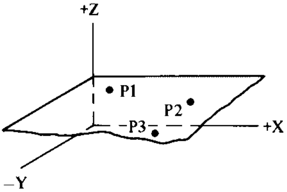
DS is used as the symbol for the Drive Surface; PS as the symbol for the Part Surface; and CS as the symbol for the Check Surface. The surfaces must be denoted in this sequence. The drive surface is the surface that the cutter will move along after coming in contact with the three surfaces. The two statements applicable to the two-surface start-up (Fig. 4b) are:

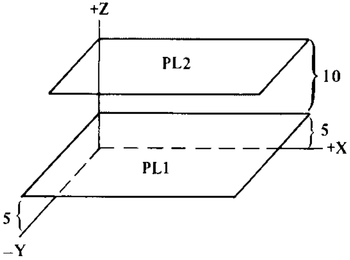
```
FROM/P1
GO/TO,DS,TO,PS
```

The one-surface start-up (Fig. 4c) is:

```
FROM/P1
GO/TO,DS
```

Planes





Planes are often used as the part surface, and are defined by three points not lying in a straight line

PL1 = PLANE/P1, P2, P3

A plane that is horizontal, or parallel to the *X-Y* plane, may be defined as:

PL1 = PLANE/0, 0, 1, 5 (0, 0, 1 does not change)

Alternatively, PL2 may be defined as a plane parallel to PL1

PL2 = PLANE/PARLEL, PL1, ZLARGE, 10

Cutter Movement Surfaces

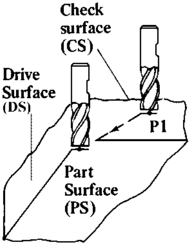


Fig. 4a.

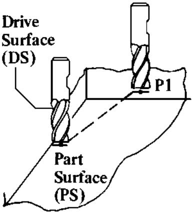


Fig. 4b.

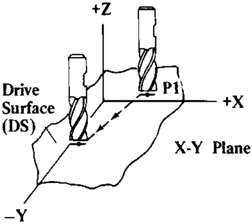
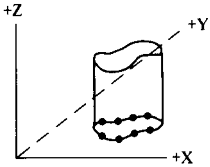
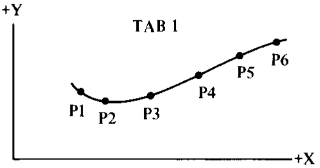


Fig. 4c.

Tabulated Cylinder





A tabulated cylinder is the line that is formed when an irregular cylinder intersects a plane. The plane intersected in the figure at the left is the *X–Y* plane.

A section of the line can be defined by a series of points on the line, as seen at the right. This line is called a TABCYL. The line must pass through all the points, therefore, it is best not to use too many. The statement to the computer would read:

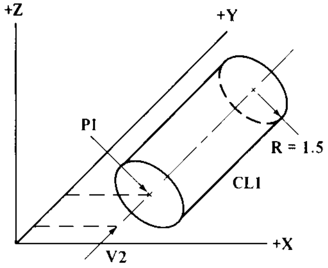
TAB1 = TABCYL/NOZ, SPLINE, P1, P2, P3, P4, P5, P6

or

TAB1 = TABCYL/NOZ, SPLINE, X., Y., X2, Y2, X3, Y3, X4, Y4, X5, Y5, X6, Y6
(where *X* and *Y* are the coordinates of the points)

3-D Geometry

Cylinder



Length of
vector = 1

A cylinder is defined by a vector, a point on the centerline, and the radius

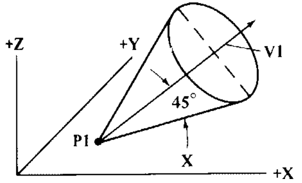
CL1 = CYLNDR/P1, V2, 1.5

where V2 is a unit vector in line with the cylinder centerline, and is described by the *X*, *Y*, and *Z* components. The cylinder centerline lies on the *X–Y* plane and is parallel to the *Y*-axis. The statement for the vector is therefore:

V2 = VECTOR/*X* component, *Y* component, *Z* component

V2 = VECTOR/0, 1, 0

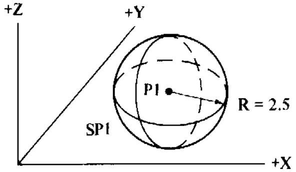
Cone



A cone is defined by its vertex, its axis as a unit vector, and the half angle (refer to cylinder for an example of a vector statement)

CON1 = CONE/P1,V1,45

Sphere



A sphere is defined by the center and the radius

SP1 = SPHERE/P1, RADIUS, 2.5

or

SP1 = SPHERE/5, 5, 3, 2.5 (where 5, 5, and 3 are the *X*, *Y*, and *Z* coordinates or P1, and 2.5 is the radius)

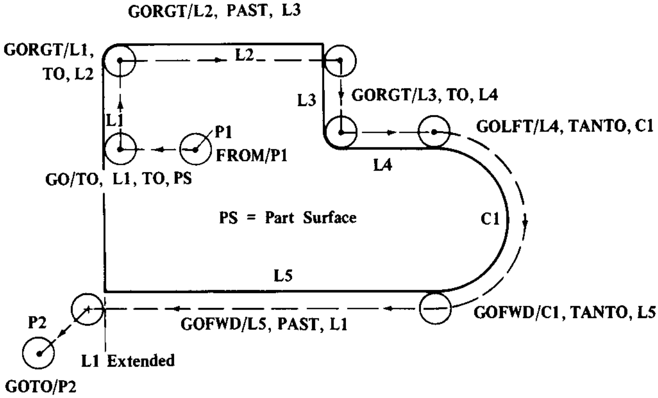


Fig. 5. A Completed Two-Surface Start-Up

Note that, in all three motion statements, the slash mark (/) lies between the GO and the TO. When the cutter is moving to a point rather than to surfaces, such as in a start-up, the statement is GOTO/ rather than GO/TO. A two-surface start-up, Fig. 3, when completed, might appear as shown in Fig. 5, which includes the motion statements needed. The motion statements, as they would appear in a part program, are shown at the left, below:

```
FROM/P1                          FROM/P1
GO/TO,L1,TO,PS                   GOTO/P2
GORGT/L1,TO,L2                   GOTO/P3
GORGT/L2,PAST,L3                 GOTO/P4
GORGT/L3,TO,L4                   GOTO/P5
GOLFT/L4,TANTO,C1                GOTO/P6
GOFWD/C1,TANTO,L5                GOTO/P7
GOFWD/L5,PAST,L1
GOTO/P2
```

GOTO statements can move the cutter throughout the range of the machine, as shown in Fig. 6. APT statements for such movements are shown at the right in the preceding example. The cutter may also be moved incrementally, as shown in Fig. 7. Here, the cutter is to move 2 inches in the + X direction, 1 inch in the + Y direction, and 1.5 inches in the + Z direction. The incremental move statement (indicated by DLTA) is:

GODLTA/2,1,1.5

The first position after the slash is the X movement; the second the Y movement, and the third, the Z movement.

Five-Axis Machining: Machining on five axes is achieved by causing the APT program to generate automatically a unit vector that is normal to the surface being machined, as shown in Fig. 8. The vector would be described by its X, Y, and Z components. These components, along with the X, Y, and Z coordinate positions of the tool tip, are fed into the postprocessor, which determines the locations and angles for the machine tool head and/or table.

APT Postprocessor Statements.—Statements that refer to the operation of the machine rather than to the geometry of the part or the motion of the cutter about the part are called postprocessor statements. APT postprocessor statements have been standardized internationally. Some common statements and an explanation of their meaning follow:

MACHIN/ Specifies the postprocessor that is to be used. Every postprocessor has an identity code, and this code must follow the slash mark (/). For example: MACHIN/LATH,82

FEDRATE/ Denotes the feed rate. If in inches per minute (ipm), only the number need be shown. If in inches per revolution (ipr), IPR must be shown, for example: FEDRAT/.005,IPR

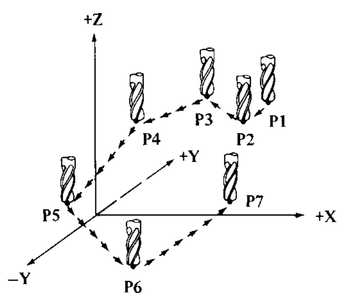


Fig. 6. A Series of GOTO Statements

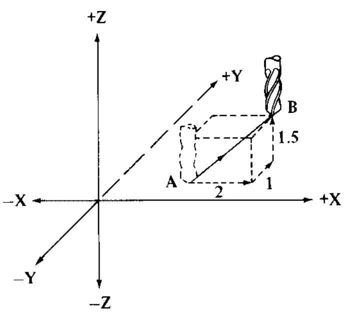


Fig. 7. Incremental Cutter Movements

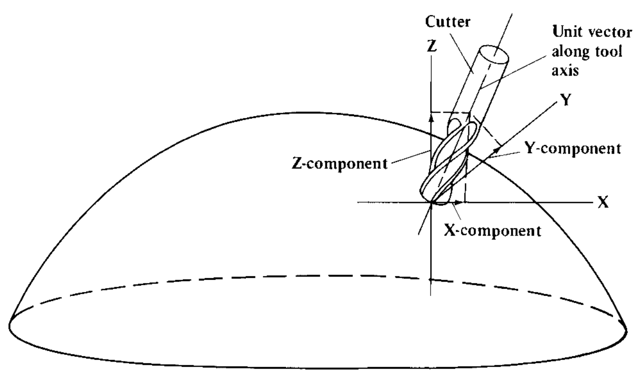


Fig. 8. Five-Axis Machining

RAPID Means rapid traverse and applies only to the statement that immediately follows it

SPINDL/ Refers to spindle speed. If in revolutions per minute (rpm), only the number need be shown. If in surface feet per minute (sfm), the letters SFM need to be shown, for example: SPINDL/ 100SFM

COOLNT/ Means cutting fluid and can be subdivided into: COOLNT/ON, COOLNT/MIST, COOLNT/FLOOD, COOLNT/OFF

TURRET/ Used to call for a selected tool or turret position

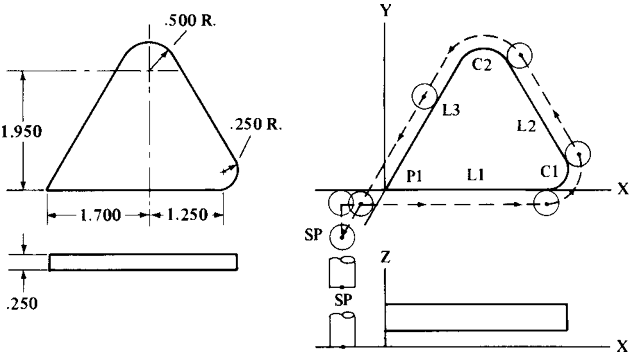


Fig. 9. Symbols for Geometrical Elements

CYCLE/ Specifies a cycle operation such as a drilling or boring cycle. An example of a drilling cycle is: CYCLE/DRILL,RAPTO,.45,FEDTO,0,IPR,.004. The next statement might be GOTO/PI and the drill will then move to P1 and perform the cycle operation. The cycle will repeat until the CYCLE/OFF statement is read

END Stops the machine but does not turn off the control system

APT Example Program.—A dimensioned drawing of a part and a drawing with the symbols for the geometry elements are shown in Fig. 9. A complete APT program for this part, starting with the statement PARTNO 47F36542 and ending with FINI, is shown at the left below.

(1) PARTNO	(1) PARTNO
(2) CUTTER/.25	(2) CUTTER/.25
(3) FEDRAT/5	(3) FEDRAT/5
(4) SP = POINT/–.5, –.5, .75	(4) SP = POINT/–.5, –.5, .75
(5) P1 = POINT/0, 0, 1	(5) P1 = POINT/0, 0, 1
(6) L1 = LINE/P1, ATANGL, 0	(6) L1 = LINE/P1, ATANGL, 0
(7) C1 = CIRCLE/(1.700 + 1.250), .250, .250	(7) C1 = CIRCLE/(1.700 + 1.250), .250, .250
(8) C2 = CIRCLE/1.700, 1.950, .5	(8) C2 = CIRCLE/1.700, 1.950, .5
(9) L2 = LINE/RIGHT, TANTO, C1, RIGHT, TANTO, C2	(9) L2 = LINE/RIGHT, TANTO, C1, RIGHT, TANTO, C2
(10) L3 = LINE/P1, LEFT, TANTO, C2	(10) L3 = LINE/P1, LEFT, TANTO, C2
(11) FROM/SP	(11) FROM/SP
(12) GO/TO, L1	(12) FRO M –.500 –.5000 .7500
(13) GORGT/L1, TANTO, C1	(13) GO/TO/, L1
(14) GOFWD/C1, TANTO, L2	(14) GT –.5000 –.1250 .0000

```
(15)  GOFWD/L2, TANTO, C2
(16)  GOFWD/C2, TANTO, L3
(17)  GOFWD/L3, PAST, L1
(18)  GOTO/SP
(19)  FINI
```

```
(15)  GORGT/L1, TANTO, C1
(16)  GT      2.9500          –.1250          .0000
(17)  GOFWD/C1, TANTO, L2
(18)  CIR     2.9500      .2500          .3750 CCLW
(19)          3.2763          .4348           .0000
(20)  GOFWD/L2, TANTO, C2
(21)  GT      2.2439          2.2580          .0000
(22)  GOFWD/C2, TANTO, L3
(23)  CIR     1.700       1.9500         .6250 CCLW
(24)          1.1584          2.2619          .0000
(25)  GOFWD/L3, PAST, L1
(26)  GT      –.2162          –.1250          .0000
(27)  GOTO/SP
(28)  GT      –.5000          –.5000          .7500
(29)  FINI
```

The numbers at the left of the statements are for reference purposes only, and are not part of the program. The cutter is set initially at a point represented by the symbol SP, having coordinates $X = –0.5$, $Y = –0.5$, $Z = 0.75$, and moves to L1 (extended) with a one-surface start-up so that the bottom of the cutter rests on the $X–Y$ plane. The cutter then moves counterclockwise around the part, past L1 (extended), and returns to SP. The coordinates of P1 are $X = 0$, $Y = 0$, and $Z = 1$.

Referring to the numbers at the left of the program:

(1) PARTNO must begin every program. Any identification can follow.

(2) The diameter of the cutter is specified. Here it is 0.25 inch.

(3) The feed rate is given as 5 inches per minute, which is contained in a postprocessor statement.

(4)–(10) Geometry statements.

(11)–(18) Motion statements.

(19) All APT programs end with FINI.

A computer printout from section II of the APT program is shown at the right, above. This program was run on a desktop personal computer. Lines (1) through (10) repeat the geometry statements from the original program. The motion statements are also repeated, and below each motion statement are shown the X, Y, and Z coordinates of the end points of the center-line (CL) movements for the cutter. Two lines of data follow those for the circular movements. For example, Line (18), which follows Line (17), GOFWD/C1,TANTO,L2, describes the X coordinate of the center of the arc, 2.9500, the Y coordinate of the center of the arc, 0.2500, and the radius of the arc required to be traversed by the cutter.

This radius is that of the arc shown on the part print, plus the radius of the cutter (0.2500 + 0.1250 = 0.3750). Line (18) also shows that the cutter is traveling in a counterclockwise (CCLW) motion. A circular motion is described in Lines (22), (23), and (24). Finally, the cutter is directed to return to the starting point, SP, and this command is noted in Line (27). The X, Y, and Z coordinates of SP are shown in Line (28).

APT for Turning.—In its basic form, APT is not a good program for turning. Although APT is probably the most suitable program for three-, four-, and five-axis machining, it is awkward for the simple two-axis geometry required for lathe operations. To overcome this problem, preprocessors have been developed especially for lathe part programming. The statements in the lathe program are automatically converted to basic APT statements in the computer and processed by the regular APT processor. An example of a lathe program, based on the APT processor and made available by the McDonnell Douglas Automation Co., is shown below. The numbers in parentheses are not part of the program, but are used only for reference. Fig. 10 shows the general set-up for the part, and Fig. 11 shows an enlarged view of the part profile with dimensions expressed along what would be the *X*- and *Y*-axes on the part print.

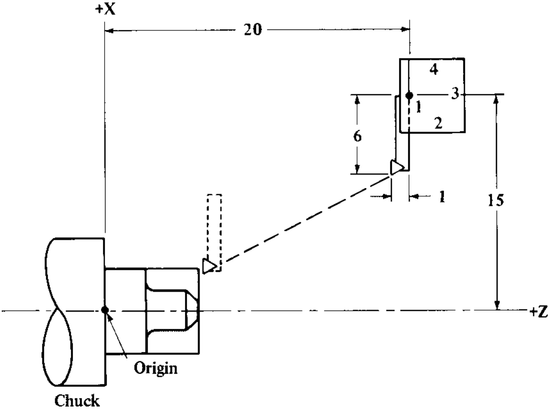


Fig. 10. Setup for APT Turning

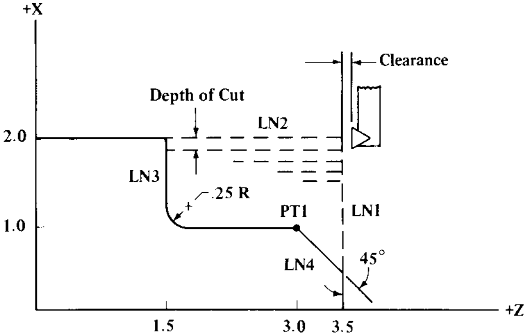


Fig. 11.

```
(1)   PARTNO LATHE EXAMPLE
(2)   MACHIN/MODEL LATHE
(3)             T1 =  TOOL/FACE, 1, XOFF, –1, YOFF, –6, RADIUS, .031
(4)        BLANK1 =  SHAPE/FACE, 3.5, TURN, 2
(5)         PART1 =  SHAPE/FACE, 3.5, TAPER, 3.5, .5, ATANGL, – 45, TURN, 1,$
                     FILLET, .25 FACE, 1.5 TURN, 2
(6)                  FROM/(20–1), (15–6)
(7)                  LATHE/ROUGH, BLANK1, PART1, STEP, .1, STOCK, .05,$
                     SFM, 300, IPR, .01, T1
(8)                  LATHE/FINISH, PART1, SFM, 400, IPR, .005, T1
(9)                  END
(10)                 FINI
```

Line (3) describes the tool. Here, the tool is located on face 1 of the turret and its tip is −1 inch "off" (offset) in the *X* direction and −6 inches "off" in the *Y* direction, when considering *X*–*Y* rather than *X*–*Z* axes. The cutting tool tip radius is also noted in this statement. Line (4) describes the dimensions of the rough material, or blank. Lines parallel to the *X*-axis are noted as FACE lines, and lines parallel to the *Z*-axis are noted as TURN lines. The FACE line (LN1) is located 3.5 inches along the *Z*-axis and parallel to the *X*-axis. The TURN line (LN2) is located 2 inches above the *Z*-axis and parallel to it. Note that in Figs. 10 and 11, the *X*-axis is shown in a vertical position and the *Z*-axis in a horizontal position. Line (5) describes the shape of the finished part. The term FILLET is used in this statement to describe a circle that is tangent to the line described by TURN, 1 and the line that is described by FACE, 1.5. The $ sign means that the statement is continued on the next line. These geometry elements must be contiguous and must be described in sequence. Line (6) specifies the position of the tool tip at the start of the operation, relative to the point of origin. Line (7) describes the roughing operation and notes that the material to be roughed out lies between BLANK1 and PART1; that the STEP, or depth of roughing cuts, is to be 0.1 inch; that 0.05 inch is to be left for the finish cut; that the speed is to be 300 sfm and the feed rate is to be 0.01 ipr; and that the tool to be used is identified by the symbol T1. Line (8) describes the finish cut, which is to be along the contour described by PART1.

Indexable Insert Holders for NC.—Indexable insert holders for numerical control lathes are usually made to more precise standards than ordinary holders. Where applicable, reference should be made to American National Standard B212.3-1986, Precision Holders for Indexable Inserts. This standard covers the dimensional specifications, styles, and designations of precision holders for indexable inserts, which are defined as tool holders that locate the gage insert (a combination of shim and insert thicknesses) from the back or front and end surfaces to a specified dimension with a ± 0.003 inch (± 0.08 mm) tolerance. In NC programming, the programmed path is that followed by the center of the tool tip, which is the center of the point, or nose radius, of the insert. The surfaces produced are the result of the path of the nose and the major cutting edge, so it is necessary to compensate for the nose or point radius and the lead angle when writing the program. Table , from B212.3, gives the compensating dimensions for different holder styles. The reference point is determined by the intersection of extensions from the major and minor cutting edges, which would be the location of the point of a sharp pointed tool. The distances from this point to the nose radius are *L*1 and *D*1; *L*2 and *D*2 are the distances from the sharp point to the center of the nose radius. Threading tools have sharp corners and do not require a radius compensation. Other dimensions of importance in programming threading tools are also given in Table 2; the data were developed by Kennametal, Inc.

Table 1. Insert Radius Compensation *ANSI B212.3-1986*

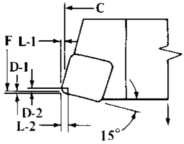
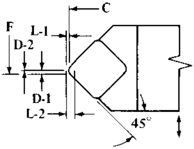
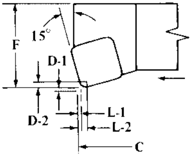
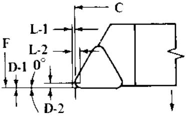
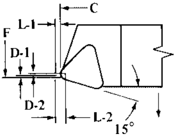
Square Profile						
B Style[a] Also Applies to R Style		Turning 15° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0035	.0191	.0009	.0110
		1/32	.0070	.0383	.0019	.0221
		3/64	.0105	.0574	.0028	.0331
		1/16	.0140	.0765	.0038	.0442
D Style[a]; Also Applies to S Style		Turning 45° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0065	.0221	.0065	0
		1/32	.0129	.0442	.0129	0
		3/64	.0194	.0663	.0194	0
		1/16	.0259	.0884	.0259	0
K Style[a];		Facing 15° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0009	.0110	.0035	.0191
		1/32	.0019	.0221	.0070	.0383
		3/64	.0028	.0331	.0105	.0574
		1/16	.0038	.0442	.0140	.0765
Triangle Profile						
G Style[a];		Turning 0° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0114	.0271	0	.0156
		1/32	.0229	.0541	0	.0312
		3/64	.0343	.0812	0	.0469
		1/16	.0458	.1082	0	.0625
B Style[a]; Also Applies to R Style		Turning and Facing 15° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0146	.0302	.0039	.0081
		1/32	.0291	.0604	.0078	.0162
		3/64	.0437	.0906	.0117	.0243
		1/16	.0582	.1207	.0156	.0324

Table 1. *(Continued)* **Insert Radius Compensation** *ANSI B212.3-1986*

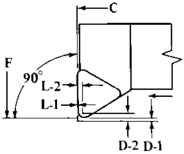
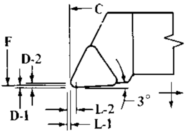
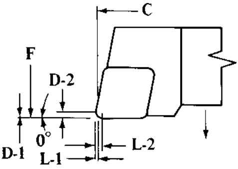
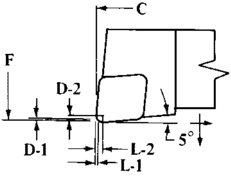
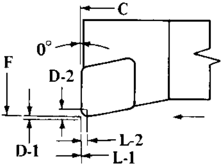
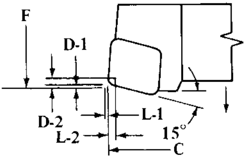
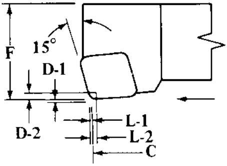
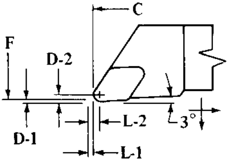
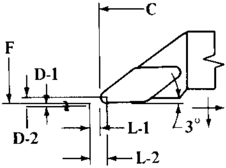
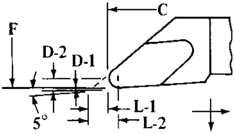
Triangle Profile *(continued)*						
F Style[a];		Facing 90° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1⁄64	0	.0156	.0114	.0271
		1⁄32	0	.0312	.0229	.0541
		3⁄64	0	.0469	.0343	.0812
		1⁄16	0	.0625	.0458	.1082
J Style[a];		Turning & Facing 3° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1⁄64	.0106	.0262	.0014	.0170
		1⁄32	.0212	.0524	.0028	.0340
		3⁄64	.0318	.0786	.0042	.0511
		1⁄16	.0423	.1048	.0056	.0681
80° Diamond Profile						
G Style[a];		Turning & Facing 0° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1⁄64	.0030	.0186	0	.0156
		1⁄32	.0060	.0312	0	.0312
		3⁄64	.0090	.0559	0	.0469
		1⁄16	.0120	.0745	0	.0625
L Style[a];		Turning & Facing 5° Reverse Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1⁄64	.0016	.0172	.0016	.0172
		1⁄32	.0031	.0344	.0031	.0344
		3⁄64	.0047	.0516	.0047	.0516
		1⁄16	.0062	.0688	.0062	.0688
F Style[a];		Facing 0° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1⁄64	0	.0156	.0030	.0186
		1⁄32	0	.0312	.0060	.0372
		3⁄64	0	.0469	.0090	.0559
		1⁄16	0	.0625	.0120	.0745

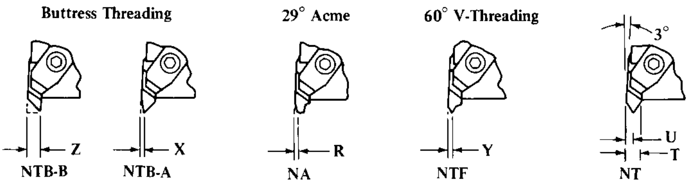
Table 1. *(Continued)* **Insert Radius Compensation** *ANSI B212.3-1986*

80° Diamond Profile *(continued)*						
R Style[a];		Turning 15° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0011	.0167	.0003	.0117
		1/32	.0022	.0384	.0006	.0234
		3/64	.0032	.0501	.0009	.0351
		1/16	.0043	.0668	.0012	.0468
K Style[a];		Facing 15° Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0003	.0117	.0011	.0167
		1/32	.0006	.0234	.0022	.0334
		3/64	.0009	.0351	.0032	.0501
		1/16	.0012	.0468	.0043	.0668
55° Profile						
J Style[a];		Profiling 3° Reverse Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0135	.0292	.0015	.0172
		1/32	.0271	.0583	.0031	.0343
		3/64	.0406	.0875	.0046	.0519
		1/16	.0541	.1166	.0062	.0687
35° Profile						
J Style[a]; Negative rake holders have 6° back rake and 6° side rake		Profiling 3° Reverse Lead Angle				
		Rad.	*L*-1	*L*-2	*D*-1	*D*-2
		1/64	.0330	.0487	.0026	.0182
		1/32	.0661	.0973	.0051	.0364
		3/64	.0991	.1460	.0077	.0546
		1/16	.1322	.1947	.0103	.0728
L Style[a];		Profiling 5° Lead Angle				
		Rad.	*L*-1	*L* -2	*D*-1	*D*-2
		1/64	.0324	.0480	.0042	.0198
		1/32	.0648	.0360	.0086	.0398
		3/64	.0971	.1440	.0128	.0597
		1/16	.1205	.1920	.0170	.0795

[a] *L*-1 and *D*-1 over sharp point to nose radius; and *L*-2 and *D*-2 over sharp point to center of nose radius. The *D*-1 dimension for the *B, E, D, M, P, S, T,* and *V* style tools are over the sharp point of insert to a sharp point at the intersection of a line on the lead angle on the cutting edge of the insert and the *C* dimension. The *L*-1 dimensions on *K* style tools are over the sharp point of insert to sharp point intersection of lead angle and *F* dimensions.

All dimensions are in inches.

Table 2. Threading Tool Insert Radius Compensation for NC Programming

Threading						
Insert Size	T	R	U	Y	X	Z
2	$\frac{5}{32}$ Wide	.040	.075	.040	.024	.140
3	$\frac{3}{16}$ Wide	.046	.098	.054	.031	.183
4	$\frac{1}{4}$ Wide	.053	.128	.054	.049	.239
5	$\frac{3}{8}$ Wide	.099	.190	…	…	…
						

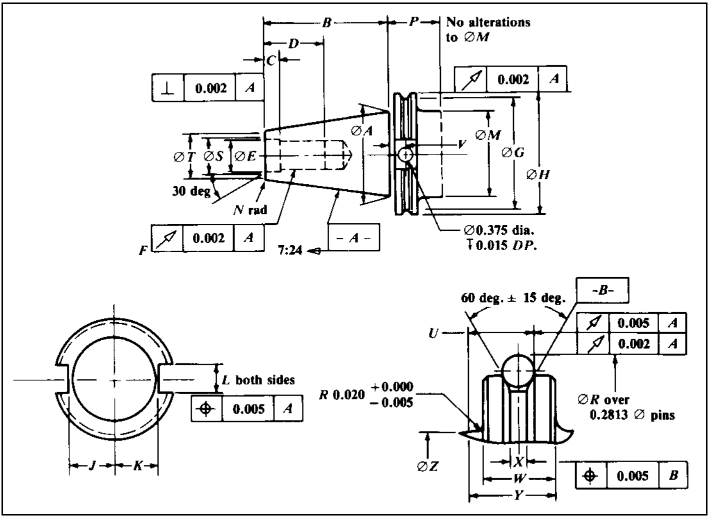
All dimensions are given in inches.

Courtesy of Kennametal, Inc.

The *C* and *F* characters are tool holder dimensions other than the shank size. In all instances, the *C* dimension is parallel to the length of the shank and the *F* dimension is parallel to the side dimension; actual dimensions must be obtained from the manufacturer. For all *K* style holders, the *C* dimension is the distance from the end of the shank to the tangent point of the nose radius and the end cutting edge of the insert. For all other holders, the *C* dimension is from the end of the shank to a tangent to the nose radius of the insert. The *F* dimension on all B, D, E, M, P, and V style holders is measured from the back side of the shank to the tangent point of the nose radius and the side cutting edge of the insert. For all A, F, G, J, K, and L style holders, the *F* dimension is the distance from the back side of the shank to the tangent of the nose radius of the insert. In all these designs, the nose radius is the standard radius corresponding to those given in the paragraph *Cutting Point Configuration* on page 732.

V-Flange Tool Shanks and Retention Knobs.—Dimensions of ANSI B5.18-1972 (R1998) standard tool shanks and corresponding spindle noses are detailed on pages 920 through 924, and are suitable for spindles used in milling and associated machines. Corresponding equipment for higher-precision numerically controlled machines, using retention knobs instead of drawbars, is usually made to the ANSI/ASME B5.50-1985 standard.

Essential Dimensions of V-Flange Tool Shanks *ANSI/ASME B5.50-1985*

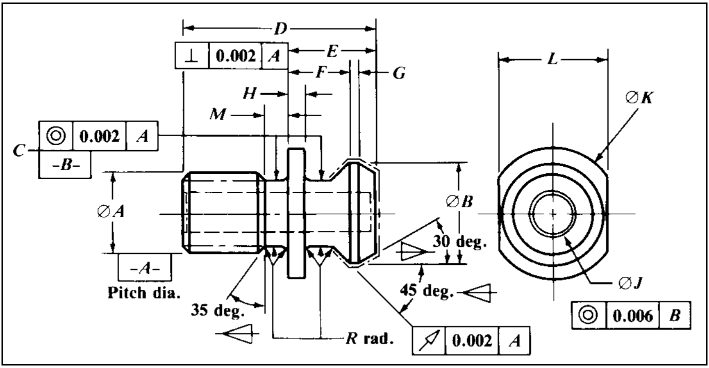


	A	*B*	*C*	*D*	*E*	*F*	*G*	*H*	*J*	*K*
	Tolerance	±0.005	±0.010	Min.	+ 0.015 −0.000	UNC 2B	±0.010	±0.002	+0.000 −0.015	+0.000 −0.015
Size	Gage Dia.									
30	1.250	1.875	0.188	1.00	0.516	0.500-13	1.531	1.812	0.735	0.640
40	1.750	2.687	0.188	1.12	0.641	0.625-11	2.219	2.500	0.985	0.890
45	2.250	3.250	0.188	1.50	0.766	0.750-10	2.969	3.250	1.235	1.140
50	2.750	4.000	0.250	1.75	1.031	1.000-8	3.594	3.875	1.485	1.390
60	4.250	6.375	0.312	2.25	1.281	1.250-7	5.219	5.500	2.235	2.140

	A	*L*	*M*	*N*	*P*	*R*	*S*	*T*	*Z*
	Tolerance	±0.001	±0.005	+0.000 −0.015	Min.	±0.002	±0.010	Min. Flat	+0.000 −0.005
Size	Gage Dia.								
30	1.250	0.645	1.250	0.030	1.38	2.176	0.590	0.650	1.250
40	1.750	0.645	1.750	0.060	1.38	2.863	0.720	0.860	1.750
45	2.250	0.770	2.250	0.090	1.38	3.613	0.850	1.090	2.250
50	2.750	1.020	2.750	0.090	1.38	4.238	1.125	1.380	2.750
60	4.250	1.020	4.250	0.120 0.200	1.500	5.683	1.375	2.04	4.250

Notes: Taper tolerance to be 0.001 in. in 12 in. applied in direction that increases rate of taper. Geometric dimensions symbols are to ANSI Y14.5M-1982. Dimensions are in inches. Deburr all sharp edges. Unspecified fillets and radii to be $0.03 \pm 0.010R$, or $0.03 \pm 0.010 \times 45$ degrees. Data for size 60 are not part of Standard. For all sizes, the values for dimensions U (tol. ± 0.005) are 0.579: for V (tol. ± 0.010), 0.440; for W (tol. ± 0.002), 0.625; for X (tol. ± 0.005), 0.151; and for Y (tol. ± 0.002), 0.750.

Essential Dimensions of V-Flange Tool Shank Retention Knobs
ANSI/ASME B5.50-1985



Size/ Totals	*A*	*B*	*C*	*D*	*E*	*F*
	UNC 2A	±0.005	±0.005	±0.040	±0.005	±0.005
30	0.500-13	0.520	0.385	1.10	0.460	0.320
40	0.625-11	0.740	0.490	1.50	0.640	0.440
45	0.750-10	0.940	0.605	1.80	0.820	0.580
50	1.000-8	1.140	0.820	2.30	1.000	0.700
60	1.250-7	1.460	1.045	3.20	1.500	1.080

Size/ Totals	*G*	*H*	*J*	*K*	*L*	*M*	*R*
	±0.010	±0.010	±0.010		+0.000 −0.010	±0.040	+0.010 −0.005
30	0.04	0.10	0.187	0.65 0.64	0.53	0.19	0.094
40	0.06	0.12	0.281	0.94 0.92	0.75	0.22	0.094
45	0.08	0.16	0.375	1.20 1.18	1.00	0.22	0.094
50	0.10	0.20	0.468	1.44 1.42	1.25	0.25	0.125
60	0.14	0.30	0.500	2.14 2.06	1.50	0.31	0.125

Notes: Dimensions are in inches. Material: low-carbon steel. Heat treatment: carburize and harden to 0.016 to 0.028 in. effective case depth. Hardness of noted surfaces to be Rockwell 56-60; core hardness Rockwell C35-45. Hole *J* shall not be carburized. Surfaces *C* and *R* to be free from tool marks. Deburr all sharp edges. Geometric dimension symbols are to

ANSI Y14.5M-1982. Data for size 60 are not part of Standard.

CAD/CAM

CAD/CAM.—CAD in engineering means computer-aided design using a computer graphics system to develop mechanical, electrical/electronic, and architectural designs. A second D (CADD) is sometimes added (computer-aided drafting and design) and simply indicates a mechanical drafting or drawing program. CAD technology is the foundation for a wide variety of engineering, design, drafting, analysis, and manufacturing activities. Often a set of drawings initially developed in the design phase of a project is also used for analyzing and optimizing the design, creating mechanical drawings of parts and assemblies and for generating NC/CNC part programs that control machining operations. Formerly, after a component had been designed with CAD, the design was passed to a part programmer who developed a program for machining the components, either manually or directly on the computer (graphic) screen, but the process often required redefining and reentering part geometry. This procedure is often regarded as the CAM part of CAD/CAM, although CAM (for computer-aided manufacturing) has a much broader meaning and involves the computer in many other manufacturing activities such as factory simulation and planning analyses. Improvements in the speed and capability of computers, operating systems, and programs (including, but not limited to CAD) have simplified the process of integrating the manufacturing process and passing drawings (revised, modified, and translated, as necessary) through the design, analysis, simulation, and manufacturing stages.

A CAD drawing is a graphic representation of part geometry data stored in a drawing database file. The drawing database generally contains the complete list of entity (line, arc, etc.) and coordinate information required to build the CAD drawing, and additional information that may be required to define solid surfaces and other model characteristics. The format of data in a drawing file depends on the CAD program used to create the file. Generally, drawings are not directly interchangeable between drawing programs, however, drawings created in one system can usually be translated into an intermediate format or file type, such as DXF, that allows some of the drawing information to be exchanged between different programs. Translation frequently results in some loss of detail or loss of other drawing information because the various drawing programs do not all have the same features. The section Drawing Exchange Standards covers some of the available methods of transferring drawing data between different CAD programs.

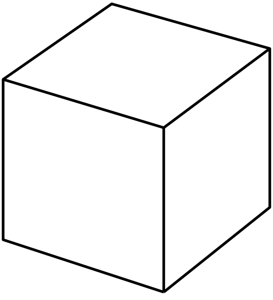


Fig. 1. Simple Wireframe Cube with Hidden Lines Automatically Removed

The simplest CAD drawings are two-dimensional and conform to normal engineering drafting practice showing orthographic (front, top, and side views, for example), exploded, isometric, or other views of a component. Depending on the complexity of the part and machining requirements, two-dimensional drawings are often sufficient for use in developing NC/CNC part programs. If a part can be programmed within a two-dimensional

CAD framework, a significant cost saving may be realized because 3-D drawings require considerably more time, drawing skill, and experience to produce than 2-D drawings.

Wireframes are the simplest two- and three-dimensional forms of drawing images and are created by defining all edges of a part and, where required, lines defining surfaces. Wireframe drawing elements consist primarily of lines and arcs that can be used in practically any combination. A wireframe drawing of a cube, as in Fig. 1, consists of 12 lines of equal length (some are hidden and thus not shown), each perpendicular to the others. Information about the interior of the cube and the character of the surfaces is not included in the drawing. With such a system, if a 1-inch cube is drawn and a 0.5-inch cylinder is required to intersect the cube's surface at the center of one of its faces, the intersection points cannot be determined because nothing is known about the area between the edges. A wireframe model of this type is ambiguous if the edges overlap or do not meet where they should. Hidden-line removal can be used to indicate the relative elevations of the drawing elements, but normally a drawing cannot be edited when hidden lines have been removed. Hidden lines can be shown dashed or can be omitted from the view.

Two-dimensional drawing elements, such as lines, arcs, and circles, are constructed by directly or indirectly specifying point coordinates, usually x and y, that identify the location, size, and orientation of the entities. Three-dimensional drawings are also made up of a collection of lines, arcs, circles, and other drawing elements and are stored in a similar manner. A third point coordinate, z, indicates the elevation of a point in 3-D drawings. On the drawing screen, working in the x-y plane, the elevation is commonly thought of as the distance of a point or object into the screen (away from the observer) or out of the viewing screen (toward the observer). Coordinate axes are oriented according to the right-hand rule: If the fingers of the right hand point in the direction from the positive x-axis to the positive y-axis, the thumb of the right hand points in the direction of the positive z-axis.

Assigning a thickness (or extruding) to objects drawn in two dimensions quickly gives some 3-D characteristics to an object and can be used to create simple prismatic 3-D shapes, such as cubes and cylinders. Usually, the greatest difficulty in creating 3-D drawings is in picking and visualizing the three-dimensional points in a two-dimensional workspace (the computer display screen). To assist in the selection of 3-D points, many CAD programs use a split or windowed screen drawing area that can simultaneously show different views of a drawing. Changes made in the current or active window are reflected in each of the other windows. A typical window setup might show three orthogonal (mutually perpendicular) views of the drawing and a perspective or 3-D view. Usually, the views shown can be changed as required to suit the needs of the operator.

If carefully constructed, wireframe images may contain enough information to completely define the external geometry of simple plane figures. Wireframe images are especially useful for visualization of 3-D objects and are effectively used during the design process to check fits, clearances, and dimensional accuracy. Parts designed to be used together can be checked for accuracy of fit by bringing them together in a drawing, superimposing the images, and graphically measuring clearances. If the parts have been designed or drawn incorrectly, the errors will frequently be obvious and appropriate corrections can be made.

A more complicated level of 3-D drawing involves solids, with sections of the part being depicted on the screen as solid geometrical structures called primitives, such as cylinders, spheres, and cubes. Primitives can be assembled on a drawing to show more complex parts. Three distinct forms of image may be generated by 3-D systems, although not all systems make use of all three.

Surface Images: A surface image defines not only the edges of the part, but also the "skin" of each face or surface. For the example mentioned previously, the intersection for the 0.5-inch cylinder would be calculated and drawn in position. Surface models are necessary for designing free-form objects such as automotive body panels and plastics injection moldings used in consumer goods. For a surface model, the computer must be provided with much more information about the part in addition to the *x*, *y*, *z* coordinates defining each point, as in a wireframe. This information may include tangent vectors, surface normals, and weighting that determines how much influence one point has on another, twists, and other mathematical data that define abstract curves, for instance. Fig. 2 shows a typical 3-D surface patch.

Shaded images may be constructed using simulated light sources, reflections, colors, and textures to make renderings more lifelike. Surface images are sometimes ambiguous, with surfaces that overlap or miss each other entirely. Information about the interior of the part, such as the center of gravity or the volume, also may not be available, depending on the CAD package.

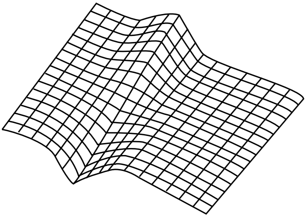


Fig. 2. A 3-D Surface Patch

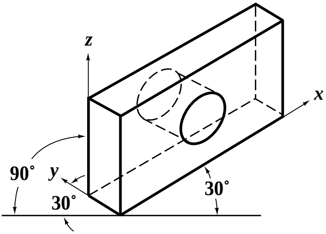


Fig. 3. Isometric Drawing Showing Orientation of Principle Drawing Axes

Solid Images: A solid image is the ultimate electronic representation of a part, containing all the necessary information about edges, surfaces, and the interior. Most solid-imaging programs can calculate volume, center of mass, centroid, and moment of inertia. Several methods are available for building a solid model. One method is to perform Boolean operations on simple shapes such as cylinders, cones, cubes, and blocks. Boolean operations are used to union (join), difference (subtract one from another), and intersect (find the common volume between two objects). Thus, making a hole in a part requires subtracting a cylinder from a rectangular block. This type of program is called constructive solid geometry (CSG).

The boundary representation type of imaging program uses profiles of 2-D shapes that it extrudes, rotates, and otherwise translates in 3-D space to create the required solid. Sometimes combinations of the above two programs are used to attain a blend of flexibility, accuracy, and performance. For more precision, greatly increased time is needed for calculations, so compromises sometimes are needed to maintain reasonable productivity. Solid images may be sliced or sectioned on the screen to provide a view of the interior. This type of image is also useful for checking fit and assembly of one part with another.

Solid images provide complete, unambiguous representation of a part, but the programs require large amounts of computer memory. Each time a Boolean operation is performed, the list of calculations that must be done to define the model becomes longer, so that computation time increases.

Drawing Projections.—Several different techniques are used to display objects on paper or a computer screen to give an accurate three-dimensional appearance. Several of these methods are commonly used in CAD drawings.

Isometric drawings, as in Fig. 3, can be used to good effect for visualizing a part because they give the impression of a 3-D view and are often much faster to create. Isometric drawings are created in 2-D space, with the *x*- and *y*-axes being inclined at 30 degrees to the horizontal, as shown in Fig. 3, and the *z*-axis as vertical. Holes and cylinders in isometric drawings become elliptical. Because of the orientation of the *x*-, *y*-, and *z*-axes, the true length of lines may not be accurately represented in isometric drawings and dimensions should not be taken directly from a print. Some CAD programs have a special set of predefined drawing axes to facilitate creating isometric drawings.

In *parallel projections,* lines that are parallel in an object, assembly, or part being portrayed remain parallel in the drawing. Parallel projections show 3-D objects in a dimensionally correct manner, so that relative and scaled dimensions may be taken directly from a drawing. However, drawings may not appear as realistic as isometric or perspective drawings.

A characteristic of *perspective drawings* is that parallel lines converge (see Fig. 4) so that objects that are farther away from the observer appear smaller. Perspective drawing techniques are used in some three-dimensional drawings to convey the true look of an object, or group of objects. Because objects in perspective drawings are not drawn to scale, dimensional information cannot be extracted from the drawings of a part. Some 3-D drawing packages have a true perspective drawing capability, and others use a simulation technique to portray a 3-D perspective.

An *axonometric projection* is a 3-D perpendicular projection of an object onto a surface, such that the object is tilted relative to its normal orientation. An axonometric projection of a cube, as in Fig. 1, shows three faces of the cube. CAD systems are adept at using this type of view, making it easy to see an object from any angle.

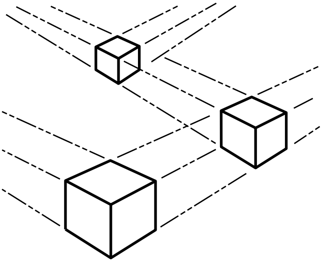


Fig. 4. Perspective Drawing of Three Equal-Size Cubes and Construction Lines

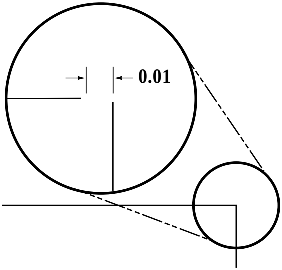


Fig. 5. A Common Positioning Error

Drawing Tips and Traps.—Images sometimes appear correct on the screen but contain errors that show up when the drawing is printed or used to produce NC/CNC part programs. In Fig. 5, the two lines within the smaller circle appear to intersect at a corner, but when the view of the intersection is magnified, as in the larger circle, it is clear that the lines actually do not touch. Although an error of this type may not be easily visible, other parts placed in the drawing relative to this part will be out of position.

A common problem that shows up in plotting, but is difficult to detect on the screen, comes from placing lines in the same spot. When two or more lines occupy exactly the same location on the screen, there is usually no noticeable effect on the display. However, when the drawing is plotted, each line is plotted separately, causing the single line visible to become thicker and darker. Likewise, if a line that appears continuous on the screen is actually made up of several segments, plotting the line will frequently result in a broken, marred, or blotted appearance to the line because the individual segments are plotted separately, and at different times. To avoid these problems and to get cleaner looking plots, replace segmented lines with single lines and avoid constructions that place one line directly on top of another.

Exact decimal values should be used when entering point coordinates from the keyboard, if possible; fractional sizes should be entered as fractions, not truncated decimals. For example, $^5⁄_{16}$ should be entered as 0.3125 or $^5⁄_{16}$, not 0.313. Accumulated rounding errors and surprises later on when parts do not fit are thus reduced. Drawing dimensions, on the other hand, should not have more significant digits or be more precise than necessary. Unnecessary precision in dimensioning leads to increased difficulty in the production stage because the part has to be made according to the accuracy indicated on the drawing.

Snap and *object snap* commands make selecting lines, arcs, circles, or other drawing entities faster, easier, and more accurate when picking and placing objects on the screen. Snap permits only points that are even multiples of the snap increment to be selected by the pointer. A $\frac{1}{8}$-inch snap setting, for example, will allow points to be picked at exactly $\frac{1}{8}$-inch intervals. Set the *snap increment* to the smallest distance increment (1 in., $\frac{1}{4}$ in., 1 ft., etc.) being used in the area of the drawing under construction and reset the snap increment frequently, if necessary. The snap feature can be turned off during a command to override the setting or to select points at a smaller interval than the snap increment allows. Some systems permit setting a different snap value for each coordinate axis.

The *object snap* selection mode is designed to select points on a drawing entity according to predefined characteristics of the entity. For example, if end-point snap is in effect, picking a point anywhere along a line will select the end point of the line nearest the point picked. Object snap modes include point, intersection, midpoint, center and quadrants of circles, tangency point (allows picking a point on an arc or circle that creates a tangent to a line), and perpendicular point (picks a point that makes a perpendicular from the base point to the object selected). When two or more object snap modes are used together, the nearest point that meets the selection criteria will be chosen. Using object snap will greatly reduce the frequency of the type of problem shown in Fig. 5.

Copy: Once drawn, avoid redrawing the same object. It is almost always faster to copy and modify a drawing than to draw it again. The basic copy commands are: copy, array, offset, and mirror. Use these, along with move and rotate and the basic editing commands, to modify existing objects. Copy and move should be the most frequently used commands. If possible, create just one instance of a drawing object and then copy and move it to create others.

To create multiple copies of an object, use the *copy, multiple* feature to copy selected objects as many times as required simply by indicating the destination points. The *array* command makes multiple copies of an object according to a regular pattern. The rectangular array produces rows and columns, and the polar array puts the objects into a circular pattern, such as in a bolt circle. *Offset* copies an entity and places the new entity a specified distance from the original and is particularly effective at placing parallel lines and curves, and for creating concentric copies of closed shapes. *Mirror* creates a mirror image copy of an object, and is useful for making right- and left-hand variations of an object as well as for copying objects from one side of an assembly to the other. In some CAD programs, a system variable controls whether text is mirrored along with other objects.

Many manufacturers distribute drawings of their product lines in libraries of CAD drawings, usually as DXF files, that can be incorporated into existing drawings. The suitability of such drawings depends on the CAD program and drawing format being used, the skill of the technician who created the drawings, and the accuracy of the drawings. A typical example, Fig. 6, shows a magnetically coupled actuator drawing distributed by Tol-O-Matic, Inc. Libraries of frequently used drawing symbols and blocks are also available from commercial sources.

Create Blocks of Frequently Used Objects: Once created, complete drawings or parts of drawings can be saved and later recalled, as needed, into another drawing. Such objects can be scaled, copied, stretched, mirrored, rotated, or otherwise modified without changing the original. When shapes are initially drawn in unit size (i.e., fitting within a 1 × 1 square) and saved, they can be inserted into any drawing and scaled very easily. One or more individual drawing elements can be saved as a group element, or *block*, that can be manipulated in a drawing as a single element. Block properties vary, depending on the drawing program, but are among the most powerful features of CAD. Typically, blocks are uniquely named and, as with simple objects, may be saved in a file on the disk. Blocks are ideal for creating libraries of frequently used drawing symbols. Blocks can be copied, moved, scaled very easily, rotated, arrayed, and inserted as many times as is required in a drawing and manipulated as one object. When scaled, each object within the block is also scaled to the same degree.

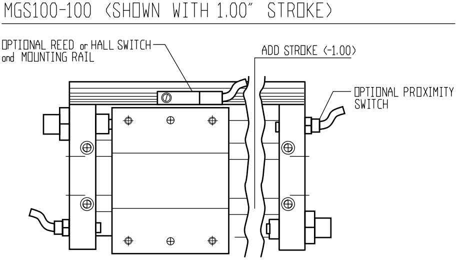


Fig. 6. Manufacturer's Drawing of a Magnetically Coupled Actuator (*Courtesy of Tol-O-Matic, Inc.*)

When a family of parts is to be drawn, create and block a single drawing of the part that fits within a unit cube of convenient size, such as $1 \times 1 \times 1$. When the block is inserted in a drawing, it is scaled appropriately in the x-, y-, and z-directions. For example, $\frac{3}{8}$-inch bolts can be drawn 1 inch long in the x-direction and $\frac{3}{8}$-inch in diameter in the y-z plane. If a 5-inch bolt is needed, insert the "bolt" block with a scale of 5 in the x-direction and a scale of 1 in the y- and z-directions.

Once blocked, the individual components of a block (lines, arcs, circles, surfaces, and text, for example) cannot be individually changed or edited. To edit a block, a copy (instance) of the block must be *exploded* (unblocked) to divide it into its original components. Once exploded, all the individual elements of the block (except other blocks) can be edited. When the required changes have been made, the block must be redefined (redeclared as a block by giving it a name and identifying its components). If the block is redefined using the same name, any previous references to the block in the drawing will be updated to match the redefined block. For example, an assembly drawing is needed that shows a mechanical frame with 24 similar control panels attached to it. Once one of the panels is drawn and defined as a block (using the name *PANEL*, for instance), the block can be inserted (or copied) into the drawing 24 times. Later, if changes need to be made to the panel design, one instance of the block *PANEL* can be exploded, modified, and redefined with the name *PANEL*. When *PANEL* is redefined, every other copy of the *PANEL* block in the drawing is also redefined, so every copy of *PANEL* in the drawing is updated. On the other hand, if the block was redefined with a different name, say, *PANEL1*, existing copies of *PANEL* remain unchanged. When redefining a block that already exists in the drawing, be sure to use the same insertion point that was used for the original definition of the block; otherwise, the positions of existing blocks with the same name will be changed.

Use of Text Attributes to Request Drawing Information Automatically: Text attributes are a useful method for attaching textual information to a particular part or feature of a drawing. An attribute is basically a text variable that has a name and can be assigned a value. Attributes are created by defining attribute characteristics such as a name, location in the drawing, text size and style, and default value. The attribute value is assigned when the attribute is inserted into a drawing as part of a block.

Fig. 7 shows two views of a title block for size A to C drawing sheets. The upper figure includes the title block dimensions (included only for reference) and the names and locations of the attributes (COMPANY, TITLE1, TITLE2, etc.). When a block containing text attributes is inserted in a drawing, the operator is asked to enter the value of each attribute.

To create this title block, first draw the frame of the title block and define the attributes (name, location and default value for: company name and address, drawing titles [2 lines], drawing size, drawing number, revision number, scale, and sheet number). Finally, create and name a block containing the title frame and all the attribute definitions (do not include the dimensions).

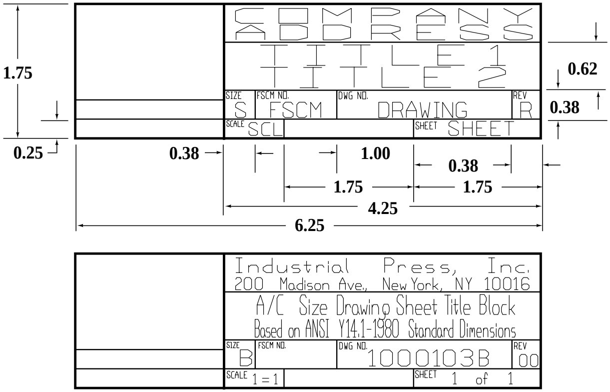


Fig. 7. Title Block for A to C Size Drawing Sheets Showing the Placement of Text Attributes. The Lower Figure Shows the Completed Block

When the block is inserted into a drawing, the operator is asked to enter the attribute values (such as company name, drawing title, etc.), which are placed into the title block at the predetermined location. The lower part of Fig. 7 shows a completed title block as it might appear inserted in a drawing. A complete drawing sheet could include several additional blocks, such as a sheet frame, a revision block, a parts list block, and any other supplementary blocks needed. Some of these blocks, such as the sheet frame, title, and parts list blocks, might be combined into a single block that could be inserted into a drawing at one time.

Define a Default Drawing Configuration: Drawing features that are commonly used in a particular type of drawing can be set up in a template file so that frequently used settings, such as text and dimension styles, text size, drawing limits, initial view, and other default settings, are automatically set up when a new drawing is started. Different configurations can be defined for each frequently used drawing type, such as assembly, parts, or printed circuit drawings. When creating a new drawing, use one of the template files as a pattern or open a template file and use it to create the new drawing, saving it with a new name.

Scaling Drawings: Normally, for fast and accurate drawing, it is easiest to draw most objects full scale, or with a 1:1 scale. This procedure greatly simplifies creation of the initial drawing, and ensures accuracy, because scale factors do not need to be calculated. If it becomes necessary to fit a large drawing onto a small drawing sheet (for example, to fit a 15 × 30 inch assembly onto a 11 × 17 inch, B-sized, drawing sheet), the drawing sheet can be scaled larger to fit the assembly size. Likewise, large drawing sheets can be scaled down to fit small drawings. The technique takes some practice, but it permits the drawing assembly to be treated full scale. If editing is required at a later date (to move something or add a hole in a particular location, for example), changes can be made without rescaling and dimensions can be taken directly from the unscaled drawing on the computer.

Scaling Text on Drawing Sheets: It is usually desirable that text, dimensions, and a few other features on drawings stay a consistent size on each sheet, even when the drawing size is very different. The following procedure ensures that text and dimensions (other features as well, if desired) will be the same size, from drawing to drawing without resorting to scaling the drawing to fit onto the drawing sheet.

Create a drawing sheet having the exact dimensions of the actual sheet to be output (A, B, C, D, or E size, for example). Use text attributes, such as the title block illustrated in Fig. 7, to include any text that needs to be entered each time the drawing sheet is used. Create a block of the drawing sheet, including the text attributes, and save the block to disk. Repeat for each size drawing sheet required.

Establish the nominal text and dimension size requirements for the drawing sheet when it is plotted full size (1:1 scale). This is the size text that will appear on a completed drawing. Use Table 1 as a guide to recommended text sizes of various drawing features.

Table 1. Standard Sizes of Mechanical Drawing Lettering *ANSI Y14.2M–1992*

Use For	Inch		Metric	
	Min. Letter Heights, (in)	Drawing Size	Min. Letter Heights, (mm)	Drawing Size
Drawing title, drawing size, CAGE Code, drawing number, and revision letter[a]	0.24	D, E, F, H, J, K	6	A0, A1
	0.12	A, B, C, G	3	A2, A3, A4
Section and view letters	0.24	All	6	All
Zone letters and numerals in borders	0.24	All	6	All
Drawing block headings	0.10	All	2.5	All
All other characters	0.12	All	3	All

[a] When used within the title block.

Test the sheet by setting the text size and dimension scale variables to their nominal values (established above) and place some text and dimensions onto the drawing sheet. Plot a copy of the drawing sheet and check that text and dimensions are the expected size.

To use the drawing sheet, open a drawing to be placed on the sheet and insert the sheet block into the drawing. Scale and move the sheet block to locate the sheet relative to the drawing contents. When scaling the sheet, try to use whole-number scale factors (3:1, 4:1, etc.), if possible; this will make setting text size and dimension scale easier later on. Set the text-size variable equal to the nominal text size multiplied by the drawing sheet insertion scale (for example, for 0.24 text height on a drawing sheet scaled 3:1, the text-size variable will be set to $3 \times 0.24 = 0.72$). Likewise, set the dimension-scale variable equal to the nominal dimension size multiplied by the drawing sheet insertion scale.

Once the text size and dimensions scale variables have been set, enter all the text and dimensions into the drawing. If text of another size is needed, multiply the *new* nominal text size by the sheet scale to get the actual size of the text to use in the drawing.

Use Appropriate Detail: Excessive detail may reduce the effectiveness of the drawing, increase the drawing time on individual commands and the overall time spent on a drawing, and reduce performance and speed of the CAD program. Whenever possible, symbolic drawing elements should be used to represent more complicated parts of a drawing unless the appearance of that particular component is essential to the drawing.

Drawing everything to scale often serves no purpose but to complicate a drawing and increase drawing time. The importance of detail depends on the purpose of a drawing, but detail in one drawing is unnecessary in another. For example, the slot size of a screw head (length and width) varies with almost every size of screw. If the purpose of a drawing is to show the type and location of the hardware, a symbolic representation of a screw is usually all that is required. The same is generally true of other screw heads, bolt threads, bolt head diameters and width across the flats, wire diameters, and many other hardware features.

Drawing Exchange Standards.—The ability to transfer working data between different CAD, CAD/CAM, design analysis, and NC/CNC programs is one of the most important requirements of engineering drawing programs. Once an engineer, designer, draftsman, or machinist enters relevant product data into his or her machine (computer or machine tool), the information defining the characteristics of the product should be available to the others involved in the project without recreating or reentering it. In view of manufacturing goals of reducing lead time and increasing productivity, concurrent engineering, and improved product performance, interchangeable data are a critical component in a CAD/CAM program. Depending on the requirements of a project, it may be entirely possible to transfer most if not all of the necessary product drawings from one drawing system to another.

IGES stands for Initial Graphics Exchange Specification and is a means of exchanging or converting drawings and CAD files for use in a different computer graphics system. The concept is shown diagrammatically in Fig. 8. Normally, a drawing prepared on the computer graphics system supplied by company A would have to be redrawn before it would operate on the computer graphics system supplied by company B. However, with IGES, the drawing can be passed through a software package called a preprocessor that converts it into a standardized IGES format that can be stored on a magnetic disk. A postprocessor at company B is then used to convert the standard IGES format to that required for their graphics system. Both firms would be responsible for purchasing or developing their own preprocessors and postprocessors, to suit their own machines and control systems. Almost all the major graphics systems manufacturing companies today either have or are developing IGES preprocessor and postprocessor programs to convert software from one system to another.

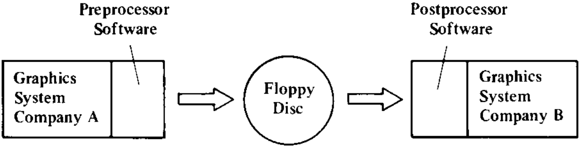


Fig. 8.

DXF stands for Drawing Exchange Format and is a pseudo-standard file format used for exchanging drawings and associated information between different CAD and design analysis programs. Nearly all two- and three-dimensional CAD programs support some sort of drawing exchange through the use of DXF files, and most can read and export DXF files. There are, however, differences in the drawing features supported and the manner in which the DXF files are handled by each program. For example, if a 3-D drawing is exported in the DXF format and imported into a 2-D CAD program, some loss of information results because all the 3-D features are not supported by the 2-D program, so that most attempts to make a transfer between such programs fail completely. Most common drawing entities (lines, arcs, etc.) will transfer successfully, although other problems may occur. For example, drawing entities that are treated as a single object in an original drawing (such as blocks, hatch patterns, and symbols) may be divided into hundreds of individual components when converted into a DXF file. Consequently, such a drawing may become much more difficult to edit after it is transferred to another drawing program.

ASCII stands for American Standard Code for Information Interchange. ASCII is a code system that describes the manner in which character-based information is stored in a computer system. Files stored in the ASCII format can be transferred easily between computers, even those using different operating systems. Although ASCII is not a drawing file format, many CAD drawing formats (DXF and IGES, for example) are ASCII files. In these files, the drawing information is stored according to a specific format using ASCII characters. ASCII files are often referred to as pure text files because they can be read and edited by simple text editors.

HPGL, for Hewlett-Packard Graphics Language, is a format that was first developed for sending vector- (line-) based drawing information to pen plotters. The format is commonly used for sending drawing files to printers and plotters for printing. Because HPGL is a character-based format (ASCII), it can be transferred between computers easily. Normally, devices that recognize the HPGL format can print the files without using the program on which the file (a drawing, for example) was created.

STL is a CAD drawing format that is primarily used to send CAD drawings to *rapid automated prototyping* machines. STL is a mnemonic abbreviation for stereo-lithography, the technique that is used to create three-dimensional solid models directly from computer-generated drawings and for which the drawing format was originally developed. Most prototyping machines use 3-D CAD drawing files in STL format to create a solid model of the part represented by a drawing.

STEP stands for Standard for Exchange of Product Model Data and is a series of existing and proposed ISO standards written to allow access to all the data that surround a product. It extends the IGES idea of providing a geometric data transfer to include all the other data that would need to be communicated about a product over its lifetime, and facilitates the use and accessibility of the product data. Although STEP is a new standard, software tools have been developed for converting data from the IGES to STEP format and from STEP to IGES.

Rapid Automated Prototyping.—Rapid automated prototyping is a method of quickly creating an accurate three-dimensional physical model directly from a computerized conception of the part. The process is accomplished without machining or the removal of any material, but rather is a method of building up the model in three-dimensional space. The process makes it possible to easily and automatically create shapes that would be difficult or impossible to produce by any other method. Currently, production methods are able to produce models with an accuracy tolerance of ± 0.005 inch. Models are typically constructed of photoreactive polymer resins, nylon, polycarbonate or other thermoplastics, and investment casting wax. The model size is limited by the capability of the modeling machines to about 1 cubic foot at the present, however, large models can be built in sections and glued or otherwise fastened together.

Much of the work and a large part of the cost associated with creating a physical model by rapid prototyping are in the initial creation of the CAD model. The model needs to be a 3-D design model, built using wireframe, surface, or solid CAD modeling techniques. Many full-featured CAD systems support translation of drawing files into the STL format, which is the preferred file format for downloading CAD models to rapid prototyping machines. CAD programs without STL file format capability can use the IGES or DXF file format. This process can be time-consuming and expensive because additional steps may have to be taken by the service bureau to recreate features lost in converting the IGES or DXF file into STL format. If the design file has to be edited by a service bureau to recreate surfaces lost in the translation, unwanted changes to the model may occur, unnoticed. The safest route is to create a CAD model and export it directly into the STL format, leaving little chance for unexpected errors. Reverse STL generators are also available that will display a file saved in STL format or convert it into a form that can be imported into a CAD program.

DNC.—DNC stands for Direct Numerical Control and refers to a method of controlling numerical control machines from a remote location by means of a link to a computer or computer network. In its simplest form, DNC consists of one NC or CNC machine linked by its serial port to a computer. The computer may be used to develop and store CNC part programs and to transfer part programs to the machine as required. DNC links are normally two-directional, meaning that the NC/CNC can be operated from a computer terminal and the computer can be operated or ordered to supply data to the NC/CNC from the machine's control panel.

The number of machines that can be connected to a DNC network depends on the network's capability; in theory, any number of machines can be attached, and controlled. The type of network depends on the individual DNC system, but most industry standard network protocols are supported, so DNC nodes can be connected to existing networks very easily. Individual NC/CNC machines on a network can be controlled locally, from a network terminal in another building, or even from a remote location miles away through phone or leased lines.

Machinery Noise.—Noise from machinery or other mechanical systems can be controlled to some degree in the design or development stage if quantified noise criteria are provided the designer. Manufacturers and consumers may also use the same information in deciding whether the noise generated by a particular machine will be acceptable for a specific purpose.

Such criteria for noise may be classified into three types: 1) those relating to the degree of interference with speech communications; 2) those relating to physiological damage to humans, especially their hearing; and 3) those relating to psychological disturbances in people exposed to noise.

Sound Level Specifications: Noise criteria generally are specified in some system of units representing sound levels. One commonly used system specifies sound levels in units called decibels on the "A" scale, written dBA. The dBA scale designates a sound level system weighted to match human hearing responses to various frequencies and loudness. For example, to permit effective speech communication, typical criteria for indoor maximum noise levels are: meeting and conference rooms, 42 dBA; private offices and small meeting rooms, 38 to 47 dBA; supervisors' offices and reception rooms, 38 to 52 dBA; large offices and cafeterias, 42 to 52 dBA; laboratories, drafting rooms, and general office areas, 47 to 56 dBA; maintenance shops, computer rooms, and washrooms, 52 to 61 dBA; control and electrical equipment rooms, 56 to 66 dBA; and manufacturing areas and foremen's offices, 66 dBA. Similarly, there are standards and recommendations for daily permissible times of exposure at various steady sound levels to avoid hearing damage. For a working shift of 8 hours, a steady sound level of 90 dBA is the maximum generally permitted, with marked reduction in allowable exposure times for higher sound levels.*

Measuring Machinery Noise.—The noise level produced by a single machine can be measured by using a standard sound level meter of the handheld type set to the dBA scale. However, when other machines are running at the same time, or when there are other background noises, the noise of the machine cannot be measured directly. In such cases, two measurements, taken as follows, can be used to calculate the noise level of the individual machine. The meter should be held at arm's length and well away from any bystanders to avoid possible significant error up to 5 dBA.

Step 1. At the point of interest, measure the total noise, T, in decibels; that is, measure the noise of the shop and the machine in question when all machines are running; Step 2. Turn off the machine in question and measure B, the remaining background noise level; Step 3. Calculate M, the noise of the machine alone, $M = 10\log_{10}[10^{(T/10)} - 10^{(B/10)}]$.

Example 1: With a machine running, the sound level meter reads 51 decibels as the total shop noise T; and with the machine shut off the meter reads 49 decibels as the remaining background noise B. What is the noise level M of the machine alone? $M = 10\log_{10}[10^{(51/10)} - 10^{(49/10)}] = 46.7$ decibels dBA.

Example 2: If in Example 1 the remaining background noise level B was 41 decibels instead of 49, what is the noise level of the machine alone? $M = 10\log_{10}[10^{(51/10)} - 10^{(41/10)}] = 50.5$ decibels dBA.

Note: From this example it is evident that when the background noise level B is approximately 10 or more decibels lower than the total noise level T measured at the machine in question, then the background noise does not contribute significantly to the sound level at the machine and, for practical purposes, $M = T$ and no calculation is required.

---

* After April 1983, if employee noise exposures equal or exceed an 8-hour, time-weighted average sound level of 85 dB, OSHA requires employers to administer an effective hearing conservation program.