

TABLE OF CONTENTS
MACHINING OPERATIONS

CUTTING SPEEDS AND FEEDS

1009 Introduction to Speeds and Feeds
 1009 Cutting Tool Materials
 1013 Cutting Speeds
 1014 Cutting Conditions
 1014 Selecting Cutting Conditions
 1014 Tool Troubleshooting
 1016 Cutting Speed Formulas
 1018 RPM for Various Cutting Speeds and Diameter

SPEED AND FEED TABLES

1022 How to Use the Tables
 1022 Principal Speed and Feed Tables
 1026 Speed and Feed Tables for Turning
 1027 Plain Carbon and Alloy Steels
 1031 Tool Steels
 1032 Stainless Steels
 1033 Ferrous Cast Metals
 1035 Speed and Tool Life Adjustments
 1037 Copper Alloys
 1038 Titanium and Titanium Alloys
 1039 Superalloys
 1040 Speed and Feed Tables for Milling
 1043 Slit Milling
 1044 Aluminium Alloys
 1045 Plain Carbon and Alloy Steels
 1049 Tool Steels
 1050 Stainless Steels
 1052 Ferrous Cast Metals
 1054 High Speed Steel Cutters
 1056 Speed Adjustment Factors
 1057 Radial Depth of Cut Adjustments
 1059 Tool Life Adjustments
 1060 Drilling, Reaming, and Threading
 1061 Plain Carbon and Alloy Steels
 1066 Tool Steels
 1067 Stainless Steels
 1068 Ferrous Cast Metals
 1070 Light Metals
 1071 Adjustment Factors for HSS
 1072 Copper Alloys
 1072 Tapping and Threading
 1074 Cutting Speed for Broaching
 1075 Spade Drills
 1075 Spade Drill Geometry
 1077 Spade Drilling
 1079 Feed Rates
 1080 Power Consumption
 1081 Trepanning

ESTIMATING SPEEDS AND MACHINING POWER

1082 Planer Cutting Speeds
 1082 Cutting Speed and Time
 1082 Planing Time
 1082 Speeds for Metal-Cutting Saws
 1082 Turning Unusual Material
 1084 Estimating Machining Power
 1084 Power Constants
 1085 Feed Factors
 1085 Tool Wear Factors
 1088 Metal Removal Rates
 1090 Estimating Drilling Thrust, Torque, and Power
 1090 Work Material Factor
 1091 Chisel Edge Factors
 1091 Feed Factors
 1091 Drill Diameter Factors

MACHINING ECONOMETRICS

1093 Tool Wear And Tool Life Relationships
 1093 Equivalent Chip Thickness (ECT)
 1094 Tool-life Relationships
 1098 The G- and H-curves
 1099 Tool-life Envelope
 1102 Forces and Tool-life
 1104 Surface Finish and Tool-life
 1106 Shape of Tool-life Relationships
 1107 Minimum Cost
 1108 Production Rate
 1108 The Cost Function
 1109 Global Optimum
 1110 Economic Tool-life
 1113 Machine Settings and Cost Calculations
 1113 Nomenclature
 1114 Cutting Formulas
 1118 Tooling And Total Cost
 1119 Optimized Data
 1122 High-speed Machining Econometrics
 1123 Chip Geometry in Milling
 1125 Chip Thickness
 1127 Forces and Tool-life
 1128 High-speed Milling
 1129 Econometrics Comparison

TABLE OF CONTENTS
MACHINING OPERATIONS

SCREW MACHINE FEEDS AND SPEEDS

1131	Automatic Screw Machine Tools
1131	Knurling
1131	Revolution for Knurling
1131	Cams for Threading
1132	Cutting Speeds and Feeds
1134	Spindle Revolutions
1135	Practical Points on Cam
1136	Stock for Screw Machine Products
1138	Band Saw Blade Selection
1139	Tooth Forms
1139	Types of Blades
1140	Band Saw Speed and Feed Rate
1141	Bimetal Band Saw Speeds
1142	Band Saw Blade Break-In

CUTTING FLUIDS

1144	Types of Fluids
1144	Cutting Oils
1144	Water-Miscible Fluids
1145	Selection of Cutting Fluids
1146	Turning, Milling, Drilling and Tapping
1147	Machining
1148	Machining Magnesium
1149	Metalworking Fluids
1149	Classes of Metalworking Fluids
1149	Occupational Exposures
1150	Fluid Selection, Use, and Application
1151	Fluid Maintenance
1152	Respiratory Protection

MACHINING NONFERROUS METALS AND NON-METALLIC MATERIALS

1153	Machining Nonferrous Metals
1153	Aluminium
1154	Magnesium
1155	Zinc Alloy Die-Castings
1155	Monel and Nickel Alloys
1156	Copper Alloys
1156	Machining Non-metals
1156	Hard Rubber
1156	Formica Machining
1157	Micarta Machining
1157	Ultrasonic Machining

GRINDING FEEDS AND SPEEDS

1158	Basic Rules
1158	Wheel life T and Grinding Ratio
1159	ECT in Grinding
1160	Optimum Grinding Data
1162	Surface Finish, Ra
1163	Spark-out Time
1164	Grinding Cutting Forces
1165	Grinding Data
1166	Grindability Groups
1166	Side Feed, Roughing and Finishing
1167	Relative Grindability
1168	Grindability Overview
1168	Procedure to Determine Data
1174	Calibration of Recommendations
1176	Optimization

GRINDING AND OTHER ABRASIVE PROCESSES

1177	Grinding Wheels
1177	Abrasive Materials
1178	Bond Properties
1178	Structure
1179	ANSI Markings
1179	Sequence of Markings
1180	ANSI Shapes and Sizes
1180	Selection of Grinding Wheel
1181	Standard Shapes Ranges
1188	Grinding Wheel Faces
1189	Classification of Tool Steels
1190	Hardened Tool Steels
1194	Constructional Steels
1195	Cubic Boron Nitride
1196	Dressing and Truing
1196	Tools and Methods for Dressing and Truing
1198	Guidelines for Truing and Dressing
1199	Diamond Truing and Crossfeeds
1200	Size Selection Guide
1200	Minimum Sizes for Single-Point Truing Diamonds

TABLE OF CONTENTS
MACHINING OPERATIONS

GRINDING AND OTHER ABRASIVE PROCESSES	GRINDING AND OTHER ABRASIVE PROCESSES
<i>(Continued)</i>	<i>(Continued)</i>
1201 Diamond Wheels	1229 Offhand Grinding
1201 Shapes	1229 Floor- and Bench-Stand Grinding
1202 Core Shapes and Designations	1229 Portable Grinding
1202 Cross-sections and Designations	1229 Swing-Frame Grinding
1203 Designations for Location	1230 Abrasive Belt Grinding
1204 Composition	1230 Application of Abrasive Belts
1205 Designation Letters	1230 Selection Contact Wheels
1206 Selection of Diamond Wheels	1230 Abrasive Cutting
1206 Abrasive Specification	1233 Cutting-Off Difficulties
1207 Handling and Operation	1233 Honing Process
1207 Speeds and Feeds	1233 Rate of Stock Removal
1207 Grinding Wheel Safety	1234 Formula for Rotative Speeds
1207 Safety in Operating	1234 Factors in Rotative Speed Formulas
1208 Handling, Storage and Inspection	1235 Eliminating Undesirable Honing Conditions
1208 Machine Conditions	1235 Tolerances
1208 Grinding Wheel Mounting	1235 Laps and Lapping
1209 Safe Operating Speeds	1235 Material for Laps
1210 Portable Grinders	1236 Laps for Flat Surfaces
1212 Cylindrical Grinding	1236 Grading Abrasives
1212 Plain, Universal, and Limited- Purpose Machines	1237 Charging Laps
1212 Traverse or Plunge Grinding	1237 Rotary Diamond Lap
1212 Work Holding on Machines	1237 Grading Diamond Dust
1213 Work-Holding Methods	1238 Cutting Properties
1213 Selection of Grinding Wheels	1238 Cutting Qualities
1214 Wheel Recommendations	1238 Wear of Laps
1214 Operational Data	1238 Lapping Abrasives
1215 Basic Process Data	1238 Effect on Lapping Lubricants
1215 High-Speed	1239 Lapping Pressures
1216 Areas and Degrees of Automation	1239 Wet and Dry Lapping
1216 Troubles and Their Correction	1239 Lapping Tests
1220 Centerless Grinding	
1221 Through-feed Method of Grinding	
1221 In-feed Method	
1221 End-feed Method	
1221 Automatic Centerless Method	
1221 Centerless Grinding	
1222 Surface Grinding	
1223 Principal Systems	
1225 Grinding Wheel Recommendations	
1226 Process Data for Surface Grinding	
1226 Basic Process Data	
1227 Faults and Possible Causes	
	KNURLS AND KNURLING
	1240 Knurls and Knurling
	1240 ANSI Standard
	1240 Preferred Sizes
	1240 Specifications
	1241 Cylindrical Tools
	1242 Flat Tools
	1242 Specifications for Flat Dies
	1242 Formulas to Knurled Work
	1243 Tolerances
	1244 Marking on Knurls and Dies
	1244 Concave Knurls

TABLE OF CONTENTS
MACHINING OPERATIONS

MACHINE TOOL ACCURACY		NUMERICAL CONTROL PROGRAMMING	
1248	Degrees of Accuracy Expected with NC Machine Tool	<i>(Continued)</i>	
1249	Part Tolerances	1292	APT Programming
NUMERICAL CONTROL		1294	APT Computational Statements
1254	Introduction	1294	APT Geometry Statements
1254	CNC Technology	1295	Points, Lines and Circles
1254	Numerical Control vs. Manual Operations	1299	APT Motion Statements
1255	Numerical Control Standards	1300	Contouring Cutter Movements
1258	Programmable Controller	1301	Circles and Planes
1262	Closed-Loop System	1303	3-D Geometry
1262	Open-Loop System	1304	APT Postprocessor Statements
1262	Adaptive Control	1306	APT Example Program
1263	Flexible Manufacturing Systems	1307	APT for Turning
1264	Flexible Manufacturing Cell	1309	Indexable Insert Holders for NC
1264	Flexible Manufacturing Module	1310	Insert Radius Compensation
1264	Axis Nomenclature	1312	Threading Tool Insert Radius
1267	Total Indicator Reading	1313	V-Flange Tool Shanks
NUMERICAL CONTROL PROGRAMMING		1314	Retention Knobs
1269	Programming	CAD/CAM	
1272	Postprocessors	1315	CAD/CAM
1272	G-Code Programming	1317	Drawing Projections
1272	Format Classification	1318	Drawing Tips and Traps
1272	Letter Addresses	1322	Sizes of Lettering on Drawing
1274	Sequence Number (N-Word)	1322	Drawing Exchange Standards
1274	Preparatory Word (G-Word)	1324	Rapid Automated Prototyping
1278	Miscellaneous Functions	1324	DNC
1279	Feed Function (F-Word)	1325	Machinery Noise
1280	Spindle Function (S-Word)	1325	Measuring Machinery Noise
1280	Tool Function (T-Word)		
1282	Linear Interpolation		
1283	Circular Interpolation		
1284	Helical and Parabolic Interpolation		
1285	Subroutine		
1287	Conditional Expressions		
1287	Fixed (Canned) Cycles		
1291	Turning Cycles		
1291	Thread Cutting		

CUTTING SPEEDS AND FEEDS**Introduction to 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 773.

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 \times 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 1022.

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 16. Tool Troubleshooting Check List

Problem	Tool Material	Remedy
Excessive flank wear—Tool life too short	Carbide	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 1. Use a coolant 2. Reduce the cutting speed 3. Reduce the feed 4. Widen the chip breaker groove
Cutting edge chipping	Carbide	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 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	<ol style="list-style-type: none"> 1. Change to a grade containing more tantalum 2. Reduce the cutting speed 3. Reduce the feed
Poor surface finish	Carbide	<ol style="list-style-type: none"> 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 16. (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. $\pi = 3.1416$.

Example: The cutting speed for turning a 4-inch (101.6-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.

$$N = \frac{12V}{\pi D} = \frac{12 \times 575}{3.1416 \times 4} = 549 \text{ rpm} \quad N = \frac{1000V}{\pi D} = \frac{1000 \times 175.3}{3.1416 \times 101.6} = 549 \text{ rpm}$$

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.

RPM FOR VARIOUS SPEEDS

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 3/8	116	146	175	204	233	262	291	349	407	466	524	582
1 5/8	111	139	167	195	222	250	278	334	389	445	500	556
1 7/8	106	133	159	186	212	239	265	318	371	424	477	530
1 9/8	102	127	153	178	204	230	254	305	356	406	457	508
1 11/8	97.6	122	146	171	195	220	244	293	342	390	439	488
1 3/4	93.9	117	141	165	188	212	234	281	328	374	421	468
1 5/4	90.4	113	136	158	181	203	226	271	316	362	407	452
1 7/4	87.3	109	131	153	175	196	218	262	305	349	392	436
1 9/4	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 1/16	72.0	90.0	108	126	144	162	180	216	252	288	324	360
2 1/8	68.0	85.5	102	119	136	153	170	204	238	272	306	340
2 3/16	64.4	80.5	96.6	113	129	145	161	193	225	258	290	322
2 1/4	61.2	76.3	91.7	107	122	138	153	184	213	245	275	306
2 5/16	58.0	72.5	87.0	102	116	131	145	174	203	232	261	290
2 3/8	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 1/16	48.8	61.0	73.2	85.4	97.6	110	122	146	171	195	219	244
3 1/8	46.8	58.5	70.2	81.9	93.6	105	117	140	164	188	211	234
3 3/16	45.2	56.5	67.8	79.1	90.4	102	113	136	158	181	203	226
3 1/4	43.6	54.5	65.5	76.4	87.4	98.1	109	131	153	174	196	218
3 5/16	42.0	52.5	63.0	73.5	84.0	94.5	105	126	147	168	189	210
3 3/8	40.8	51.0	61.2	71.4	81.6	91.8	102	122	143	163	184	205
3 7/16	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 3/8	34.0	42.4	51.0	59.4	67.9	76.3	84.8	102	119	136	153	170
4 5/8	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 1/16	29.1	36.4	43.6	50.9	58.2	65.4	72.7	87.2	102	116	131	145
5 1/8	27.8	34.7	41.7	48.6	55.6	62.5	69.4	83.3	97.2	111	125	139
5 3/8	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 3/8	23.5	29.4	35.2	41.1	47.0	52.8	58.7	70.4	82.2	93.9	106	117
6 5/8	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 3/8	20.4	25.4	30.5	35.6	40.7	45.8	50.9	61.1	71.0	81.4	91.6	102
7 5/8	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											
$\frac{1}{4}$	3438	3820	4202	4584	4966	5348	5730	6112	6493	6875	7639	8403
$\frac{3}{16}$	2750	3056	3362	3667	3973	4278	4584	4889	5195	5501	6112	6723
$\frac{3}{8}$	2292	2546	2801	3056	3310	3565	3820	4074	4329	4584	5093	5602
$\frac{7}{16}$	1964	2182	2401	2619	2837	3056	3274	3492	3710	3929	4365	4802
$\frac{1}{2}$	1719	1910	2101	2292	2483	2675	2866	3057	3248	3439	3821	4203
$\frac{9}{16}$	1528	1698	1868	2037	2207	2377	2547	2717	2887	3056	3396	3736
$\frac{5}{8}$	1375	1528	1681	1834	1987	2139	2292	2445	2598	2751	3057	3362
$\frac{11}{16}$	1250	1389	1528	1667	1806	1941	2084	2223	2362	2501	2779	3056
$\frac{3}{4}$	1146	1273	1401	1528	1655	1783	1910	2038	2165	2292	2547	2802
$\frac{13}{16}$	1058	1175	1293	1410	1528	1646	1763	1881	1998	2116	2351	2586
$\frac{7}{8}$	982	1091	1200	1310	1419	1528	1637	1746	1855	1965	2183	2401
$\frac{15}{16}$	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
$1\frac{1}{16}$	809	899	988	1078	1168	1258	1348	1438	1528	1618	1798	1977
$1\frac{1}{8}$	764	849	933	1018	1103	1188	1273	1358	1443	1528	1698	1867
$1\frac{1}{4}$	724	804	884	965	1045	1126	1206	1287	1367	1448	1609	1769
$1\frac{3}{8}$	687	764	840	917	993	1069	1146	1222	1299	1375	1528	1681
$1\frac{1}{2}$	654	727	800	873	946	1018	1091	1164	1237	1309	1455	1601
$1\frac{3}{4}$	625	694	764	833	903	972	1042	1111	1181	1250	1389	1528
$1\frac{7}{8}$	598	664	730	797	863	930	996	1063	1129	1196	1329	1461
$1\frac{1}{2}$	573	636	700	764	827	891	955	1018	1082	1146	1273	1400
$1\frac{9}{16}$	550	611	672	733	794	855	916	978	1039	1100	1222	1344
$1\frac{5}{8}$	528	587	646	705	764	822	881	940	999	1057	1175	1293
$1\frac{11}{16}$	509	566	622	679	735	792	849	905	962	1018	1132	1245
$1\frac{3}{4}$	491	545	600	654	709	764	818	873	927	982	1091	1200
$1\frac{13}{16}$	474	527	579	632	685	737	790	843	895	948	1054	1159
$1\frac{7}{8}$	458	509	560	611	662	713	764	815	866	917	1019	1120
$1\frac{15}{16}$	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
$2\frac{1}{8}$	404	449	494	539	584	629	674	719	764	809	899	988
$2\frac{1}{4}$	382	424	468	509	551	594	636	679	721	764	849	933
$2\frac{3}{8}$	362	402	442	482	522	563	603	643	683	724	804	884
$2\frac{1}{2}$	343	382	420	458	496	534	573	611	649	687	764	840
$2\frac{5}{8}$	327	363	400	436	472	509	545	582	618	654	727	800
$2\frac{3}{4}$	312	347	381	416	451	486	520	555	590	625	694	763
$2\frac{7}{8}$	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
$3\frac{1}{8}$	274	305	336	366	397	427	458	488	519	549	611	672
$3\frac{1}{4}$	264	293	323	352	381	411	440	470	499	528	587	646
$3\frac{3}{8}$	254	283	311	339	367	396	424	452	481	509	566	622
$3\frac{1}{2}$	245	272	300	327	354	381	409	436	463	490	545	600
$3\frac{5}{8}$	237	263	289	316	342	368	395	421	447	474	527	579
$3\frac{3}{4}$	229	254	280	305	331	356	382	407	433	458	509	560
$3\frac{7}{8}$	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
$4\frac{1}{4}$	202	224	247	269	292	314	337	359	383	404	449	494
$4\frac{1}{2}$	191	212	233	254	275	297	318	339	360	382	424	466
$4\frac{3}{4}$	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
$5\frac{1}{4}$	163	181	199	218	236	254	272	290	308	327	363	399
$5\frac{1}{2}$	156	173	190	208	225	242	260	277	294	312	347	381
$5\frac{3}{4}$	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
$6\frac{1}{4}$	137	152	168	183	198	213	229	244	259	274	305	336
$6\frac{1}{2}$	132	146	161	176	190	205	220	234	249	264	293	322
$6\frac{3}{4}$	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
$7\frac{1}{4}$	118	131	144	158	171	184	197	210	223	237	263	289
$7\frac{1}{2}$	114	127	139	152	165	178	190	203	216	229	254	279
$7\frac{3}{4}$	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	86.4	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 Speed and Feed 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.

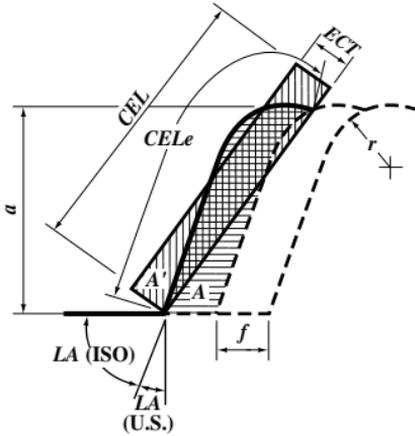
The combined feed/speed portion of the speed tables gives two sets of feed and speed data for each material represented. These feed/speed pairs are the *optimum* and *average* data (identified by *Opt.* and *Avg.*); the *optimum* set is always on the left side of the column and the *average* set is on 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.



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

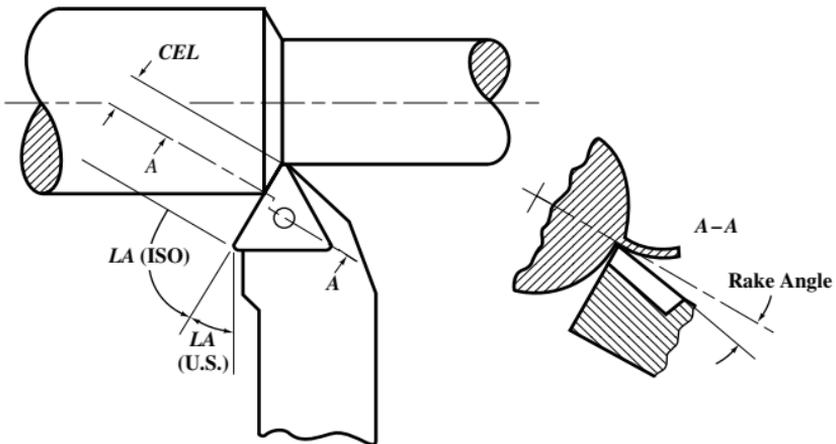


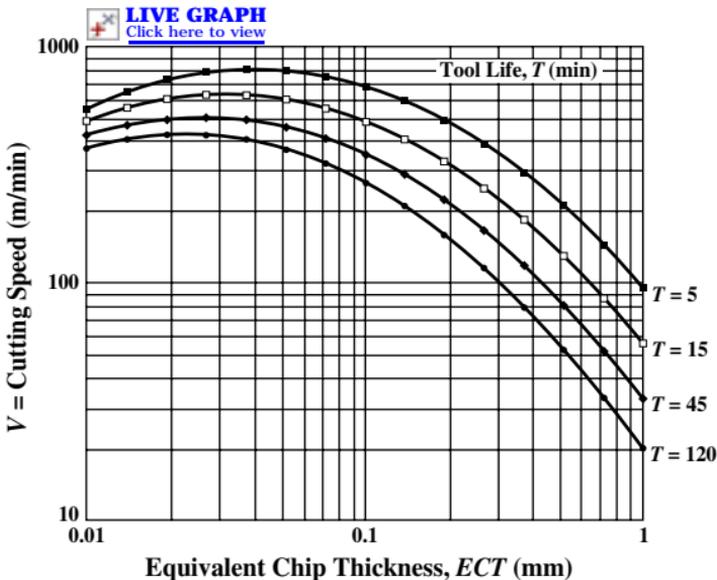
Fig. 2. Cutting Geometry for Turning

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 \times 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 $F_f = 1.07$ 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 1036*)

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				Cermets		
			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 plain carbon steels (resulfurized): 1212, 1213, 1215	100-150	150	f s	17 805	8 1075	36 405	17 555	17 1165	8 1295	28 850	13 1200	15 3340	8 4985	15 1670	8 2500	7 1610	3 2055
	150-200	160	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
1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	100-150	130	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
	150-200	120	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		
1132, 1137, 1139, 1140, 1144, 1146, 1151	275-325	75															
	325-375	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		
(Leaded): 11L17, 11L18, 12L13, 12L14	100-150	140	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
	150-200	145	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		
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 s	17 805	8 1075	36 405	17 555	17 1165	8 1295	28 850	13 1200	15 3340	8 4985	15 1670	8 2500	7 1610	3 2055
	125-175	110	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	90															
	225-275	70	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		

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					
				Hard		Tough		Hard		Tough		Hard		Tough		Cermet	
		Speed (fpm)		Opt.		Avg.		Opt.		Avg.		Opt.		Avg.		Opt.	
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	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	85	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		
	225-275	70	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		
	275-325	60	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		
	325-375	40	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	30	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		
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	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
	175-225	80	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	1365	1695
	225-275	65	f	17	8	36	17	17	8	28	13	10	5	10	5		
			s	615	815	300	405	865	960	755	960	1400	1965	1170	1640		
	275-325	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		
	325-375	35	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	30	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		
Free-machining alloy steels, (resulfurized): 4140, 4150	175-200	110	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
	200-250	90	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
	250-300	65	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
	300-375	50	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
	375-425	40	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	Tool Material															
		HSS		Uncoated Carbide				Coated Carbide				Ceramic				Cermets	
				Hard		Tough		Hard		Tough		Hard		Tough			
		Speed (fpm)		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>		<i>Opt.</i> <i>Avg.</i>	
Free-machining alloy steels: (leadeds): 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	75															
300-375	55	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	50																
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	100	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
	175-225	90	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
	225-275	70	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	
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	50	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	
375-425	30 (20)																
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	40 (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	
375-425	30 (20)																

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		
		Speed (fpm)	Hard		Tough		Hard		Tough		Hard		Tough		Cermet		
			f = feed (0.001 in./rev), s = speed (ft/min)														
		Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	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 s	17 220	8 295	36 100	17 150	20 355	10 525	28 600	13 865			10 660	5 810	7 570	3 740
	350–400	35	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
	43–48 Rc	25															
	48–52 Rc	10	f s			17 55†	8 90						7 385	3 645	10 270	5 500	
Maraging steels (not AISI): 18% Ni, Grades 200, 250, 300, and 350	250–325	60	f s	17 220	8 295	36 100	17 150	20 355	10 525	28 600	13 865	660	810	10 570	5 740	7	3
	50–52 Rc	10	f s			17 55†	8 90					7 385‡	3 645	10 270	5 500		
Nitriding steels (not AISI): Nitalloy 125, 135, 135 Mod., 225, and 230, Nitalloy N, Nitalloy EZ, Nitrex 1	200–250	70	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
	300–350	30	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

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

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

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 (Austenitic): 203EZ, 303, 303Se, 303MA, 303Pb, 303Cu, 303 Plus X	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
	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	110	f	13	7	36	17			28	13	7	3
	185–240	100	s	520	640	310				625	815	695	875
	275–325	60	f	13	7	36	17			28	13		
	375–425	30	s	210	260	85	135			130	165		
Stainless steels (Ferritic): 405, 409 429, 430, 434, 436, 442, 446, 502 (Austenitic): 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348 (Austenitic): 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	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
	135–185 225–275	75 65											
(Martensitic): 403, 410, 420, 501	135–185	70	f s	13 520	7 640	36 310	17 345			28 625	13 165	7 695	3 875
	135–175	95											
	175–225	85											
	275–325	55											
(Martensitic): 414, 431, Greek Ascoloy, 440A, 440B, 440C	375–425	35											
	225–275	55–60	f	13	7	36	17			28	13	13	7
	275–325	45–50	s	210	260	85	135			130	165	200†	230
	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	13	7	36	17			28	13	13	7
	275–325	50	s	520	640	310	345			625	815	695	875
	325–375	40	f	13	7	36	17						
	375–450	25	s	195	240	85	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 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																
		HSS	Uncoated Carbide				Coated Carbide				Ceramic				Cermet		CBN	
			Tough		Hard		Tough		Hard		Tough		Hard		Tough			
		f = feed (0.001 in./rev), s = speed (ft/min)																
Speed (fpm)		Opt.		Avg.		Opt.		Avg.		Opt.		Avg.		Opt.		Avg.		
Gray Cast Iron																		
ASTM Class 20	120–150	120	f	28	13	28	13	28	13	15	8	15	8	8	4	24	11	
ASTM Class 25	160–200	90	s	240	365	665	1040	585	945	1490	2220	1180	1880	395	510	8490	36380	
ASTM Class 30, 35, and 40	190–220	80																
ASTM Class 45 and 50	220–260	60	f	28	13	28	13	28	13	11	6	11	6	8	4	24	11	
ASTM Class 55 and 60	250–320	35	s	160	245	400	630	360	580	1440	1880	1200	1570	335	420	1590	2200	
ASTM Type 1, 1b, 5 (Ni resist)	100–215	70	f	28	13			28	13	15	8	15	8	8	4			
ASTM Type 2, 3, 6 (Ni resist)	120–175	65	s	110	175			410	575	1060	1590	885	1320	260	325			
ASTM Type 2b, 4 (Ni resist)	150–250	50																
Malleable Iron																		
(Ferritic): 32510, 35018	110–160	130	f	28	13	28	13	28	13	15	8	15	8					
			s	180	280	730	940	660	885	1640	2450	1410	2110					
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160–200	95	f	28	13	28	13	28	13	13	7	13	7					
	200–240	75	s	125	200	335	505	340	510	1640	2310	1400	1970					
(Martensitic): 53004, 60003, 60004	200–255	70																
(Martensitic): 70002, 70003	220–260	60	f	28	13			28	13	11	6	11	6					
(Martensitic): 80002	240–280	50	s	100	120			205	250	1720	2240	1460	1910					
(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

Material	Brinell Hardness	Tool Material														
		Uncoated HSS	Uncoated Carbide				Coated Carbide				Ceramic				Cermets	
			Hard		Tough		Hard		Tough		Hard		Tough			
		f = feed (0.001 in./rev), s = speed (ft/min)														
Speed (fpm)	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		28	13	28	13	28	13	15	8	15	8	8	4
			s		200	325	490	700	435	665	970	1450	845	1260	365	480
(Ferritic-Pearlitic): 80-55-06	190-225	80	f		28	13	28	13	28	13	11	6	11	6	8	4
	225-260	65	s		130	210	355	510	310	460	765	995	1260	1640	355	445
(Pearlitic-Martensitic): 100-70-03	240-300	45	f													
	270-330	30	s		28	13			28	13	10	5	10	5	8	4
(Martensitic): 120-90-02	300-400	15	f		40	65			145	175	615	750	500	615	120	145
			s													
Cast Steels																
(Low-carbon): 1010, 1020	100-150	110	f	17	8	36	17	17	8	28	13	15	8		7	3
	125-175	100	s	370	490	230	285	665	815	495	675	2090	3120		625	790
(Medium-carbon): 1030, 1040, 1050	175-225	90	f													
	225-300	70	s													
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150-200	90	f	17	8	36	17	17	8	28	13	15	8		7	3
	200-250	80	s	370	490	150	200	595	815	410	590	1460	2170		625	790
	250-300	60	f													
	175-225	80	s													
(Medium-carbon alloy): 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	225-250	70	f	17	8	36	17	17	8			15	8			
	250-300	55	s	310	415	115	150	555	760			830	1240			
	300-350	45	f			28	13					15	8			
			s			70†	145					445	665			
	350-400	30	f			28	13			28	13			15	8	
			s			115†	355			335	345			955	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, Cermet, 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 $\leq \frac{1}{2}f_{opt}$, the cutting speed for desired tool life is $V_T = f_s \times V_{15}$; for medium feeds where $\frac{1}{2}f_{opt} < \text{feed} < \frac{3}{4}f_{opt}$, $V_T = f_m \times V_{15}$; and for larger feeds where $\frac{3}{4}f_{opt} \leq \text{feed} \leq f_{opt}$, $V_T = f_l \times V_{15}$. 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		Feed Factor F_f	Depth of Cut		Depth-of-Cut Factor F_d
in.	mm		in.	mm	
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 $\frac{1}{2}f_{opt}$, $\frac{3}{4}f_m$ 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	Speed (fpm)	HSS	Uncoated Carbide	Polycrystalline Diamond	
			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)		
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, 8V-5Fe-1Al	320–370 320–380	30 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 1 Al-8V-5Fe	375–420 375–440	20 20			
Beta Alloys					
13V-11Cr-3Al, 8Mo-8V-2Fe-3Al, 3Al-8V-6Cr-4Mo-4Zr, 11.5Mo-6Zr-4.5Sn	275–350 375–440	25 20	f s	17 55	8 150

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.	
Aluminum Die-Casting Alloys							
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 ST and A	600 500	f s	36 2820	17 4570		
All aluminum sand and permanent mold casting alloys	AC ST and A	750 600					
Alloys 308.0 and 319.0	—	—	f s	36 865	17 1280	11 5890 ^a	8 8270
Alloys 390.0 and 392.0	AC ST and A	80 60	f s	24 2010	11 2760	8 4765	4 5755
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 AC	100 125	f s	36 630	17 1060	11 7560	6 9930

^aThe 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

Material Description	Tool Material										
	HSS Turning		Uncoated Carbide		Ceramic				CBN		
	Rough	Finish	Tough		Hard		Tough				
	Speed (fpm)		f = feed (0.001 in./rev), s = speed (ft/min)								
			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	
Refractalloy 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									
S-816, V-36	10-15	15-20									
S-590	10-20	15-30									
Udimet 630		20-25									
N-155		15-25	f	20	10				20	10	
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	15-20	20-25	s	75	125	11	6	11	6	230	400
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									
Rene 41	10-15	12-20									
Incoloy 901	10-20	20-35									
Waspaloy	10-30	25-35	f	28	13	11	6	10	5	20	10
Inconel 625, 702, 706, 718 (wrought), 721, 722, X750, 751, 901, 600, and 604	15-20	20-35	s	20	40	895	2230	345	815	185	315
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									
Udimet 500, 700, and 710	10-15	12-20									
Astrolloy	5-10	5-15									
Mar-M200, M246, M421, and Rene 77, 80, and 95 (forged)		10-12	f	28	13	11	6	10	5	20	10
		10-15	s	15	15	615	1720	290	700	165	280
B-1900, GMR-235 and 235D, IN 100 and 738, Inconel 713C and 718 (cast), M252 (cast)	8-10	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 $\frac{3}{64}$ 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.

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, $\frac{3}{64}$ -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 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.

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

$$\begin{aligned} f_m &= f_t n_t N = 0.008 \times 8 \times 108 \\ &= 7 \text{ ipm (approximately)} \end{aligned}$$

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 $\frac{3}{4}$ is maintained (i.e., $\frac{9}{8} = \frac{3}{4}$). However, if a different feed or axial depth of cut is required, or if the ar/D ratio is not equal to $\frac{3}{4}$, 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 .

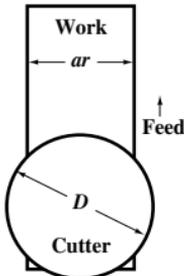


Fig. 3.

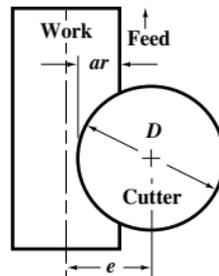


Fig. 4.

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:

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

$$F_f = (1.25 + 1.43)/2 = 1.34$$

$$F_d = 1.0$$

$$F_{ar} = 1.0$$

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

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: $ff_{opt} = 3/15 = 0.2$ 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 $V = V_{opt} \times F_f$.

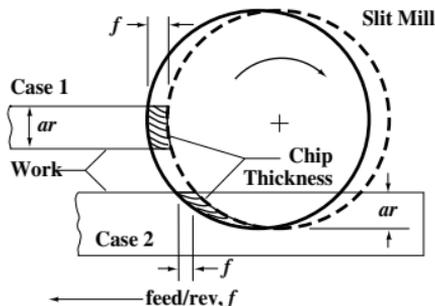


Fig. 5. Determination of Radial Depth of Cut or in Slit Milling

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	CD ST and A	f s	15 8	8 100	15 620	8 2020	39 755	20 1720	8 3750	4 8430	16 1600	8 4680	39 840	20 2390
All aluminum sand and permanent mold casting alloys	CD ST and A	f s	15 165	8 850	15 620	8 2020	39 755	20 1720	8 3750	4 8430	16 1600	8 4680	39 840	20 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	—	f			15	8	39	20	8	4			39	20
		s			355	1385	405	665	2320	4755			500	1680
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, $\frac{3}{4}$ -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 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

Material	Brinell Hardness	HSS	End Milling						Face Milling				Slit Milling				
			HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		
		Speed (fpm)	f	s	Opt.		Avg.		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
(Resulfurized): 1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	150–200	130	f	7	4						39	20					
			s	35	100						215	405					
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151	100–150	130	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
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151	150–200	115	f	7	4									39	20		
			s	30	85									185	350		
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151	275–325	70															
			f	7	4	7	4	7	4	39	20	39	20	39	20	39	20
(Resulfurized): 1132, 1137, 1139, 1140, 1144, 1146, 1151	325–375	45	s	25	70	210	435	300	560	90	170	175	330	90	235	135	325
			s	35	100												
(Leaded): 11L17, 11L18, 12L13, 12L14	100–150	140	f	7	4						39	20					
			s	35	100						215	405					
(Leaded): 11L17, 11L18, 12L13, 12L14	150–200	130	f	7	4						39	20					
			s	30	85						185	350					
(Leaded): 11L17, 11L18, 12L13, 12L14	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
Plain carbon steels: 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	125–175	110	f	7	4						39	20					
			s	35	100						215	405					
Plain carbon steels: 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	175–225	90	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	225–275	65	f	7	4						39	20					
			s	30	85						185	350					

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

Material	Brinell Hardness	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)															
			Speed (fpm)	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.
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 s	7 35	4 100							39 215	20 405					
	175–225	85	f s	7 30	4 85							39 185	20 350					
	225–275	70																
	275–325	55																
	325–375	35	f s	7 25	4 70	7 210	4 435	7 300	4 560	39 90	20 170	39 175	20 330	39 90	20 235	39 135	20 325	
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 s	7 30	4 85	7 325	4 565	7 465	4 720	39 140	20 220	39 195	20 365	39 170	20 350	39 245	20 495	
	175–225	75																
	225–275	60	f s	7 30	4 85							39 185	20 350					
	275–325	45																
	325–375	30	f s	7 25	4 70	7 210	4 435	7 300	4 560	39 90	20 170	39 175	20 330	39 90	20 235	39 135	20 325	
375–425	15																	
Free-machining alloy steels (Resulfurized): 4140, 4150	175–200	100	f s	15 7	8 30	15 105	8 270	15 270	8 450			39 295	20 475	39 135	20 305	7 25	4 70	
	200–250	90																
	250–300	60	f s	15 6	8 25	15 50	8 175	15 85	8 255			39 200	20 320	39 70	20 210	7 25	4 70	
	300–375	45	f s	15 5	8 20	15 40	8 155	15 75	8 225			39 175	20 280					
375–425	35																	

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

Material	Brinell Hardness	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)															
			Speed (fpm)	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 s	7 30	4 85	7 325	4 565	7 465	4 720	39 140	20 220	39 195	20 365	39 170	20 350	39 245	20 495	
	200–250	95	f s	7 30	4 85							39 185	20 350					
	250–300	70																
	300–375	50	f s	7 25	4 70	7 210	4 435	7 300	4 560	39 90	20 170	39 175	20 330	39 90	20 235	39 135	20 325	
	375–425	40																
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	100	f s	15 7	8 30	15 105	8 270	15 220	8 450			39 295	20 475	39 135	20 305	39 265	20 495	
	175–225	90																
	225–275	60	f s	15 6	8 25	15 50	8 175	15 85	8 255			39 200	20 320	39 70	20 210	39 115	20 290	
	275–325	50	f s	15 5	8 20	15 45	8 170	15 80	8 240			39 190	20 305					
	325–375	40	f s	15 5	8 20	15 40	8 155	15 75	8 225			39 175	20 280					
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 s	15 5	8 30	15 105	8 270	15 220	8 450			39 295	20 475	39 135	20 305	39 265	20 495	
	225–275	60	f s	15 5	8 25	15 50	8 175	15 85	8 255			39 200	20 320	39 70	20 210	39 115	20 290	
	275–325	50 (40)	f s	15 5	8 25	15 45	8 170	15 80	8 240			39 190	20 305					
	325–375	35 (30)																
	375–425	20	f s	15 5	8 20	15 40	8 155	15 75	8 225			39 175	20 280					

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

Material	Brinell Hardness	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)													
			Speed (fpm)	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.
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	60	f		8	4	8	4								
	300–350	45	s		165	355	300	480								
	350–400	20	f	8 4	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

Material	Brinell Hardness	HSS	End Milling						Face Milling				Slit Milling				
			HSS		Uncoated Carbide		Coated Carbide		Uncoated Carbide		CBN		Uncoated Carbide		Coated Carbide		
		Speed (fpm)	f = feed (0.001 in./tooth), 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																
Shock resisting: S1, S2, S5, S6, S7	175-225																
Cold work, oil hardening: O1, O2, O6, O7	175-225	50	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
Cold work, high carbon, high chromium: D2, D3, D4, D5, D7	200-250	40															
Cold work, air hardening: A2, A3, A8, A9, A10 A4, A6 A7	200-250	50	f s							39 255	20 385						
	200-250	45															
	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 50-52 Rc 52-56 Rc	— — —	f s			5 20†	3 55					39 50	20 135	39 5†	20 15		
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	100-150 150-200	75 60															
High-speed steel: M1, M2, M6, M10, T1, T2, T6 M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8 T15, M3-2	200-250	50															
	225-275	40	f s							39 255	20 385						
	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, \dagger = 15; coated carbide = 10; CBN = 1.

Table 13. Cutting Feeds and Speeds for Milling Stainless Steels

Material	Brinell Hardness	HSS	End Milling						Face Milling		Slit Milling				
			HSS		Uncoated Carbide		Coated Carbide		Coated Carbide		Uncoated Carbide		Coated Carbide		
			Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	
															f = feed (0.001 in./tooth), s = speed (ft/min)
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		
	185-240	100													
(Martensitic): 416, 416Se, 416 Plus X, 420F, 420FSe, 440F, 440FSe	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													
	175-225	85													
(Martensitic): 403, 410, 420, 501	275-325	55													
	375-425	35													

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

Material	Brinell Hardness	HSS	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 Ascology, 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 s	7 20	4 55	7 210	4 585					39 75	20 240	
	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, $\frac{3}{4}$ -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 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	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.				
Gray Cast Iron																		
ASTM Class 20	120-150	100	f	5	3	5	3		39	20	39	20	39	20	39	20		
ASTM Class 25	160-200	80	s	35	90	520	855		140	225	285	535	1130	1630	200	530	205	420
ASTM Class 30, 35, and 40	190-220	70																
ASTM Class 45 and 50	220-260	50	f	5	3	5	3		39	20	39	20	39	20	39	20	39	20
ASTM Class 55 and 60	250-320	30	s	30	70	515	1100		95	160	185	395	845	1220	150	400	145	380
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	5	3	5	3		39	20	39	20	39	20			39	20
			s	30	70	180	250		120	195	225	520	490	925			85	150
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160-200	80	f	5	3	5	3		39	20	39	20	39	20			39	20
	200-240	65	s	25	65	150	215		90	150	210	400	295	645			70	125
(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	7	4	7	4		39	20	39	20	39	20			39	20
			s	15	35	125	240		100	155	120	255	580	920			60	135
(Ferritic-Pearlitic): 80-55-06	190-225	60	f	7	4	7	4		39	20	39	20	39	20			39	20
	225-260	50	s	10	30	90	210		95	145	150	275	170	415			40	100
(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					
			Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.					
			f = feed (0.001 in./tooth), s = speed (ft/min)																		
Cast Steels																					
(Low carbon): 1010, 1020	100–150	100	f	7	4	7	4	7	4							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		
	150–200	85	s	20	55	160†	400	345	560							205‡	340	65†	180	180	370
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	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, $\frac{3}{4}$ -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	Hardness, 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
	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
AISI 1033 to 1095; 1524 to 1566	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
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	.003	.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)	Hardness, 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							
		1/2	3/4	1 and up	1/4	1/2	3/4	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 cast003	.004	.005	.002	.004	.005	.006	.005-.016	.006	.005-.020	.004-.012
Cast aluminum alloys—hardened003	.004	.005	.002	.003	.004	.005	.004-.012	.005	.005-.020	.004-.012
Wrought aluminum alloys—cold drawn003	.004	.005	.002	.003	.004	.005	.004-.014	.005	.005-.020	.004-.012
Wrought aluminum alloys—hardened002	.003	.004	.001	.002	.003	.004	.003-.012	.004	.005-.020	.004-.012
Magnesium alloys003	.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							Depth of Cut and Lead Angle									
	V_{avg}/V_{opt}							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 f_{opt} from the speed tables. For example, given two feed/speed pairs $\frac{7}{15}$ and $\frac{4}{45}$ 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 $\frac{1}{2}$ -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_f 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							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 = $\frac{3}{8}$). 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 $\leq \frac{1}{2}f_{opt}$, the cutting speed for the desired tool life T is $V_T = f_s \times V_{15}$; for medium feeds, where $\frac{1}{2}f_{opt} < \text{feed} < \frac{3}{4}f_{opt}$, $V_T = f_m \times V_{15}$; and for larger feeds, where $\frac{3}{4}f_{opt} \leq \text{feed} \leq f_{opt}$, $V_T = f_l \times V_{15}$. 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	—
Cermets	6	CC690	KYON3000	—	Q6
	7	CT515	KT125	CM	VC605
Polycrystalline	8	CT525	KT150	CR	VC610
	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 779 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 $\frac{1}{8}$ in, 0.002 to 0.006 in./rev for $\frac{1}{8}$ - to $\frac{1}{4}$ -in drills; 0.004 to 0.010 in./rev for $\frac{1}{4}$ - to $\frac{1}{2}$ -in drills; 0.007 to 0.015 in./rev for $\frac{1}{2}$ - 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 21 s 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 16 s 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 8 s 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 8 s 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 21 s 55	11 125	8 310	4 620	36 140	18 185	83 140	20 185
	125–175	90	60								
	175–225	70	45	f 8 s 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								
	175–225	75	50								
	225–275	60	40	f 8 s 365			4 735				
	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.	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			8	4				
				s			365	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			8	4				
				s			355	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	16	8	8	4	27	14	83	20
	200-250	90	60	s	50	95	370	740	105	115	90	115
	250-300	65	40	f			8	4				
				s			365	735				
	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			8	4				
				s			355	600				
	275-325	50	30	f	11	6	8	4	19	10	83	20
	325-375	35	25	s	50	85	335	570	95	135	60	95
	375-425	25	15	f			8	4				
				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.	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 s	16 75	8 140	8 410	4 685	26 150	13 160	83 125	20 160
	225–275	60 (50)	40 (30)	f s			8 355	4 600				
	275–325	45 (35)	30 (25)	f s	11 50	6 85	8 335	4 570	19 95	10 135	83 60	20 95
	325–375 375–425	30 (30) 20 (20)	15 (20) 15 (10)	f s			8 310	4 525				
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 s			8 325	4 545				
	300–350	35	20	f s			8 270	4 450				
	350–400	20	10	f s			8 325	4 545				
Maraging steels (not AISI): 18% Ni Grade 200, 250, 300, and 350	250–325	50	30	f s			8 325	4 545				
Nitriding steels (not AISI): Nitralloy 125, 135, 135 Mod., 225, and 230, Nitralloy N, Nitralloy EZ, Nitrex I	200–250	60	40	f s	16 75	8 140	8 410	4 685	26 150	13 160	83 125	20 160
	300–350	35	20	f s			8 310	4 525				

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, $\frac{25}{32}$ -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 radial depth of cut to reamer diameter (i.e., $f_d = 1.00$). Speed adjustment factors to modify tool life are found in Table 15c.

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 5a 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 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.

<i>Drill diameter = 0.079 inch</i>	<i>Drill diameter = 4.0 inches</i>
$F_f = 1.40$	$F_f = 1.40$
$F_d = (0.34 + 0.38)/2 = 0.36$	$F_d = (1.95 + 1.73)/2 = 1.85$
$V = 50 \times 1.4 \times 0.36 = 25.2$ 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		HSS	
		Speed (fpm)		f = feed (0.001 in./rev), s = speed (ft/min)									
		Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.
Water hardening: W1, W2, W5	150–200	85	55										
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	f s	15 45	7 85	8 360	4 605	24 90	12 95	83 75	20 95	
(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	150–200 200–250	60 50	40 30	f s			8 270	4 450					
(Tungsten type): H21, H22, H23, H24, H25, H26	150–200 200–250	55 40	35 25	f s									
(Molybdenum type): H41, H42, H43	150–200 200–250	45 35	30 20										
Special-purpose, low alloy: L2, L3, L6	150–200	60	40	f s	15 45	7 85	8 360	4 605	24 90	12 95	83 75	20 95	
Mold steel: P2, P3, P4, P5, P6P20, P21	100–150 150–200	75 60	50 40	f s									
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.	Opt.	Avg.
Free-machining stainless steels (Ferritic): 430F, 430FSe	135–185	90	60	f 15 s 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 15 s 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 15 s 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 15 s 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								
	135–175	75	50								
	175–225	65	45								
	275–325	40	25								
(Martensitic): 403, 410, 420, 501	375–425	25	15								
	225–275	50	30								
	275–325	40	25								
(Martensitic): 414, 431, Greek Ascology	375–425	25	15								
	225–275	45	30								
	275–325	40	25								
(Martensitic): 440A, 440B, 440C	375–425	25	15								
	225–275	45	30								
	275–325	40	25								
(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 15 s 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.	Opt.	Avg.	
ASTM Class 20	120–150	100	65											
ASTM Class 25	160–200	90	60	f	16	8	11	6	11	6	26	13	83	20
ASTM Class 30, 35, and 40	190–220	80	55	s	80	90	85	180	235	485	85	65	90	80
ASTM Class 45 and 50	220–260	60	40	f	13	6	11	6	11	6	21	10	83	20
ASTM Class 55 and 60	250–320	30	20	s	50	50	70	150	195	405	50	30	55	45
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	19	10			11	6	30	16	83	20
				s	80	100			270	555	95	80	100	85
(Pearlitic): 40010, 43010, 45006, 45008, 48005, 50005	160–200	80	55	f	14	7	11	6	11	6	22	11	83	20
	200–240	70	45	s	65	65	85	180	235	485	65	45	70	60
(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	17	9	11	6	11	6	28	14	83	20
				s	70	80	85	180	235	485	80	60	80	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 (Ferritic-Pearlitic): 80-55-06 (Pearlitic-Martensitic): 100-70-03	270-330	25	15														
	330-400	10	5														
	190-225	70	45	f	13	6	11	6	11	6	21	11	83	20			
	225-260	50	30	s	60	60	70	6	150	195	6	405	55	40	83	60	20
	240-300	40	25														
Cast Steels																	
(Low carbon): 1010, 1020	100-150	100	65	f	18	9					29	15	83	20			
				s	35	70					75	85	65	85			
(Medium carbon): 1030, 1040, 1050	125-175	90	60														
	175-225	70	45														
	225-300	55	35	f	15	7			8	4	24	12	83	20			
(Low-carbon alloy): 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150-200	75	50	s	35	60			195†	475	65	70	55	70			
	200-250	65	40														
	250-300	50	30														
	175-225	70	45	f					8	4							
(Medium-carbon alloy): 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	225-250	60	35	s					130†	315							
	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		HSS	
		Speed (fpm)		f = feed (0.001 in./rev), s = speed (ft/min)									
				Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.	Opt.	Avg.
All wrought aluminum alloys, 6061-T651, 5000, 6000, 7000 series	CD	400	400	f s	31 390	16 580	11 3235	6 11370	52 610	26 615	83 635	20 565	
	ST and A	350	350										
All aluminum sand and permanent mold casting alloys	AC	500	500	f s	31 390	16 580	11 3235	6 11370	52 610	26 615	83 635	20 565	
	ST and A	350	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	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	300	f s	24 65	12 85	11 555	6 1955	40 85	20 80	83 85	20 80	
		70	70										
Alloys 413	ST and A	—	—	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	f s	—	—	—	—	—	—	—	—	
All cast magnesium alloys	A,AC, ST and A	450	450	f s	—	—	—	—	—	—	—	—	

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								
								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_{avg}/V_{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 15c](#).

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, $\frac{25}{32}$ -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 $\frac{3}{64}$ 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 2*D* 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 taper 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.

Spade Drills

Spade drills are used to produce holes ranging in size from about 1 inch to 6 inches diameter, and even larger. Very deep holes can be drilled and blades are available for core drilling, counterboring, and for bottoming to a flat or contoured shape. There are two principal parts to a spade drill, the blade and the holder. The holder has a slot into which the blade fits; a wide slot at the back of the blade engages with a tongue in the holder slot to locate the blade accurately. A retaining screw holds the two parts together. The blade is usually made from high-speed steel, although cast nonferrous metal and cemented carbide-tipped blades are also available. Spade drill holders are classified by a letter symbol designating the range of blade sizes that can be held and by their length. Standard stub, short, long, and extra long holders are available; for very deep holes, special holders having wear strips to support and guide the drill are often used. Long, extra long, and many short length holders have coolant holes to direct cutting fluid, under pressure, to the cutting edges. In addition to its function in cooling and lubricating the tool, the cutting fluid also flushes the chips out of the hole. The shank of the holder may be straight or tapered; special automotive shanks are also used. A holder and different shank designs are shown in Fig. 1; Figs. 2a through Fig. 2f show some typical blades.

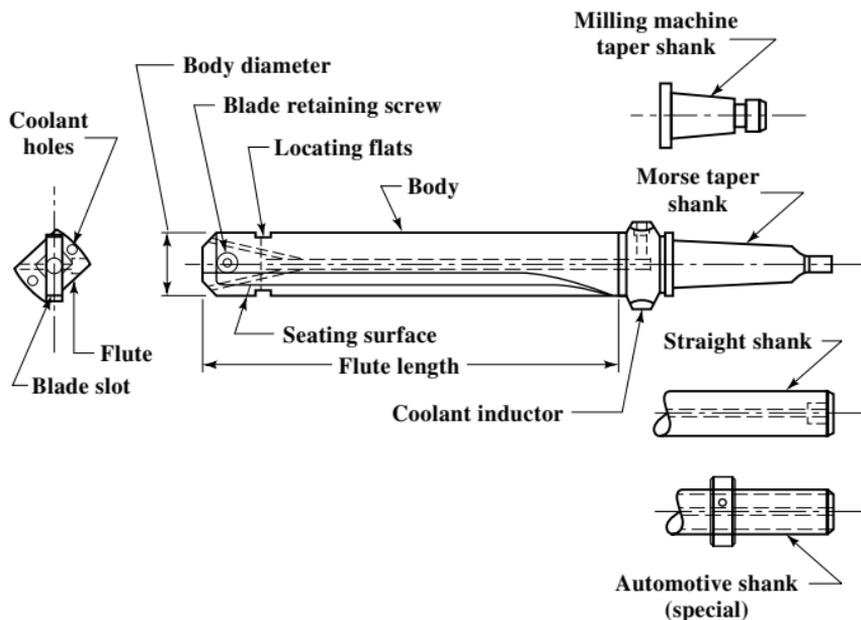


Fig. 1. Spade Drill Blade Holder

Spade Drill Geometry.—Metal separation from the work is accomplished in a like manner by both twist drills and spade drills, and the same mechanisms are involved for each. The two cutting lips separate the metal by a shearing action that is identical to that of chip formation by a single-point cutting tool. At the chisel edge, a much more complex condition exists. Here the metal is extruded sideways and at the same time is sheared by the rotation of the blunt wedge-formed chisel edge. This combination accounts for the very high thrust force required to penetrate the work. The chisel edge of a twist drill is slightly rounded, but on spade drills, it is a straight edge. Thus, it is likely that it is more difficult for the extruded metal to escape from the region of the chisel edge with spade drills. However, the chisel edge is shorter in length than on twist drills and the thrust for spade drilling is less.

Typical Spade Drill Blades

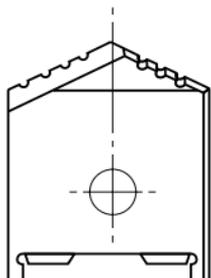


Fig. 2a. Standard blade

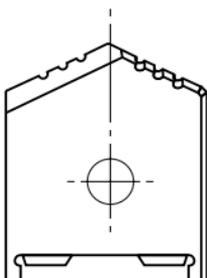


Fig. 2b. Standard blade with corner chamfer

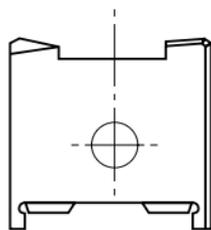


Fig. 2c. Core drilling blade

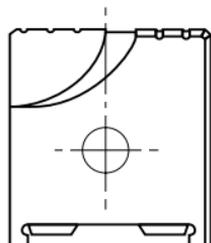


Fig. 2d. Center cutting facing or bottoming blade

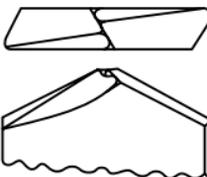


Fig. 2e. Standard blade with split point or crankshaft point

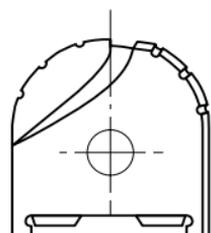


Fig. 2f. Center cutting radius blade

Basic spade drill geometry is shown in Fig. 3. Normally, the point angle of a standard tool is 130 degrees and the lip clearance angle is 18 degrees, resulting in a chisel edge angle of 108 degrees. The web thickness is usually about $\frac{1}{4}$ to $\frac{5}{16}$ as thick as the blade thickness. Usually, the cutting edge angle is selected to provide this web thickness and to provide the necessary strength along the entire length of the cutting lip. A further reduction of the chisel edge length is sometimes desirable to reduce the thrust force in drilling. This reduction can be accomplished by grinding a secondary rake surface at the center or by grinding a split point, or crankshaft point, on the point of the drill.

The larger point angle of a standard spade drill—130 degrees as compared with 118 degrees on a twist drill—causes the chips to flow more toward the periphery of the drill, thereby allowing the chips to enter the flutes of the holder more readily. The rake angle facilitates the formation of the chip along the cutting lips. For drilling materials of average hardness, the rake angle should be 10 to 12 degrees; for hard or tough steels, it should be 5 to 7 degrees; and for soft and ductile materials, it can be increased to 15 to 20 degrees. The rake surface may be flat or rounded, and the latter design is called radial rake. Radial rake is usually ground so that the rake angle is maximum at the periphery and decreases uniformly toward the center to provide greater cutting edge strength at the center. A flat rake surface is recommended for drilling hard and tough materials in order to reduce the tendency to chipping and to reduce heat damage.

A most important feature of the cutting edge is the chip splitters, which are also called chip breaker grooves. Functionally, these grooves are chip dividers; instead of forming a single wide chip along the entire length of the cutting edge, these grooves cause formation of several chips that can be readily disposed of through the flutes of the holder. Chip splitters must be carefully ground to prevent the chips from packing in the grooves, which greatly reduces their effectiveness. Splitters should be ground perpendicular to the cutting lip and parallel to the surface formed by the clearance angle. The grooves on the two cut-

ting lips must not overlap when measured radially along the cutting lip. Fig. 4 and the accompanying table show the groove form and dimensions.

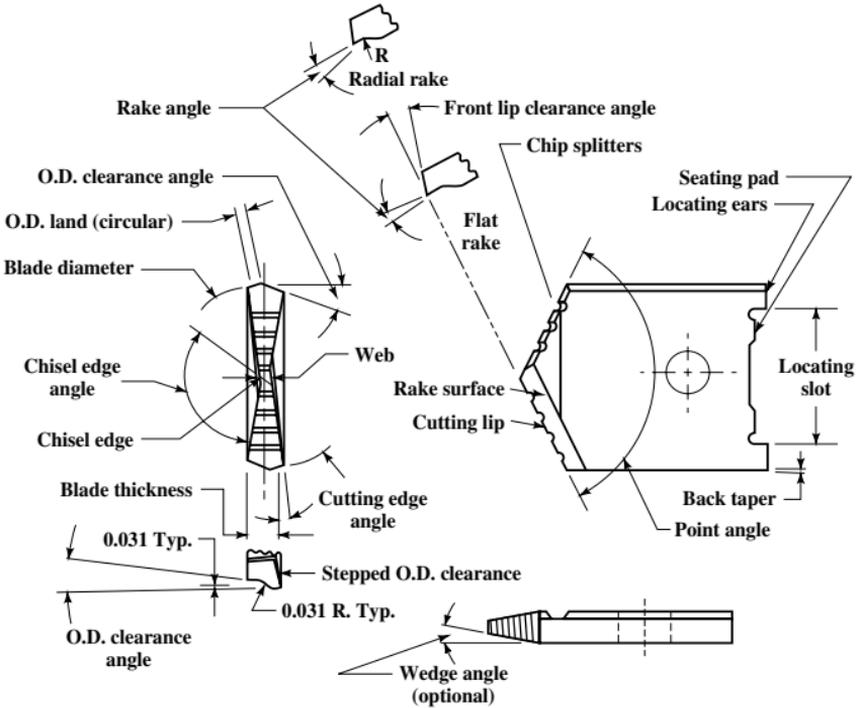


Fig. 3. Spade Drill Blade

On spade drills, the front lip clearance angle provides the relief. It may be ground on a drill grinding machine but usually it is ground flat. The normal front lip clearance angle is 8 degrees; in some instances, a secondary relief angle of about 14 degrees is ground below the primary clearance. The wedge angle on the blade is optional. It is generally ground on thicker blades having a larger diameter to prevent heel dragging below the cutting lip and to reduce the chisel edge length. The outside-diameter land is circular, serving to support and guide the blade in the hole. Usually it is ground to have a back taper of 0.001 to 0.002 inch per inch per side. The width of the land is approximately 20 to 25 per cent of the blade thickness. Normally, the outside-diameter clearance angle behind the land is 7 to 10 degrees. On many spade drill blades, the outside-diameter clearance surface is stepped about 0.030 inch below the land.

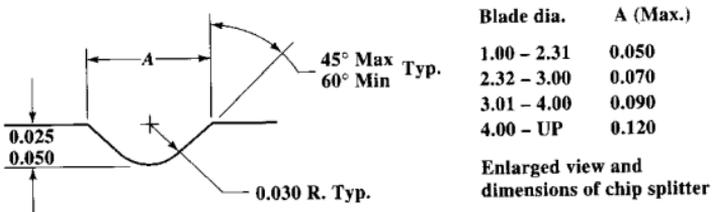


Fig. 4. Spade Drill Chip Splitter Dimensions

Spade Drilling.—Spade drills are used on drilling machines and other machine tools where the cutting tool rotates; they are also used on turning machines where the work

rotates and the tool is stationary. Although there are some slight operational differences, the methods of using spade drills are basically the same. An adequate supply of cutting fluid must be used, which serves to cool and lubricate the cutting edges; to cool the chips, thus making them brittle and more easily broken; and to flush chips out of the hole. Flood cooling from outside the hole can be used for drilling relatively shallow holes, of about one to two and one-half times the diameter in depth. For deeper holes, the cutting fluid should be injected through the holes in the drill. When drilling very deep holes, it is often helpful to blow compressed air through the drill in addition to the cutting fluid to facilitate ejection of the chips. Air at full shop pressure is throttled down to a pressure that provides the most efficient ejection. The cutting fluids used are light and medium cutting oils, water-soluble oils, and synthetics, and the type selected depends on the work material.

Starting a spade drill in the workpiece needs special attention. The straight chisel edge on the spade drill has a tendency to wander as it starts to enter the work, especially if the feed is too light. This wander can result in a mispositioned hole and possible breakage of the drill point. The best method of starting the hole is to use a stub or short-length spade drill holder and a blade of full size that should penetrate at least $\frac{1}{8}$ inch at full diameter. The holder is then changed for a longer one as required to complete the hole to depth. Difficulties can be encountered if spotting with a center drill or starting drill is employed because the angles on these drills do not match the 130-degree point angle of the spade drill. Longer spade drills can be started without this starting procedure if the drill is guided by a jig bushing and if the holder is provided with wear strips.

Chip formation warrants the most careful attention as success in spade drilling is dependent on producing short, well-broken chips that can be easily ejected from the hole. Straight, stringy chips or chips that are wound like a clock spring cannot be ejected properly; they tend to pack around the blade, which may result in blade failure. The chip splitters must be functioning to produce a series of narrow chips along each cutting edge. Each chip must be broken, and for drilling ductile materials they should be formed into a "C" or "figure 9" shape. Such chips will readily enter the flutes on the holder and flow out of the hole.

Proper chip formation is dependent on the work material, the spade drill geometry, and the cutting conditions. Brittle materials such as gray cast iron seldom pose a problem because they produce a discontinuous chip, but austenitic stainless steels and very soft and ductile materials require much attention to obtain satisfactory chip control. Thinning the web or grinding a split point on the blade will sometimes be helpful in obtaining better chip control, as these modifications allow use of a heavier feed. Reducing the rake angle to obtain a tighter curl on the chip and grinding a corner chamfer on the tool will sometimes help to produce more manageable chips.

In most instances, it is not necessary to experiment with the spade drill blade geometry to obtain satisfactory chip control. Control usually can be accomplished by adjusting the cutting conditions; i.e., the cutting speed and the feed rate.

Normally, the cutting speed for spade drilling should be 10 to 15 per cent lower than that for an equivalent twist drill, although the same speed can be used if a lower tool life is acceptable. The recommended cutting speeds for twist drills on [Tables 17 through 23](#), starting on page [1061](#), can be used as a starting point; however, they should be decreased by the percentage just given. It is essential to use a heavy feed rate when spade drilling to produce a thick chip, and to force the chisel edge into the work. In ductile materials, a light feed will produce a thin chip that is very difficult to break. The thick chip on the other hand, which often contains many rupture planes, will curl and break readily. [Table 1](#) gives suggested feed rates for different spade drill sizes and materials. These rates should be used as a starting point and some adjustments may be necessary as experience is gained.

Table 1. Feed Rates for Spade Drilling

Material	Hardness, Bhn	Feed—Inches per Revolution					
		Spade Drill Diameter—Inches					
		1-1¼	1¼-2	2-3	3-4	4-5	5-8
Free Machining Steel	100-240	0.014	0.016	0.018	0.022	0.025	0.030
	240-325	0.010	0.014	0.016	0.020	0.022	0.025
	100-225	0.012	0.015	0.018	0.022	0.025	0.030
Plain Carbon Steels	225-275	0.010	0.013	0.015	0.018	0.020	0.025
	275-325	0.008	0.010	0.013	0.015	0.018	0.020
	150-250	0.014	0.016	0.018	0.022	0.025	0.030
Free Machining Alloy Steels	250-325	0.012	0.014	0.016	0.018	0.020	0.025
	325-375	0.010	0.010	0.014	0.016	0.018	0.020
	125-180	0.012	0.015	0.018	0.022	0.025	0.030
Alloy Steels	180-225	0.010	0.012	0.016	0.018	0.022	0.025
	225-325	0.009	0.010	0.013	0.015	0.018	0.020
	325-400	0.006	0.008	0.010	0.012	0.014	0.016
Tool Steels							
Water Hardening	150-250	0.012	0.014	0.016	0.018	0.020	0.022
Shock Resisting	175-225	0.012	0.014	0.015	0.016	0.017	0.018
Cold Work	200-250	0.007	0.008	0.009	0.010	0.011	0.012
Hot Work	150-250	0.012	0.013	0.015	0.016	0.018	0.020
Mold	150-200	0.010	0.012	0.014	0.016	0.018	0.018
Special-Purpose	150-225	0.010	0.012	0.014	0.016	0.016	0.018
High-Speed	200-240	0.010	0.012	0.013	0.015	0.017	0.018
	110-160	0.020	0.022	0.026	0.028	0.030	0.034
Gray Cast Iron	160-190	0.015	0.018	0.020	0.024	0.026	0.028
	190-240	0.012	0.014	0.016	0.018	0.020	0.022
	240-320	0.010	0.012	0.016	0.018	0.018	0.018
	140-190	0.014	0.016	0.018	0.020	0.022	0.024
Ductile or Nodular Iron	190-250	0.012	0.014	0.016	0.018	0.018	0.020
	250-300	0.010	0.012	0.016	0.018	0.018	0.018
Malleable Iron							
Ferritic	110-160	0.014	0.016	0.018	0.020	0.022	0.024
Pearlitic	160-220	0.012	0.014	0.016	0.018	0.020	0.020
	220-280	0.010	0.012	0.014	0.016	0.018	0.018
Free Machining Stainless Steel							
Ferritic	...	0.016	0.018	0.020	0.024	0.026	0.028
Austenitic	...	0.016	0.018	0.020	0.022	0.024	0.026
Martensitic	...	0.012	0.014	0.016	0.016	0.018	0.020
Stainless Steel							
Ferritic	...	0.012	0.014	0.018	0.020	0.020	0.022
Austenitic	...	0.012	0.014	0.016	0.018	0.020	0.020
Martensitic	...	0.010	0.012	0.012	0.014	0.016	0.018
Aluminum Alloys	...	0.020	0.022	0.024	0.028	0.030	0.040
Copper Alloys	(Soft)	0.016	0.018	0.020	0.026	0.028	0.030
	(Hard)	0.010	0.012	0.014	0.016	0.018	0.018
Titanium Alloys	...	0.008	0.010	0.012	0.014	0.014	0.016
High-Temperature Alloys	...	0.008	0.010	0.012	0.012	0.014	0.014

Power Consumption and Thrust for Spade Drilling.—In each individual setup, there are factors and conditions influencing power consumption that cannot be accounted for in a simple equation; however, those given below will enable the user to estimate power consumption and thrust accurately enough for most practical purposes. They are based on experimentally derived values of unit horsepower, as given in Table 2. As a word of caution, these values are for sharp tools. In spade drilling, it is reasonable to estimate that a dull tool will increase the power consumption and the thrust by 25 to 50 per cent. The unit horsepower values in the table are for the power consumed at the cutting edge, to which must be added the power required to drive the machine tool itself, in order to obtain the horsepower required by the machine tool motor. An allowance for power to drive the machine is provided by dividing the horsepower at the cutter by a mechanical efficiency factor, e_m . This factor can be estimated to be 0.90 for a direct spindle drive with a belt, 0.75 for a back gear drive, and 0.70 to 0.80 for geared head drives. Thus, for spade drilling the formulas are

$$hp_c = uhp \left(\frac{\pi D^2}{4} \right) f N$$

$$B_s = 148,500 \text{ uhp} f D$$

$$hp_m = \frac{hp_c}{e_m}$$

$$f = \frac{f_m}{N}$$

where hp_c = horsepower at the cutter hp_m = horsepower at the motor

B_s = thrust for spade drilling in pounds

uhp = unit horsepower

D = drill diameter in inches

f = feed in inches per revolution

f_m = feed in inches per minute

N = spindle speed in revolutions per minute

e_m = mechanical efficiency factor

Table 2. Unit Horsepower for Spade Drilling

Material	Hardness	uhp	Material	Hardness	uhp
Plain Carbon and Alloy Steel	85–200 Bhn	0.79	Titanium Alloys	250–375 Bhn	0.72
	200–275	0.94	High-Temp Alloys	200–360 Bhn	1.44
	275–375	1.00	Aluminum Alloys	...	0.22
	375–425	1.15	Magnesium Alloys	...	0.16
	45–52 Rc	1.44	Copper Alloys	20–80 Rb	0.43
Cast Irons	110–200 Bhn	0.5		80–100 Rb	0.72
	200–300	1.08			
Stainless Steels	135–275 Bhn	0.94			
	30–45 Rc	1.08			

Example: Estimate the horsepower and thrust required to drive a 2-inch diameter spade drill in AISI 1045 steel that is quenched and tempered to a hardness of 275 Bhn. From Table 17 on page 1061, the cutting speed, V , for drilling this material with a twist drill is 50 feet per minute. This value is reduced by 10 per cent for spade drilling and the speed selected is thus $0.9 \times 50 = 45$ feet per minute. The feed rate (from Table 1, page 1079) is 0.015 in/rev. and the unit horsepower from Table 2 above is 0.94. The machine efficiency factor is estimated to be 0.80 and it will be assumed that a 50 per cent increase in the unit horsepower must be allowed for dull tools.

Step 1. Calculate the spindle speed from the following formula:

$$N = \frac{12V}{\pi D}$$

where: N = spindle speed in revolutions per minute

V = cutting speed in feet per minute

D = drill diameter in inches

$$\text{Thus, } N = \frac{12 \times 45}{\pi \times 2} = 86 \text{ revolutions per minute}$$

Step 2. Calculate the horsepower at the cutter:

$$\text{hp}_c = \text{uhp} \left(\frac{\pi D^2}{4} \right) f N = 0.94 \left(\frac{\pi \times 2^2}{4} \right) 0.015 \times 86 = 3.8$$

Step 3. Calculate the horsepower at the motor and provide for a 50 per cent power increase for the dull tool:

$$\text{hp}_m = \frac{\text{hp}_c}{e_m} = \frac{3.8}{0.80} = 4.75 \text{ horsepower}$$

$$\text{hp}_m \text{ (with dull tool)} = 1.5 \times 4.75 = 7.125 \text{ horsepower}$$

Step 4. Estimate the spade drill thrust:

$$B_s = 148,500 \times \text{uhp} \times f D = 148,500 \times 0.94 \times 0.015 \times 2 = 4188 \text{ lb (for sharp tool)}$$

$$B_s = 1.5 \times 4188 = 6282 \text{ lb (for dull tool)}$$

Trepanning.—Cutting a groove in the form of a circle or boring or cutting a hole by removing the center or core in one piece is called trepanning. Shallow trepanning, also called face grooving, can be performed on a lathe using a single-point tool that is similar to a grooving tool but has a curved blade. Generally, the minimum outside diameter that can be cut by this method is about 3 inches and the maximum groove depth is about 2 inches. Trepanning is probably the most economical method of producing deep holes that are 2 inches, and larger, in diameter. Fast production rates can be achieved. The tool consists of a hollow bar, or stem, and a hollow cylindrical head to which a carbide or high-speed steel, single-point cutting tool is attached. Usually, only one cutting tool is used although for some applications a multiple cutter head must be used; e.g., heads used to start the hole have multiple tools. In operation, the cutting tool produces a circular groove and a residue core that enters the hollow stem after passing through the head. On outside-diameter exhaust trepanning tools, the cutting fluid is applied through the stem and the chips are flushed around the outside of the tool; inside-diameter exhaust tools flush the chips out through the stem with the cutting fluid applied from the outside. For starting the cut, a tool that cuts a starting groove in the work must be used, or the trepanning tool must be guided by a bushing. For holes less than about five diameters deep, a machine that rotates the trepanning tool can be used. Often, an ordinary drill press is satisfactory; deeper holes should be machined on a lathe with the work rotating. A hole diameter tolerance of ± 0.010 inch can be obtained easily by trepanning and a tolerance of ± 0.001 inch has sometimes been held. Hole runout can be held to ± 0.003 inch per foot and, at times, to ± 0.001 inch per foot. On heat-treated metal, a surface finish of 125 to 150 μm AA can be obtained and on annealed metals 100 to 250 μm AA is common.

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:

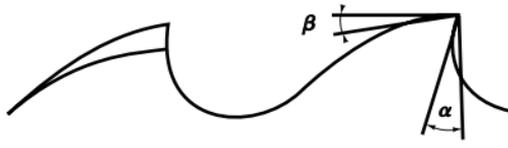
$$T = \frac{W}{F} \left[L \times \left(\frac{1}{V_c} + \frac{1}{V_r} \right) + 0.025 \right]$$

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 = $3.142 \times$ blade diameter in inches \times rpm of saw shaft $\div 12$).

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)			
			$\frac{1}{4}$ - $\frac{3}{4}$	$\frac{3}{4}$ -1 $\frac{1}{2}$	1 $\frac{1}{2}$ -2 $\frac{1}{2}$	2 $\frac{1}{2}$ -3 $\frac{1}{2}$
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 3a](#), and [3b](#) can be used for all machining operations except drilling and grinding. Values given are for sharp tools.

Table 3a. Power Constants, K_p , Using Sharp Cutting Tools

Material	Brinell Hardness	K_p Inch Units	K_p Metric Units	Material	Brinell Hardness	K_p Inch Units	K_p Metric Units
Ferrous Cast Metals							
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				
Alloy Cast Iron	150–175	0.30	0.82	Cast Steel	150–175	0.62	1.69
	175–200	0.63	1.72		175–200	0.78	2.13
	200–250	0.92	2.51		200–250	0.86	2.35
			

Table 3a. (Continued) Power Constants, K_p , Using Sharp Cutting Tools

Material	Brinell Hardness	K_p Inch Units	K_p Metric Units	Material	Brinell Hardness	K_p Inch Units	K_p Metric Units
High-Temperature Alloys, Tool Steel, Stainless Steel, and Nonferrous Metals							
High-Temperature Alloys					150-175	0.60	1.64
A286	165	0.82	2.24	Stainless Steel {	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
	175-200	0.75	2.05	Aluminum			
	200-250	0.88	2.40	Cast	...	0.25	0.68
Tool Steel {	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

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 4](#). 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 5](#). 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 3b. Power Constants, K_p , Using Sharp Cutting Tools

Material	Brinell Hardness	K_p Inch Units	K_p Metric Units	Material	Brinell Hardness	K_p Inch Units	K_p SI Metric Units
Wrought Steels							
Plain Carbon Steels							
All Plain Carbon Steels	80–100	0.63	1.72	All Plain Carbon Steels	220–240	0.89	2.43
	100–120	0.66	1.80		240–260	0.92	2.51
	120–140	0.69	1.88		260–280	0.95	2.59
	140–160	0.74	2.02		280–300	1.00	2.73
	160–180	0.78	2.13		300–320	1.03	2.81
	180–200	0.82	2.24		320–340	1.06	2.89
	200–220	0.85	2.32		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	AISI 1137, 1138, 1139, 1140, 1141, 1144, 1145, 1146, 1148, 1151	180–200	0.51	1.39
	120–140	0.42	1.15		200–220	0.55	1.50
	140–160	0.44	1.20		220–240	0.57	1.56
	160–180	0.48	1.31		240–260	0.62	1.69
	180–200	0.50	1.36	
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	AISI 4130, 4320, 4615, 4620, 4626, 5120, 8615, 8617, 8620, 8622, 8625, 8630, 8720	140–160	0.56	1.53
	160–180	0.65	1.77		160–180	0.59	1.61
	180–200	0.69	1.88		180–200	0.62	1.69
	200–220	0.72	1.97		200–220	0.65	1.77
	220–240	0.76	2.07		220–240	0.70	1.91
	240–260	0.80	2.18		240–260	0.74	2.02
	260–280	0.84	2.29		260–280	0.77	2.10
	280–300	0.87	2.38		280–300	0.80	2.18
	300–320	0.91	2.48		300–320	0.83	2.27
	320–340	0.96	2.62		320–340	0.89	2.43
	340–360	1.00	2.73	
AISI 1330, 1335, 1340, E52100	160–180	0.79	2.16	
	180–200	0.83	2.27	
	200–220	0.87	2.38	

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 6. Formulas for calculating the metal removal rate, Q , for different machining operations are given in Table 7. These formulas are used together with others given below. The following formulas can be used with either customary inch or with SI metric units.

$$P_c = K_p C Q W \quad (1)$$

$$P_m = \frac{P_c}{E} = \frac{K_p C Q W}{E} \quad (2)$$

where P_c = power at the cutting tool; hp, or kW

Table 4. 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

^aTurning, in/rev; milling, in/tooth; planing and shaping, in/stroke; broaching, in/tooth.

^bTurning, mm/rev; milling, mm/tooth; planing and shaping, mm/stroke; broaching, mm/tooth.

Table 5. 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
Planing and Shaping	Use values given for turning	

P_m = power at the motor; hp, or kW

K_p = power constant (see Tables 3a and 3b)

Q = metal removal rate; in³/min or cm³/s (see Table 7)

C = feed factor for power constant (see Table 4)

W = tool wear factor (see Table 5)

E = machine tool efficiency factor (see Table 6)

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; inch, or mm
 d = depth of cut; inch, or mm
 n_t = number of teeth on milling cutter
 n_c = number of teeth engaged in work
 w = width of cut; inch, or mm

Table 6. 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

Table 7. Formulas for Calculating the Metal Removal Rate, Q

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

Example: A 180–200 Bhn AISI 4130 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:

$$K_p = 0.62 \text{ (from Table 3b)}$$

$$C = 0.94 \text{ (from Table 4)}$$

$$W = 1.30 \text{ (from Table 5)}$$

$$E = 0.80 \text{ (from Table 6)}$$

$$Q = 12 Vfd = 12 \times 350 \times 0.016 \times 0.100 \text{ (from Table 7)}$$

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

$$P_c = K_p C Q W = 0.62 \times 0.94 \times 6.72 \times 1.30 = 5.1 \text{ hp}$$

$$P_m = \frac{P_c}{E} = \frac{5}{0.80} = 6.4 \text{ hp}$$

SI metric units:

$$K_p = 1.69 \text{ (from Table 3b)}$$

$$C = 0.94 \text{ (from Table 4)}$$

$$W = 1.30 \text{ (from Table 5)}$$

$$E = 0.80 \text{ (from Table 6)}$$

$$Q = \frac{V}{60}fd = \frac{107}{60} \times 0.40 \times 2.54 = 1.81 \text{ cm}^3/\text{s} \text{ (from Table 7)}$$

$$P_c = K_p C Q W = 1.69 \times 0.94 \times 1.81 \times 1.30 = 3.74 \text{ kW}$$

$$P_m = \frac{P_c}{E} = \frac{3.74}{0.80} = 4.677 \text{ kW}$$

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.

$$K_p = 0.85 \text{ (From Table 3b)}$$

$$C = 0.92 \text{ (From Table 4)}$$

$$W = 1.30 \text{ (From Table 5)}$$

$$E = 0.80 \text{ (From Table 6)}$$

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

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

$$V = \frac{Q_{max}}{12fd} = \frac{7.87}{12 \times 0.018 \times 0.125} \quad (Q = 12Vfd)$$

$$= 291 \text{ fpm}$$

Example: A 160–180 Bhn gray iron casting that is 6 inches wide is to have $\frac{1}{8}$ 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 inch) will be removed in one cut. Estimate the cutting speed that will utilize the maximum power available on the machine.

$$K_p = 0.52 \text{ (From Table 3a)}$$

$$C = 1.00 \text{ (From Table 4)}$$

$$W = 1.20 \text{ (From Table 5)}$$

$$E = 0.80 \text{ (From Table 6)}$$

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

$$f_m = \frac{Q_{max}}{w d} = \frac{12.82}{6 \times 0.125} = 17.1 \text{ in/min} \quad (Q = f_m w d)$$

$$N = \frac{f_{max}}{f_t n_t} = \frac{17}{0.012 \times 10} = 142.4 \text{ rpm} \quad (f_m = f_t n_t N)$$

$$V = \frac{\pi D N}{12} = \frac{\pi \times 8 \times 142}{12} = 298.3 \text{ fpm} \quad \left(N = \frac{12 V}{\pi D} \right)$$

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 8 through 11 are for sharp drills, and tool wear factors are given in Table 5. 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 9. 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 8. Work Material Factor, K_d , for Drilling with a Sharp Drill

Work Material	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 Ti6Al4V 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 9. Chisel Edge Factors for Torque and Thrust

<i>c/d</i>	Approx. <i>w/d</i>	Torque Factor <i>A</i>	Thrust Factor <i>B</i>	Thrust Factor <i>J</i>	<i>c/d</i>	Approx. <i>w/d</i>	Torque Factor <i>A</i>	Thrust Factor <i>B</i>	Thrust Factor <i>J</i>
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 $c/d = 0.18$; for split point drills, use $c/d = 0.03$

c/d = Length of Chisel Edge \div Drill Diameter.

w/d = Web Thickness at Drill Point \div Drill Diameter.

For inch units only:

$$T = 2K_d F_f F_T B W + K_d D^2 J W \quad (1)$$

$$M = K_d F_f F_M A W \quad (2)$$

$$P_c = MN / 63.025 \quad (3)$$

For SI metric units only:

$$T = 0.05 K_d F_f F_T B W + 0.007 K_d D^2 J W \quad (4)$$

$$M = \frac{K_d F_f F_M A W}{40,000} = 0.000025 K_d F_f F_M A W \quad (5)$$

$$P_c = MN / 9550 \quad (6)$$

Use with either inch or metric units:

$$P_m = \frac{P_c}{E} \quad (7)$$

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 8)

F_f = Feed factor (See Table 10)

F_T = Thrust factor for drill diameter (See Table 11)

F_M = Torque factor for drill diameter (See Table 11)

A = Chisel edge factor for torque (See Table 9)

B = Chisel edge factor for thrust (See Table 9)

J = Chisel edge factor for thrust (See Table 9)

W = Tool wear factor (See Table 5)

N = Spindle speed; rpm

E = Machine tool efficiency factor (See Table 6)

D = Drill diameter; in., or mm

c = Chisel edge length; in., or mm (See Table 9)

w = Web thickness at drill point; in., or mm (See Table 9)

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:

$$K_d = 24,000 \text{ (From Table 8)} \quad F_f = 0.021 \text{ (From Table 10)}$$

$$F_T = 0.899 \text{ (From Table 11)} \quad F_M = 0.786 \text{ (From Table 11)}$$

$$A = 1.085 \text{ (From Table 9)} \quad B = 1.355 \text{ (From Table 9)} \quad J = 0.030 \text{ (From Table 9)}$$

Table 10. 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 11. 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

$W = 1.30$ (From Table 5)

$$T = 2K_d F_f F_T B W + K_d d^2 J W$$

$$= 2 \times 24,000 \times 0.21 \times 0.899 \times 1.355 \times 1.30 + 24,000 \times 0.875^2 \times 0.030 \times 1.30$$

$$= 2313 \text{ lb}$$

$$M = K_d F_f F_M A W$$

$$= 24,000 \times 0.021 \times 0.786 \times 1.085 \times 1.30 = 559 \text{ in. lb}$$

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

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.

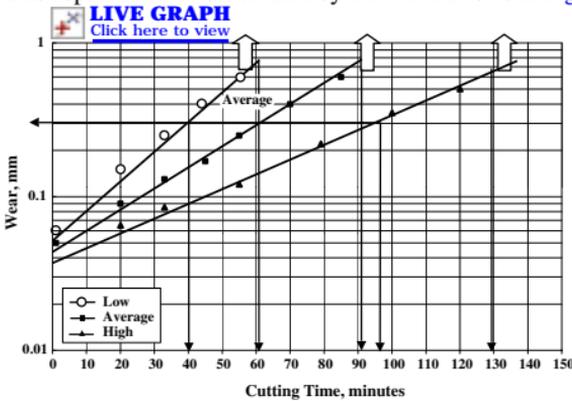


Fig. 1. Flank Wear as a Function of Cutting Time

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 \times depth of cut), (mm² or inch²)

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 $CEL = 4$ mm², 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:

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

Example 2: Using above data and a cutting speed of $V = 250$ m/min specific metal removal rate becomes $SMRR = 0.6 \times 250 = 150$ (cm³/min/mm).

ECT in Grinding: In grinding *ECT* is defined as in the other metal cutting processes, and is approximately equal to $ECT = V_w \times ar \div V$, where V_w is the work speed, ar is the depth of cut, and $A = V_w \times ar$. Wheel life is constant no matter how depth ar , or work speed V_w , is selected at $V = \text{constant}$ (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:

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

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 1158.

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.

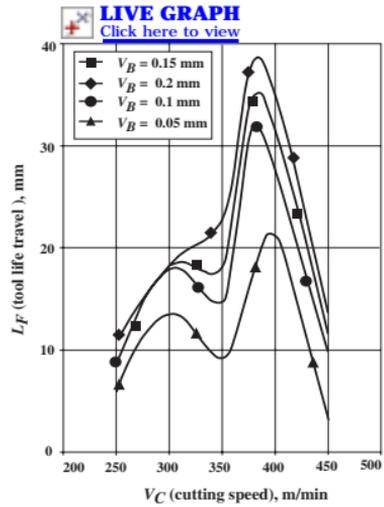
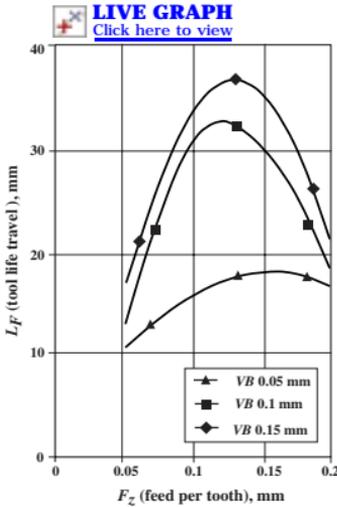


Fig. 2. Influence of feed per tooth on cutting time Fig. 3. Influence of cutting speed on tool-life

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:

$$V \times T^n = C \quad (1a)$$

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

$$\ln V + n \times \ln T = \ln C \quad (1b)$$

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.

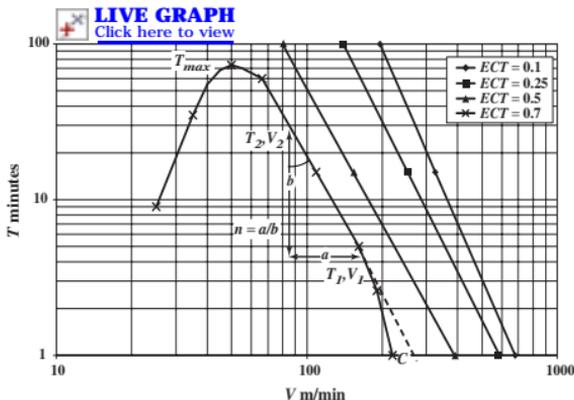


Fig. 4. Definition of slope n and constant C in Taylor's equation

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

$$T^n = A \times f^m \times a^p \times LA^q \times r^s \quad (2)$$

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 ($f, a, LA, r, \text{etc.}$) 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 $x = \ln ECT$ $y = \ln V$ $z = \ln T$

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: $n = N_0 - L \times \ln(ECT)$

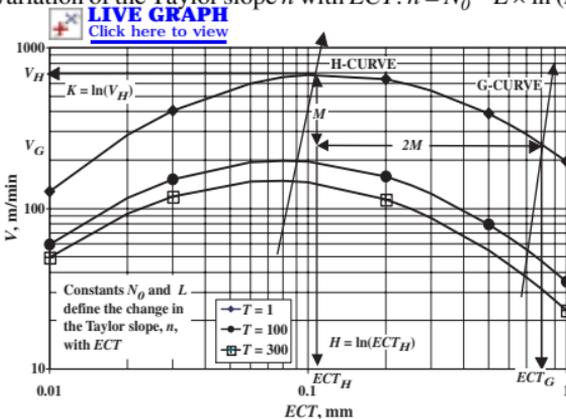


Fig. 5. Definitions of the constants $H, K, L, M,$ and N_0 for tool-life equation in the V -ECT plane with tool-life constant

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 1110.

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.

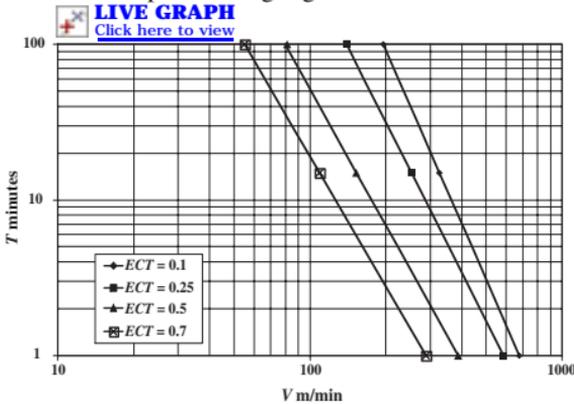


Fig. 6a. Tool-life vs. cutting speed T - V , ECT plotted

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.

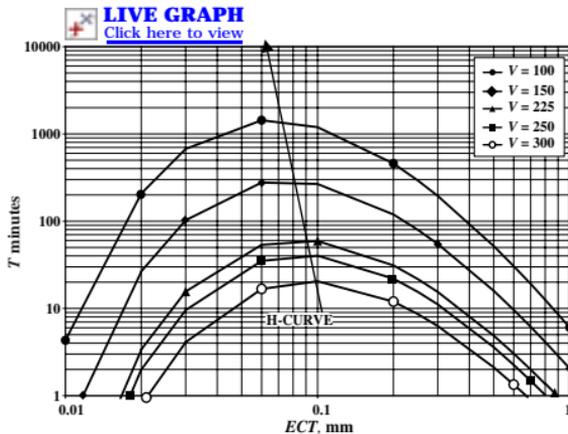


Fig. 6b. Tool-life vs. ECT , T - ECT , cutting speed plotted

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.

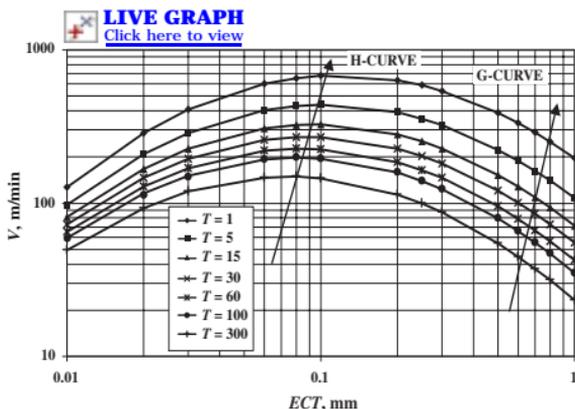


Fig. 6c. Cutting speed vs. ECT , V - ECT , tool-life plotted

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 AE and BC. The border curve, line AB, shows a lower limit of tool-life, $T_{MIN} = 5$ minutes, and border curve, line 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.

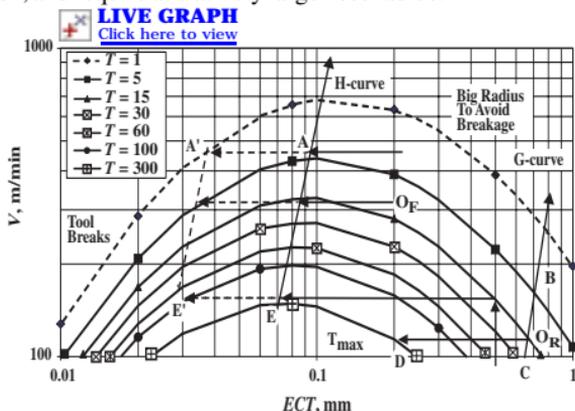


Fig. 7. Cutting speed vs. ECT , V - ECT , tool-life plotted

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 approx-

imately 0.03 mm should not be allowed. In Fig. 7, the ECT_{MIN} boundary is indicated by contour line $A'E'$.

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 1122 for more on this topic.

Fig. 7 demonstrates, in principle, minimum cost conditions for roughing at point O_R , and for finishing at point O_F , where surface finish or tolerances have set a limit. Maintaining the speed at 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 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.

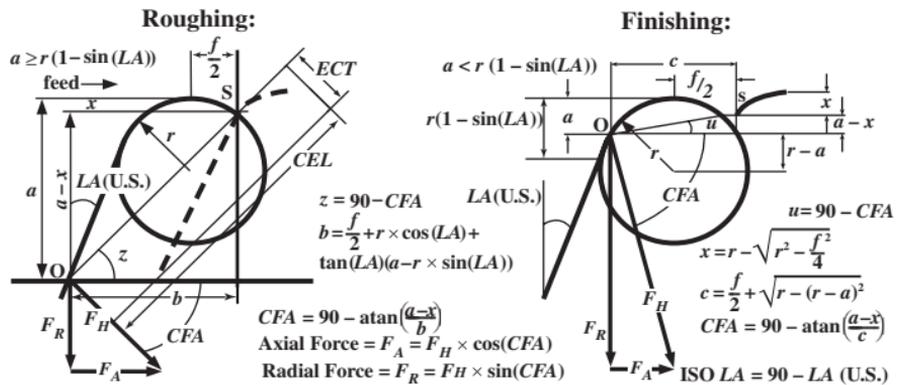


Fig. 8. Definitions of equivalent chip thickness, ECT , and chip flow angle, CFA .

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 1082.

Specific Cutting Force, K_c : The specific cutting force, or the specific energy to cut, K_c , is defined as the ratio between the cutting force F_C and the chip cross sectional area, A , thus, $K_c = F_C \div A$ N/mm².

The value of K_c decreases when ECT increases, and when the cutting speed V increases. Usually, K_c is written in terms of its value at $ECT = 1$, called K_{c1} , and neglecting the effect of cutting speed, thus $K_c = K_{c1} \times ECT^B$, where B = slope in log-log coordinates

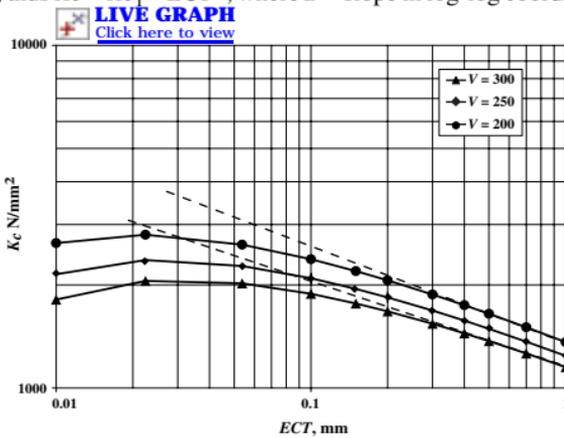


Fig. 9. K_c vs. ECT , cutting speed plotted

A more accurate relationship is illustrated in Fig. 9, where K_c 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, K_c varies with ECT from about 1900 to 1300 N/mm² when ECT increases from 0.1 to 0.7 mm. Generally the speed effect on the magnitude of K_c is approximately 5 to 15 percent when using economic speeds.

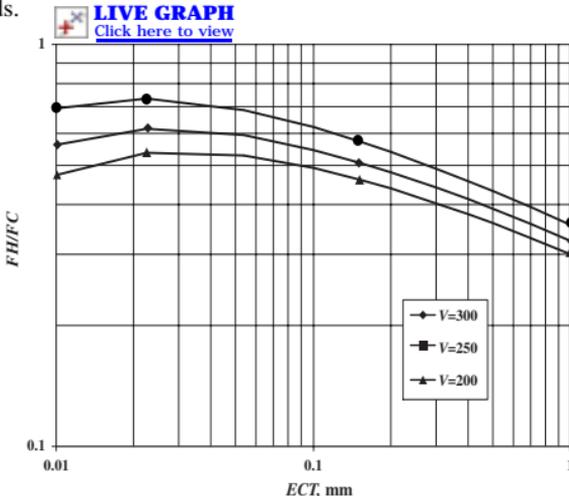


Fig. 10. F_H/F_C vs. ECT , cutting speed plotted

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 and the cutting force F_C , in turning.

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

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

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:

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

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

Finishing:

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

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

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.

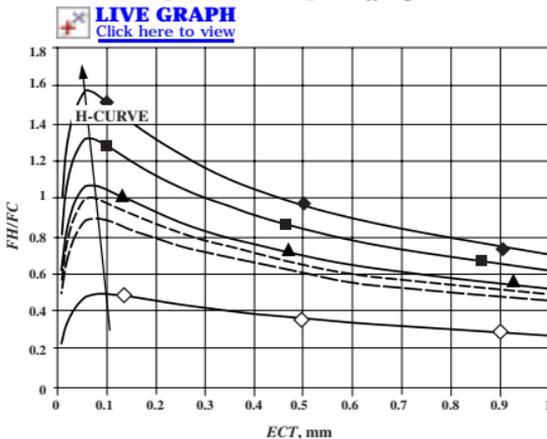


Fig. 11a. F_H/F_C vs. ECT

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 .

 [LIVE GRAPH](#)
[Click here to view](#)

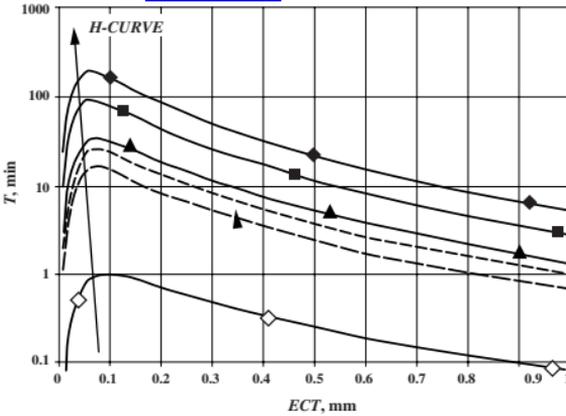


Fig. 11b. Tool-life vs. ECT

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).

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

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 its 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.

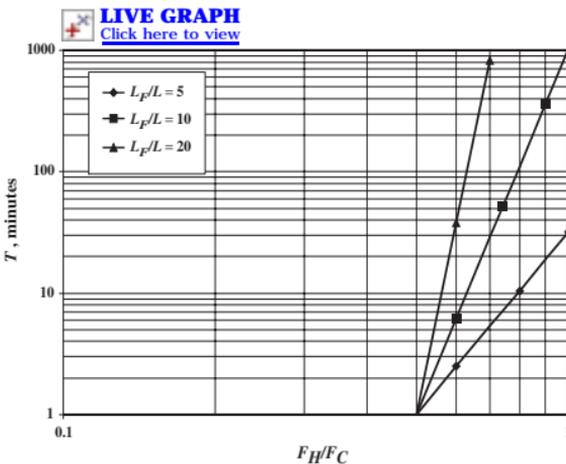


Fig. 12. Tool-life vs. F_H/F_C

Tool-life varies with F_H/F_C with a 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 $L_F/L = 5, 10, \text{ and } 20$.

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.

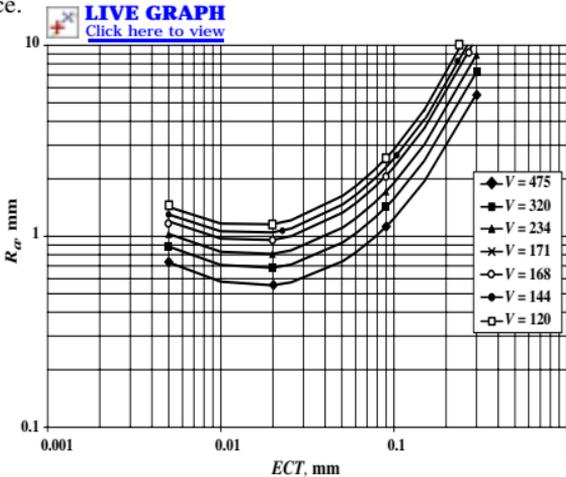


Fig. 13. R_a vs. ECT , nose radius r constant

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,

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

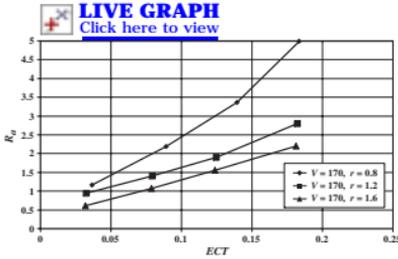


Fig. 14. R_a vs. ECT
cutting speed constant, nose radius r varies

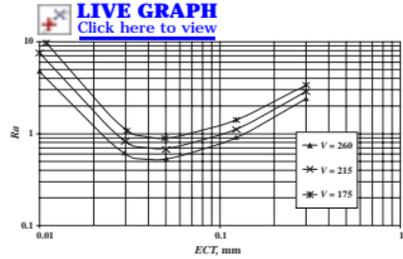


Fig. 15. R_a vs. ECT ,
cutting speed and nose radius r constant

In roughing, multiply the finishing values found using Equation (6) by 1.5, thus, $R_{a(Rough)} = 1.5 \times R_{a(Finish)}$ 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.

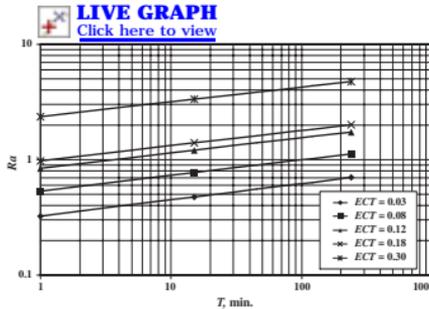


Fig. 16. R_a vs. T , holding ECT constant

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

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

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.

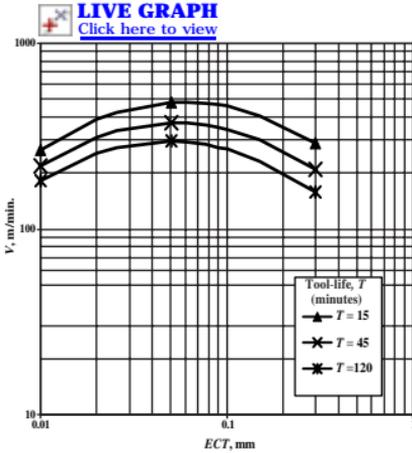


Fig. 17a. Tool-life for turning cast iron using coated carbide

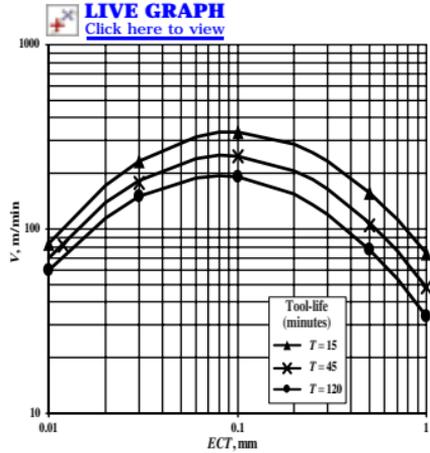


Fig. 17b. Tool-life for turning low-alloy steel using coated carbide

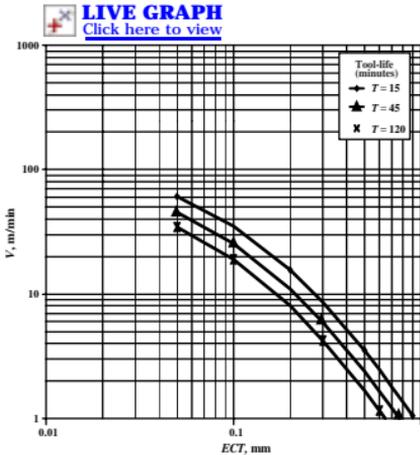


Fig. 17c. Tool-life for end-milling AISI4140 steel using high-speed steel

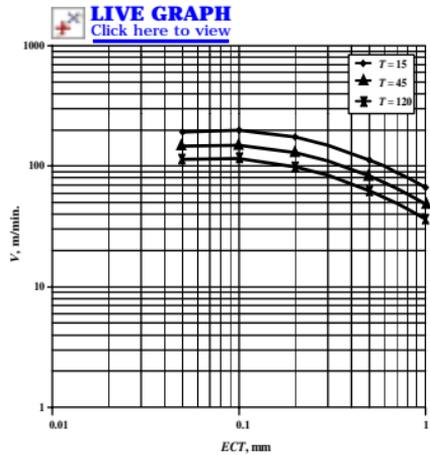


Fig. 17d. Tool-life for end-milling low-alloy steel using uncoated carbide

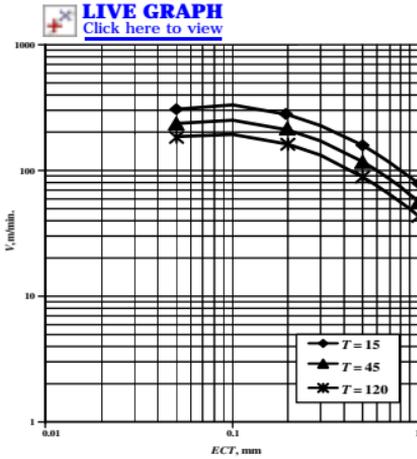


Fig. 17e. Tool-life for end-milling low-alloy steel using coated carbide

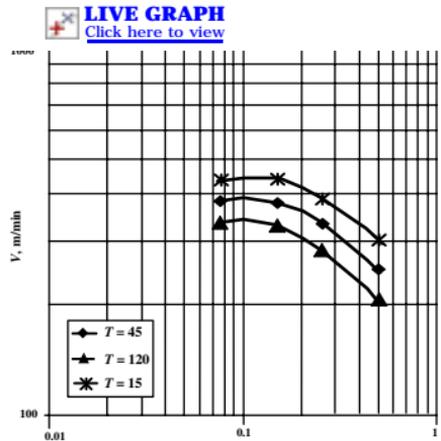


Fig. 17f. Tool-life for face-milling SAE 1045 steel using coated carbide

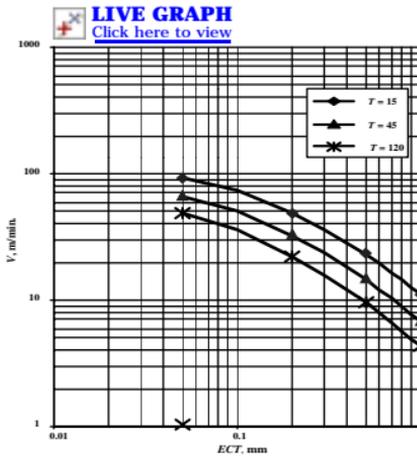


Fig. 17g. Tool-life for solid carbide drill

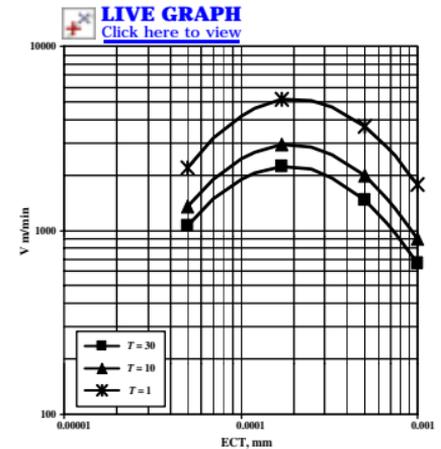


Fig. 17h. Wheel-life in grinding M4 tool-steel

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 $C_{TOT} = \text{cost of machining time} + \text{tool changing cost} + \text{tooling cost}$. 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.

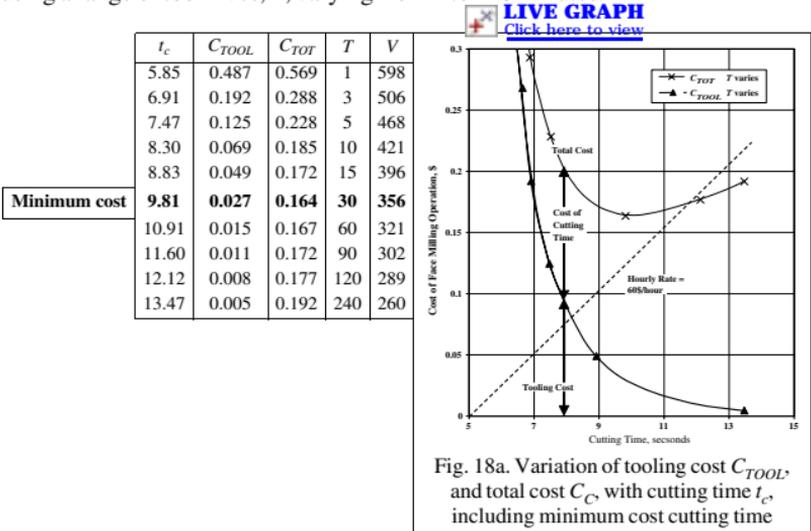


Fig. 18a. Variation of tooling cost C_{TOOL} and total cost C_C with cutting time t_c , including minimum cost cutting time

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 .

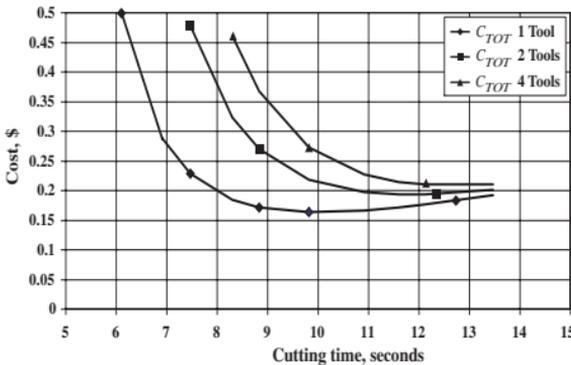


Fig. 18b. Total cost vs. cutting time for simultaneously cutting with 1, 2, and 4 tools

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				
f_z	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
0.60	119	4.903	0.0186	0.0867
0.70	91	4.924	0.0187	0.0871
Maximum Production Rate, $T = 5$ minutes				
f_z	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
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

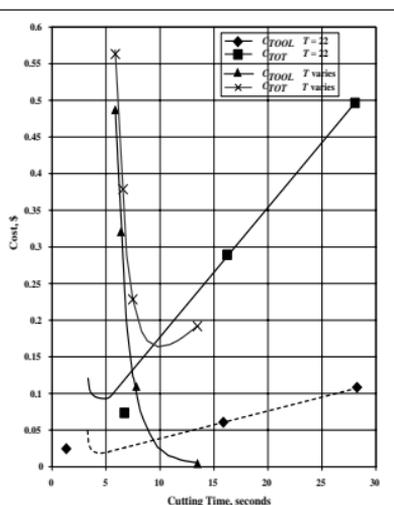


Fig. 19. Variation of tooling and total cost with cutting time, comparing global optimum with minimum cost at $f_z = 0.1$ mm

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

1115). 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: $V \times T = C$ and differentiating C_{TOT} with respect to T yields:

Economic tool-life:

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

Economic cutting speed:

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

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 1027 and handbook Table 5b on page 1035 for turning, and Table 15e on page 1059 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.

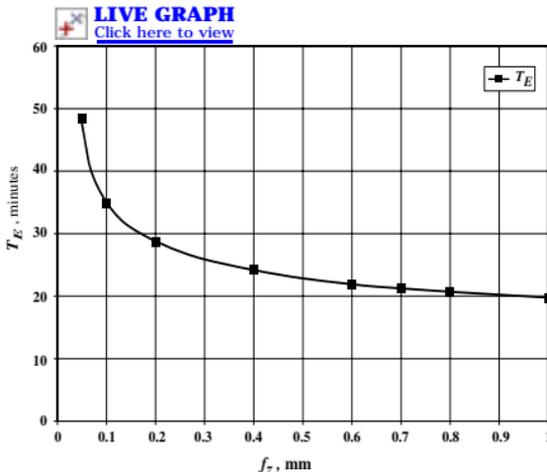


Fig. 20a. Economic tool-life, T_E vs. feed per tooth, f_z

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.

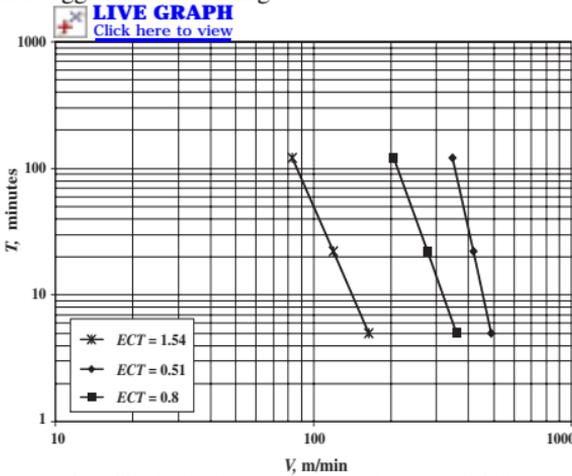


Fig. 20b. Tool-life vs. cutting speed, constant ECT

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 $H_R = \$60/\text{hour}$, 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,

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

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: $T=1, 5, 15, 30, 60$ 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 $T=1, 5, \text{etc.}$, 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.

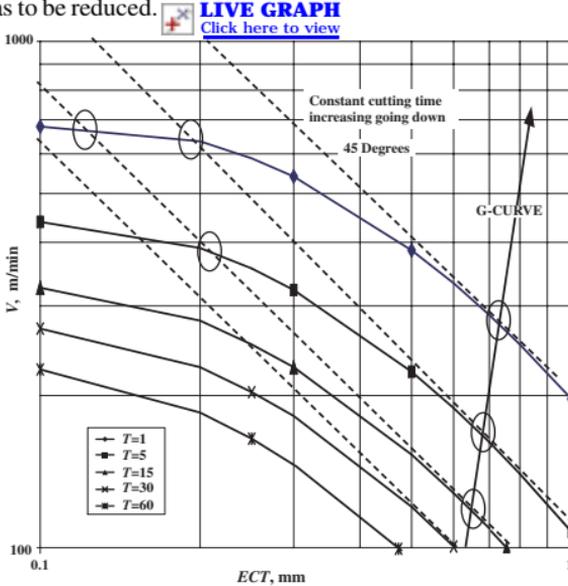


Fig. 21. Constant cutting time in the V - ECT plane, tool-life constant

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 1022 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 guess-work, 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: $F_R = f_1 \times \text{rpm}_1 = f_2 \times \text{rpm}_2 = f_3 \times \text{rpm}_3 \dots$, 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 \times 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) $\times t_c$, 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:

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

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,

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

Cost of Tooling + Tool Changes per Batch:

Including cutting tools, holders, and tool changing costs,

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

Total Cost of Cutting per Batch:

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

Equivalent Tooling-cost Time, T_V :

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

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, $T_V = T_{RPL} + \frac{60C_E}{H_R} = 1 + \frac{60(6.8)}{50} = 9.16$ 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, $rpm_E = (1000V_E)/(\pi D) = (1000 \times 278)/(3.1416 \times 50) = 1770$ rpm

Economic feed rate, $F_{RE} = f \times rpm_E = 0.25 \times 1770 = 443$ 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 \times 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 Speed and Feed Tables* on page 1022, by using the f_{avg} values converted to mm as follows: feed (mm/rev) = feed (inch/rev) \times 25.4 (mm/inch), thus 0.001 inch/rev = 0.001 \times 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) \times 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 \times (0.001 inch/rev) = 0.017 and 0.009 inch/rev, respectively. Convert the given feeds to mm/rev.

$$\text{feed} = 0.017 \times 25.4 = 17 \times 0.0254 = 0.4318 \text{ mm/rev}$$

$$\text{feed} = 0.008 \times 25.4 = 8 \times 0.0254 = 0.2032 \text{ mm/rev}$$

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 [1027](#), and for ASTM Class 20 grey cast iron using data from [Table 4a](#) on page [1033](#), as follows:

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

$$n = \frac{\ln(V_1/V_2)}{\ln(T_2/T_1)} = \frac{\ln(525/705)}{\ln(15/45)} = 0.27 \quad C = V_1 \times (T_1)^n = 1458$$

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

$$n = \frac{\ln(V_1/V_2)}{\ln(T_2/T_1)} = \frac{\ln(235/320)}{\ln(15/45)} = 0.28 \quad C = V_1 \times (T_1)^n = 685$$

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

$$n = \frac{\ln(V_1/V_2)}{\ln(T_2/T_1)} = \frac{\ln(330/440)}{\ln(15/45)} = 0.26 \quad C = V_1 \times (T_1)^n = 894$$

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

$$n = \frac{\ln(V_1/V_2)}{\ln(T_2/T_1)} = \frac{\ln(125/175)}{\ln(15/45)} = 0.31 \quad C = V_1 \times (T_1)^n = 401$$

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

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

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.

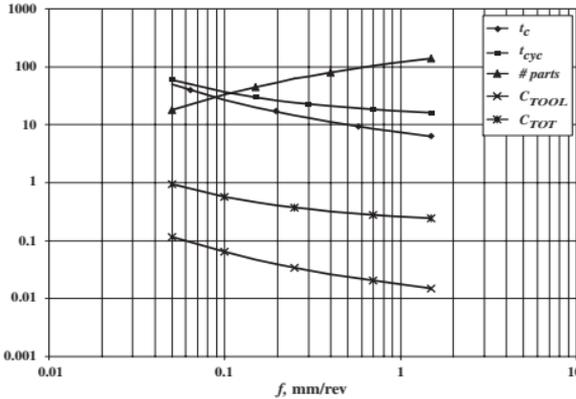


Fig. 22. Cutting time, cycle time, number of parts before tool change, tooling cost, and total cost vs. feed for tool-life = 15 minutes, idle time = 10 s, and batch size = 1000 parts

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 $n = 0.35$ and $C = 280$.

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:

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

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

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

$$T_V = T_{RPL} + 60 \times C_E/H_R = 2.24 + 60 \times 6/50 = 9.44 \text{ minutes}$$

3) Determine economic tool-life T_E

$$T_E = T_V \times (1/n - 1) = 9.44 \times (1/0.35 - 1) = 17.5 \text{ minutes}$$

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

$$V_E = C/T_E^n \text{ m/min} = 280 / 17.5^{0.35} = 103 \text{ m/min}$$

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. [LIVE GRAPH](#)
[Click here to view](#)

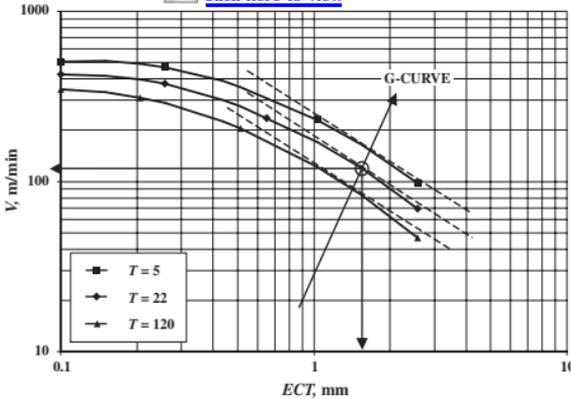


Fig. 23a. Cutting speed vs. ECT, tool-life constant

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.

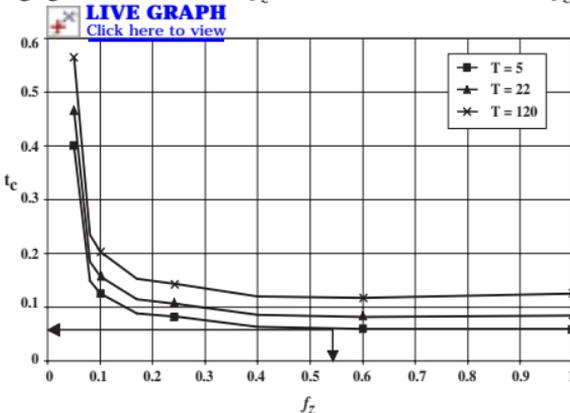


Fig. 23b. Cutting time per part vs. feed per tooth

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.

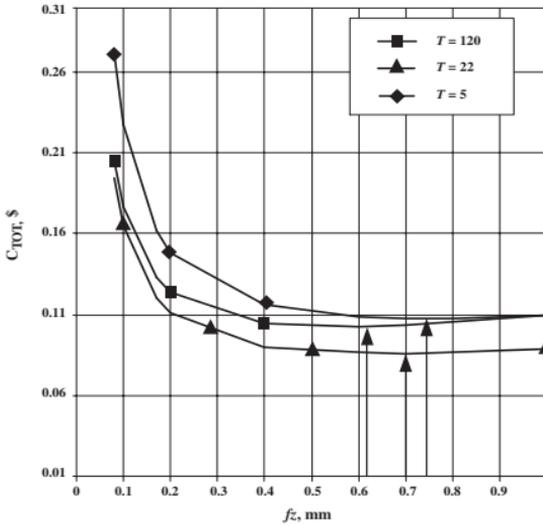


Fig. 23c. Total cost vs. feed/tooth

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.

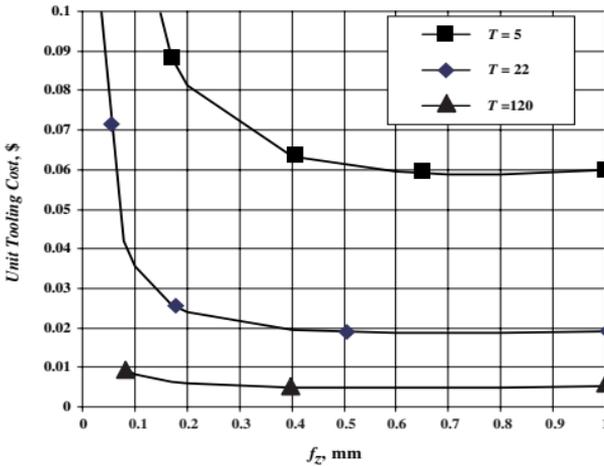


Fig. 23d. Tooling cost versus feed/tooth

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.

MACHINING ECONOMETRICS

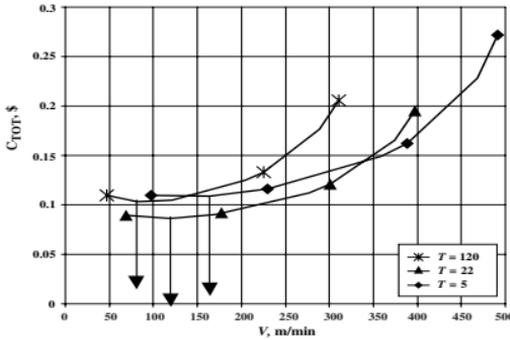


Fig. 23e. Total cost vs. cutting speed at 3 constant tool-lives, feed varies

The total cost curves in Fig. 23e. 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 f_z , and Cutting Speed V , at Tool-lives 5, 22, and 120 minutes

f_z	ECT	$T = 5$ minutes			$T = 22$ minutes			$T = 120$ minutes		
		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 1044.

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 1096).

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 1042, and Fig. 5 on page 1043. 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.

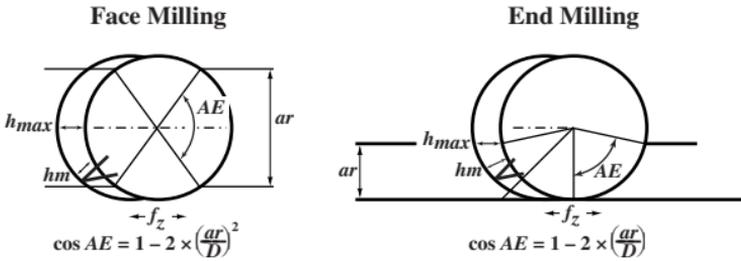


Fig. 24.

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.

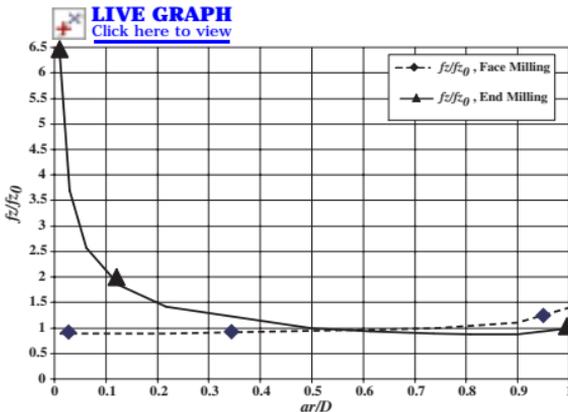


Fig. 25. Feed/tooth versus ar/D for face and end milling

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

ar/D	Face Milling					End Milling (straight)				
	eccentricity $e = 0$ $z = 8$ $f_{z0} = 0.017$ $\cos AE = 1 - 2 \times (ar/D)^2$					$z = 2$ $f_{z0} = 0.017$ $\cos AE = 1 - 2 \times (ar/D)$				
	AE	hmf_z	hm	ECT/hm	f_z/f_{z0}	AE	hmf_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 :

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

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:

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

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

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:

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

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

ar/D	Face Milling				End Milling (straight)			
	hm	f_z/f_{z0}	ECT	ECT_0 corrected for f_z/f_{z0}	hm	f_z/f_{z0}	ECT	ECT_0 corrected for f_z/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 $LA = 0^\circ$ (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:

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

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 $LA = 0^\circ$ (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:

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

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.

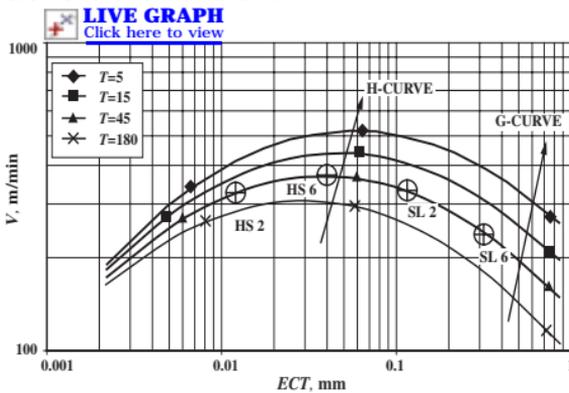


Fig. 26a. Cutting speed vs. ECT , tool-life plotted, for end milling

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:

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

$$\text{rpm} = 1000 \times V/3.1416/D, \text{ where } D \text{ is diameter of cutter.}$$

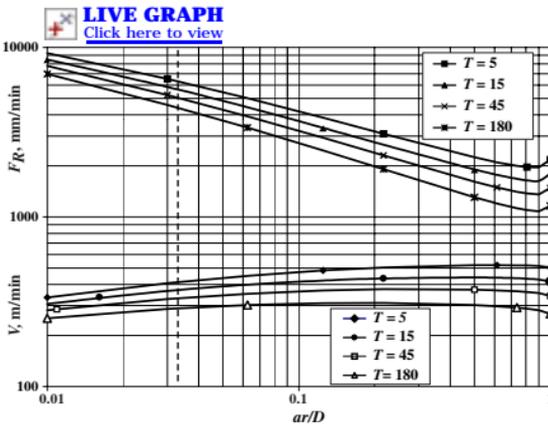


Fig. 26b. High speed feed rate and cutting speed versus ar/D at $T = 5, 15, 45,$ and 180 minutes

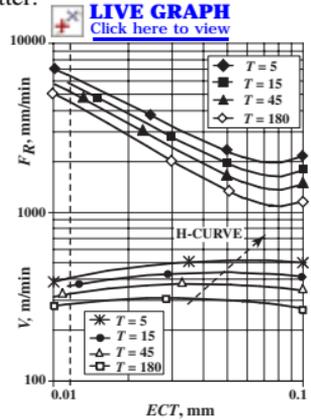


Fig. 26c. High speed feed rate and cutting speed versus $ECT, ar/D$ plotted at $T = 5, 15, 45,$ and 180 minutes

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 $f_z/f_{z=0}$ ratio to maintain a mean chip thickness = 0.108 mm changes F_R in a different proportion than the cutting speed.

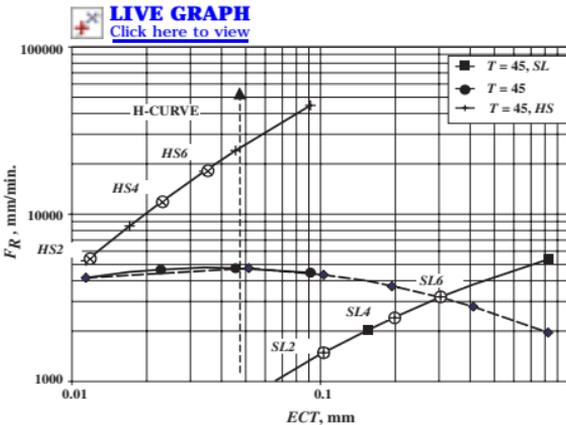


Fig. 26d. Feed rate versus ECT comparison of slot milling ($ar/D = 1$) and high-speed milling (at $ar/D = 0.03$) for 2, 4, and 6 teeth at $T = 45$ minutes

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 (1417 ipm) 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).

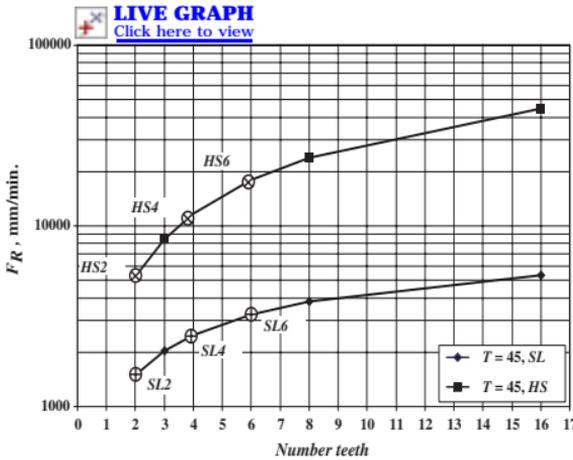


Fig. 26e. Feed rate versus number of teeth comparison of slot milling ($ar/D = 1$) and high-speed milling at ($ar/D = 0.03$) for 2, 4, and 6 teeth at $T = 45$ minutes

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 = $3.4 \times 0.25 = 0.8$ 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 1113). 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.

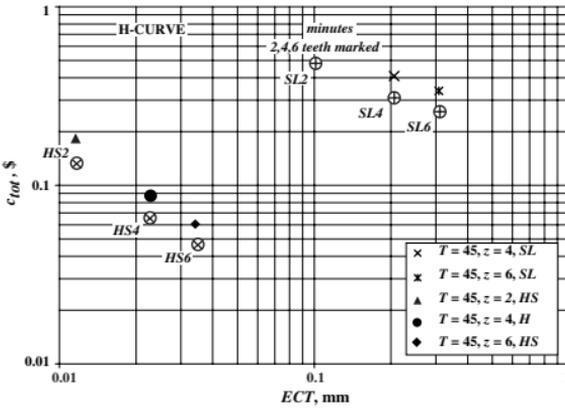


Fig. 27. Cost comparison of slot milling ($ar/D = 1$) and high-speed milling at $ar/D = 0.03$ for 2, 4, and 6 teeth at $T = 45$ minutes

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 $F_R = 5250, z = 2$

$$c_{tot} = 50 \times (1000/5250) \times (1 + 6/45)/60 = \$0.180 \text{ per part}$$

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 1240.

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 1134 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

Tool	Cut		Material to be Machined						
	Width or Depth, Inches	Dia. of Hole, Inches	Brass ^a	Mild or Soft Steel		Tool Steel, 0.80–1.00% C			
			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
Boring tools	0.005	0.008	50	110	0.004	30	60
Box tools, roller rest Single chip finishing	$\frac{1}{32}$...	0.012	0.010	70	150	0.005	40	75
	$\frac{1}{16}$...	0.010	0.008	70	150	0.004	40	75
	$\frac{1}{8}$...	0.008	0.007	70	150	0.003	40	75
	$\frac{3}{16}$...	0.008	0.006	70	150	0.002	40	75
	$\frac{1}{4}$...	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 $\frac{1}{8}$	0.003	0.0015	50	110	0.001	30	75
	...	Over $\frac{1}{8}$	0.006	0.0035	50	110	0.002	30	75
Cutoff tools {	Angular	...	0.0015	0.0006	80	150	0.0004	50	85
	Circular	$\frac{3}{64}$ – $\frac{1}{8}$	0.0035	0.0015	80	150	0.001	50	85
	Straight	$\frac{1}{16}$ – $\frac{1}{8}$	0.0035	0.0015	80	150	0.001	50	85
Stock diameter under $\frac{1}{4}$ 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
	...	$\frac{1}{16}$	0.004	0.002	40	60	0.0012	30	45
	...	$\frac{3}{32}$	0.006	0.0025	40	60	0.0016	30	45
	...	$\frac{1}{8}$	0.009	0.0035	40	75	0.002	30	60
	...	$\frac{3}{16}$	0.012	0.004	40	75	0.003	30	60
	...	$\frac{1}{4}$	0.014	0.005	40	75	0.003	30	60
	...	$\frac{5}{16}$	0.016	0.005	40	75	0.0035	30	60
	...	$\frac{3}{8}$ – $\frac{5}{8}$	0.016	0.006	40	85	0.004	30	60
	Form tools, circular	$\frac{1}{8}$...	0.002	0.0009	80	150	0.0006	50
$\frac{1}{4}$...	0.002	0.0008	80	150	0.0005	50	85
$\frac{3}{8}$...	0.0015	0.0007	80	150	0.0004	50	85
$\frac{1}{2}$...	0.0012	0.0006	80	150	0.0004	50	85
$\frac{5}{8}$...	0.001	0.0005	80	150	0.0003	50	85
$\frac{3}{4}$...	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						
	Width or Depth, Inches	Dia. of Hole, Inches	Brass ^a	Mild or Soft Steel			Tool Steel, 0.80–1.00% C		
			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 $\frac{5}{32}$ in. {	$\frac{1}{32}$	0.012	0.010	70	150	0.008	40	85
		$\frac{1}{16}$	0.010	0.009	70	150	0.006	40	85
		$\frac{1}{8}$	0.017	0.014	70	150	0.010	40	85
	Turned diam. over $\frac{5}{32}$ in. {	$\frac{1}{16}$	0.015	0.012	70	150	0.008	40	85
		$\frac{1}{8}$	0.012	0.010	70	150	0.008	40	85
		$\frac{3}{16}$	0.010	0.008	70	150	0.006	40	85
Knee tools		$\frac{1}{4}$	0.009	0.007	70	150	0.0045	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	$\frac{1}{8}$ or less	0.010 – 0.007	0.008 – 0.006	70	105	0.006 – 0.004	40	60
	0.004 – 0.008	$\frac{1}{8}$ 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 {	...	0.0025	0.002	70	105	0.0015	40	60
		$\frac{1}{16}$ – $\frac{1}{8}$...	0.0008	0.0006	70	105	0.0004	40
Swing tools, forming		$\frac{1}{8}$	0.002	0.0007	70	150	0.0005	40	85
		$\frac{1}{4}$	0.012	0.0005	70	150	0.0003	40	85
		$\frac{3}{8}$	0.001	0.0004	70	150	0.0002	40	85
		$\frac{1}{2}$	0.0008	0.0003	70	150	0.0002	40	85
Turning, straight and taper ^b		$\frac{1}{32}$	0.008	0.006	70	150	0.0035	40	85
		$\frac{1}{16}$	0.006	0.004	70	150	0.003	40	85
		$\frac{1}{8}$	0.005	0.003	70	150	0.002	40	85
		$\frac{3}{16}$	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 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														
$\frac{1}{16}$	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
$\frac{1}{8}$	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	...
$\frac{3}{16}$	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
$\frac{1}{4}$	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
$\frac{5}{16}$	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
$\frac{3}{8}$	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
$\frac{7}{16}$	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
$\frac{1}{2}$	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
$\frac{9}{16}$	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
$\frac{5}{8}$	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
$\frac{11}{16}$	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
$\frac{3}{4}$	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 $15 \div 400 = 0.0375$, 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 $810 \div 413 = 1.96$ 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 1132 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: $W = D^2 \times 2.675 \times F$

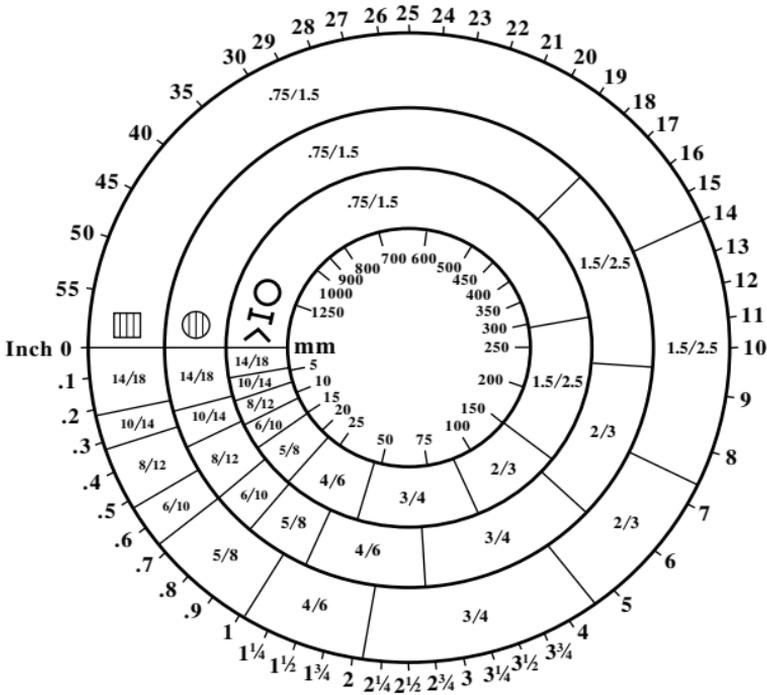
For iron stock: $W = D^2 \times 2.65 \times F$

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 $\frac{1}{4}$ inch and a minimum tooth spacing of about $\frac{1}{6}$ 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 $\pi/4 \times (4^2 - 3^2) = 5.5 \text{ in.}^2$. 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.

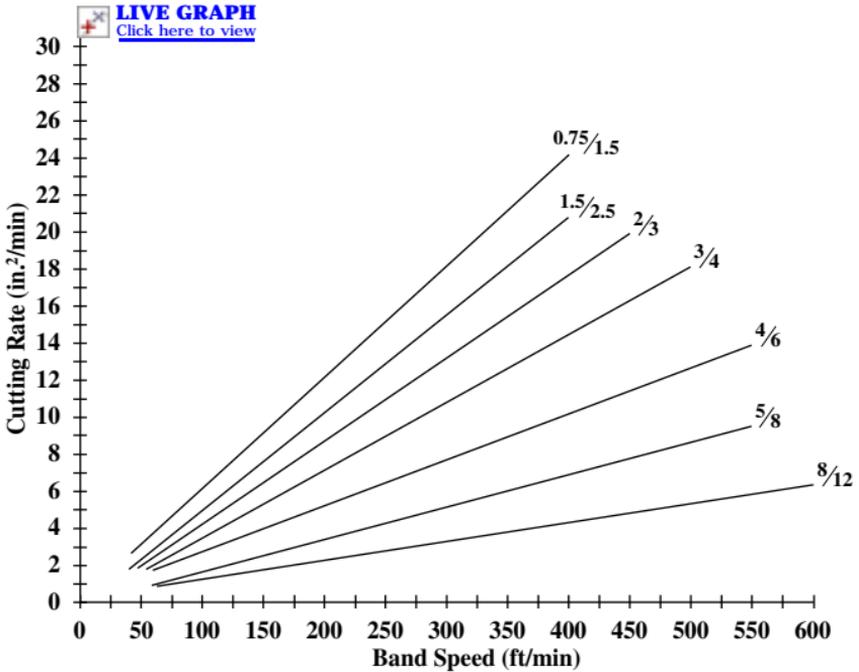
Tooth Forms.—Band saw teeth are characterized by a tooth form that includes the shape, spacing (pitch), rake angle, and gullet capacity of the tooth. Tooth form affects the cutting efficiency, noise level, blade life, chip-carrying capacity, and the surface finish quality of the cut. The rake angle, 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 gulletted (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
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 Ascology	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-4Zr, Ti-8Mo-8V-2Fe-3Al	80
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).

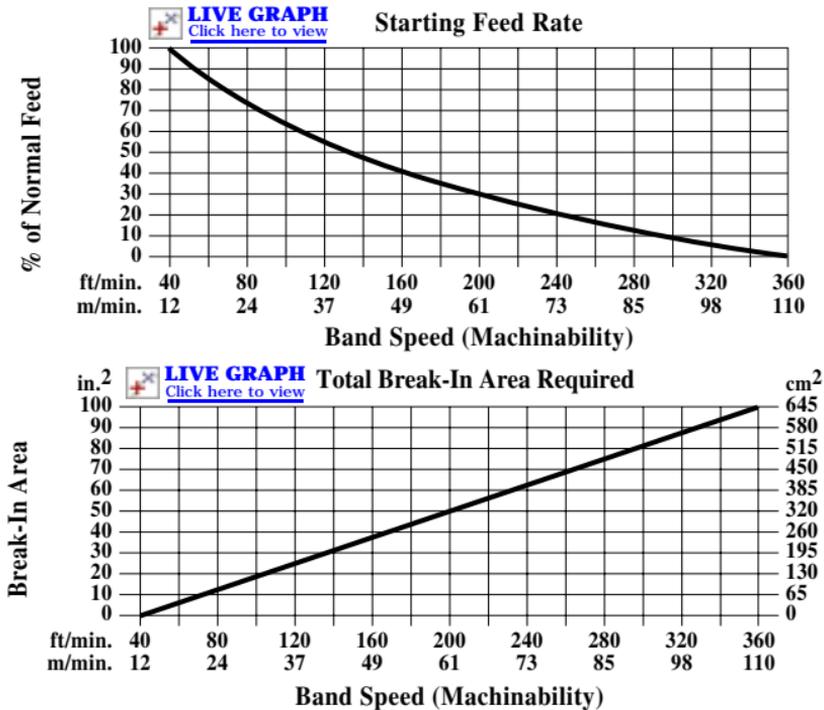
Table data are based on material provided by LENOX Blades, American Saw & Manufacturing Co.

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 cut-

ting 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-car- bon 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 ^c	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-car- bon 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 Min- eral 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 4½ 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.

^dWhen 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.

^eSulfurized 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 Metal working 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 E1687-98, “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 recom-

mended 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 slide-way 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 an air motion sufficient to overcome room air currents and any airflow generated by the process. This airflow cap-

tures 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 a 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 AND NON-METALLIC MATERIALS

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 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 [Table 2](#) on page 556 and [Table 3](#) on page 560. 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 Non-metals

Machining Hard Rubber.—Tools suitable for steel may be used for hard rubber, with no top or side rake angles and 10 to 20 degree 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 degree 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 teeth 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.

Formica Machining.—Blanks can be cut from sheets of "Formica" either by a band saw or by trepanning tools in a boring mill or a drill press. To saw blanks, first describe a circle as a guide line, then use a 21-gage 3½-point saw running at a speed of 5000 feet per minute. The saw should be sharp, with a ¼-inch set on both sides. In drilling, use an ordinary high-speed drill whose point is ground to an included angle of 55 to 60 degrees. Another method is to grind the drill point slightly off center. The feed must be rapid and caution used to prevent the drill from lagging in its work, and the speed must be 1200 revolutions per minute. For all machining operations on "Formica" gear material, provision must be made in grinding for the tools to clear themselves. For reaming, the entry of the reamer and the reaming process must be rapid. There must not be a lag between the end of the reaming operation and the withdrawal of the reamer. In turning the outside diameter and the sides of blanks,

MACHINING MICARTA

1157

the tools must be sharp and have 3 to 5 degrees more rake than is common practice for metal. A cutting speed of 750 feet per minute, which is equal to 720 revolutions per minute on a 4-inch diameter blank, is recommended. The depth of the cut can be $\frac{1}{16}$ to $\frac{1}{8}$ inch, but the feed should be 0.010 inch, regardless of the depth of the cut. Teeth may be cut on a hobbing machine, shaper, or milling machine. The speed of the cutter should be 150 feet per minute, and the feed from 0.023 to 0.040 inch per revolution. It is advisable to back up the blank to prevent fraying or breaking out of the material as the cutter comes through. The backing plates can be economically made from hard wood.

Micarta Machining.—In cutting blanks from sheets of “micarta” a band saw running at a speed of 350 revolutions per minute has been found suitable. The saw should be of the bevel-tooth type, seven teeth to the inch. For large quantities a trepanning tool should be used. In trepanning blanks, the tool should be fed so as to cut part way through all of the “layouts”; then the micarta plate should be turned over, and the cutting completed from the reverse side.

Turning tools should be of high-speed steel cutting at speeds similar to those used for bronze or cast iron. If two cuts are taken, about 0.010 inch of stock should be left for the finishing cut.

Drilling at right angles to the layers is done with a standard drill, which should be backed off sufficiently to provide plenty of clearance. When drilling parallel to layers, a “flat” or “bottom” drill should be used. In rough-drilling, the hole should preferably be drilled partly through the material from each side to prevent possible splitting as the tool protrudes. If this is impracticable, the hole can be drilled all the way through the material, provided the material is “backed up” with wood, stiff cardboard, or any other material that is sufficiently rigid to support the under surface at the point where the drill comes through.

The methods described for drilling apply as well to tapping, except that when the tapping is done parallel to the layers, it is advisable to clamp the material to equalize the stress on the layers and prevent possible splitting.

In milling, a standard tool may be used at a speed and feed corresponding to that used in working bronze or soft steel. The cutting angle of the cutter will give better results if ground with a slight rake.

While there is a wide range of practice as to feeds and speeds in cutting gears on hobbing machines, a hob speed of not less than 140 revolutions per minute, has given satisfaction. In machining gear teeth on a gear shaper, a speed of about 100 to 130 strokes per minute with a fairly fine feed has given good results. Backing-up plates should be used in machining micarta gears.

Ultrasonic Machining.—This method of cutting and engraving hard materials such as glass, precious stones, and carbides uses a transducer (vibratory unit) to obtain the necessary mechanical vibrations needed. The transducer converts the input energy, in this case electrical, into another form of energy, in this case mechanical.

A tool of the required size and shape is made of brass or other soft material and is attached to the transducer. The tool is lowered until it just barely touches the work, and current is applied. At the same time, a slurry of water and fine abrasive, usually boron carbide, is pumped over the work. The tool does not actually touch the work, but the vibrations literally hammer the particles of abrasive into the surface and chip off tiny fragments. Some wear does take place in the tool, but it is very slight and, as it is equally distributed, it does not change the shape. The method is quite commonly applied to cutting designs in the stones of signet rings, but it is also applied to cutting intricately shaped holes in carbide or hardened steel.