

Cutting Tool Applications

Cutting Tool Applications

By George Schneider, Jr. CMfgE



Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

Chapter 1

Cutting-Tool

Materials

1.1 Introduction

Many types of tool materials, ranging from high carbon steel to ceramics and diamonds, are used as cutting tools in today's metalworking industry. It is important to be aware that differences do exist among tool materials, what these differences are, and the correct application for each type of material.

The various tool manufacturers assign many names and numbers to their products. While many of these names and numbers may appear to be similar, the applications of these tool materials may be entirely different. In most cases the tool manufacturers will provide tools made of the proper material for each given application. In some particular applications, a premium or higher priced material will be justified.

This does not mean that the most expensive tool is always the best tool. Cutting tool users cannot afford to ignore the constant changes and advancements that are being made in the field of tool material technology. When a tool change is needed or anticipated, a performance comparison should be made before selecting the tool for the job. The optimum tool is not necessarily the least expensive or the most expensive, and it is not always the same tool that was used for the job last time. The best tool is the one that has been carefully chosen to get the job done quickly, efficiently and economically.

Author's Note

I wish to express my sincere appreciation to Prentice Hall and to Stephen Helba in particular, for giving me permission to use some of the information, graphs and photos recently published in Applied Manufacturing Process Planning authored by Donald H. Nelson and George Schneider, Jr.

The author also wishes to thank over 40 companies who have provided technical information and photo exhibits...their contributions have made this reference text possible.

*And finally, I would like to express my appreciation to **Tooling & Production's** Stan Modic and Joe McKenna for giving me the opportunity to make this information available to the general public.*

George Schneider, Jr.



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: www.ltu.edu

Prentice Hall: www.prenhall.com

A cutting tool must have the following characteristics in order to produce good quality and economical parts:

Hardness: Hardness and strength of the cutting tool must be maintained at elevated temperatures also called Hot Hardness

Toughness: Toughness of cutting tools is needed so that tools don't chip or fracture, especially during interrupted cutting operations.

Wear Resistance: Wear resistance means the attainment of acceptable tool life before tools need to be replaced.

The materials from which cutting

tures, led to the development of high speed tool steels (HSS). The major difference between high speed tool steel and plain high carbon steel is the addition of alloying elements to harden and strengthen the steel and make it more resistant to heat (hot hardness).

Some of the most commonly used alloying elements are: manganese, chromium, tungsten, vanadium, molybdenum, cobalt, and niobium (columbium). While each of these elements will add certain specific desirable characteristics, it can be generally stated that they add deep hardening capability, high hot

(PM) form. The difference between powdered and conventional metals is in the method by which they are made. The majority of conventional high speed steel is poured into an ingot and then, either hot or cold, worked to the desired shape. Powdered metal is exactly as its name indicates. Basically the same elements that are used in conventional high speed steel are prepared in a very fine powdered form. These powdered elements are carefully blended together, pressed into a die under extremely high pressure, and then sintered in an atmospherically controlled furnace. The PM method of manufacturing cutting tools is explained in Section 1.3.1 Manufacture of Carbide Products.

HSS Surface Treatment: Many surface treatments have been developed in an attempt to extend tool life, reduce power consumption, and to control other factors which affect operating conditions and costs. Some of these treatments have been used for many years and have proven to have some value. For example, the black oxide coatings which commonly appear on drills and taps are of value as a deterrent to build-up on the tool. The black oxide is basically a 'dirty' surface which discourages the build-up of work material.

One of the more recent developments in coatings for high speed steel is titanium nitride by the physical vapor deposition (PVD) method. Titanium nitride is deposited on the tool surface in one of several different types of furnace at relatively low temperature, which does not significantly affect the heat treatment (hardness) of the tool being coated. This coating is known to extend the life of a cutting tool significantly or to allow the tool to be used at higher operating speeds. Tool life can be extended by as much as three times, or operating speeds can be increased up to fifty percent.

Cast Alloys: The alloying elements in high speed steel, principally cobalt, chromium and tungsten, improve the cutting properties sufficiently, that metallurgical researchers developed the cast alloys, a family of these materials without iron.

A typical composition for this class of tool material was 45 percent cobalt, 32 percent chromium, 21 percent tungsten, and 2 percent carbon. The purpose of such alloying was to obtain a cutting tool with hot hardness superior to high

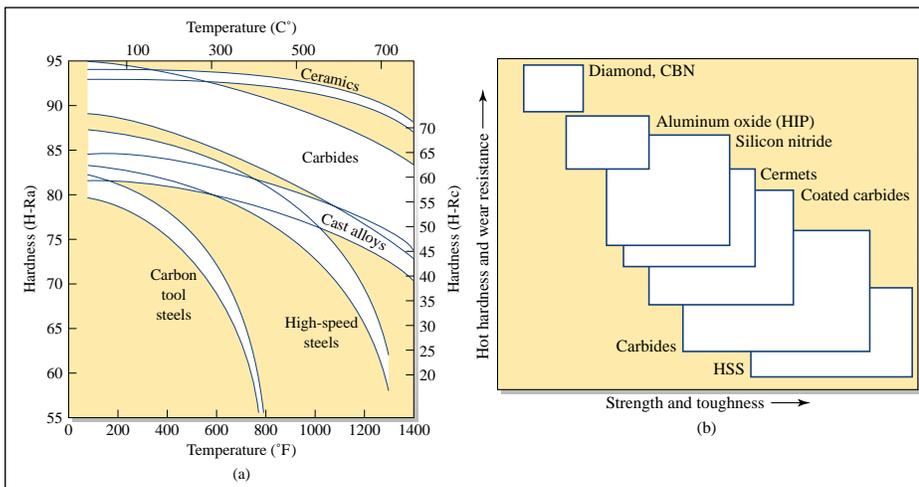


Figure 1.1. (a) Hardness of various cutting-tool materials as a function of temperature. (b) Ranges of properties of various groups of materials.

tools are made are all characteristically hard and strong. There is a wide range of tool materials available for machining operations, and the general classification and use of these materials are of interest here.

1.2 Tool Steels and Cast Alloys

Plain carbon tool steel is the oldest of the tool materials dating back hundreds of years. In simple terms it is a high carbon steel (steel which contains about 1.05% carbon). This high carbon content allows the steel to be hardened, offering greater resistance to abrasive wear. Plain high carbon steel served its purpose well for many years. However, because it is quickly over tempered (softened) at relatively low cutting temperatures, (300 to 500 degrees F), it is now rarely used as cutting tool material except in files, saw blades, chisels, etc. The use of plain high carbon steel is limited to low heat applications.

High Speed Tool Steel: The need for tool materials which could withstand increased cutting speeds and tempera-

tures, resistance to abrasive wear, and strength, to high speed tool steel. These characteristics allow relatively higher machining speeds and improved performance over plain high carbon steel.

The most common high speed steels used primarily as cutting tools are divided into the M and T series. The M series represents tool steels of the molybdenum type and the T series represents those of the tungsten type. Although there seems to be a great deal of similarity among these high speed steels, each one serves a specific purpose and offers significant benefits in its special application.

An important point to remember is that none of the alloying elements for either series of high speed tool steels is in abundant supply and the cost of these elements is skyrocketing. In addition, U.S. manufacturers must rely on foreign countries for supply of these very important elements.

Some of the high speed steels are now available in a powdered metal

speed steel.

When applying cast alloy tools, their brittleness should be kept in mind and sufficient support should be provided at all times. Cast alloys provide high abrasion resistance and are thus useful for cutting scaly materials or those with hard inclusions.

1.3 Cemented Tungsten Carbide

Tungsten carbide was discovered by Henri Moissan in 1893 during a search for a method of making artificial diamonds. Charging sugar and tungsten oxide, he melted tungsten sub-carbide in an arc furnace. The carbonized sugar reduced the oxide and carburized the tungsten. Moissan recorded that the tungsten carbide was extremely hard, approaching the hardness of diamond and exceeding that of sapphire. It was more than 16 times as heavy as water. The material proved to be extremely brittle and seriously limited its industrial use.

Commercial tungsten carbide with 6 percent cobalt binder was first produced and marketed in Germany in 1926. Production of the same carbide began in the United States in 1928 and in Canada in 1930.

At this time, hard carbides consisted of the basic tungsten carbide system with cobalt binders. These carbides exhibited superior performance in the machining of cast iron, nonferrous, and non metallic materials, but were disappointing when used for the machining of steel.

Most of the subsequent developments in the hard carbides have been modifi-

cations of the original patents, principally involving replacement of part or all of the tungsten carbide with other carbides, especially titanium carbide and/or tantalum carbide. This led to the development of the modern multi-carbide cutting tool materials permitting the high speed machining of steel.

A new phenomenon was introduced with the development of the cemented carbides, again making higher speeds possible. Previous cutting tool materials, products of molten metallurgy, depended largely upon heat treatment for their properties and these properties could, in turn, be destroyed by further heat treatment. At high speeds, and consequently high temperatures, these products of molten metallurgy failed.

A different set of conditions exists with the cemented carbides. The hardness of the carbide is greater than that of most other tool materials at room temperature and it has the ability to retain it hardness at elevated temperatures to a greater degree, so that greater speeds can be adequately supported.

1.3.1 Manufacture of Carbide Products

The term "tungsten carbide" describes a comprehensive family of hard carbide compositions used for metal cutting tools, dies of various types, and wear parts. In general, these materials are composed of the carbides of tungsten, titanium, tantalum or some combination of these, sintered or cemented in a matrix binder, usually cobalt.

Blending: The first operation after reduction of the tungsten compounds to tungsten metal powder is the milling of tungsten and carbon prior to the carburizing operation. Here, 94 parts by weight of tungsten and 6 parts by weight of carbon, usually added in the form of lamp black, are blended together in a rotating mixer or ball mill. This operation must be performed under carefully controlled conditions in order to insure optimum dispersion of the carbon in the tungsten. Carbide Blending Equipment, better known as a Ball Mill, is shown in Figure 1.2.

In order to provide the necessary strength, a binding agent, usually cobalt (Co) is added to the tungsten (WC) in powder form and these two are ball milled together for a period of several days, to form a very intimate mixture. Careful control of conditions, including



Figure 1.3. Blended tungsten carbide powder is produced by mixing tungsten carbide (WC) with a cobalt (Co) binder in a ball milling process. (Courtesy American National Carbide Co)

time, must be exercised to obtain a uniform, homogeneous product. Blended Tungsten Carbide Powder is shown in Figure 1.3.

Compacting: The most common compacting method for grade powders involves the use of a die, made to the shape of the eventual product desired. The size of the die must be greater than the final product size to allow for dimensional shrinkage which takes place in the final sintering operation. These dies are expensive, and usually made with tungsten carbide liners. Therefore sufficient number of the final product (compacts) are required, to justify the expense involved in manufacturing a special die. Carbide



Figure 1.2. Carbide blending equipment, better known as ball mill is used to ensure optimum dispersion of the carbon within the tungsten. (Courtesy American National Carbide Co)

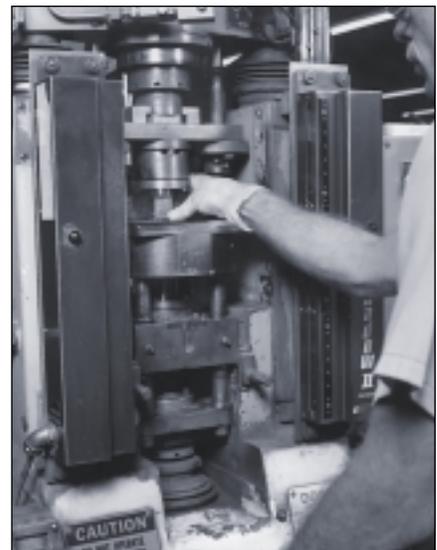


Figure 1.4. Carbide compacting equipment, better known as a pill press, is used to produce carbide products in various shapes. (Courtesy American National Carbide Co)



Figure 1.5. Various carbide compacts, which are produced with special dies mounted into pill presses. (Courtesy American National Carbide Co)

Compacting Equipment, better known as a Pill Press, is shown in Figure 1.4. Various pill pressed carbide parts are shown in Figure 1.5.

If the quantities are not high, a larger briquette, or billet may be pressed. This billet may then be cut up (usually after pre-sintering) into smaller units and shaped or preformed to the required configuration, and again, allowance must be made to provide for shrinkage. Ordinarily pressures used in these cold compacting operations are in the neighborhood of 30,000 PSI. Various carbide preformed parts are shown in Figure 1.6.

A second compacting method is the hot pressing of grade powders in graphite dies at the sintering temperature. After cooling, the part has attained

full hardness. Because the graphite dies are expendable, this system is generally used only when the part to be produced is too large for cold pressing and sintering.

A third compacting method, usually used for large pieces, is isostatic pressing. Powders are placed into a closed, flexible container which is then suspended in a liquid in a closed pressure vessel. Pressure in the liquid is built up to the point where the powders become properly compacted. This system is advantageous for pressing large pieces, because the pressure acting on the powders operates equally from all directions, resulting in a compact of uniform pressed density.

Sintering: Sintering of tungsten-cobalt (WC-Co) compacts is performed with the cobalt binder in liquid phase. The compact is heated in hydrogen atmosphere or vacuum furnaces to temperatures ranging from 2500 to 2900 degrees Fahrenheit, depending on the composition. Both time and temperature must be carefully adjusted in combination, to effect optimum control over properties and geometry. The compact will shrink approximately 16 percent on linear dimensions, or 40 percent in volume. The exact amount of shrinkage depends on several factors including particle size of the powders and the composition of the grade. Control of size and shape is most important and is least predictable during the cooling cycle. This is particularly true with

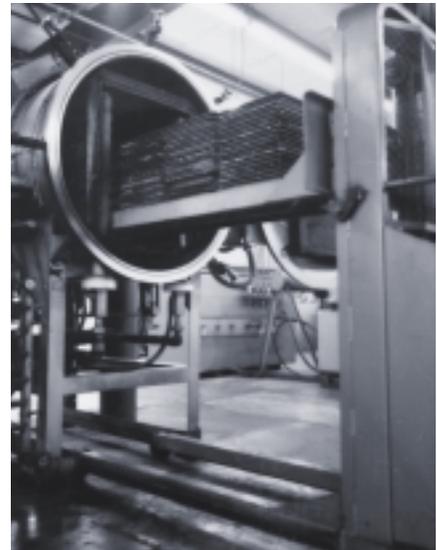


Figure 1.7. Carbide parts are loaded into a sintering furnace, where they are heated to temperatures ranging from 2500° to 2900°F. (Courtesy American National Carbide Co)

those grades of cemented carbides with higher cobalt contents.

With cobalt having a lesser density than tungsten, it occupies a greater part of the volume than would be indicated by the rated cobalt content of the grade; and because cobalt contents are generally a much higher percentage of the mass in liquid phase, extreme care is required to control and predict with accuracy the magnitude and direction of shrinkage. Figure 1.7 shows carbide parts being loaded into a Sintering Furnace.

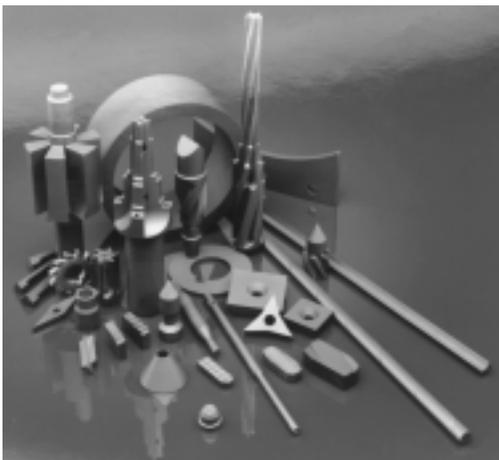


Figure 1.6. If quantities are not high, pre-sintered billets are shaped or preformed into required shapes. (Courtesy Duramet Corporation)

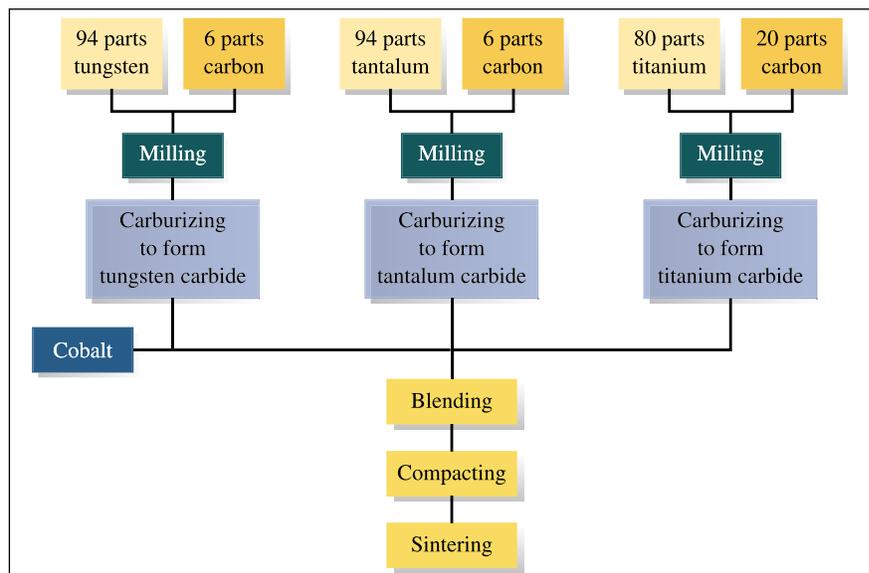


Figure 1.8. Schematic diagram of the cemented tungsten carbide manufacturing process.

A more detailed schematic diagram of the cemented tungsten carbide manufacturing process is shown in Figure 1.8.

1.3.2 Classification of Carbide Tools

Cemented carbide products are classified into three major grades:

Wear Grades: Used primarily in dies, machine and tool guides, and in such everyday items as the line guides on fishing rods and reels; anywhere good wear resistance is required.

Impact Grades: Also used for dies, particularly for stamping and forming, and in tools such as mining drill heads.

Cutting Tool Grades: The cutting tool grades of cemented carbides are divided into two groups depending on their primary application. If the carbide is intended for use on cast iron which is a nonductile material, it is graded as a cast iron carbide. If it is to be used to cut steel, a ductile material, it is graded as a steel grade carbide.

Cast iron carbides must be more resistant to abrasive wear. Steel carbides require more resistance to cratering and heat. The tool wear characteristics of various metals are different, thereby requiring different tool properties. The high abrasiveness of cast iron causes mainly edge wear to the tool. The long chip of steel, which flows across the tool at normally higher cutting speeds, causes mainly cratering and heat deformation to the tool. Tool wear characteristics and chip formation will be discussed in Chapter 2.

It is important to choose and use the correct carbide grade for each job application. There are several factors that make one carbide grade different from another and therefore more suitable for a specific application. The carbide grades may appear to be similar, but the difference between the right and wrong carbide for the job, can mean the difference between success and failure.

Figure 1.8 illustrates how carbide is manufactured, using pure tungsten carbide with a cobalt binder. The pure tungsten carbide makes up the basic carbide tool and is often used as such, particularly when machining cast iron. This is because pure tungsten carbide is extremely hard and offers the best resistance to abrasive wear.

Large amounts of tungsten carbide are present in all of the grades in the two cutting groups and cobalt is always used as the binder. The more common alloy-

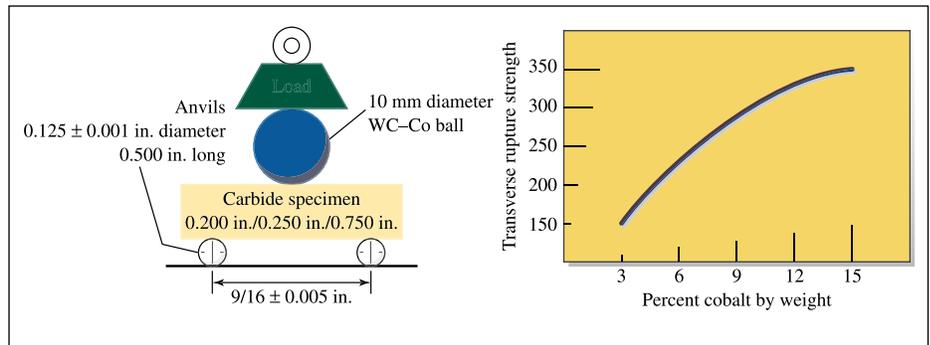


Figure 1.9. The method used to measure Transverse Rupture Strength (TRS) is shown as well as the relationship of TRS to cobalt (Co) content.

ing additions to the basic tungsten/cobalt material are: tantalum carbide, and titanium carbide.

While some of these alloys may be present in cast iron grades of cutting tools, they are primarily added to steel grades. Pure tungsten carbide is the most abrasive-resistant and will work most effectively with the abrasive nature of cast iron. The addition of the alloying materials such as tantalum carbide and titanium carbide offers many benefits:

- The most significant benefit of titanium carbide is that it reduces cratering of the tool by reducing the tendency of the long steel chips to erode the surface of the tool.
- The most significant contribution of tantalum carbide is that it increases the hot hardness of the tool which, in turn, reduces thermal deformation.

Varying the amount of cobalt binder in the tool material largely affects both the cast iron and steel grades in three ways. Cobalt is far more sensitive to heat than the carbide around it. Cobalt is also more sensitive to abrasion and chip welding. Therefore, the more cobalt present, the softer the tool is, making it more sensitive to heat deformation, abrasive wear, and chip welding

and leaching which causes cratering. On the other hand, cobalt is stronger than carbide. Therefore more cobalt improves the tool strength and resistance to shock. The strength of a carbide tool is expressed in terms of 'Transverse Rupture Strength' (TRS). Figure 1.9 shows how Transverse Rupture Strength is measured.

The third difference between the cast iron and steel grade cutting tools, is carbide grain size. The carbide grain size is controlled by the ball mill process. There are some exceptions, such as micro-grain carbides, but generally the smaller the carbide grains, the harder the tool. Conversely, the larger the carbide grain, the stronger the tool. Carbide grain sizes at 1500x magnification are shown in Exhibits 1.10 and 1.11.

In the C- classification method (Figure 1.12), grades C-1 through C-4 are for cast iron and grades C-5 through C-8 for steel. The higher the C- number in each group, the harder the grade, the lower the C- number, the stronger the grade. The harder grades are used for finish cut applications; the stronger grades are used for rough cut applications.

Many manufacturers produce and

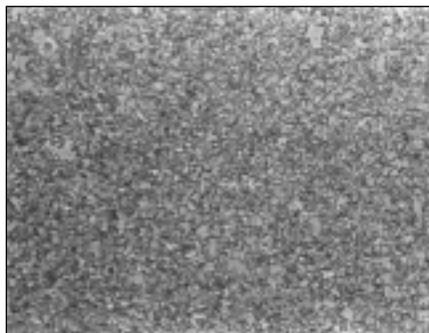


Figure 1.10. Carbide grain size (0.8 micron WC @ 1500x) consisting of 90% WC and 10% Co.

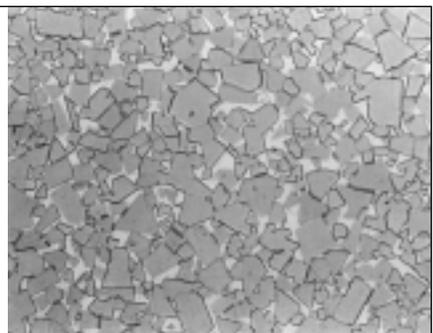


Figure 1.11. Carbide grain size (7 microns WC @ 1500x) consisting of 90% WC and 10% Co.

Classification Number	Materials to be Machined	Machining Operation	Type of Carbide	Characteristics Of		Typical Properties	
				Cut	Carbide	Hardness H-Ra	Transverse Rupture Strength (MPa)
C-1	Cast iron, nonferrous metals, and nonmetallic materials requiring abrasion resistance	Roughing cuts	Wear-resistant grades; generally straight WC-Co with varying grain sizes	Increasing cutting speed	Increasing hardness and wear resistance	89.0	2,400
C-2		General purpose		↓ ↑	↓ ↑	92.0	1,725
C-3		Finishing		↓ ↑	↓ ↑	92.5	1,400
C-4		Precision boring and fine finishing		Increasing feed rate	Increasing strength and binder content	93.5	1,200
C-5	Steels and steel-alloys requiring crater and deformation resistance	Roughing cuts	Crater-resistant grades; various WWC-Co compositions with TiC and/or TaC alloys	Increasing cutting speed	Increasing hardness and wear resistance	91.0	2,070
C-6		General purpose		↓ ↑	↓ ↑	92.0	1,725
C-7		Finishing		↓ ↑	↓ ↑	93.0	1,380
C-8		Precision boring and fine finishing		Increasing feed rate	Increasing strength and binder content	94.0	1,035

Figure 1.12. Classification, application, characteristics, and typical properties of metal-cutting carbide grades.

distribute charts showing a comparison of their carbide grades with those of other manufacturers. These are not equivalency charts, even though they may imply that one manufacturer's carbide is equivalent to that of another manufacturer. Each manufacturer knows his carbide best and only the manufacturer of that specific carbide can accurately place that carbide on the C-chart. Many manufacturers, especially those outside the U. S., do not use the C-classification system for carbides. The placement of these carbides on a C-chart by a competing company is based upon similarity of application and is, at best an 'educated guess'. Tests have shown a marked difference in performance among carbide grades that manufacturers using the C-classification system have listed in the same category.

1.3.3 Coated Carbide Tools

While coated carbides have been in existence since the late 1960's, they did not reach their full potential until the mid 1970's. The first coated carbides were nothing more than standard carbide grades which were subjected to a coating process. As the manufacturers gained experience in producing coated carbides, they began to realize that the coating was only as good as the base carbide under the coating (known as the substrate).

It is advisable to consider coated carbides for most applications. When the

proper coated carbide, with the right edge preparation is used in the right application, it will generally outperform any uncoated grade. The microstructure of a coated carbide insert at 1500x magnification is shown in Figure 1.13.

Numerous types of coating materials are used, each for a specific application. It is important to observe the do's and don't's in the application of coated carbides. The most common coating materials are:

- Titanium Carbide
- Titanium Nitride
- Ceramic Coating
- Diamond Coating
- Titanium Carbo-Nitride

In addition, multi-layered combinations of these coating materials are

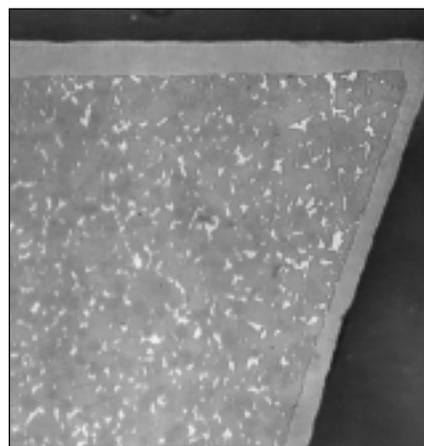


Figure 1.13. Microstructure of a coated carbide insert at 1500x magnification. (Courtesy of Kennametal Inc.)

used. The microstructure of a multi-layered coated carbide insert at 1500x magnification is shown in Figure 1.14.

In general the coating process is accomplished by chemical vapor deposition (CVD). The substrate is placed in an environmentally controlled chamber having an elevated temperature. The coating material is then introduced into the chamber as a chemical vapor. The coating material is drawn to and deposited on the substrate by a magnetic field around the substrate. It takes many hours in the chamber to achieve a coating of 0.0002 to 0.0003 inch on the substrate. Another process is Physical Vapor Deposition (PVD).

Titanium Carbide Coating: Of all the coatings, titanium carbide is the most widely used. Titanium carbide is used on many different substrate materials for cutting various materials under varying conditions. Titanium carbide coatings allow the use of higher cutting speeds because of their greater resistance to abrasive wear and cratering and higher heat resistance.

Titanium Nitride Coating - Gold Color: Titanium nitride is used on many different substrate materials. The primary advantage of titanium nitride is its resistance to cratering. Titanium nitride also offers some increased abrasive wear resistance and a significant increase in heat resistance permitting higher cutting speeds. It is also said that titanium nitride is more slippery, allowing chips to pass over it, at the cutting interface, with less friction.

Ceramic Coating - Black Color: Because aluminum oxide (ceramic) is extremely hard and brittle, it is not opti-



Figure 1.14. Microstructure of a multi-layered coated carbide insert at 1500x magnification. (Courtesy of Kennametal Inc.)

mal for interrupted cuts, scaly cuts, and hard spots in the workpiece. This is not to say that it will never work under these conditions, but it may be more subject to failure by chipping. Even with these limitations, aluminum oxide is probably the greatest contributor to the coated carbides. Aluminum oxide ceramic allows much higher cutting speeds than other coated carbides because of its outstanding resistance to abrasive wear and its resistance to heat and chemical interaction.

Diamond Coating: A recent development concerns the use of diamond polycrystalline as coating for tungsten carbide cutting tools. Problems exist regarding adherence of the diamond film to the substrate and the difference in thermal expansion between diamond and substrate materials. Thin-film diamond coated inserts are now available using either PVD (Physical Vapor Deposition) or CVD (Chemical Vapor Deposition) coating methods. Diamond coated tools are effective in machining abrasive materials, such as aluminum alloys containing silicon, fiber reinforced materials, and graphite. Improvements in tool life of as much as tenfold have been obtained over other coated tools.

Titanium Carbo-Nitride - Black Color Multilayered Coatings: Titanium carbo-nitride normally appears as the intermediate layer of two or three phase coatings. The role of titanium carbo-nitride is one of neutrality, helping the other coating layers to bond into a sandwich-like structure (Figure 1-14). Other multi-layer coating combinations are being developed to effectively machine stainless steels and aerospace alloys. Chromium-based coatings such as chromium carbide have been found to be effective in machining softer metals such as aluminum, copper, and titanium.

There are a few important points to remember about using coated carbides. Coated carbides will not always out-perform uncoated grades but because of the benefits offered by coated carbides, they should always be a first consideration when selecting cutting tools.

When comparing the cost between coated and uncoated

carbides there will be little difference when the benefits of coated carbides are considered. Because coated carbides are more resistant to abrasive wear, cratering, and heat, and because they are more resistant to work material build-up at lower cutting speeds, tool life is extended, reducing tool replacement costs. Coated carbides permit operation at higher speeds, reducing production costs.

All coated carbides have an edge hone to prevent coating build-up during the coating process. This is because the coating will generally seek sharp edges. The edge hone is usually very slight and actually extends tool life. However, a coated insert should never be reground or honed. If a special edge preparation is required the coated carbides must be ordered that way. The only time the edge hone may be of any disadvantage is when making a very light finishing cut. Carbide insert edge preparations will be discussed in Chapter 2.

1.4 Ceramic and Cermet Tools

Ceramic Aluminum Oxide (Al_2O_3) material for cutting tools was first developed in Germany sometime around 1940. While ceramics were slow to develop as tool materials, advancements made since the mid 1970's have greatly improved their usefulness. Cermets are basically a combination of ceramic and titanium carbide. The word cermet is derived from the words 'ceramic' and 'metal'.

Ceramic Cutting Tools: Ceramics are non-metallic materials. This puts them in an entirely different category than HSS and carbide tool materials. The use of ceramics as cutting tool

material has distinct advantages and disadvantages. The application of ceramic cutting tools is limited because of their extreme brittleness. The transverse rupture strength (TRS) is very low. This means that they will fracture more easily when making heavy or interrupted cuts. However, the strength of ceramics under compression is much higher than HSS and carbide tools.

There are two basic types of ceramic material; hot pressed and cold pressed. In hot pressed ceramics, usually black or gray in color, the aluminum oxide grains are pressed together under extremely high pressure and at a very high temperature to form a billet. The billet is then cut to insert size. With cold pressed ceramics, usually white in color, the aluminum oxide grains are pressed together, again under extremely high pressure but at a lower temperature. The billets are then sintered to achieve bonding. This procedure is similar to carbide manufacture, except no metallic binder material is used. While both hot and cold pressed ceramics are similar in hardness, the cold pressed ceramic is slightly harder. The hot pressed ceramic has greater transverse rupture strength. Various shapes of both hot and cold pressed ceramic inserts are shown in Figure 1.15.

The brittleness, or relative strength, of ceramic materials is their greatest disadvantage when they are compared to HSS or carbide tools. Proper tool geometry and edge preparation play an important role in the application of ceramic tools and help to overcome their weakness. Some of the advantages of ceramic tools are:

- High strength for light cuts on very hard work materials.
- Extremely high resistance to abrasive wear and cratering.
- Capability of running at speeds in excess of 2000 SFPM.
- Extremely high hot hardness.
- Low thermal conductivity.

While ceramics may not be the all-around tool for the average shop, they can be useful in certain applications. Ceramic tools have been alloyed with zirconium (about 15%) to increase their strength.

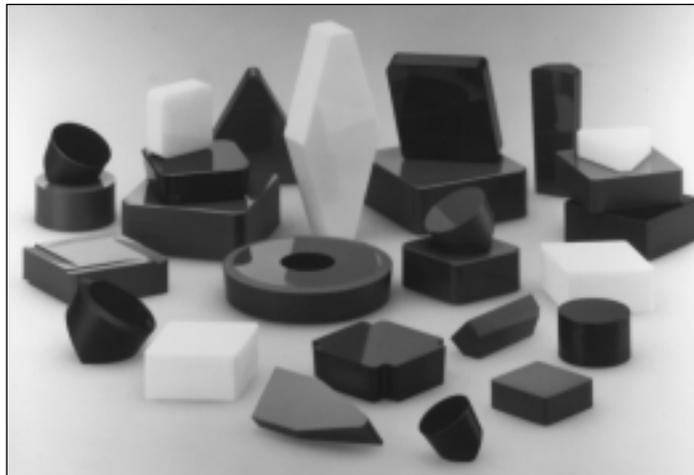


Figure 1.15. Various sizes and shapes of hot- and cold-pressed ceramic inserts. (Courtesy Greenleaf Corp.)

Many ceramic tool manufacturers are recommending the use of ceramic tools for both rough cutting and finishing operations. Practical shop experience indicates that these recommendations are somewhat optimistic. To use ceramic tools successfully, insert shape, work material condition, machine tool capability, set-up, and general machining conditions must all be correct. High rigidity of the machine tool and set-up is also important for the application of ceramic tools. Ceramics are being developed to have greater strength (higher TRS). Some manufacturers are offering ceramic inserts with positive geometry and even formed chip breaker grooves.

Cermet Cutting Tools: The manufacturing process for cermets is similar to the process used for hot pressed ceramics. The materials, approximately 70 percent ceramic and 30 percent titanium carbide, are pressed into billets under extremely high pressure and temperature. After sintering, the billets are sliced to the desired tool shapes. Subsequent grinding operations for final size and edge preparation, complete the manufacturing process.

The strength of cermets is greater than that of hot pressed ceramics. Therefore, cermets perform better on interrupted cuts. However, when compared to solid ceramics, the presence of the 30 percent titanium carbide in cermets decreases the hot hardness and resistance to abrasive wear. The hot hardness and resistance to abrasive wear of cermets are high when compared to HSS and carbide tools. The greater strength of cermets allows them to be available in a significantly larger selection of geometries, and to be used in standard insert holders for a greater variety of applications. The geometries include many positive/negative, and chip breaker configurations.

Silicon-Nitride Base Ceramics: Developed in the 1970's, silicon-nitride (SiN) base ceramic tool materials consist of silicon nitride with various addi-

tions of aluminum oxide, yttrium oxide, and titanium carbide. These tools have high toughness, hot hardness and good thermal shock resistance. Sialon for example is recommended for machining cast irons and nickel base superalloys at intermediate cutting speeds.

1.5 Diamond, CBN and Whisker-Reinforced Tools

The materials described here are not commonly found in a heavy metal working environment. They are most commonly used in high speed automatic production systems for light finishing of precision surfaces. To complete the inventory of tool materials, it is important to note the characteristics and general applications of these specialty materials.

Diamond: The two types of diamonds being used as cutting tools are industrial grade natural diamonds, and synthetic polycrystalline diamonds. Because diamonds are pure carbon, they have an affinity for the carbon of ferrous metals. Therefore, they can only be used on non-ferrous metals.

Some diamond cutting tools are made of a diamond crystal compaction (many small crystals pressed together) bonded to a carbide base (Fig. 1.16). These diamond cutting tools should only be used for light finishing cuts of precision surfaces. Feeds should be very light and speeds are usually in excess of 5000 surface feet per minute (SFPM). Rigidity in the machine tool and the set-up is very critical because of the extreme hardness and brittleness of diamond.

Cubic Boron Nitride: Cubic boron nitride (CBN) is similar to diamond in its polycrystalline structure and is also bonded to a carbide base. With the

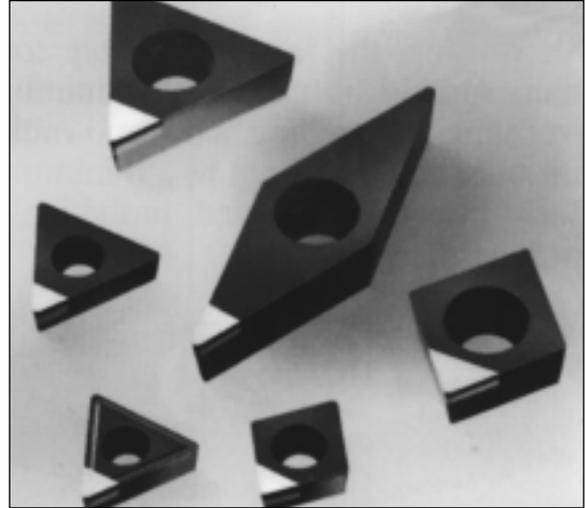


Figure 1.16. Polycrystalline diamond material bonded to a carbide base of various sizes and shapes. (Courtesy of Sandvik Coromant Co.)

exception of titanium, or titanium alloyed materials, CBN will work effectively as a cutting tool on most common work materials. However, the use of CBN should be reserved for very hard and difficult-to-machine materials. CBN will run at lower speeds, around 600 SFPM, and will take heavier cuts with higher lead angles than diamond. Still, CBN should mainly be considered as a finishing tool material because of its extreme hardness and brittleness. Machine tool and set-up rigidity for CBN as with diamond, is critical.

Whisker-Reinforced Materials: In order to further improve the performance and wear resistance of cutting tools to machine new work materials and composites, whisker-reinforced composite cutting tool materials have been developed. Whisker-reinforced materials include silicon-nitride base tools and aluminum-oxide base tools, reinforced with silicon-carbide (SiC) whiskers. Such tools are effective in machining composites and nonferrous materials, but are not suitable for machining irons and steels.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>
Prentice Hall: <http://www.prenhall.com>

Chapter 2

Metal Removal

Methods

2.1 Introduction

The process of metal removal, a process in which a wedge-shaped tool engages a workpiece to remove a layer of material in the form of a chip, goes back many years. Even with all of the sophisticated equipment and techniques used in today's modern industry, the basic mechanics of forming a chip remain the same. As the cutting tool engages the workpiece, the material directly ahead of the tool is sheared and deformed under tremendous pressure. The deformed material then seeks to relieve its stressed condition by fracturing and flowing into the space above the tool in the form of a chip. A turning tool holder generating a chip is shown in Figure 2.1.

2.2 Cutting Tool Forces

The deformation of a work material means that enough force has been exerted by the tool to permanently reshape or fracture the work material. If a material is reshaped, it is said to have exceeded its plastic limit. A chip is a combination of reshaping and fracturing. The deformed chip is separated from the parent material by fracture. The cutting action and the chip formation can be more easily analyzed if the edge of the tool is set perpendicular to the relative motion of the material, as shown in Figure 2.2. Here the undeformed chip thickness t_1 is the value of the depth of cut, while t_2 is the thickness of the deformed chip after leaving the workpiece. The major deformation starts at the shear zone and diameter determines the angle of shear.

A general discussion of the forces acting in metal cutting is presented by using the example of a typical turning operation. When a solid bar is turned, there are three



FIGURE 2.1 A turning toolholder insert generating a chip. (Courtesy Kennametal Inc.)

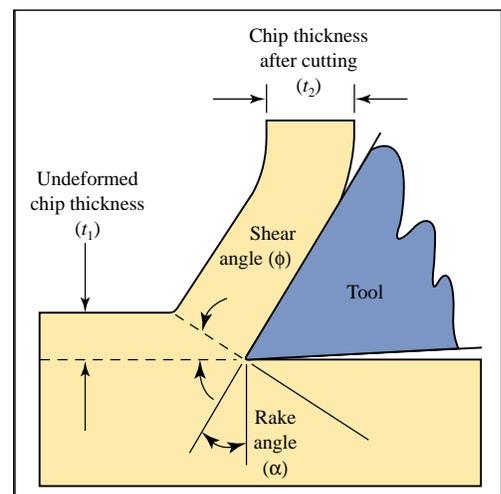


FIGURE 2.2 Chip formation showing the deformation of the material being machined.

forces acting on the cutting tool (Fig. 2.3):

Tangential Force: This acts in a direction tangential to the revolving workpiece and represents the resistance to the rotation of the workpiece. In a normal operation, tangential force is the highest of the three forces and accounts for about 98 percent of the total power required by the operation.

Longitudinal Force: Longitudinal force acts in the direction parallel to the axis of the work and represents the resistance to the longitudinal feed of the tool. Longitudinal force is usually about 50 percent as great as tangential force. Since feed velocity is usually very low in relation to the velocity of the rotating workpiece, longitudinal force accounts for only about 1 percent of total power required.

Radial Force: Radial force acts in a radial direction from the center line of the workpiece. The radial force is generally the smallest of the three, often about 50 percent as large as longitudinal force. Its effect on power requirements is very small because velocity in the radial direction is negligible.

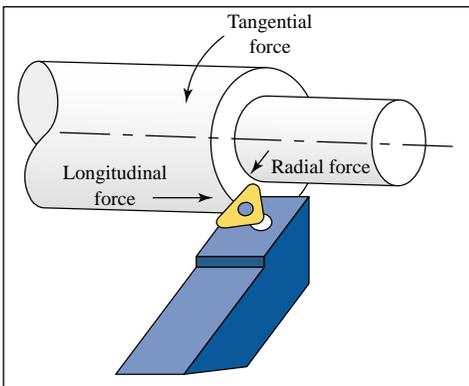


FIGURE 2.3 Typical turning operation showing the forces acting on the cutting tool.

2.3 Chip Formation and Tool Wear

Regardless of the tool being used or the metal being cut, the chip forming process occurs by a mechanism called plastic deformation. This deformation can be visualized as shearing. That is when a metal is subjected to a load exceeding its elastic limit. The crystals of the metal elongate through an action of slipping or shearing, which takes place within the crystals and between adjacent crystals. This action, shown in Figure 2.4 is similar to the action that takes place when a deck of cards is

given a push and sliding or shearing occurs between the individual cards.

Metals are composed of many crystals and each crystal in turn is composed of atoms arranged into some definite pattern. Without getting into a complicated discussion on the atomic makeup and characteristics of metals, it should be noted, that the slipping of the crystals takes place along a plane of greatest ionic density.

Most practical cutting operations, such as turning and milling, involve two or more cutting edges inclined at various angles to the direction of the cut. However, the basic mechanism of cutting can be explained by analyzing cutting done with a single cutting edge.

Chip formation is simplest when a continuous chip is formed in orthogonal cutting (Fig. 2.5a). Here the cutting edge of the tool is perpendicular to the line of tool travel, tangential, longitudinal, and radial forces are in the same plane, and only a single, straight cutting edge is active. In oblique cutting, (Fig. 2.5b), a single, straight cutting edge is inclined in the direction of tool travel. This inclination causes changes in the direction of chip flow up the face of the tool. When the cutting edge is inclined, the chip flows across the tool face with a sideways movement that produces a helical form of chip.

2.3.1 Chip Formation

Metal cutting chips have been classified

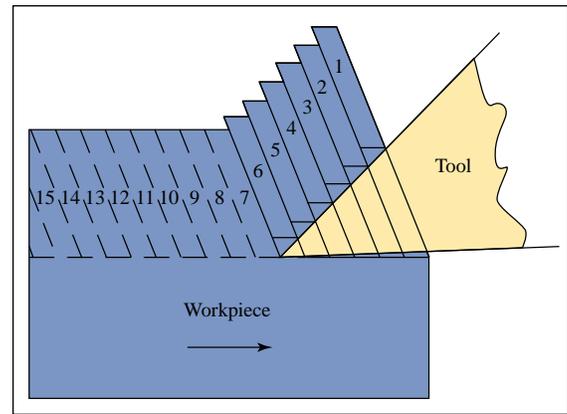


FIGURE 2.4 Chip formation compared to a sliding deck of cards.

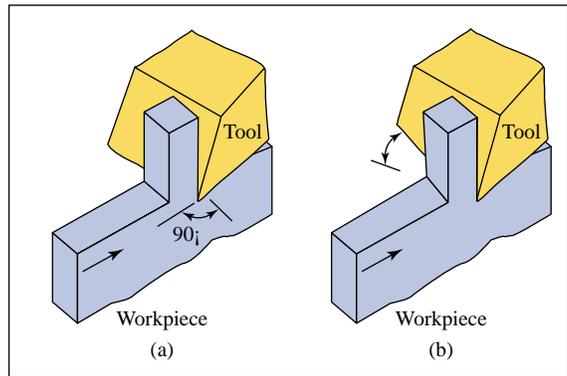


FIGURE 2.5 Chip formation showing both (a) orthogonal cutting and (b) oblique cutting.

into three basic types:

- discontinuous or segmented
- continuous
- continuous with a built-up edge.

All three types of chips are shown in Figure 2.6 a,b, and c.

Discontinuous Chip - Type 1:

Discontinuous or segmented chips are produced when brittle metal such as cast iron and hard bronze are cut or when some ductile metals are cut under poor cutting conditions. As the point of the cutting tool contacts the metal, some compression occurs, and the chip begins

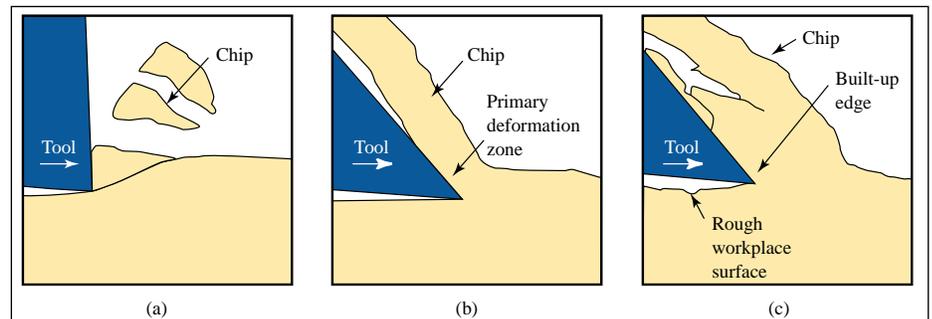


FIGURE 2.6 Types of chip formations: (a) discontinuous, (b) continuous, (c) continuous with built-up edge (BUE).

flowing along the chip-tool interface. As more stress is applied to brittle metal by the cutting action, the metal compresses until it reaches a point where rupture occurs and the chip separates from the unmachined portion. This cycle is repeated indefinitely during the cutting operation, with the rupture of each segment occurring on the shear angle or plane. Generally, as a result of these successive ruptures, a poor surface is produced on the workpiece.

Continuous Chip - Type 2: The Type 2 chip is a continuous ribbon produced when the flow of metal next to the tool face is not greatly restricted by a built-up edge or friction at the chip tool interface. The continuous ribbon chip is considered ideal for efficient cutting action because it results in better finishes.

Unlike the Type 1 chip, fractures or ruptures do not occur here, because of the ductile nature of the metal. The crystal structure of the ductile metal is elongated when it is compressed by the action of the cutting tool and as the chip separates from the metal. The process of chip formation occurs in a single plane, extending from the cutting tool to the unmachined work surface. The area where plastic deformation of the crystal structure and shear occurs, is called the shear zone. The angle on which the chip separates from the metal is called the shear angle, as shown in Figure 2.2.

Continuous Chip with a Built-up Edge (BUE)- Type 3: The metal ahead of the cutting tool is compressed and forms a chip which begins to flow along the chip-tool interface. As a result of the high temperature, the high pressure, and the high frictional resistance against the flow of the chip along the chip-tool interface, small particles of metal begin adhering to the edge of the cutting tool while the chip shears away. As the cutting process continues, more particles adhere to the cutting tool and a larger build-up results, which affects the cutting action. The built-up edge increases in size and becomes more unstable. Eventually a point is reached where fragments are torn off. Portions of these fragments which break off, stick to both the chip and the workpiece. The build-up and breakdown of the built-up edge occur rapidly during a cutting action and cover the machined surface with a multitude of built-up fragments. These fragments adhere to and score the

machined surface, resulting in a poor surface finish.

Shear Angle: Certain characteristics of continuous chips are determined by the shear angle. The shear angle is the plane where slip occurs, to begin chip formation (Figure 2.2). In Figure 2.7 the distortion of the work material grains in the chip, as compared to the parent material, is visible. Each fracture line in the chip as it moves upward over the tool surface can be seen, as well as the distorted surface grains where the tool has already passed. In certain work materials, these distorted surface grains account for work hardening.

Regardless of the shear angle, the compressive deformation caused by the tool force against the chip, will cause the chip to be thicker and shorter than the layer of workpiece material removed. The work or energy required to deform the material usually accounts for the largest portion of forces and power involved in a metal removing operation. For a layer of work material of given dimensions, the thicker the chip, the greater the force required to produce it.

Heat in Metal Cutting: The mechanical energy consumed in the cutting area is converted into heat. The main sources of heat are, the shear zone, the interface between the tool and the chip where the friction force generates heat, and the lower portion of the tool tip which rubs against the machined surface. The interaction of these heat sources, combined with the geometry of the cutting area, results in a complex temperature distribution, as shown in Figure 2.8.

The temperature generated in the shear plane is a function of the shear energy and the specific heat of the material. Temperature increase on the tool face depends on the friction conditions at the interface. A low coefficient of friction is, of course, desirable.

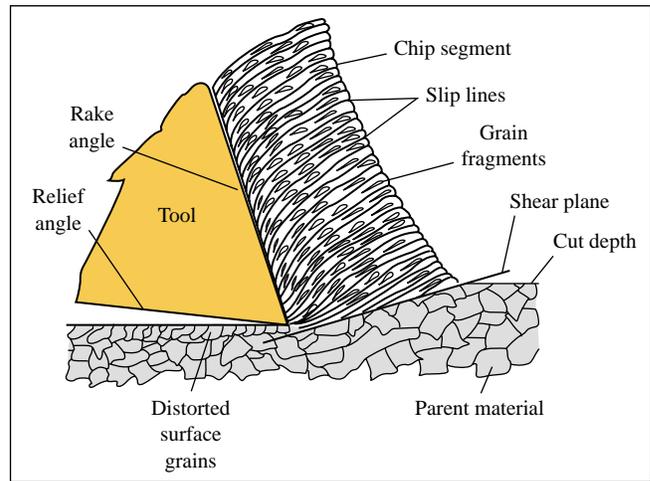


FIGURE 2.7 Distribution of work material during chip formation.

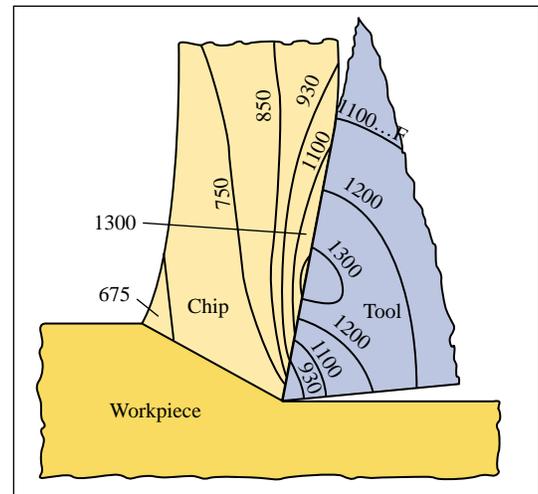


FIGURE 2.8 Typical temperature distribution in the cutting zone.

Temperature distribution will be a function of, among other factors, the thermal conductivities of the workpiece and the tool materials, the specific heat, cutting speed, depth of cut, and the use of a cutting fluid. As cutting speed increases, there is little time for the heat to be dissipated away from the cutting area and so the proportion of the heat carried away by the chip increases.

In Chapter 3 - Machinability of Metals - this topic is discussed in more detail.

2.3.2 Cutting Tool Wear

Cutting tool life is one of the most important economic considerations in metal cutting. In roughing operations, the tool material, the various tool angles, cutting speeds, and feed rates, are usually chosen to give an economi-

cal tool life. Conditions giving a very short tool life will not be economical because tool-grinding, indexing, and tool replacement costs will be high. On the other hand, the use of very low speeds and feeds to give long tool life will not be economical because of the low production rate. Clearly any tool or work material improvements that increase tool life without causing unacceptable drops in production, will be beneficial. In order to form a basis for such improvements, efforts have been made to understand the behavior of the tool, how it physically wears, the wear mechanisms, and forms of tool failure.

While the tool is engaged in the cutting operation, wear may develop in one or more areas on and near the cutting edge:

Crater Wear: Typically, cratering occurs on the top face of the tool. It is essentially the erosion of an area parallel to the cutting edge. This erosion process takes place as the chip being cut, rubs the top face of the tool. Under very high-speed cutting conditions and when machining tough materials, crater wear can be the factor which determines the life of the tool. Typical crater wear patterns are shown in Figures 2.9 and 2.10a. However, when tools are used under economical conditions, the edge wear and not the crater wear is more commonly the controlling factor in the life of the tool

Edge Wear: Edge wear occurs on the clearance face of the tool and is mainly caused by the rubbing of the newly machined workpiece surface on the contact area of the tool edge. This type of wear occurs on all tools while cutting any type of work material. Edge wear begins along the lead cutting edge and generally moves downward, away from the cutting edge. Typical edge wear patterns are shown in Figures 2.9 and 2.10b. The edge wear is also commonly known as the wearland.

Nose Wear: Usually observed after a considerable cutting time, nose wear appears when the tool has already exhibited land and/or crater wear. Wear on the nose of the cutting edge usually affects the quality of the surface finish on the workpiece.

Cutting tool material in general, and carbide tools in particular, exhibit different types of wear and/or failure:

Plastic Deformation: Edge depres-

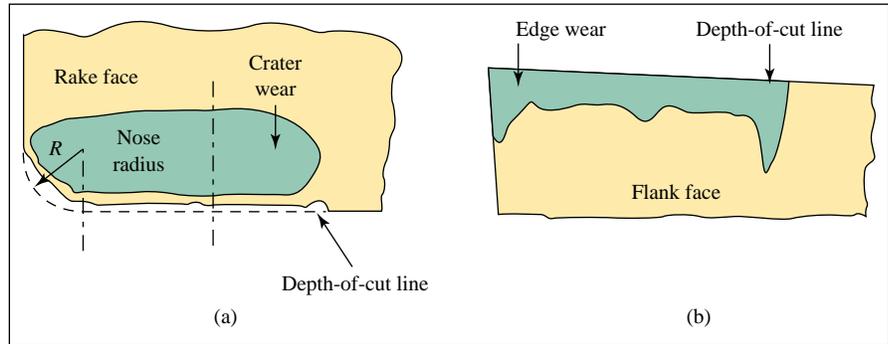


FIGURE 2.9 Carbide insert wear patterns: (a) crater wear, (b) edge wear.

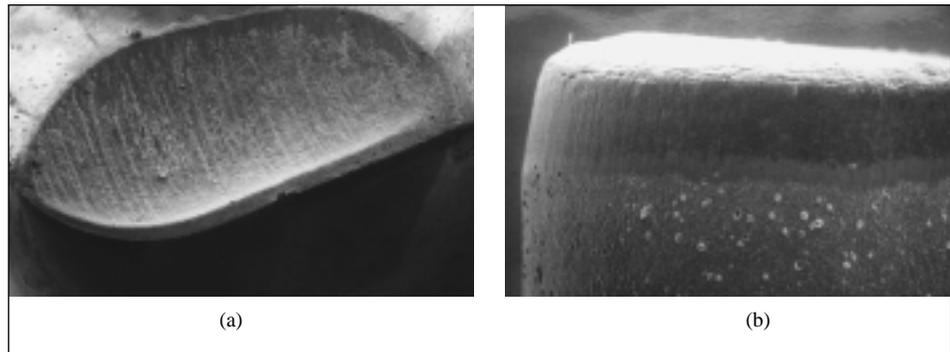


FIGURE 2.10 Carbide insert wear patterns: (a) crater wear, (b) edge wear. (Courtesy Kennametal Inc.)

sion and body bulging appear, due to excessive heat. The tool loses strength and consequently flows plastically.

Mechanical Breakage: Excessive force may cause immediate failure. Alternatively, the mechanical failure (chipping) may result from a fatigue-type failure. Thermal shock also causes mechanical failure.

Gradual Wear: The tool assumes a form of stability wear due to interaction between tool and work, resulting in crater wear. Four basic wear mechanisms affecting tool material have been categorized as:

Abrasion: Because hard inclusions in the workpiece microstructure plow into the tool face and flank surfaces, abrasion wear predominates at relatively low cutting temperatures. The abrasion resistance of a tool material is proportional to its hardness.

Adhesion: Caused by formation and subsequent destruction of minute welded junctions, adhesion wear is commonly observed as built-up edge (BUE) on the top face of the tool. This BUE may eventually disengage from the tool, causing a crater like wear. Adhesion

can also occur when minute particles of the tool surface are instantaneously welded to the chip surface at the tool-chip interface and carried away with the chip.

Diffusion: Because of high temperatures and pressures in diffusion wear, microtransfer on an atomic scale takes place. The rate of diffusion increases exponentially with increases in temperature.

Oxidation: At elevated temperature, the oxidation of the tool material can cause high tool wear rates. The oxides that are formed are easily carried away, leading to increased wear.

The different wear mechanisms as well as the different phenomena contributing to the attritious wear of the cutting tool, are dependent on the multitude of cutting conditions and especially on the cutting speeds and cutting fluids.

Aside from the sudden premature breakage of the cutting edge (tool failure), there are several indicators of the progression of physical wear. The machine operator can observe these factors prior to total rupture of the edge.

The indicators are:

- Increase in the flank wear size above a predetermined value.
- Increase in the crater depth, width or other parameter of the crater, in the rake face.
- Increase in the power consumption, or cutting forces required to perform the cut.
- Failure to maintain the dimensional quality of the machined part within a specified tolerance limit.
- Significant increase in the surface roughness of the machined part.
- Change in the chip formation due to increased crater wear or excessive heat generation.

2.4 Single Point Cutting Tools

The metal cutting tool separates chips from the workpiece in order to cut the part to the desired shape and size. There is a great variety of metal cutting tools, each of which is designed to perform a particular job or a group of metal cutting operations in an efficient manner. For example, a twist drill is designed to drill a hole of a particular size, while a turning tool might be used to turn a variety of cylindrical shapes.

2.4.1 Cutting Tool Geometry

The shape and position of the tool, relative to the workpiece, have an important effect on metal cutting. The most important geometric elements, relative to chip formation, are the location of the cutting edge and the orientation of the tool face with respect to the workpiece and the direction of cut. Other shape considerations are concerned primarily with relief or clearance, i.e., taper applied to tool surfaces to prevent rubbing or dragging against the work.

Terminology used to designate the surfaces, angles and radii of single point tools, is shown in Figure 2.11. The tool shown here is a brazed-tip type, but the same definitions apply to indexable tools.

T & P TO PLACE FIG. 2.11 HERE

The Rake Angle: The basic tool geometry is determined by the rake angle of the tool as shown in Figure 2.12. The rake angle is always at the top side of the tool. With the tool tip at the center line of the workpiece, the rake angle is determined by the angle of the tool as it goes away from the workpiece center line location. The neutral, posi-

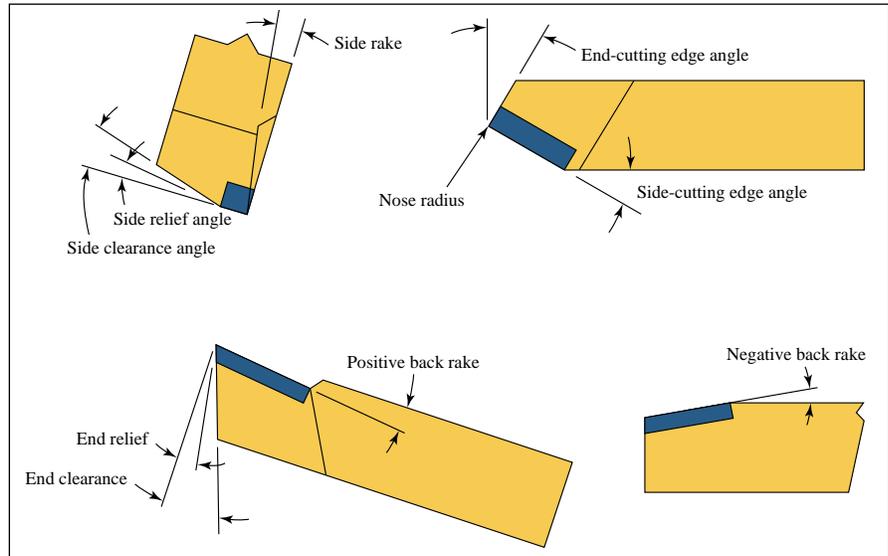


FIGURE 2.11 Terminology used to designate the surfaces, angles, and radii of single-point tools.

tive, and negative rakes are seen in (a), (b), and (c) in Figure 2.12. The angle for these geometries is set by the position of the insert pocket in the tool holder. The positive/negative (d) and double positive (e) rake angles are set by a combination of the insert pocket in the tool holder and the insert shape itself.

There are two rake angles: back rake as shown in Figure 2.12, and side rake as shown in Figure 2.13. In most turning and boring operations, it is the side rake that is the most influential. This is because the side rake is in the direction of the cut.

Rake angle has two major effects during the metal cutting process. One

major effect of rake angle is its influence on tool strength. An insert with negative rake will withstand far more loading than an insert with positive rake. The cutting force and heat are absorbed by a greater mass of tool material, and the compressive strength of carbide is about two and one half times greater than its transverse rupture strength.

The other major effect of rake angle is its influence on cutting pressure. An insert with a positive rake angle reduces cutting forces by allowing the chips to flow more freely across the rake surface.

Negative Rake: Negative rake tools

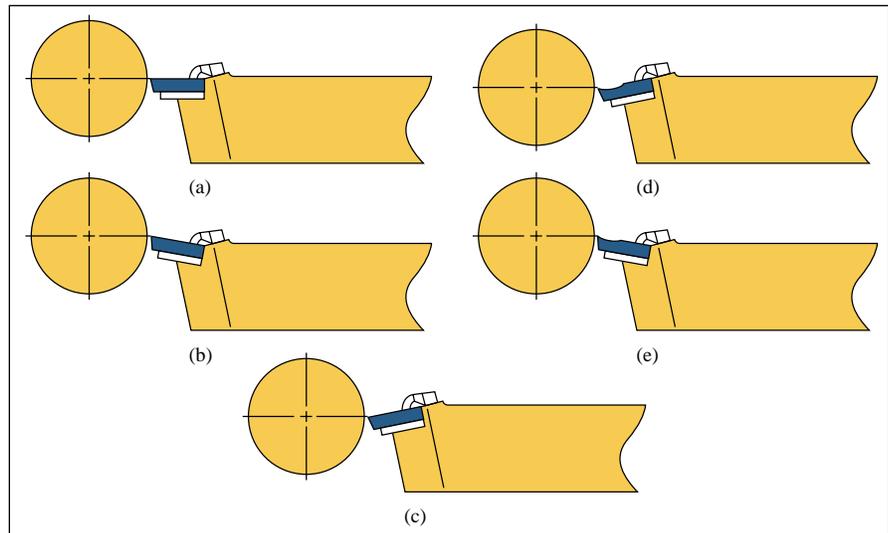


FIGURE 2.12 With the cutting tool on center, various back rake angles are shown: (a) neutral, (b) positive, (c) negative, (d) positive/negative, (e) double positive.

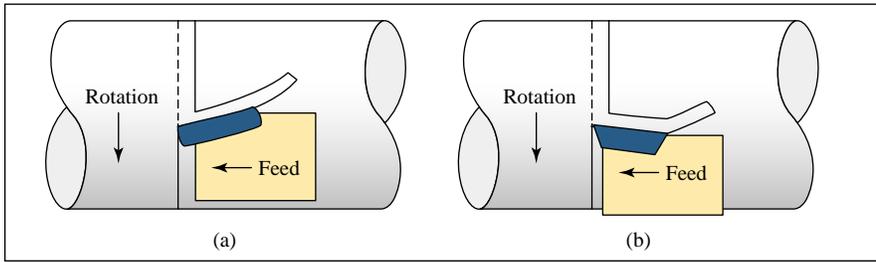


FIGURE 2.13 Side-rake-angle variations: (a) negative, (b) positive.

should be selected whenever workpiece and machine tool stiffness and rigidity allow. Negative rake, because of its strength, offers greater advantage during roughing, interrupted, scaly, and hard-spot cuts. Negative rake also offers more cutting edges for economy and often eliminates the need for a chip breaker. Negative rakes are recommended on insert grades which do not possess good toughness (low transverse rupture strength)

Negative rake is not, however, without some disadvantages. Negative rake requires more horsepower and maximum machine rigidity. It is more difficult to achieve good surface finishes with negative rake. Negative rake forces the chip into the workpiece, generates more heat into the tool and workpiece, and is generally limited to boring on larger diameters because of chip jamming.

Positive Rake: Positive rake tools should be selected only when negative rake tools can't get the job done. Some areas of cutting where positive rake may prove more effective are, when cutting tough, alloyed materials that tend to 'work-harden', such as certain stainless steels, when cutting soft or gummy metals, or when low rigidity of workpiece, tooling, machine tool, or fixture allows chatter to occur. The shearing action and free cutting of positive rake tools will often eliminate problems in these areas.

One exception that should be noted when experiencing chatter with a positive rake is, that at times the preload effect of the higher cutting forces of a negative rake tool will often dampen out chatter in a marginal situation. This may be especially true during lighter cuts when tooling is extended or when the machine tool has excessive backlash.

Neutral Rake: Neutral rake tools are

seldom used or encountered. When a negative rake insert is used in a neutral rake position, the end relief (between tool and workpiece) is usually inadequate. On the other hand, when a positive insert is used at a neutral rake, the tip of the insert is less supported, making the insert extremely vulnerable to breakage.

Positive/Negative Rake: The positive/negative rake is generally applied using the same guidelines as a positive rake. The major advantages of a positive/negative insert are that it can be used in a negative holder, it offers greater strength than a positive rake, and it doubles the number of cutting edges when using a two-sided insert.

The positive/negative insert has a ten degree positive rake. It is mounted in the normal five degree negative pocket which gives it an effective five degree positive rake when cutting. The positive/negative rake still maintains a cutting attitude which keeps the carbide under compression and offers more mass for heat dissipation. The positive/negative insert also aids in chip

breaking on many occasions, as it tends to curl the chip.

Double Positive Rake: The double positive insert is the weakest of all inserts. It is free cutting, and generally used only when delicate, light cuts are required which exert minimum force against the workpiece, as in the case of thin wall tubing, for example. Other uses of double positive inserts are for very soft or gummy work materials, such as low carbon steel and for boring small diameter holes when maximum clearance is needed.

Side Rake Angles: In addition to the back rake angles there are side rake angles as shown in Figure 2.13. These angles are normally determined by the tool manufacturers. Each manufacturer's tools may vary slightly, but usually an insert from one manufacturer can be used in the tool holder from another. The same advantage of positive and negative geometry that was discussed for back rake, applies to side rake. When back rake is positive so is side rake and when back rake is negative so is side rake.

Side and End Relief Angles: Relief angles are for the purpose of helping to eliminate tool breakage and to increase tool life. The included angle under the cutting edge must be made as large as practical. If the relief angle is too large, the cutting tool may chip or break. If the angle is too small, the tool will rub against the workpiece and generate excessive heat, and this will in turn, cause premature dulling of the cutting tool.

Small relief angles are essential when

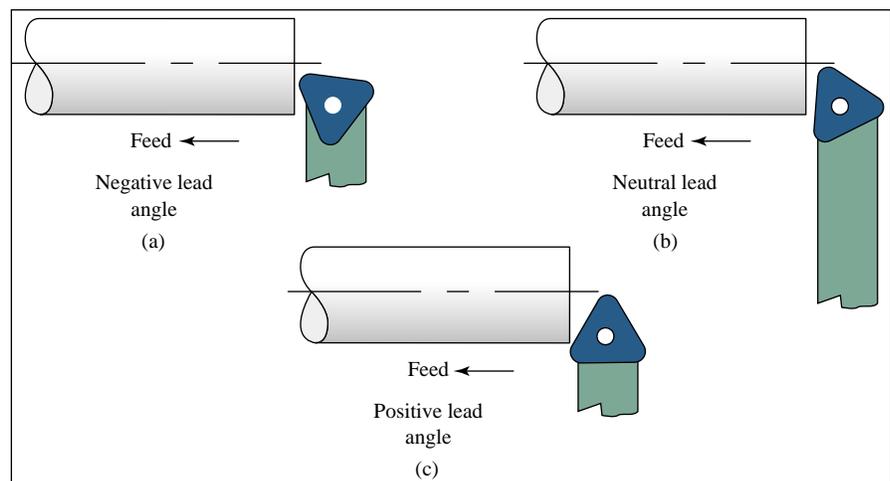


FIGURE 2.14 Lead-angle variations: (a) negative, (b) neutral, (c) positive.

machining hard and strong materials, and they should be increased for the weaker and softer materials. A smaller angle should be used for interrupted cuts or heavy feeds, and a larger angle for semi-finish and finish cuts.

Lead Angle: Lead angle (Fig. 2.14) is determined by the tool holder which must be chosen for each particular job. The insert itself can be used in any appropriate holder, for that particular insert shape, regardless of lead angle.

Lead angle is an important consideration when choosing a tool holder. A positive lead angle is the most commonly used and should be the choice for the majority of applications. Positive lead angle performs two main functions:

- It thins the chip
- It protects the insert

The undeformed chip thickness decreases when using a positive lead angle.

Positive lead angles vary, but the most common lead angles available on standard holders are 10, 15, 30 and 45 degrees. As seen in Figure 2.15, the volume of chip material is about the same in each case but the positive lead angle distributes the cutting force over a greater area of the tool's edge. This allows a substantial increase in feed rate without reducing the tool life because of excessive loading. The greater the lead angle, the more the feed rate can be increased.

Positive lead angle also reduces the longitudinal force (direction of feed) on the workpiece. But positive lead angle increases the radial force because the cutting force is always approximately perpendicular to the cutting edge (Fig. 2.16). This may become a problem when machining a workpiece that is not well supported. Care must be taken in cases where an end support, such as a tail stock center is not used.

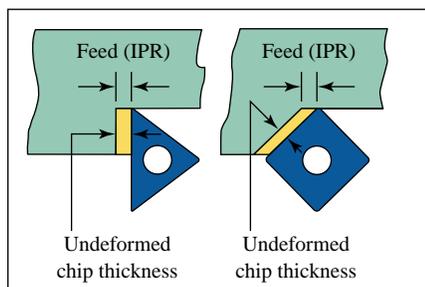


FIGURE 2.15 Lead angle vs. chip thickness. A positive lead angle thins the chip and protects the insert.

A heavy positive lead angle also has a tendency to induce chatter because of a greater tool contact area. This chatter is an amplification of tool or workpiece deflection resulting from the increased contact. In this situation it is appropriate to decrease the positive lead angle.

A positive lead angle protects the tool and promotes longer tool life. As shown in Figure 2.17 the tool comes in contact with the workpiece well away from the tool tip, which is the weakest point of the tool. As the tool progresses into the cut, the load against the tool gradually increases, rather than occurring as a sudden shock to the cutting edge. The positive lead angle also reduces the wear on the cutting edge caused by a layer of hardened material or scale, by thinning the layer and spreading it over a greater area. These advantages are extremely beneficial during interrupted cuts. Another way that positive lead angle helps to extend tool life is by allowing intense heat build-up to dissipate more rapidly, since more of the tool is in contact with the workpiece.

Neutral and negative lead angle tools also have some benefits. A neutral angle offers the least amount of tool contact, which will sometimes reduce the tendency to chatter, and lowers longitudinal forces. This is important on less stable workpieces or set-ups. Negative lead angles permit machining to a shoulder or a corner and are useful for facing. Cutting forces tend to pull the insert out of the seat, leading to erratic size control. Therefore, negative lead angles should be avoided if at all possible.

2.4.2 Edge Preparation

Edge preparation is a step taken to prolong tool life or to enhance tool performance. There are four basic approaches to edge preparation:

- Edge hone
- Edge “L” land
- Edge chamfer
- Combinations of the above

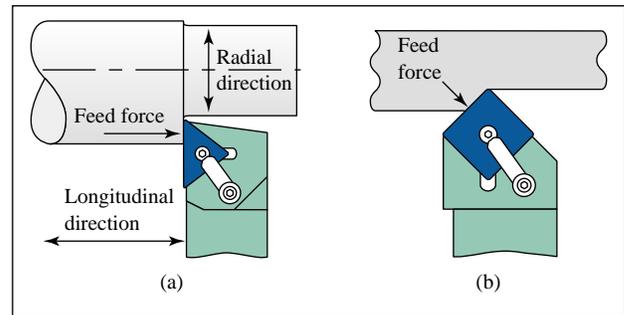


FIGURE 2.16 Lead angles and their effects on longitudinal and radial cutting-tool feed forces.

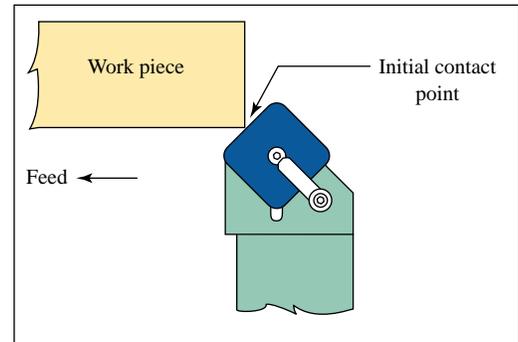


FIGURE 2.17 Gradual feed/workpiece contact protects the cutting tool by slowing increasing the load.

Many inserts, including carbide, ceramic, etc., are purchased with a standard edge preparation, normally an edge hone. The primary purpose of edge preparation is to increase the insert's resistance to chipping, breaking, and wear. Figure 2.18 illustrates the basic edge preparations.

Tool materials such as carbide and ceramic are very hard and brittle. Therefore, a lead sharp cutting edge on inserts made of these materials is extremely prone to chipping and breaking. Once a cutting edge is chipped, the wear rate is greatly accelerated or breakage occurs. A prepared edge eliminates the sharp edge and provides other benefits such as redistributing cutting forces.

Edge Hone: The edge hone is by far the most commonly used edge preparation. Many inserts are automatically provided with an edge hone at the time of purchase, especially larger inserts that will be exposed to heavy cutting. An edge hone on a ground or precision insert must usually be specially requested. A standard light hone in the United States usually has a radius of 0.001 to 0.003 inch; A standard heavy hone has a radius of 0.004 to 0.007 inch. Heavier

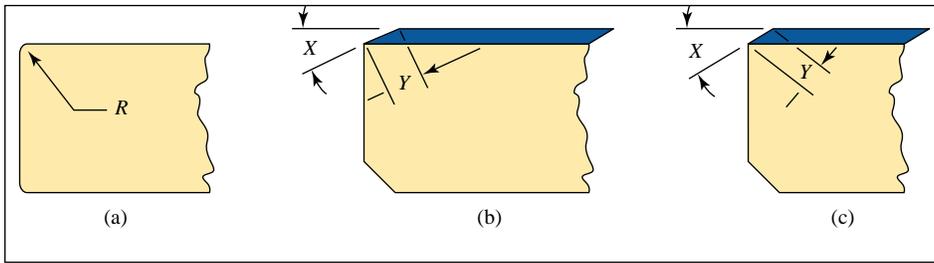


FIGURE 2.18 The three basic edge preparations are (a) edge hone, (b) L land, (c) edge chamfer.

hones are available on request. The heavier the hone, the more resistance an edge has to chipping and breaking, especially in heavy roughing cuts, interrupted cuts, hard spot cuts, and scaly cuts.

It is standard practice of all manufacturers to hone inserts that are to be coated before the inserts are subjected to the coating process. The reason for this is that during the coating process, the coating material tends to build up on sharp edges. Therefore it is necessary to hone those edges to prevent build-up.

‘L’ Land: The ‘L’ land edge preparation adds strength to the cutting edge of an insert. Essentially, the ‘L’ land amplifies the advantages of negative rake by diverting a greater amount of cutting force into the body of the insert. The ‘L’ land amplifies this condition because the included angle at the insert’s edges is 110 degrees as opposed to 90 degrees. The ‘L’ land is particularly beneficial when engaging severe scale, interruptions, and roughing.

The ‘L’ land configuration is normally 20 degrees by two thirds of the feedrate. The feedrate should exceed the land width by about one third. This is not a hard and fast rule, but it does serve as a good starting point. If the land width is greater than the feedrate, severe jamming of the chips, excessive high pressures, and high heat will likely occur, resulting in rapid tool failure.

Something other than a 20 degree land angle may be considered, with varying land width. Some experimentation may prove beneficial, however, if the land angle is varied from 20 degrees it should probably be less rather than more than 20 degrees to keep from jamming the chips.

An ‘L’ land is normally used only on negative, flat top inserts placed at a negative rake angle. To use an ‘L’ land on a positive or a positive/negative insert

would defeat the purpose of positive cutting action.

Chamfer: A chamfer is a compromise between a heavy hone and an ‘L’ land. A chamfer will also increase an insert’s resistance to chipping and breaking. In a shop situation a chamfer is easier and quicker to apply than a heavy hone, because it can be applied with a grinder rather than a hand hone. When a chamfer is applied it should be very slight, 45 degrees by 0.005 to 0.030 inch.

Normally a chamfer presents a negative cutting situation which can result in some problems. The area of application for chamfers is limited and caution must be exercised. A slight chamfer is often used on a hard and brittle tool for making a very light finishing cut on hard work material. In this instance, the chamfer will strengthen the cutting edge.

Combinations: Any time that a sharp edge can be eliminated the life of an insert will likely be extended. When an ‘L’ land or chamfer is put on an insert, it will make a dramatic improvement in performance, but the ‘L’ land or chamfer will leave some semi-sharp corners. To get the maximum benefit from an ‘L’ land or chamfer, it will help to add a slight hone to each semi-sharp corner. This will be of significant value in extending tool life, particularly when a large ‘L’ land is used.

Nose Radius: The nose radius of an insert has a great influence in the metal cutting process. The primary function of the nose radius is to provide strength to the tip of the tool. Most of the other functions and the size of the nose radius are just as important. The choice of nose radius will affect the results of the cutting operation; however, inserts are provided with various standard radii and, in most cases, one of these will meet each specific cutting need.

The larger the radius, the stronger the tool tip will be. However, a large radius causes more contact with the work surface and can cause chatter. The cutting forces will increase with a large radius for the same reason, increased contact with the work surface. When taking a shallow cut, a depth approximately equal to the radius or less, the radius acts as a positive lead angle, thinning the chip. A large radius will allow the

cutting heat to dissipate more quickly into the insert body, reducing the temperature build-up at the cutting edge.

One of the most important influences of a large radius is that of surface finish. The larger the radius, the better the surface finish will be at an equal feedrate. A larger radius will allow a faster feedrate and yet obtain a satisfactory finish. During a finishing cut, the feedrate should not exceed the radius if a reasonable surface finish is required.

2.4.3 Chip Breakers

Breaking the chip effectively when machining with carbide tools is of the utmost importance, not only from the production viewpoint, but also from the safety viewpoint. When machining steel at efficient carbide cutting speeds, a continuous chip flows away from the work at high speed.

If this chip is allowed to continue, it may wrap around the toolpost, the workpiece, the chuck, and perhaps around the operator’s arm. Not only is the operator in danger of receiving a nasty laceration, but if the chip winds around the workpiece and the machine, he must spend considerable time in removing it. A loss of production will be encountered. Therefore it is imperative that this chip be controlled and broken in some manner.

With the advent of numerical control (NC) machining and automatic chip handling systems, the control of chips is becoming more important than ever. The control of chips on any machine tool, old or new, helps to avoid jam-ups with tooling and reduces safety hazards from flying chips. There is a great deal of research and development being conducted in chip control, much of which has been very successful.

There are two basic types of chip control being used with indexable insert tooling: the mechanical chip breaker,



FIGURE 2.19 Mechanical chip breaker.

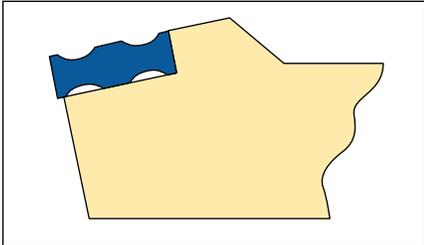


FIGURE 2.20 Sintered chip breaker.

Figure 2.19, and the sintered chip breaker, Figure 2.20. Mechanical chip breakers are not as commonly used as sintered chip breakers. There are more parts involved with the mechanical chip breaker, which increases the cost, and the chip breaker hampers changing and indexing the insert. However, mechanical chip breakers are extremely effective in controlling chips during heavy metal removing operations.

There are two groups of mechanical chip breakers, solid and adjustable as shown in Figure 2.21. Solid chip breakers are available in various lengths and angles, to suit each metal cutting application. The adjustable chip breaker can eliminate the need for stocking various sizes of solid chip breakers.

Sintered chip breakers are available in many different configurations, some designed for light feeds, some for heavy feeds, and still others for handling both light and heavy feeds. Figure 2.22 shows examples of the various sintered chip breaker configurations available from a single manufacturer. There are single sided and double sided designs of sintered chip breaker inserts.

Many of the designs will significantly reduce cutting forces as well as control chips. Normally it would be more economical to use a double sided insert

because of the additional cutting edges available. However, this is not always true. While a double sided insert is more economical under moderate and finish cutting conditions because of its additional cutting edges, a single sided design will justify itself, from a cost standpoint,

through more effective chip control and reduced cutting forces in certain situations. Figure 2.23 shows five common

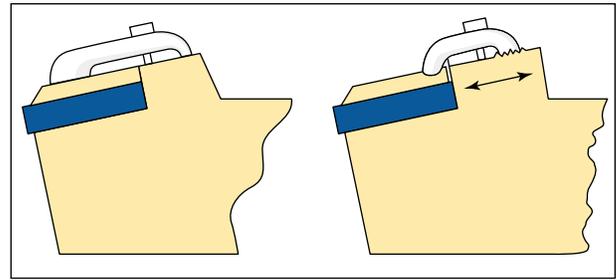


FIGURE 2.21 Solid and adjustable chip breaker.

insert styles with sintered chip breakers. Figure 2.22 illustrates that a single sided insert is flat on the bottom as com-

Double-Sided General-Purpose Groove Geometries		
.004— .020 ipr feed range		Offers excellent mix of low cost per cutting edge and effective chip control. Designed for general-purpose use at low feed rates.
.005— .065 ipr feed range		Offers excellent mix of low cost per cutting edge and effective chip control. Designed for general-purpose use at medium feed rates
.012— .070 ipr feed range		Offers excellent mix of low cost per cutting edge and effective chip control. Designed for general-purpose use at high feed rates
Single-Side Low Force Groove Geometries		
.005— .045 ipr feed range		Offers lower cutting forces than general-purpose grooves in medium feed range applications. Insert has 11° clearance angle for use in positive rake tool holder.
.006— .050 ipr feed range		Generates about 25% less cutting force than general-purpose chip grooves. Designed for medium-feed applications where force reduction, particularly in the radial direction, is important.
.012— .078 ipr feed range		Generates about 25% less cutting force than general-purpose chip grooves. Designed for ultra-high-feed applications where force reduction is important.
Double-Sided Low Feed Groove Geometries		
.003— .024 ipr feed range		Offers excellent chip control at ultra-low feed rates. Positive/negative design provides some force reducing advantages. Low cost per cutting edge.
.004— .032 ipr feed range		Positive/negative design provides lower cutting forces than general-purpose grooves in low- to medium-feed range. Offers low cost per cutting edge than other force-reducing geometries.

FIGURE 2.22 Various sintered chip breaker configurations, with application recommendations.

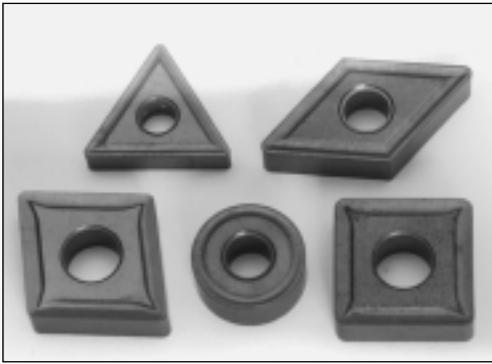


FIGURE 2.23 Five common insert shapes with various sintered chip-breaker configurations. (Courtesy American National Carbide Co.)

pared to a double sided insert. This flat bottom provides a single sided insert with better support under the cutting edge in a severe cutting situation. The single sided insert, because of its added support, has the ability to remove larger amounts of material with greater ease and efficiency, making it more economical to use. Another reason the single sided insert may be more economical is that, under heavy machining conditions, it is rare that all of the cutting edges of a double sided insert can be used. The intense thermal and mechanical shock to the insert will normally damage it to the point where the opposite cutting edge is not usable and in a sense, wasted. Figure 2.24 a,b shows two square inserts with special purpose chip breakers.

Statistics have proven that under severe conditions a single sided insert is more often the most economical choice because its higher efficiency will remove more metal in less time.

Additionally, if half of the available cutting edges of a double sided insert are unusable, for reasons stated before, then the more efficient single sided insert, having essentially the same num-

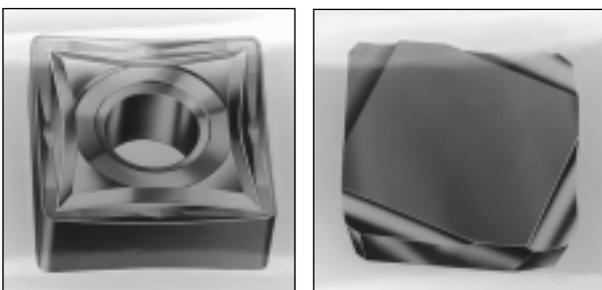


FIGURE 2.24 Two square inserts with one-sided special-purpose chip breakers. (Courtesy Iscar Metals, Inc.)

ber of usable cutting edges, is the most economical insert to use.

There are many other configurations of chip breaker designs than the ones shown in Figure 2.22. Each manufacturer has its own. The recommended application areas are generally listed in each manufacturer's catalog. However, for specific recommendations and special applications, it is best to consult the manufacturer.

Figure 2.25 shows the various types of chips that are encountered every day. Examining the chips that are coming off a workpiece will give a lot of information as to

how well the job is going, how tool wear is progressing, and why premature tool failure or short tool life is occurring.

Straight Chips: Straight chips are usually the most troublesome. They string out all over the machine tool, they get snarled in the tool, workpiece, and fixturing, they cause tooling to break, they jam up chip handling equipment, they are difficult to remove, and they are dangerous, especially when they begin to whip around. Soft gummy low carbon and tough steels usually cause this type of chip. One of the quickest ways to eliminate the straight chip, is to increase the feedrate, because a thicker chip breaks more easily. Other ways to eliminate straight chips are to decrease the lead angle, which would also thicken the chip, increase the speed, use a negative rake tool, or use a chip breaker insert.

Snarling Chips: Snarling chips are continuous chips much the same as straight chips. They are generally caused by the same conditions as straight chips and create the same problems. It stands to reason, therefore, that to correct a snarling chip situation, the same methods would be used as with straight chips. In addition, cooling the chips with a flood or mist coolant as they come off the tool, will frequently help to break them.

Infinite Helix Chips: Infinite helix chips are chips that are near the breaking point. The problems this type of chip creates are similar

Straight chips	
Snarling chips	
Infinite helix chips	
Full turns	
Half turns	
Tight chips	

FIGURE 2.25 Various types of chip formations.

to those created by straight chips. Infinite helix chips are common when machining very ductile material, such as leaded or resulfurized steels, and other soft materials. They will most often occur when making light cuts with positive rake tools. Using a sintered chip breaker insert, that will force the natural chip flow direction to change, is often effective in breaking the infinite helix chip. An increase in feed or speed will also help break the chip.

Full Turn Chips: Full turn chips are not usually a problem so long as they are consistent and without occasional stringers. A consistent full turn chip is near the ideal half turn chip.

Half Turn Chip: If there is such a thing as a perfect chip, it is the half turn or '6' shape chip. This is the chip shape that the machinist strives for in his cutting operation. The half turn chip is known as the classic chip form. The 'Half turn' or just about perfect chip is shown in Figure 2.26.

Tight Chips: Tight chips do not present a problem from a handling or inter-



FIGURE 2.26 Half-turn chip or "perfect" chip. (Courtesy Kennametal Inc.)

facing point of view, but these tight chips are a sign that poor tool life or premature tool failure may occur. The tight chip is formed by very high pressure and causes intense heat, deflection of the tool and workpiece, and rapid tool failure. A tight chip is a jammed chip, meaning that its flow path is overly restricted. Causes include; too high a feed rate, too negative a rake angle, improper chip breaker selection or setting, or a worn insert.

Many times a straight, snarled or infinite helix chip will be generated at the start of a cutting operation, when the insert is new. As the insert begins to wear, the chip gradually becomes well shaped and properly broken. It may even progress into a tight chip and eventually cause catastrophic tool failure. This is caused by a type of insert wear known as cratering. (see Figures 2.9a and 2.10a) In cratering, a groove is worn into the insert causing a false chip breaker groove to be formed. This is a definite sign of a problem, such as the insert is not of the correct carbide grade, is not the correct geometry, or that the cutting speed may be too fast.

2.5 Indexable Type Tooling

One of the more recent developments in cutting tool design is the indexable insert which is mechanically held in a toolholder. Inserts are available in several thicknesses and a variety of sizes and shapes. The round, square, triangle, and diamond account for the greatest percentage. Many other shapes, including the parallelogram, hexagon, and pentagon, are used to meet specific machining requirements. Each shape has its advantages and limitations since the operational, as well as the economical factors must be considered in tooling selection. The most common insert shapes were shown in Figure 2.23.

2.5.1 Indexable Insert Shapes

Indexable inserts have certainly established their position and potential in the metal working industry. The elimination of regrinding, accuracy of tool geometry, reduced inventory tool costs, and down time for tool changes, are some of the advantages resulting from the use of this tooling.

There are four basic shapes and a variety of special shapes. Because approximately 95 percent of all machining is done with the four basic shapes,

these are the ones of interest here. The four basic shapes are:

- square
- triangle
- diamond
- round

These shapes are available in many different configurations for almost any job. Each shape can be obtained for positive, negative, or positive/negative rake, with or without chip breaker grooves, with or without holes, with various edge preparations, in various tolerances, and in various radii and sizes. A variety of insert shapes and configurations are shown in Figure 2.27

Choosing a particular shape or insert requires a great deal of planning and thought. The choice of insert shape must be based on such factors as the workpiece configuration and tolerance, workpiece material, amount of material to be removed, machine tool capability, and economics.

The insert shape also has an influence on insert strength. As shown in Figure 2.28, the greater the included angle at the insert tip, the greater the strength. The round insert and the 100 degree corner of the first diamond shaped insert are shown as the strongest. Because of the higher cutting forces and the possibility of chatter, these inserts are more limited in use than the square shape. Therefore, for practical purposes, the square insert is the strongest for general use. Triangle and diamond inserts should only be used when a square cannot be used, such as when machining to a corner or a shoulder.

The Round Insert: Round or button inserts give a good finish at heavy feeds, and they are also ideal for forming inside corner radii. Their shape provides the greatest geometric strength, and they offer the maximum number of indexes when light cuts are being taken. The solid button type which is held in place by means of a clamp, generally has edges at 90 degrees to the surfaces for use in negative rake holders, thereby providing cutting edges on both sides of the insert. The CDH button type is made in larger sizes and has a counterbored hole. This button has clearance and is normally held in the toolholder with neutral

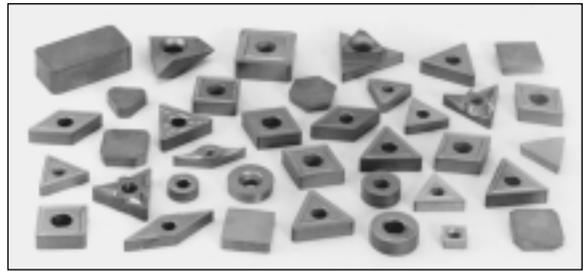


FIGURE 2.27 Various insert shapes, with and without holes, with and without chip breakers. (Courtesy American National Carbide Co.)

rake. A typical application is for tracing or contouring, where the tool must generate forms which require a large portion of the cutting edge to be in the cut.

Round inserts have their limitations, however, since the large nose radius thins the chips and increases the forces between the tool and workpiece for a given size cut. Very high radial forces are usually incurred as compared with normal cutting, particularly at normal feed rates. Chatter and deflection often result, especially when machining long-chip materials. For this reason, button inserts are applied with greater success on cast iron and the other short-chip, low-strength materials, although heavy feed rates will often improve the cutting action on ductile materials.

The Square Insert: Square inserts provide four or eight cutting edges, depending on the design of the toolholder. Positive rakes mean that relief angles must be ground on the insert, thereby eliminating the use of one side.

Square inserts are preferred for most machining jobs, where the workpiece and tool design relationships allow their use. Their shape provides strength close to that of the round insert, but with the economy of four or eight cutting edges, and also permits a reduction in the side cutting edge angle and the problem related to the chip-thinning action of the round. Economical tool application dictates the use of an insert shape which gives the maximum number of cutting

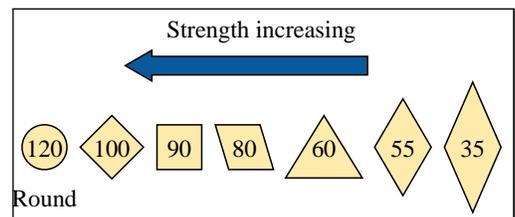


FIGURE 2.28 Various insert shapes as related to strength.

edges and is compatible with the machining operation. If the operation requires machining to a square shoulder, the square insert would be eliminated because of the design of an 'A' style tool. Since end cutting edge angle (ECEA) is required so that the tool will clear the machined surface, something less than a 90 degree included angle between the side and end of the tool is mandatory.

The Triangular Insert: Owing to design and application requirements, one of which has just been pointed out, the triangular insert has assumed an important place in indexable tooling. The triangle provides three or six cutting edges, depending on whether relief angles are required on the insert for use in a positive rake holder. The 60 degree included angle is not as strong as the 90 degree of the square, or the radius of the button, yet many machining operations are performed satisfactorily with triangular inserts. Turning to a shoulder, plunging and contouring, and numerous other operations require a generous end cutting edge angle which the triangle can provide. The 60 degree included angle is also suitable for threading operations.

Because of its fewer cutting edges and lower strength, the triangular insert and holder should only be used when other geometric shapes will not meet the job requirements.

The Pentagon Insert: A pentagon or five-sided insert is a means of providing one or two more cutting edges per insert, and the extra edges are the main reason for this design. There is, of course, a strength advantage over the square and triangle in the 108 degree included angle. As in the case of the square, the pentagonal shape sets up certain design and application limitations. The tool must always cut with a side cutting edge angle (SCEA), which thins the chip and improves tool life. However, SCEA cannot always be used owing to the requirements of the finished part's shape or because the increased radial forces cause chatter and deflection of the workpiece. The minimum SCE angle which can be used is 24 degrees. This then leaves 6 degrees end cutting edge (ECE) angle. An SCEA of 33 degrees results in 15 degrees of ECEA which is the same as that used on standard 'B' style tools and is quite adequate.

The Diamond Insert: The trend in lathe design is toward machines which generate the form on the workpiece. This is accomplished by guiding the tool so it faces, plunges, turns, and forms radii, chamfers, and machines other configurations. In order for a tool to satisfy the requirements of these complex maneuvers, it must meet certain design standards. Since the tool often plunges along an angle, a great amount of ECEA is needed. Back facing is also a common operation on such setups, and this requires negative SCEA.

The diamond insert was developed specifically for tracing operations. The industry's standard marking system includes designations for diamond inserts with included angles of 86, 80, 55, and 35 degrees. By far the most popular size is the 55 degree included angle diamond. This geometry apparently meets the requirements of most tracing operations. When the insert is positioned in the toolholder and tool block so that it cuts with 3 degree negative SCEA, it will back face with depths of cut up to 0.020 inch and in most toolholders will be able to plunge at an angle of 20 degrees with adequate clearance.

Holding the insert securely in the holder so that duplication of workpiece size to tolerances specified is achieved, has been a problem. The tendency for the insert to twist in the pocket on turning and plunging operations, and to be pulled out of the pocket on back facing operations, has resulted in design changes by some manufacturers. Diamond tracer inserts are made in regular and elongated shapes. The elongated diamond provides greater resistance to the twisting action set up by the cutting forces.

Further developments are still being made in tracer inserts and holders so that they will meet the exacting requirements of tracing operations better. In some designs the diamond shaped insert, either regular or elongated, is locked into the pocket with an eccentric pin. This gives a positive holding action and locates the insert against the back walls of the pocket, minimizing the chances for movement during the contouring operations.

The selection of a tool for a tracing operation should begin with an analysis of the requirements of the contouring

operations. The tool selected should be the one which provides the strongest geometric shape and still meets the contouring requirements. Many tracing jobs can be done satisfactorily with a triangular insert. If no back facing is included in the operations, no negative SCEA is needed and a standard 'A' style tool can be used. In some cases it is possible to use a tool designed to cut with SCEA. Generally, better tool life will be realized with lower cost per cutting edge, when tools without negative SCEA can be used.

The Parallelogram Insert: The parallelogram-shaped insert provides some advantages which make its use justified in certain applications. When a long side cutting edge is needed, it is sometimes more economical and advantageous from a machining standpoint, to use a parallelogram rather than a square or triangle.

The parallelogram also permits the construction of an 'A' style tool with greater geometric strength than is possible with a triangular insert. A limitation of the parallelogram design is the number of usable cutting edges. A negative rake insert can be used on two corners in a right or left-hand holder. To use the remaining two cutting edges, the opposite hand holder is required. Unless all four corners can be used, the use of the parallelogram insert may not be economically justifiable.

The Hexagonal Insert: A versatile tool makes use of a hexagonal shaped insert. Turning, facing, and chamfering can all be done from a number of positions. Its shape provides strong cutting edges as in the case of the pentagon, but also necessitates cutting with considerable SCEA. The number of usable cutting edges in this design makes it a most economical insert where it can be applied.

The On-Edge Insert: The on-edge insert concept (Fig. 2.29), has only been in use for a short time, but is becoming more common. The on-edge insert was first developed for milling operations. The main reason for its development was to provide the strength needed to withstand the constant interruption of milling cuts. The on-edge concept is now becoming more popular for turning inserts as well.

The main use of the on-edge insert is for rough cutting when cutting forces are high and the interruptions are often

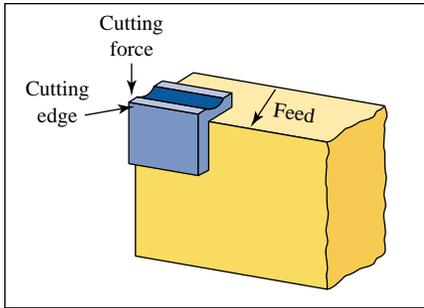


FIGURE 2.29 On-edge turning tool design.

severe.

The extra thickness of the on-edge insert offers more protection from heat and shock damage to the opposite side cutting edge during heavy roughing, than is common with standard inserts. A milling cutter section with on-edge inserts is shown in Figure 2.30.



FIGURE 2.30 On-edge milling cutter section. (Courtesy Ingersoll Cutting Tool Co.)

2.5.2 Indexable Inserts – Classes and Sizes

Inserts are commercially available with various degrees of dimensional tolerances, such as the inscribed circle of a triangle, the measurement across the flats of a square or elongated diamond, thickness, nose radii, and tangency. All these dimensions, and several other factors, contribute to the ability of an insert to be accurately indexed and to machine a given material to a specific size. The need for inserts with different tolerances depends not so much on the dimensional size of the finished part, but more on how the insert is to be used in the machining operation.

Unground Inserts: Through improved manufacturing techniques, many carbide producers can supply inserts that are to the required specifica-

tions, thus eliminating the grinding operation. Cutting edges produced by this method are not only metallurgically sound in structure, but are also honed to give them geometric increase in strength.

Utility Inserts: This type of insert is ground on the top and bottom faces only.

Precision

Inserts: These are ground all over and to close tolerances.

Honed Inserts: The development of production honing techniques for inserts has made standard inserts available to the machining industry in the pre-honed condition. These inserts have the advantage of not only having the cracked crystal layer removed from the cutting edge area, but also from the cutting tool surfaces. Lighter finishing cuts taken with finishing grades of carbide should have small amounts of honing performed on the cutting edge. Roughing grades should, conversely, be honed heavily. Carbide Insert Honing Equipment is shown in Figure 2.31.

Insert Size: The size of an insert is determined by its inscribed circle (I.C.). Every insert has an I.C. regardless of the insert shape (Fig. 2.32). The I.C. is designated in fractions of an inch in the United States, normally in 1/8 inch increments. The thickness of the insert is designated by its actual thickness in increments of 1/16 inch, and the nose radius is designated in increments of 1/64 inch.

The thickness of the insert is usually standard to a particular I.C. Sometimes however, a choice of thickness will be available. In these situations, the thickness that is appropriate to the amount of cutting force that will be applied is the optimum choice. If a thin insert is chosen, a thicker shim should be used to keep the cutting edge at the workpiece centerline.

2.5.3 Indexable Insert Identification System



FIGURE 2.31 Carbide-insert honing equipment. (Courtesy American National Carbide Co.)

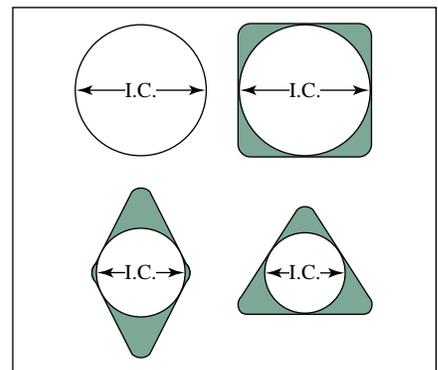


FIGURE 2.32 The size of an insert is determined by its inscribed circle (I.C.).

A standard marking system, proposed by the Cemented Carbide Producers Association and approved by the American National Standards Institute (ANSI), has been adopted by the cemented carbide manufacturers. A new identification and numbering system became necessary, due to the addition of an expanded range of types and sizes of inserts incorporating a wide variety of detail. Under this new system, the insert number, with the manufacturer's grade of carbide, is all that is needed to describe the insert. (See Fig. 2.33). The eight sequences of marking indexable inserts are:

- Shape
- Thickness
- Cutting Point
- Other Conditions
- Size
- Clearance Angle
- Class
- Type

Insert Economics: The cost of carbide and other tool materials as well as

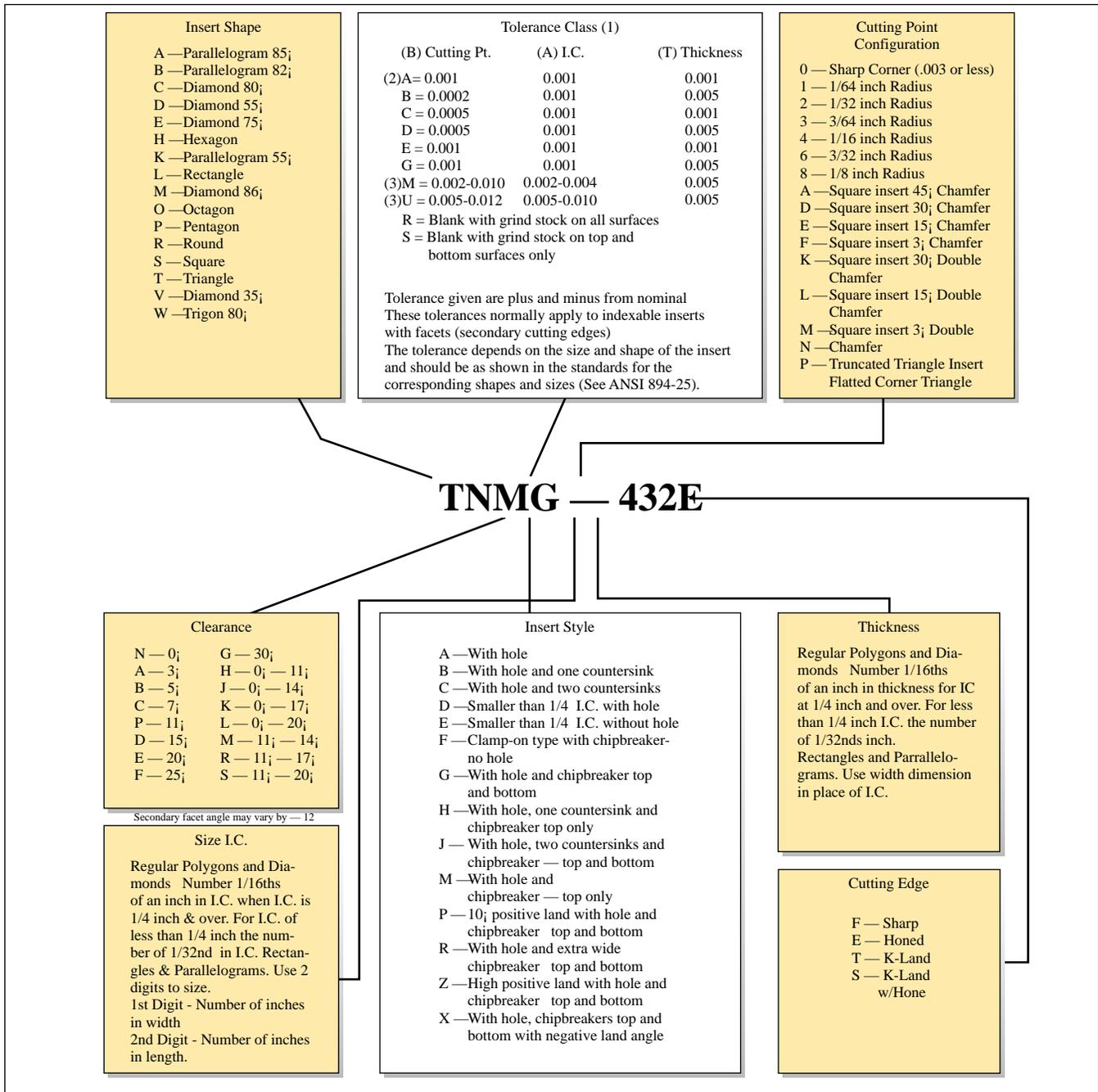


FIGURE 2.33 Standard Identification System for indexable inserts. (Courtesy Cemented Carbide Producers Association)

the cost of preparing these materials into cutting tools is relatively high and continuing to increase. Therefore, it is most important to choose tool inserts wisely. Here are some important things to consider when making the choice:

- Chose a shape that offers the most cutting edges.

Examples:

A negative insert has twice as many

cutting edges as a positive insert. A square insert has 25 percent more cutting edges than a triangle insert. A double sided chip breaker insert has twice as many cutting edges as a single sided insert.

- Choose an I.C. appropriate to the amount of material to be removed.

Examples:

A 1 inch I.C. square insert for a 1/4

inch depth of cut would be wasteful, because a large piece of expensive carbide would be used where a smaller piece would achieve the same result.

- Choose an insert tolerance that is appropriate to the job being done. In most cases an unground utility grade will do the job. The closer the tolerance, the higher the cost. Tight insert

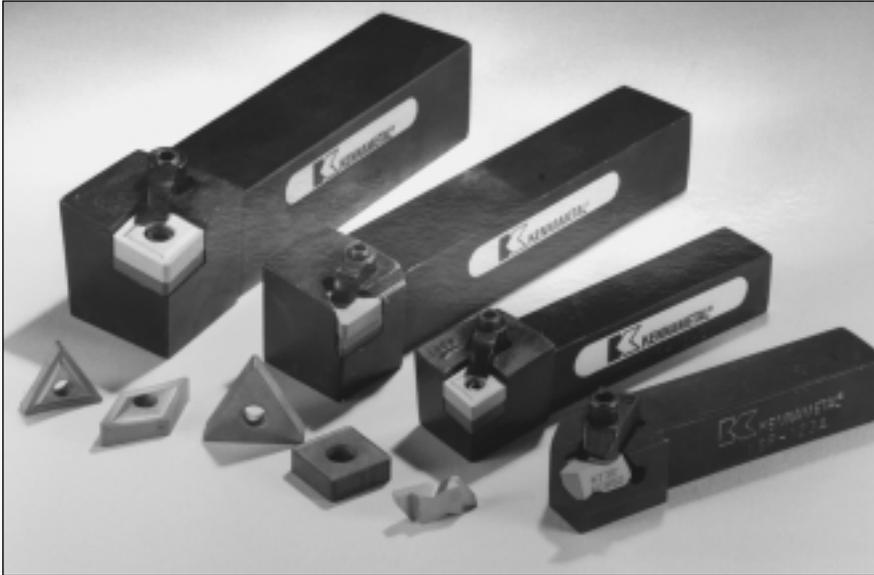


FIGURE 2.34 Four toolholders with various insert styles and sizes (Courtesy Kennametal Inc.)

tolerances are normally required only when the indexability of an insert is critical.

Example:

A 'C' tolerance insert used for finishing to a workpiece tolerance of plus or minus 0.010 inch would not be necessary. An 'M' or even a 'U' tolerance insert would be satisfactory.

- Choose a single sided insert when conditions make its efficiency more economical.

Example:

A heavy roughing cut has made the second side of a less efficient double sided insert unusable because of heat and shock damage.

2.5.4 Mechanical Tool Holders

The revolution of the indexable insert has resulted in the availability of a wide range and variety of tool holders. A number of tool holders with inserts are shown in Figure 2.34.

To select or recommend the best holder for every machining application would be a formidable task. The practice in many manufacturing plants is to standardize on one or two designs, so that a minimum of repair parts and accessories need to be carried in inventory. There are basic designs and construction elements common to all holders.

- The Shank
- The Seat
- The Clamp or Locking Device

Turning toolholders have been standardized as shown in Figure 2.35.

The Shank: The shank is the basic element of the toolholder and its purpose is to hold and present the cutting edge to the workpiece. It usually has drilled and tapped holes, slots and cutouts, and it must provide a firm support for the carbide cutting edge. Generally shanks are made of high-carbon or low-alloy steel, heat treated to give physical properties that will resist thread damage, chip erosion and deformation under the tool-block clamping

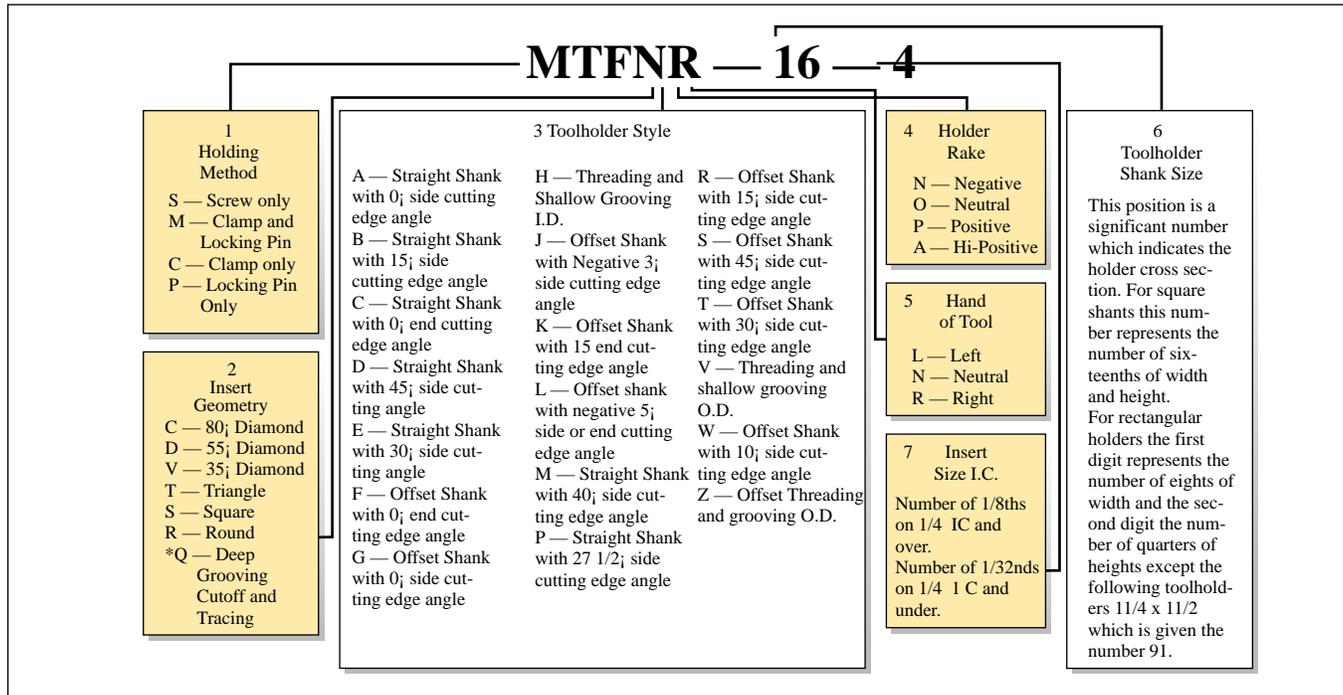


FIGURE 2.35 Standard identification system for turning toolholders. (Courtesy Cemented Carbide Producers Association)

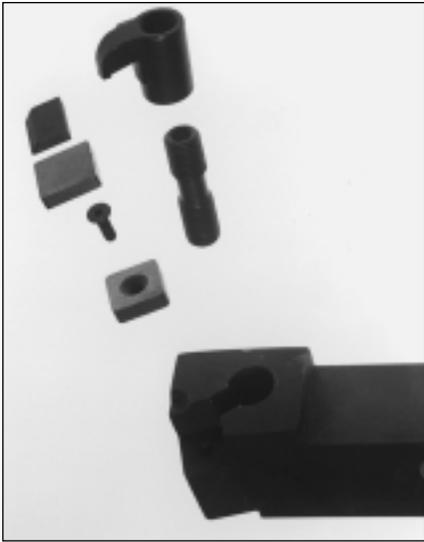


FIGURE 2.36 Tool shank with basic components. (Courtesy Kennametal Inc.)

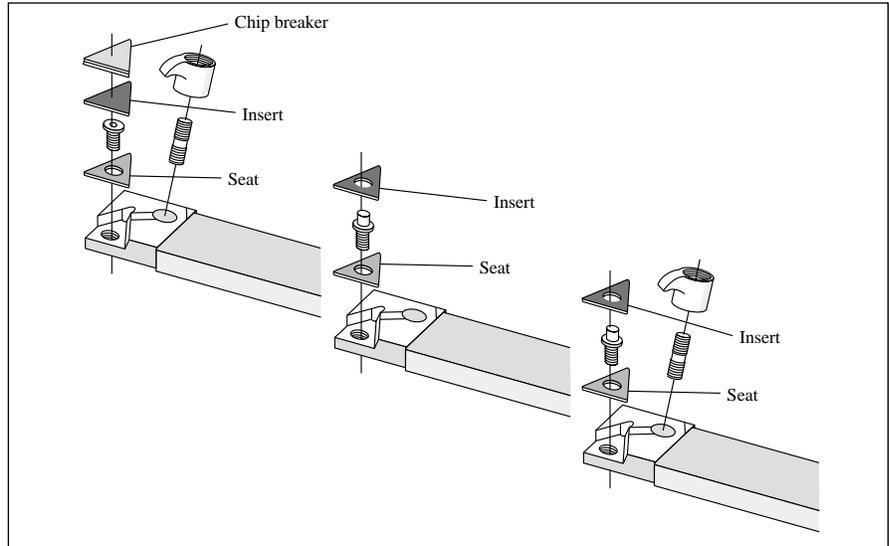


FIGURE 2.37 Schematic drawing of various insert-locking positions.

screws. Some designs and sizes which do not make use of a carbide seat are made of high alloy steel to resist deformation under the insert.

The machined area for the seat and insert is one of the most critical areas and must be flat to provide the proper support for the carbide seat and insert. Common practice is to relieve the inside corner for seat and insert clearance. The intersections of the sides and bottom of the pocket usually have a small radius, since sharp corners may be the source of cracks during heat treatment. A tool shank with basic components is shown in Figure 2.36.

The Seat: Most toolholders for indexable inserts use a carbide seat or pad as support for the insert. Cemented tungsten carbide has a high compressive strength, is hard, and can be ground to a smooth flat surface. While hardened steel has been used, and still is in some designs, a strong preference for carbide seats prevails.

The seats shown in Figure 2.37 are typical and will serve to illustrate the basic design. The periphery is chamfered at one face to clear any radius in the steel shank pocket area. If the seat or pad is held in place by a screw, the hole will be deeply countersunk so that the head of the screw will be well below the surface. If the screw head projects above the seat surface and the insert is clamped down on it, breakage of the latter could result.

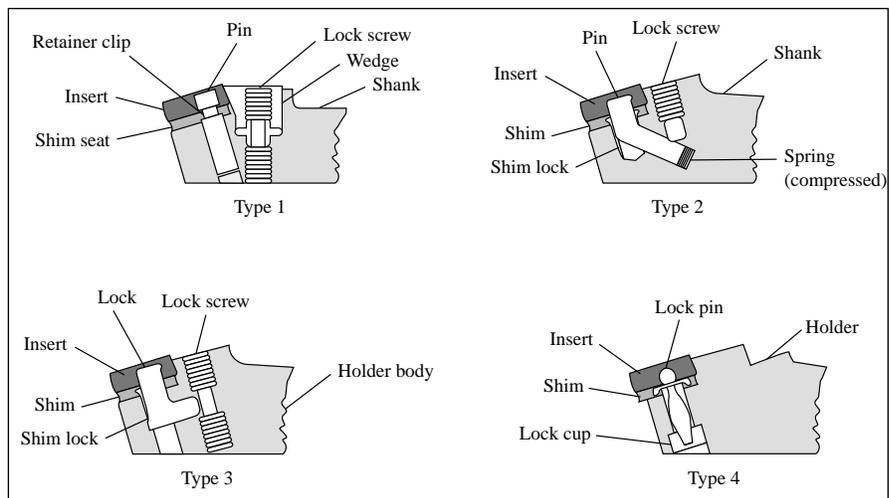


FIGURE 2.38 Various pin-type holder-locking options.

The seat is attached to the shank only for convenience and to prevent its loss when inserts are removed and replaced, or if the holder is used vertically as in a vertical turret lathe or upside down as in the rear tool post of a turret lathe.

Seat flatness is one of the most critical requirements of tool holders. Application tests have shown that an out-of-flatness condition, of as little as 0.001 inch, can result in insert breakage. Regardless of the design of the toolholder selected, the pocket and seat flatness specifications should be carefully examined and the highest standards

insisted upon.

The Clamp or Locking Device: Many clamping and locking arrangements have been developed for holding the insert in a toolholder and there is probably no one best method or design, since specific application requirements vary so greatly. There are a number of features and construction elements, however, which warrant consideration and should influence the selection of a toolholder (Fig. 2.38).

The main function of the clamping mechanism is to hold the insert securely in position and many methods of doing

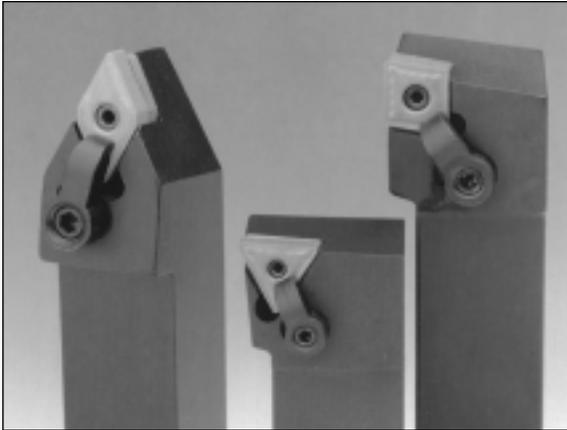


FIGURE 2.39 Three toolholders in which inserts are held by both pins and clamps. (Courtesy Sandvik Coromant Corp.)

so are in use. On normal turning and facing operations, the insert in most styles of toolholders is held in the pock-

The suitability of the clamp design to the machine toolholding blocks and to the workpiece configuration should be

et by the cutting pressures, and the load on the clamp is very light except as affected by the chip. Tracing and threading operations change the direction and amount of the load applied to the insert, and there is more tendency to twist or pull the insert out of the pocket. The ability of the clamping mechanism, to perform satisfactorily under such conditions, should be carefully evaluated. The use of a pin or lever mechanism has been incorporated in some designs to give a more positive holding action against the insert.

considered. Bulky club heads, high clamps on clamping screws, or intricate adjusting mechanisms may be in the way, especially when tools must be ganged up, or when machine and workpiece clearances are small. A toolholder which is not easily accessible and must be removed from the machine so the insert can be indexed or the chip breaker adjusted, should not be considered suitable for the application. A number of tool holders are shown in Figure 2.39 where indexable inserts are being held by both pins and clamps.

Tools which are positioned upside down should have a wrench socket in the lower end of the clamping screw so that it can be easily reached. Chip-breaker plates and clamp parts should be secured so that they will not be dropped in the chip pan when loosened for insert changing.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

Chapter 3

Machinability of Metals

3.1 Introduction

The condition and physical properties of the work material have a direct influence on the machinability of a work material. The various conditions and characteristics described as 'condition of work material', individually and in combinations, directly influence and determine the machinability. Operating conditions, tool material and geometry, and workpiece requirements exercise indirect effects on machinability and can often be used to overcome difficult conditions presented by the work material. On the other hand, they can create situations that increase machining difficulty if they are ignored. A thorough understanding of all of the factors affecting machinability and machining will help in selecting material and workpiece designs to achieve the optimum machining combinations critical to maximum productivity.

3.2 Condition of Work Material

The following eight factors determine the condition of the work material: microstructure, grain size, heat treatment, chemical composition, fabrication, hardness, yield strength, and tensile strength.

Microstructure: The microstructure of a metal refers to its crystal or grain structure as shown through examination of etched and polished surfaces under a microscope. Metals whose microstructures are similar have like machining properties. But there can be variations in the microstructure of the same workpiece, that will affect machinability.

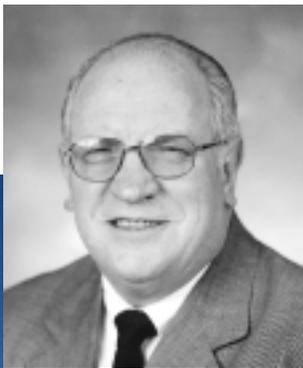
Grain Size: Grain size and structure of a metal serve as general indicators of its machinability. A metal with small undistorted grains tends to cut easily and finish easily. Such a metal is ductile, but it is also 'gummy'. Metals of an intermediate grain size represent a compromise that permits both cutting and finishing machinability. Hardness of a metal must be correlated with grain size and it is generally used as an indicator of machinability.

Heat Treatment: To provide desired properties in metals, they are sometimes put through a series of heating and cooling operations when in the solid state. A material may be treated to reduce brittleness, remove stress, to obtain ductility or toughness, to increase strength, to obtain a definite microstructure, to change hardness, or to make other changes that affect machinability.

Chemical Composition: Chemical composition of a metal is a major factor in determining its machinability. The effects of composition though, are not always clear, because the elements that make up an alloy metal, work both singly and collectively. Certain generalizations about chemical composition of steels in relation to machinability can be made, but non-ferrous alloys are too numerous and varied to permit such generalizations.

Fabrication: Whether a metal has been hot rolled, cold rolled, cold drawn, cast, or forged will affect its grain size, ductility, strength, hardness, structure - and therefore - its machinability.

The term 'wrought' refers to the hammering or forming of materials into premanufac-



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>
Prentice Hall: <http://www.prenhall.com>

tured shapes which are readily altered into components or products using traditional manufacturing techniques. Wrought metals are defined as that group of materials which are mechanically shaped into bars, billets, rolls, sheets, plates or tubing.

Casting involves pouring molten metal into a mold to arrive at a near component shape which requires minimal, or in some cases no machining. Molds for these operations are made from sand, plaster, metals and a variety of other materials.

Hardness: The textbook definition of hardness is the tendency for a material to resist deformation. Hardness is often measured using either the Brinell or Rockwell scale. The method used to measure hardness involves embedding a specific size and shaped indenter into the surface of the test material, using a predetermined load or weight. The distance the indenter penetrates the material surface will correspond to a specific Brinell or Rockwell hardness reading. The greater the indenter surface penetration, the lower the ultimate Brinell or Rockwell number, and thus the lower the corresponding hardness level. Therefore, high Brinell or Rockwell numbers or readings represent a minimal amount of indenter penetration into the workpiece and thus, by definition, are an indication of an extremely hard part. Figure 3.1 shows how hardness is measured.

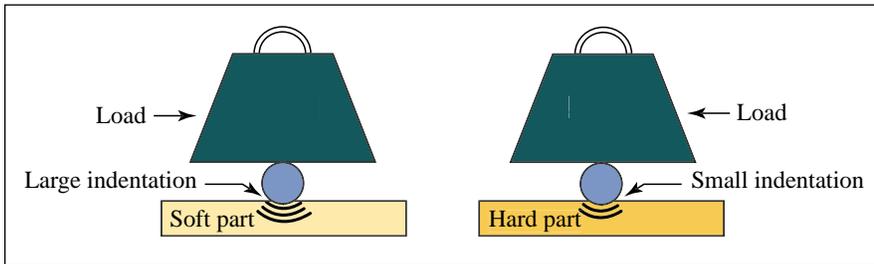


Figure 3.1 Hardness is measured by depth of indentations made.

The Brinell hardness test involves embedding a steel ball of a specific diameter, using a kilogram load, in the surface of a test piece. The Brinell Hardness Number (BHN) is determined by dividing the kilogram load by the area (in square millimeters) of the circle created at the rim of the dimple or impression left in the workpiece surface. This standardized approach provides a consistent method to make com-

parative tests between a variety of workpiece materials or a single material which has undergone various hardening processes.

The Rockwell test can be performed with various indenter sizes and loads. Several different scales exist for the Rockwell method or hardness testing. The three most popular are outlined below in terms of the actual application the test is designed to address:

Rockwell Scale	Testing Application
A	For tungsten carbide and other extremely hard materials & thin, hard sheets.
B	For medium hardness low and medium carbon steels in the annealed condition.
C	For materials > than Rockwell 'B' 100.

In terms of general machining practice, low material hardness enhances productivity, since cutting speed is often selected based on material hardness (the lower the hardness, the higher the speed). Tool life is adversely affected by an increase in workpiece hardness, since the cutting loads and temperatures rise for a specific cutting speed with part hardness, thereby reducing

tool life. In drilling and turning, the added cutting temperature is detrimental to tool life, since it produces excess heat causing accelerated edge wear. In milling, increased material hardness produces higher impact loads as inserts enter the cut, which often leads to a premature breakdown of the cutting edge.

Yield Strength: Tensile test work is used as a means of comparison of metal material conditions. These tests can

establish the yield strength, tensile strength and many other conditions of a material based on its heat treatment. In addition, these tests are used to compare different workpiece materials. The tensile test involves taking a cylindrical rod or shaft and pulling it from opposite ends with a progressively larger force in a hydraulic machine. Prior to the start of the test, two marks either two or eight inches apart are made on the rod or shaft. As the rod is systematically subjected to increased loads, the marks begin to move farther apart. A material is in the so-called 'elastic zone' when the load can be removed from the rod and the marks return to their initial distance apart of either two or eight inches. If the test is allowed to progress, a point is reached where, when the load is removed the marks will not return to their initial distance apart. At this point, permanent set or deformation of the test specimen has taken place. Figure 3.2 shows how yield strength is measured.

Yield strength is measured just prior to the point before permanent deformation takes place. Yield strength is stated in pounds per square inch (PSI) and is determined by dividing the load just prior to permanent deformation by the cross sectional area of the test specimen. This material property has been referred to as a condition, since it can be altered during heat treatment. Increased part hardness produces an increase in yield strength and therefore, as a part becomes harder, it takes a larger force to produce permanent deformation of the part. Yield strength should not be confused with fracture strength, cracking or the actual breaking of the material into pieces, since these properties are quite different and unrelated to the current subject.

By definition, a material with high yield strength (force required per unit of area to create permanent deformation) requires a high level of force to initiate chip formation in a machining operation. This implies that as a material's yield strength increases, stronger insert shapes as well as less positive cutting geometries are necessary to combat the additional load encountered in the cutting zone. Material hardness and yield strength increase simultaneously during heat treatment. Therefore, materials with relatively high yield strengths will be more difficult to machine and will reduce tool life when compared to mate-

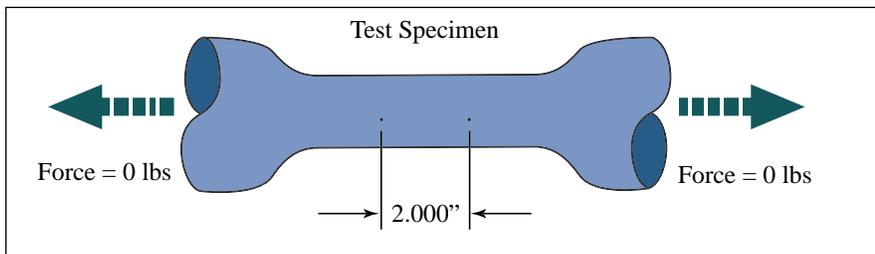


Figure 3.2 Yield strength is measured by pulling a test specimen as shown.

rials with more moderate strengths.

Tensile Strength: The tensile strength of a material increases along with yield strength as it is heat treated to greater hardness levels. This material condition is also established using a tensile test. Tensile strength (or ultimate strength) is defined as the maximum load that results during the tensile test, divided by the cross-sectional area of the test specimen. Therefore, tensile strength, like yield strength, is expressed in PSI. This value is referred to as a material condition rather than a property, since its level just like yield strength and hardness, can be altered by heat treatment. Therefore, based on the material selected, distinct tensile and yield strength levels exist for each hardness reading.

Just as increased yield strength implied higher cutting forces during machining operations, the same could be said for increased tensile strength. Again, as the workpiece tensile strength is elevated, stronger cutting edge geometries are required for productive machining and acceptable tool life.

3.3 Physical Properties of Work Materials

Physical properties will include those characteristics included in the individual material groups, such as the modulus of elasticity, thermal conductivity, thermal expansion and work hardening.

Modulus of Elasticity: The modulus of elasticity can be determined during a tensile test in the same manner as the previously mentioned conditions. However, unlike hardness, yield or tensile strength, the modulus of elasticity is a fixed material property and, therefore, is unaffected by heat treatment. This particular property is an indicator of the rate at which a material will deflect when subjected to an external force. This property is stated in PSI and typical values are several million PSI for

metals. A 2" x 4" x 8 ft. wood beam supported on either end, with a 200 pound weight hanging in the middle, will sag 17 times more than a beam of the same dimensions made out of steel and subjected to the same load. The difference is not because steel is harder or stronger, but because steel has a modulus of elasticity which is 17 times greater than wood.

General manufacturing practice dictates that productive machining of a workpiece material with a relatively moderate modulus of elasticity normally requires positive or highly positive raked cutting geometries. Positive cutting geometries produce lower cutting forces and, therefore chip formation is enhanced on elastic material using these types of tools. Sharp positive cutting edges tend to bite and promote shearing of a material, while blunt negative geometries have a tendency to create large cutting forces which impede chip formation by severely pushing or deflecting the part as the tool enters the cut.

Thermal Conductivity: Materials are frequently labeled as being either heat conductors or insulators. Conductors tend to transfer heat from a hot or cold object at a high rate, while insulators impede the flow of heat. Thermal conductivity is a measure of how efficiently a material transfers heat. Therefore, a material which has a relatively high thermal conductivity would be considered a conductor, while one with a relatively low level would be regarded as an insulator.

Metals which exhibit low thermal conductivities will not dissipate heat freely and therefore, during the machining of these materials, the cutting tool and workpiece become extremely hot. This excess heat accelerates wear at the cutting edge and reduces tool life. The proper application of sufficient amounts of coolant directly in the cutting zone

(between the cutting edge and workpiece) is essential to improving tool life in metals with low thermal conductivities.

Thermal Expansion: Many materials, especially metals, tend to increase in dimensional size as their temperature rises. This physical property is referred to as thermal expansion. The rate at which metals expand varies, depending on the type or alloy of material under consideration. The rate at which metal expands can be determined using the material's expansion coefficient. The greater the value of this coefficient, the more a material will expand when subjected to a temperature rise or contract when subjected to a temperature reduction. For example, a 100 inch bar of steel which encounters a 100 degree Fahrenheit rise in temperature would measure 100.065 inches. A bar of aluminum exposed to the same set of test conditions would measure 100.125 inches. In this case, the change in the aluminum bar length was nearly twice that of the steel bar. This is a clear indication of the significant difference in thermal expansion coefficients between these materials.

In terms of general machining practice, those materials with large thermal expansion coefficients will make holding close finish tolerances extremely difficult, since a small rise in workpiece temperature will result in dimensional change. The machining of these types of materials requires adequate coolant supplies for thermal and dimensional stability. In addition, the use of positive cutting geometries on these materials will also reduce machining temperatures.

Work Hardening: Many metals exhibit a physical characteristic which produces dramatic increases in hardness due to cold work. Cold work involves changing the shape of a metal object by bending, shaping, rolling or forming. As the metal is shaped, internal stresses develop which act to harden the part. The rate and magnitude of this internal hardening varies widely from one material to another. Heat also plays an important role in the work hardening of a material. When materials which exhibit work hardening tendencies are subjected to increased temperature, it acts like a catalyst to produce higher hardness levels in the workpiece.

The machining of workpiece materi-

als with work hardening properties should be undertaken with a generous amount of coolant. In addition, cutting speeds should correlate specifically to the material machined and should not be recklessly altered to meet a production rate. The excess heat created by unusually high cutting speeds could be extremely detrimental to the machining process by promoting work hardening of the workpiece. Low chip thicknesses should be avoided on these materials, since this type of inefficient machining practice creates heat due to friction, which produces the same type of effect mentioned earlier. Positive low force cutting geometries at moderate speeds and feeds are normally very effective on these materials.

3.4 Metal Machining

The term ‘machinability’ is a relative measure of how easily a material can be machined when compared to 160 Brinell AISI B1112 free machining low carbon steel. The American Iron and Steel Institute (AISI) ran turning tests of this material at 180 surface feet and compared their results for B1112 against several other materials. If B1112 represents a 100% rating, then materials with a rating less than this level would be decidedly more difficult to machine, while those that exceed 100% would be easier to machine.

The machinability rating of a metal takes the normal cutting speed, surface finish and tool life attained into consideration. These factors are weighted and combined to arrive at a final machin-

ability rating. The following chart shows a variety of materials and their specific machinability ratings:

3.4.1 Cast Iron

All metals which contain iron (Fe) are known as ferrous materials. The word ‘ferrous’ is by definition, ‘relating to or containing iron’. Ferrous materials include cast iron, pig iron, wrought iron, and low carbon and alloy steels. The extensive use of cast iron and steel workpiece materials, can be attributed to the fact that iron is one of the most frequently occurring elements in nature.

When iron ore and carbon are metallurgically mixed, a wide variety of workpiece materials result with a fairly unique set of physical properties. Carbon contents are altered in cast irons and steels to provide changes in hardness, yield and tensile strengths. The physical properties of cast irons and steels can be modified by changing the amount of the iron-carbon mixtures in these materials as well as their manufacturing process.

Pig iron is created after iron ore is mixed with carbon in a series of furnaces. This material can be changed further into cast iron, steel or wrought iron depending on the selected manufacturing process.

Cast iron is an iron carbon mixture which is generally used to pour sand castings, as opposed to making billets or bar stock. It has excellent flow properties and therefore, when it is heated to extreme temperatures, is an ideal material for complex cast shapes and intricate molds. This material is often used for automotive engine blocks, cylinder heads, valve bodies, manifolds, heavy equipment oil pans and machine bases.

Gray Cast Iron: Gray cast iron is an extremely versatile, very machinable relatively low strength cast iron used for pipe, automotive engine blocks, farm implements and fittings. This material receives its dark gray color from the excess carbon in the form of graphite flakes which give it its name.

Gray cast iron workpieces have relatively low hardness and strength levels. However, double negative or negative (axial) positive (radial) rake angle geometries are used to machine these materials because of their tendency to produce short discontinuous chips. When this type of chip is produced during the machining of these workpieces,

the entire cutting force is concentrated on a very narrow area of the cutting edge and therefore, double positive rake tools normally chip prematurely on these types of materials due to their lower edge strength.

White Cast Iron: White cast iron occurs when all of the carbon in the casting is combined with iron to form cementite. This is an extremely hard substance which results from the rapid cooling of the casting after it is poured. Since the carbon in this material is transformed into cementite, the resulting color of the material when chipped or fractured is a silvery white. Thus the name white cast iron. However, white cast iron has almost no ductility, and therefore when it is subjected to any type of bending or twisting loads, it fractures. The hard brittle white cast iron surface is desirable in those instances where a material with extreme abrasion resistance is required. Applications of this material would include plate rolls in a mill or rock crushers.

Due to the extreme hardness of white cast iron, it is very difficult to machine. Double negative insert geometries are almost exclusively required for these materials, since their normal hardness is 450 - 600 Brinell. As stated earlier with gray iron, this class of cast material subjects the cutting edge to extremely concentrated loads, thus requiring added edge strength.

Malleable Cast Iron: When white cast iron castings are annealed (softened by heating to a controlled temperature for a specific length of time), malleable iron castings are formed. Malleable iron castings result when hard, brittle cementite in white iron castings is transformed into tempered carbon or graphite in the form of rounded nodules or aggregate. The resulting material is a strong, ductile, tough and very machinable product which is used on a broad scope of applications.

Malleable cast irons are relatively easy to machine when compared to white iron castings. However, double negative or negative (axial) positive(radial) rake angle geometries are also used to machine these materials as with gray iron, because of their tendency to produce short discontinuous chips.

Nodular Cast Iron: Nodular or ‘ductile’ iron is used to manufacture a

Material Hardness Machinability

Rating

6061-T	
Aluminum —	190%
7075-T	
Aluminum —	120%
B1112	
Steel 160 BHN	100%
416 Stainless	
Steel 200 BHN	90%
1120 Steel 160 BHN	
80%	
1020 Steel 148 BHN	

wide range of automotive engine components including cam shafts, crank shafts, bearing caps and cylinder heads. This materials is also frequently used for heavy equipment cast parts as well as heavy machinery face plates and guides. Nodular iron is strong, ductile, tough and extremely shock resistant.

Although nodular iron castings are very machinable when compared with gray iron castings of the same hardness, high strength nodular iron castings can have relatively low machinability ratings. The cutting geometry selected for nodular iron castings is also dependent on the grade to be machined. However, double negative or positive (radial) and negative (axial) rake angles are normally used.

3.4.2 Steel

Steel materials are comprised mainly of iron and carbon, often with a modest mixture of alloying elements. The biggest difference between cast iron materials and steel is the carbon content. Cast iron materials are compositions of iron and carbon, with a minimum of 1.7 percent carbon to 4.5 percent carbon. Steel has a typical carbon content of .05 percent to 1.5 percent.

The commercial production of a significant number of steel grades is further evidence of the demand for this versatile material. Very soft steels are used in drawing applications for automobile fenders, hoods and oil pans, while premium grade high strength steels are used for cutting tools. Steels are often selected for their electrical properties or resistance to corrosion. In other applications, non magnetic steels are selected for wrist watches and minesweepers.

Plain Carbon Steel: This category of steels includes those materials which are a combination of iron and carbon with no alloying elements. As the carbon content in these materials is increased, the ductility (ability to stretch or elongate without breaking) of the material is reduced. Plain carbon steels are numbered in a four digit code according to the AISI or SAE system (i.e. 10XX). The last two digits of the code indicate the carbon content of the material in hundredths of a percentage point. For example, a 1018 steel has a .18% carbon content.

The machinability of plain carbon steels is primarily dependent on the car-

bon content of the material and its heat treatment. Those materials in the low carbon category are extremely ductile, which creates problems in chip breaking on turning and drilling operations. As the carbon content of the material rises above .30%, reliable chip control is often attainable. These materials should be milled with a positive (radial) and negative (axial) rake angle geometry. In turning and drilling operations on these materials, negative or neutral geometries should be used whenever possible. The plain carbon steels as a group are relatively easy to machine; they only present machining problems when their carbon content is very low (chip breaking or built up edge), or when they have been heat treated to an extreme (wear, insert breakage or depth of cut notching).

Alloy Steels: Plain carbon steels are made up primarily of iron and carbon, while alloy steels include these same elements with many other elemental additions. The purpose of alloying steel is either to enhance the material's physical properties or its ultimate manufacturability. The physical property enhancements include improved toughness, tensile strength, hardenability, (the relative ease with which a higher hardness level can be attained), ductility and wear resistance. The use of alloying elements can alter the final grain size of a heat treated steel, which often results in a lower machinability rating of the final product. The primary types of alloyed steel are: nickel, chromium, manganese, vanadium, molybdenum, chrome-nickel, chrome-vanadium, chrome-molybdenum, and nickel-molybdenum. The following summaries detail some of the differences in these alloys in terms of their physical as well as mechanical properties for alloyed carbon steels:

- **Nickel** - This element is used to increase the hardness and ultimate strength of the steel without sacrificing ductility.
- **Chromium** - Chromium will extend the hardness and strength gains which can be realized with nickel. However, these gains are offset by a reduction in ductility.
- **Manganese** - This category of alloyed steels possesses a greater strength level than nickel alloyed steels and improved toughness when compared to chromium alloyed steels.

- **Vanadium** - Vanadium alloyed steels are stronger, harder and tougher than their manganese counterparts. This group of materials however, loses a significant amount of its ductility when compared to the manganese group to benefit from these other physical properties.

- **Molybdenum** - This group of alloyed steels benefit from increased strength and hardness without adversely affecting ductility. These steels are often considered very tough, with an impact strength which approaches the vanadium steels.

- **Chrome-Nickel** - The alloying elements present in the chrome nickel steels produce a very ductile, tough, fine grain, wear resistant material. However, they are relatively unstable when heat treated and tend to distort, especially as their chromium and nickel content is increased.

- **Chrome-Vanadium** - This combination of alloying elements produces hardness, impact strength and toughness properties which exceed those of the chrome-nickel steels. This alloyed steel has a very fine grain structure and, therefore, improved wear resistance.

- **Chrome-Molybdenum** - This alloyed steel has slightly different properties than a straight molybdenum alloy due to the chromium content of the alloy. The final hardness and wear resistance of this alloy exceeds that of a normal molybdenum alloy steel.

- **Nickel-Molybdenum** - The properties of this material are similar to chrome-molybdenum alloyed steels except for one, its increased toughness.

The machinability of alloy steels varies widely, depending on their hardness and chemical compositions. The correct geometry selection for these materials is often totally dependent on the hardness of the part. Double positive milling or turning geometries should be selected for these materials only when either the workpiece, machine or fixturing lacks the necessary rigidity to use stronger higher force generating geometries. In milling, positive (radial) negative (axial) geometries are preferred on alloyed steels due to their strength and toughness. In turning operations, double negative or neutral geometries should be used on softer alloy steels. Lead angled tools should be used on these

materials whenever possible to minimize the shock associated with cutter entry into the cut.

Tool Steels: This group of high strength steels is often used in the manufacture of cutting tools for metals, wood and other workpiece materials. In addition, these high strength materials are used as die and punch materials due to their extreme hardness and wear resistance after heat treatment. The key to achieving the hardness, strength and wear resistance desired for any tool steel is normally through careful heat treatment. These materials are available in a wide variety of grades with a substantial number of chemical compositions designed to satisfy specific as well as general application criteria.

Tool steels are highly alloyed and therefore, quite tough; However, they can often be readily machined prior to heat treatment. Negative cutting geometries will extend tool life when machining these materials, provided the system (machine, part and fixturing) is able to withstand the additional tool force.

Stainless Steels: As the name implies, this group of materials is designed to resist oxidation and other forms of corrosion, in addition to heat in some instances. These materials tend to have significantly greater corrosion resistance than their plain or alloy steel counterparts due to the substantial additions of chromium as an alloying element. Stainless steels are used extensively in the food processing, chemical and petroleum industries to transfer corrosive liquids between processing and storage facilities. Stainless steels can be cold formed, forged, machined, welded or extruded. This group of materials can attain relatively high strength levels when compared to plain carbon and alloy steels. Stainless steels are available in up to 150 different chemical compositions. The wide selection of these materials is designed to satisfy the broad range of physical properties required by potential customers and industries.

Stainless steels fall into four distinct metallurgical categories. These categories include: austenitic, ferritic, martensitic, and precipitation hardening. Austenitic (300 series) steels are generally difficult to machine. Chatter could be a problem, thus requiring machine tools with high stiffness.

However, ferritic stainless steels (also 300 series) have good machinability. Martensitic (400 series) steels are abrasive and tend to form built-up edge, and require tool materials with high hot hardness and crater-wear resistance. Precipitation-hardening stainless steels are strong and abrasive, requiring hard and abrasion-resistant tool materials.

3.4.3 Nonferrous Metals and Alloys

Nonferrous metals and alloys cover a wide range of materials from the more common metals such as aluminum, copper, and magnesium, to high-strength high-temperature alloys such as tungsten, tantalum, and molybdenum. Although more expensive than ferrous metals, nonferrous metals and alloys have important applications because of their numerous properties, such as corrosion resistance, high thermal and electrical conductivity, low density, and ease of fabrication.

Aluminum: The relatively extensive use of aluminum as an industrial as well as consumer based material revolves around its many unique properties. For example, aluminum is a very lightweight metal (1/3 the density when compared to steel), yet it possesses great strength for its weight. Therefore, aluminum has been an excellent material for framing structures in military and commercial aircraft. The corrosive resistance of aluminum has made it a popular material selection for the soft drink industry (cans) and the residential building industry (windows and siding). In addition, most grades of aluminum are easily machined and yield greater tool life and productivity than many other metals.

Aluminum is a soft, machinable metal and the limitations on speeds are governed by the capacity of the machine and good safe practices. Chips are of the continuous type and frequently they are a limiting safety factor because they tend to bunch up. Aluminum has been machined at such high speeds that the chip becomes an oxide powder. To increase its strength and hardness, aluminum is alloyed with silicon, iron, manganese, nickel, chromium, and other metals. These materials should be machined with positive cutting geometries.

Copper: Copper is a very popular material which is widely used for its superior electrical conductivity, corro-

sion resistance and ease in formability. In addition, when alloyed properly, copper alloys can exhibit a vast array of strength levels and unique mechanical properties.

Several copper alloys are now in widespread commercial use including: copper nickels, brasses, bronzes, copper-nickel-zinc alloys, leaded copper and many special alloys. Brass and bronze are the most popular copper alloys in use.

The machinability of copper and its alloys varies widely. Pure copper and high copper alloys are very tough, abrasive, and prone to tearing. To limit and prevent tearing, these materials should be machined with positive cutting geometries. Positive geometries should also be used on bronze and bronze alloys due to their toughness and ductility. Negative axial and positive radial rake angle geometries should be used on brass alloys, since they have greater levels of machinability and in a cast state their chip formation is similar to cast iron.

Nickel: Nickel is often used as an alloying element to improve corrosion and heat resistance and the strength of many materials. When nickel is alloyed or combined with copper (Monels), chromium (Inconels and Hastelloys) or chromium and cobalt (Waspalloys), it provides a vast array of alloys which exhibit a wide range of physical properties. Other important alloys belonging to this group of materials include: Rene, Astroloy, Udimet, Incolloys, and several Haynes alloys. The machinability of nickel based alloys is generally quite low.

Most nickel based alloys should be machined using positive cutting geometries. Since these materials are machined with carbide at 120 SFPM or less, positive rake angle geometries are required to minimize cutting forces and heat generation. In the machining of most materials, increased temperature enhances chip flow and reduces the physical force on the cutting edge. Adequate clearance angles must be utilized on these materials, since many of them are very ductile and prone to work hardening. When a tool is stopped and left to rub on the workpiece, hardening of the workpiece surface will often occur. To avoid this condition, care should be taken to insure that as long as the cutting edge and part are touching,

the tool is always feeding.

Titanium and Titanium alloys:

Titanium is one of the earth's most abundant metals. Thus, its application is fairly widespread from a cutting tool material to the struts and framing members on jet aircraft. Titanium and its alloys are often selected to be used in aerospace applications due to their high strength to weight ratio and ductility.

The machining of titanium and its alloys involves the careful selection of cutting geometry and speed. Positive rake tools are often preferred on these materials to minimize part deflection and to reduce cutting temperatures in the cutting zone. The generous use of coolants on titanium and its alloys is strongly advised to maintain thermal stability and thus avoid the disastrous effects of accelerated heat and temperature buildup which leads to workpiece galling or tool breakage (drilling) and rapid edge wear. Type machinability rating for titanium and its alloys is approximately 30% or less.

Refractory Alloys: The group of materials designated as refractory alloys includes those metals which contain high concentrations of either tungsten (W), tantalum (Ta), molybdenum (Mo) or columbium (Co). This group of materials is known for its heat resistance properties which allows them to operate in extreme thermal environments without permanent damage. In addition, these materials are known for their extremely high melting points and abrasiveness. Most of these materials are quite brittle, thus, they possess very low machinability ratings when also considering their heat resistance and extreme melting properties. The machining of this group of materials is characterized by extremely low cutting speeds and feed rates when utilizing carbide cutting tools.

Cast molybdenum has a machinability rating of approximately 30 percent while pure tungsten has a rating of only 5 percent. The machinability of tantalum and columbium is at a more moderate level and thus falls between these two figures. Generally speaking, these materials should be machined at moderate to low speeds at light depths of cut using positive rake tools.

3.5 Judging Machinability

The factors affecting machinability have been explained; four methods

used to judge machinability are discussed below:

Tool Life: Metals which can be cut without rapid tool wear are generally thought of as being quite machinable, and vice versa. A workpiece material with many small hard inclusions may appear to have the same mechanical properties as a less abrasive metal. It may require no greater power consumption during cutting. Yet, the machinability of this material would be lower because its abrasive properties are responsible for rapid wear on the tool, resulting in higher machining costs.

One problem arising from the use of tool life as a machinability index is its sensitivity to the other machining variables. Of particular importance is the effect of tool material. Machinability ratings based on tool life cannot be compared if a high speed steel tool is used in one case and a sintered carbide tool in another. The superior life of the carbide tool would cause the machinability of the metal cut with the steel tool to appear unfavorable. Even if identical types of tool materials are used in evaluating the workpiece materials, meaningless ratings may still result. For example, cast iron cutting grades of carbide will not hold up when cutting steel because of excessive cratering, and steel cutting grades of carbide are not hard enough to give sufficient abrasion resistance when cutting cast iron.

Tool life may be defined as the period of time that the cutting tool performs efficiently. Many variables such as material to be machined, cutting tool material, cutting tool geometry, machine condition, cutting tool clamping, cutting speed, feed, and depth of cut, make cutting tool life determination very difficult.

The first comprehensive tool life data were reported by F.W. Taylor in 1907, and his work has been the basis for later studies. Taylor showed that the relationship between cutting speed and tool life can be expressed empirically by:

$$VT^n = C$$

where: V = cutting speed, in feet per minute

T = tool life, in minutes

C = a constant depending on work material, tool material, and other machine variables. Numerically it is the

cutting speed which would give 1 minute of tool life.

n = a constant depending on work and tool material.

This equation predicts that when plotted on log-log scales, there is a linear relationship between tool life and cutting speed. The exponent n has values ranging from 0.125 for high speed steel (HSS) tools, to 0.70 for ceramic tools.

Tool Forces and Power Consumption:

The use of tool forces or power consumption as a criterion of machinability of the workpiece material comes about for two reasons. First, the concept of machinability as the ease with which a metal is cut, implies that a metal through which a tool is easily pushed should have a good machinability rating. Second, the more practical concept of machinability in terms of minimum cost per part machined, relates to forces and power consumption, and the overhead cost of a machine of proper capacity.

When using tool forces as a machinability rating, either the cutting force or the thrust force (feeding force) may be used. The cutting force is the more popular of the two since it is the force that pushes the tool through the workpiece and determines the power consumed. Although machinability ratings could be listed according to the cutting forces under a set of standard machining conditions, the data are usually presented in terms of specific energy. Workpiece materials having a high specific energy of metal removal are said to be less machinable than those with a lower specific energy.

The use of net power consumption during machining as an index of the machinability of the workpiece is similar to the use of cutting force. Again, the data are most useful in terms of specific energy. One advantage of using specific energy of metal removal as an indication of machinability, is that it is mainly a property of the workpiece material itself and is quite insensitive to tool material. By contrast, tool life is strongly dependent on tool material.

The metal removal factor is the reciprocal of the specific energy and can be used directly as a machinability rating if forces or power consumption are used to define machinability. That is, metals

Steel



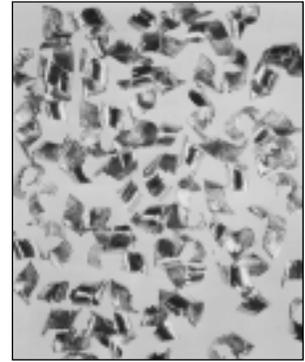
1018 Steel



1045 Steel

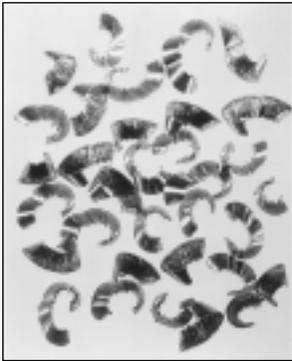


4340 Steel



Tool Steel

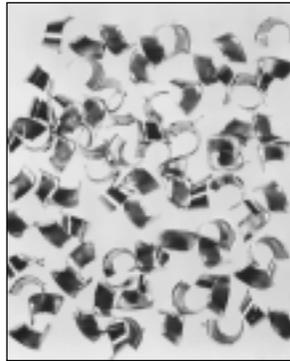
Stainless Steel



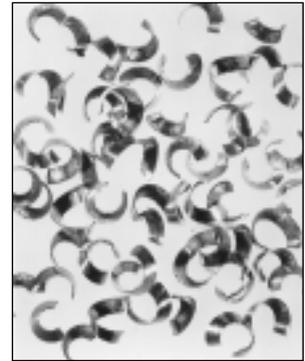
316 Stainless



17-4 PH Stainless

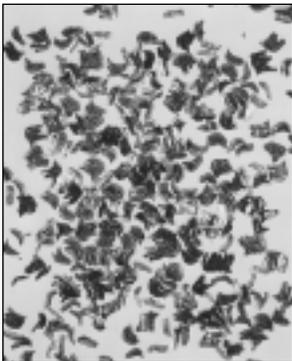


Inconel 718



Ti-6Al-4V

Cast Iron



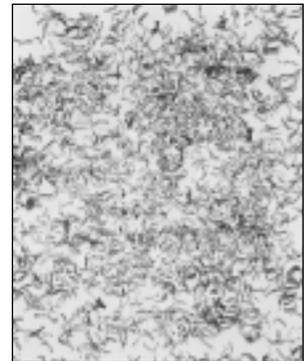
Gray Iron



80-55-06 Nodular Iron



A356 Aluminium



Brass

Figure 3.3 Ideal chips developed from a variety of common materials. (Courtesy Valenite Inc.)

with a high metal removal factor could be said to have high machinability.

Cutting tool forces were discussed in Chapter 2. Tool force and power consumption formulas and calculations are beyond the scope of this article; they are discussed in books which are more theoretical in their approach to discussing machinability of metals.

Surface Finish: The quality of the surface finish left on the workpiece during a cutting operation is sometimes useful in determining the machinability

rating of a metal. Some workpieces will not 'take a good finish' as well as others. The fundamental reason for surface roughness is the formation and sloughing off of parts of the built-up edge on the tool. Soft, ductile materials tend to form a built-up edge rather easily. Stainless steels, gas turbine alloy, and other metals with high strain hardening ability, also tend to machine with built-up edges. Materials which machine with high shear zone angles tend to minimize built-up edge effects. These

include the aluminum alloys, cold worked steels, free-machining steels, brass, and titanium alloys. If surface finish alone is the chosen index of machinability, these latter metals would rate higher than those in the first group.

In many cases, surface finish is a meaningless criterion of workpiece machinability. In roughing cuts, for example, no attention to finish is required. In many finishing cuts, the conditions producing the desired dimension on the part will inherently

provide a good finish within the engineering specification.

Machinability figures based on surface finish measurements do not always agree with figures obtained by force or tool life determinations. Stainless steels would have a low rating by any of these standards, while aluminum alloys would be rated high. Titanium alloys would have a high rating by finish measurements, low by tool life tests, and intermediate by force readings.

The machinability rating of various materials by surface finish are easily determined. Surface finish readings are taken with an appropriate instrument after standard workpieces of various materials are machined under controlled cutting conditions. The machinability rating varies inversely with the instrument reading. A low reading means good finish, and thus high machinability.

Relative ratings may be obtained by comparing the observed value of surface finish with that of a material chosen as the reference.

Chip Form: There have been machinability ratings based on the type of chip that is formed during the machining operation. The machinability might be judged by the ease of handling and disposing of chips. A material that produces long stringy chips would receive a low rating, as would one which produces fine powdery chips. Materials which inherently form nicely broken chips, a half or full turn of the normal chip helix, would receive top rating. Chip handling and disposal can be quite expensive. Stringy chips are a menace to the operator and to the finish on the freshly machined surface. However, chip formation is a function of the machine variables as well as the

workpiece material, and the ratings obtained by this method could be changed by provision of a suitable chip breaker.

Ratings based on the ease of chip disposal are basically qualitative, and would be judged by an individual who might assign letter gradings of some kind. Wide use is not made of this method of interpreting machinability. It finds some application in drilling, where good chip formation action is necessary to keep the chips running up the flutes. However, the whipping action of long coils once they are clear of the hole is undesirable. Chip formation and tool wear were discussed in Chapter 2; Figure 3.3 shows ideal chips developed from a variety of common materials.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

Chapter 4

Turning Tools & Operations

4.1 Introduction

Turning is a metal cutting process used for the generation of cylindrical surfaces. Normally the workpiece is rotated on a spindle and the tool is fed into it radially, axially, or both ways simultaneously, to give the required surface. The term ‘turning’, in the general sense, refers to the generation of any cylindrical surface with a single point tool. More specifically it is often applied just to the generation of external cylindrical surfaces oriented primarily parallel to the workpiece axis. The generation of surfaces oriented primarily perpendicular to the workpiece axis are called ‘facing’. In turning the direction of the feeding motion is predominantly axial with respect to the machine spindle. In facing a radial feed is dominant. Tapered and contoured surfaces require both modes of tool feed at the same time often referred to as ‘profiling’.

Turning facing and profiling operations are shown in Figure 4.1

The cutting characteristics of most turning applications are similar. For a given surface only one cutting tool is used. This tool must overhang its holder to some extent to enable the holder to clear the rotating workpiece. Once the cut starts, the tool and the workpiece are usually in contact until the surface is completely generated. During this time the cutting speed and cut dimensions will be constant when a cylindrical surface is being turned. In the case of facing operations the cutting speed is proportional to the work diameter, the speed decreasing as the center of the piece is approached. Sometimes a spindle speed changing mechanism is provided to increase the rotating speed of the workpiece as the tool moves to the center of the part.

In general, turning is characterized by steady conditions of metal cutting. Except at the beginning and end of the cut, the forces on the cutting tool and the tool tip temperature are essentially constant. For the special case of facing, the varying cutting speed will affect the tool tip temperature. Higher temperatures will be encountered at the larger diameters on the workpiece. However, since cutting speed has only a small effect on cutting forces, the forces acting on a facing tool may be expected to remain almost constant during the cut.

4.2 Related Turning Operations

A variety of other machining operations can be performed on a lathe in addition to turning and facing. These include the following, as shown in Figure 4.2a through 4.2f.

Single point tools are used in most operations performed on a lathe. A short description of six additional lathe operations are given below:

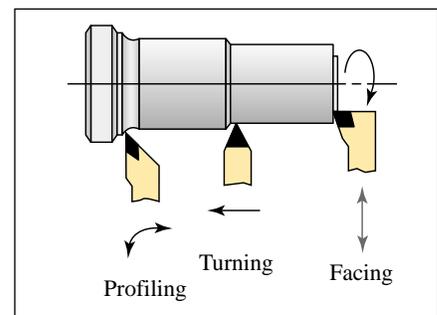


FIGURE 4.1: Diagram of the most common lathe operations: facing, turning, and profiling.



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>
Prentice Hall: <http://www.prenhall.com>

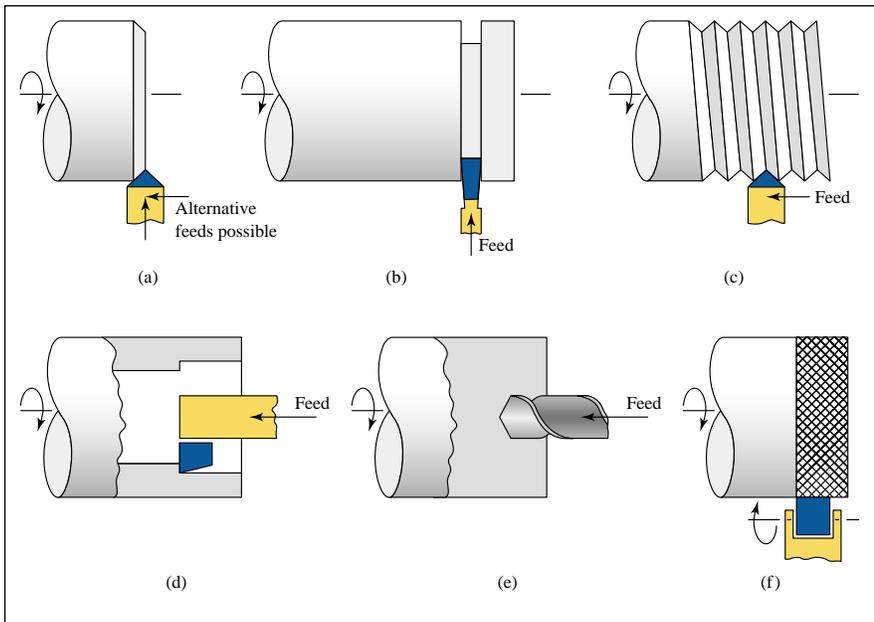


FIGURE 4.2: Related turning operations: (a) chamfering, (b) parting, (c) threading, (d) boring, (e) drilling, (f) knurling.

Chamfering: The tool is used to cut an angle on the corner of a cylinder.

Parting: The tool is fed radially into rotating work at a specific location along its length to cut off the end of a part.

Threading: A pointed tool is fed linearly across the outside or inside surface of rotating parts to produce external or internal threads.

Boring: Enlarging a hole made by a previous process. A single point tool is fed linearly and parallel to the axis

of rotation.

Drilling: Producing a hole by feeding the drill into the rotating work along its axis. Drilling can be followed by reaming or boring to improve accuracy and surface finish.

Knurling: Metal forming operation used to produce a regular cross-hatched pattern in work surfaces.

Chamfering and profiling operations are shown in Figures 4.3a and 4.3b respectively.



FIGURE 4.3: Chamfering (a) and profiling (b) operations, typically performed on a lathe or a machining center. (Courtesy Valenite Inc.)

4.3 Turning Tool Holders

Mechanical Tool Holders and the ANSI Identification System for Turning Tool Holders and indexable inserts were introduced in Chapter 2. A more detailed discussion of Toolholder Styles and their application will be presented here.

4.3.1 Toolholder Styles

The ANSI numbering system for turning toolholders has assigned letters to specific geometries in terms of lead angle and end cutting edge angle. The primary lathe machining operations of turning, facing, grooving, threading and cutoff are covered by one of the seven basic tool styles outlined by the ANSI system. The designations for the seven primary tool styles are A, B, C, D, E, F, and G.

A STYLE - Straight shank with 0 degree side cutting edge angle, for turning operations.

B STYLE - Straight shank with 15 degree side cutting edge angle, for turning operations.

C STYLE - Straight shank with 0 degree end cutting edge angle, for cutoff and grooving operations.

D STYLE - Straight shank with 45 degree side cutting edge angle, for turning operations.

E STYLE - Straight shank with 30 degree side cutting edge angle, for threading operations.

F STYLE - Offset shank with 0 degree end cutting edge angle, for facing operations.

G STYLE - Offset shank with 0 degree side cutting edge angle; this tool is an 'A' style tool with additional clearance built in for turning operations close to the lathe chuck.

There are many other styles of turning tools available in addition to those shown here, as detailed by the ANSI numbering system (see Figure 2.35). The seven basic tools are shown in operation in Figure 4.4

Right and Left Hand Toolholders

The toolholder styles discussed here and shown above represent a fraction of those standard styles available from most indexable cutting tool manufacturers. ANSI standard turning tools can be purchased in either right or left hand styles. The problem of identifying a right hand tool from a left hand tool can be resolved by remembering that when

holding the shank of a right hand tool as shown in Figure 4.5 (insert facing upward), will cut from left to right.

4.3.2 Turning Insert Shapes

Indexable turning inserts are manufactured in a variety of shapes, sizes, and thicknesses, with straight holes, with countersunk holes, without holes, with chipbreakers on one side, with chipbreakers on two sides or without chipbreakers. The selection of the appropriate turning toolholder geometry accompanied by the correct insert shape and chip breaker geometry, will ultimately have a significant impact on the productivity and tool life of a specific turning operation.

Insert strength is one important factor in selecting the correct geometry for a workpiece material or hardness range. Triangle inserts are the most popular shaped inserts primarily because of their wide application range. A triangular insert can be utilized in any of the seven basic turning holders mentioned earlier. Diamond shaped inserts are used for profile turning operations while squares are often used on lead angle tools. The general rule for rating an insert's strength based on its shape is: 'the larger the included angle on the insert corner, the greater the insert strength'.

The following list describes the different insert shapes from strongest to weakest. The relationship between insert shapes and insert strength was shown in Chapter 2. (see Fig. 2.28)

Insert Letter Designation	Insert Description	Insert Included Angle
R	Round	N/A
O	Octagon	135
H	Hexagon	120
P	Pentagon	108
S	Square	90
C	Diamond	80
T	Triangle	60
D	Diamond	55
V	Diamond	35

Six common turning tool holders are shown in Figure 4.6a and five common indexable insert shapes with molded chip breakers are shown in Figure 4.6b.

4.4 Operating Conditions

Operating conditions control three important metal cutting variables: metal removal rate, tool life, and surface

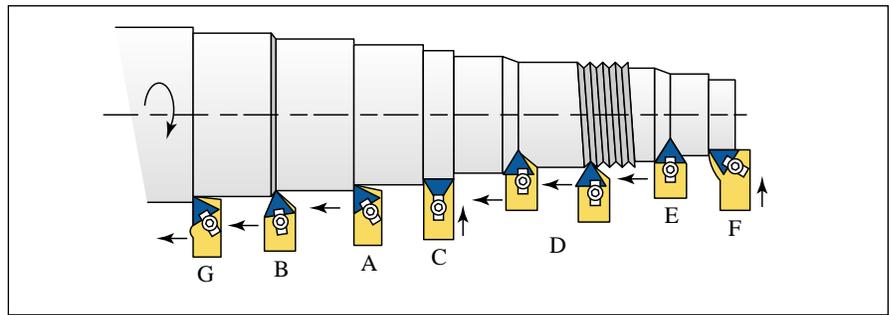


FIGURE 4.4: The primary lathe machining operations of turning, facing, grooving, threading and cut-off are performed with one of seven basic toolholder styles.

finish. Correct operating conditions must be selected to balance these three variables and to achieve the minimum machining cost per piece, the maximum production rate, and/or the best surface finish whichever is desirable for a particular operation.

The success of any machining operation is dependent on the set-up of the workpiece and the cutting tool. Set-up becomes especially important when the workpiece is not stiff or rigid and when the tooling or machine tool components must be extended to reach the area to be machined.

Deflection of the workpiece, the cutting tool, and the machine, is always present and can never be eliminated totally. This deflection is usually so minimal that it has no influence on an operation, and often goes unnoticed. The deflection only becomes a problem when it results in chatter, vibration, or distortion. It is therefore, very important to take the necessary time and effort to ensure that the set-up is as rigid as possible for the type of operation to be performed. This is especially important when making heavy or interrupted cuts.

Balancing should be considered when machining odd-shaped workpieces, especially those workpieces that have uneven weight distribution and those which are loaded off-center. An unbalanced situation can be a safety hazard and can cause work inaccuracies, chatter, and damage to the machine tool. While unbalance problems may not be apparent, they may exist at low speed operations and will become increasingly severe as the speed is increased. Unbalance condi-

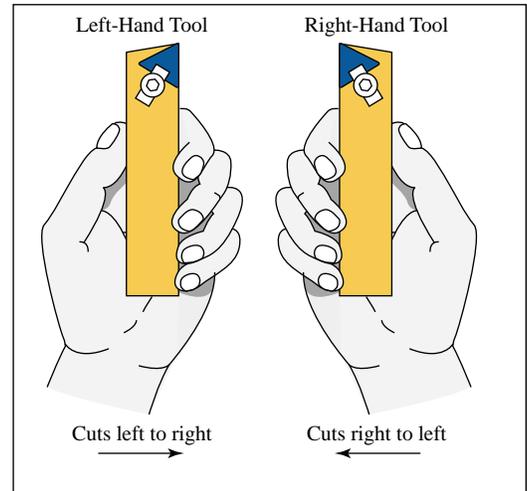


FIGURE 4.5: Identification method for right- and left-hand turning toolholders.

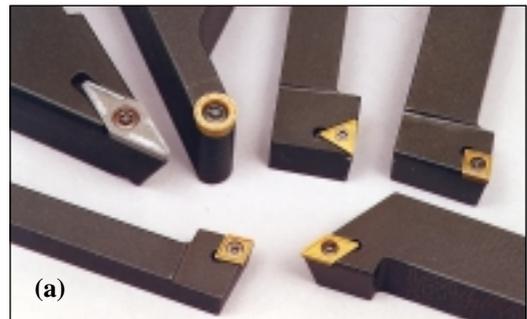


FIGURE 4.6: Common turning toolholders (a) and common indexable insert shapes (b) with molded chipbreakers are shown. (Courtesy Valenite Inc.)



FIGURE 4.7: Operating conditions become very important when machining very large parts. (Courtesy Sandvik Coromant Corp.)

tions most often occur when using turntables and lathe face plates.

As material is removed from the workpiece, the balance may change. If a series of roughing cuts causes the workpiece to become unbalanced, the problem will be compounded when the speed is increased to take finishing cuts. As a result, the reasons for problems in achieving the required accuracy and surface finish may not be apparent until the machining operation has progressed to the finishing stage. Operating conditions become very important when machining very large parts as shown in Figure 4.7.

4.4.1 Work Holding Methods

In lathe work the three most common work holding methods are:

- Held in a chuck



FIGURE 4.8: Many of the various work-holding devices used on a lathe for turning operations. (Courtesy Kitagawa Div. Sumikin Bussan International Corp.)

- Held between centers

Many of the various work holding devices used on a lathe are shown in Figure 4.8.

Chucks: The most common method of work holding, the chuck, has either three or four jaws (Fig. 4.9) and is mounted on the end of the main

spindle. A three jaw chuck is used for gripping cylindrical workpieces when the operations are such that the machined surface is concentric with the work surfaces.

The jaws have a series of teeth that mesh with spiral grooves on a circular plate within the chuck. This plate can be rotated by the key inserted in the square socket, resulting in simultaneous radial motion of the jaws. Since the jaws maintain an equal distance from the chuck axis, cylindrical workpieces are automatically centered when gripped.

Three jaw chucks, as shown in Figure 4.10, are often used to automatically clamp cylindrical parts using either electric or hydraulic power.

With the four jaw chuck, each jaw can be adjusted independently by rotation of the radially mounted threaded screws. Although accurate mounting of a workpiece can be time consuming, a four jaw chuck is often necessary for non-cylindrical workpieces. Both three and four jaw chucks are shown in Figure 4.8.

Between Centers: For accurate turning operations or in cases where the work surface is not truly cylindrical, the workpiece can be turned between centers. This form of work holding is illustrated in Fig. 4.11. Initially the workpiece has a conical center hole drilled at each end to provide location for the lathe centers. Before supporting the workpiece between the centers (one in the headstock

and one in the tailstock) a clamping device called a ‘dog’ is secured to the workpiece. The dog is arranged so that the tip is inserted into a slot in the drive plate mounted on the main spindle, ensuring that the workpiece will rotate with the spindle.

Lathe centers support the workpiece between the headstock and the tailstock. The center used in the headstock spindle is called the ‘live’ center. It rotates with the headstock spindle. The ‘dead’ center is located in the tailstock spindle. This center usually does not rotate and must be hardened and lubricated to withstand the wear of the revolving work. Shown in figure 4.12 are three kinds of dead

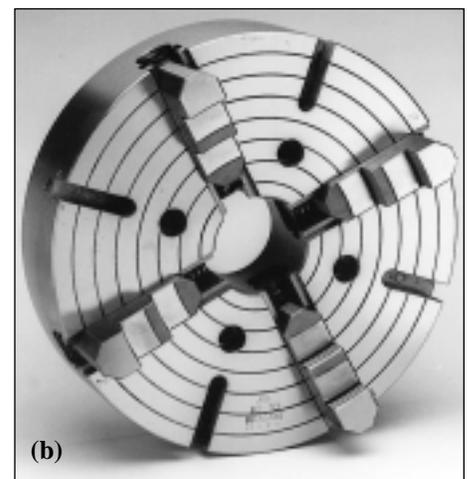


FIGURE 4.9: The most common method of work holding, the chuck, has either three jaws (a) or four jaws (b). (Courtesy Kitagawa Div. Sumikin Bussan International Corp.)

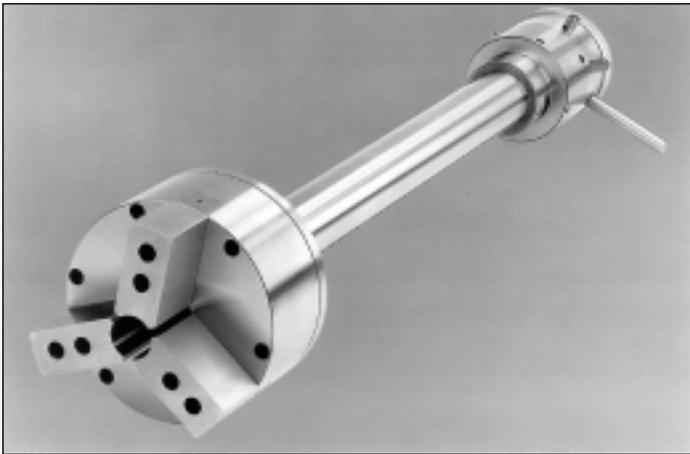


FIGURE 4.10: Three-jaw chucks are often used in automated machining systems pneumatically or hydraulically clamp cylindrical parts. (Courtesy Royal Products)



FIGURE 4.12: Hardened “dead” centers are mounted in the tailstock; they do not rotate with the workpiece and must be lubricated. (Courtesy Stark Industrial, Inc.)

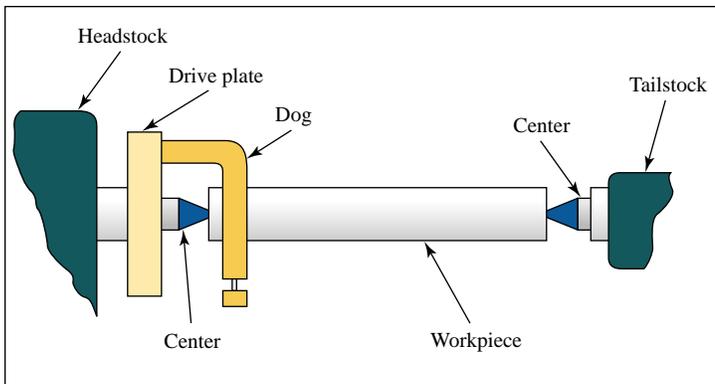


FIGURE 4.11: For accurate machining, cylindrical parts can be turned between centers.

centers.

As shown in Figure 4.13, some manufacturers are making a roller-bearing or ball-bearing center in which the center revolves.

The hole in the spindle into which the center fits, is usually of a Morse standard taper. It is important that the hole in the spindle be kept free of dirt and also that the taper of the center be clean and free of chips or burrs. If the taper of the live center has particles of dirt or a burr on it, it will not run true. The centers play a very important part in lathe operation. Since they give support to the workpiece, they must be properly ground and in perfect alignment with each other. The workpiece must have perfectly drilled and countersunk holes to receive the centers. The center must have a 60 degree point.

Collets: Collets are used when smooth bar stock, or workpieces that have been machined to a given diameter, must be held more accu-

rately than normally can be achieved in a regular three or four jaw chuck. Collets are relatively thin tubular steel bushings that are split into three longitudinal segments over about two thirds of their length (See Fig. 4.14a). The smooth internal surface of the split end is shaped to fit the piece of stock that is to be held. The external surface at the split end is a taper that fits within an internal taper of a collet sleeve placed in the spindle hole. When the collet is pulled inward into the spindle, by means of the draw bar that engages threads on the inner end of the collet, the action of the two mating

tapers squeezes the collet segments together, causing them to grip the workpiece (Fig. 4.14b).

Collets are made to fit a variety of symmetrical shapes. If the stock surface is smooth and accurate, collets will provide accurate centering; maximum runout should be less than 0.0005 inch. However, the work should be no more than 0.002 inch larger or 0.005 inch smaller than the nominal size of the collet. Consequently, collets are used only on drill rod, cold drawn, extruded, or previously machined material.

Another type of collet has a size range of about 1/8 inch. Thin strips of hardened steel are bonded together on their sides by synthetic rubber to form a truncated cone with a central hole. The collet fits into a tapered spindle sleeve so that the outer edges of the metal strips are in contact with the inner taper of the sleeve. The inner edges bear against the workpiece. Pulling the collet into the adapter sleeve causes the strips to grip the work. Because of their greater size range, fewer of these collets

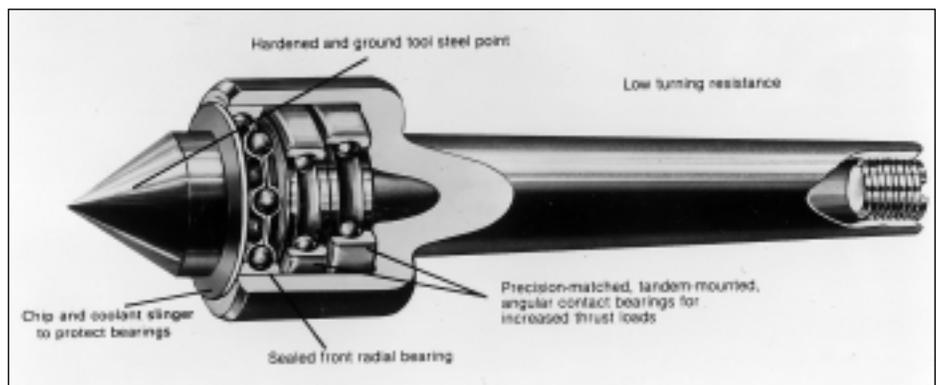
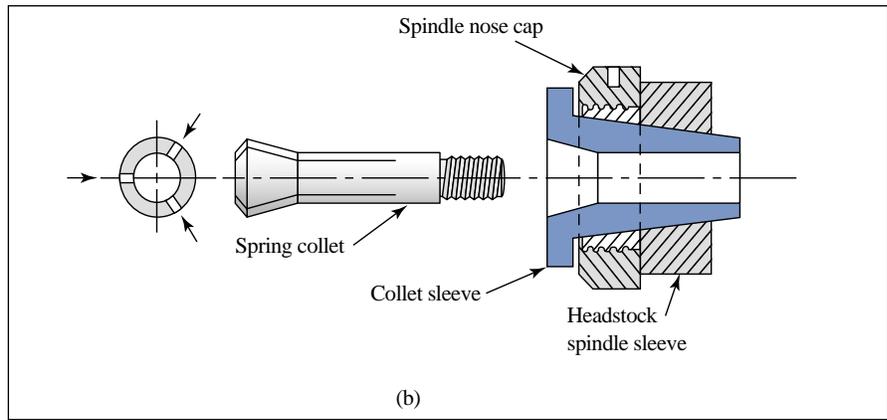


FIGURE 4.13: Hardened “live” centers are mounted in the tailstock; they rotate with the workpiece and do not need lubrication. (Courtesy Royal Products)



(a)

FIGURE 4.14: A collet (a) and a collet mounting assembly (b) are shown here. (Courtesy Lyndex Corp.)



(b)

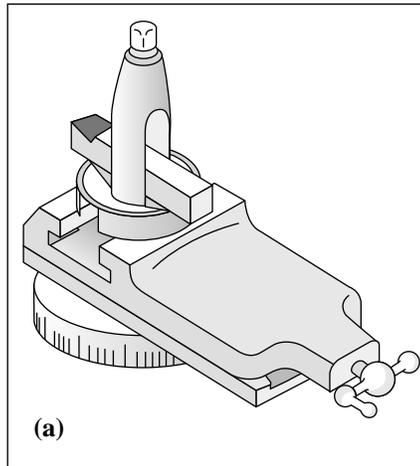
are required than with the ordinary type.

4.4.2 Tool Holding Devices

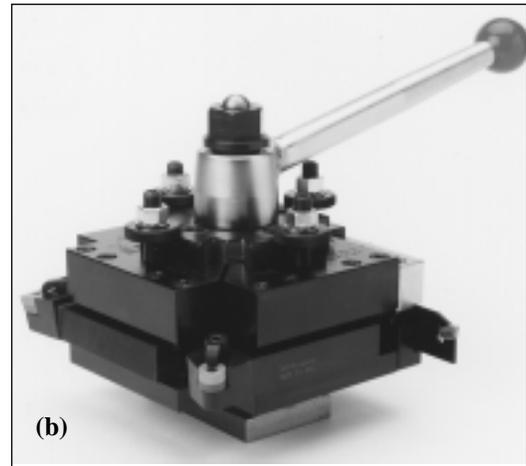
The simplest form of tool holder or post is illustrated in Figure 4.15a and is suitable for holding one single-point tool. Immediately below the tool is a curved block resting on a concave spherical surface. This method of support provides an easy way of inclining the tool so that its corner is at the correct height for the machining operation. In Figure 4.15a the tool post is shown mounted on a compound rest. The rest is a small slideway that can be clamped in any angular position in the horizontal plane and is mounted on the cross slide of the lathe. The compound rest allows the tool to be hand fed at an oblique angle to the lathe bed and is required in operations like screw-threading and the machining of short tapers or chamfers.

Another common form of tool post, the square turret, is shown in Figure 4.15b. It also is mounted on the compound rest. As its name suggests, this four-way tool post can accommodate as many as four cutting tools. Any cutting tool can be quickly brought into position by unlocking the tool post with the lever provided, rotating the tool post, and then reclamping with the lever.

All standard tool holders are designed to cut with the cutting point located on the centerline of the machine and workpiece. If the cutting point is not on the centerline, as shown in Figure 4.16a, the clearance angle between the tool holders and the workpiece will be reduced. The lack of clearance will lead to poor tool life and poor surface finish. It will also force the workpiece away from the tool when working with small diameters.



(a)



(b)

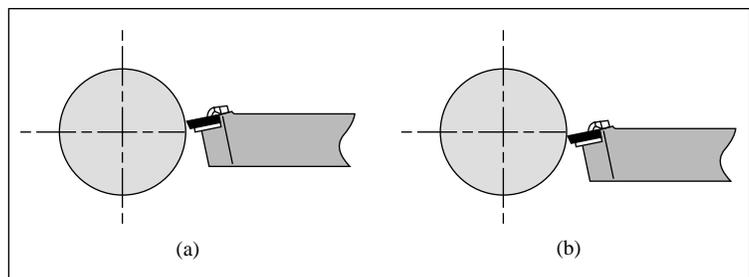
FIGURE 4.15: A toolpost for single-point tools (a) and a quick change indexing square turret, which can hold up to four tools (b). (Courtesy Dorian Tool)

On the other hand, if the cutting edge is positioned below the centerline, as shown in Figure 4.16b, the rake angle becomes more negative. Very high cutting forces will be generated and the chip will be directed into a tight curl. Insert fracture can very easily occur and a small diameter workpiece can even climb over the top of the tool and be torn from the machine.

Occasionally, however, moving the cutting point off centerline can solve a problem. An example is in situations when machining flimsy parts or when deep grooving chatter is a constant threat. Moving the tool slightly above centerline

(2% to 4% of the workpiece diameter) will change the rake angle slightly, and this in turn, will reduce cutting forces and make chatter less of a danger.

Interrupted cuts present special problems, particularly when machining large diameter workpieces. It is best to position the cutting point slightly below the centerline to present the insert in a stronger cutting position. A lead angle should also be used whenever possible. Moving the cutting point slightly below the centerline and using a lead angle, allows the workpiece to contact the tool



(a)

(b)

FIGURE 4.16: Cutting edge above workpiece centerline (a) and cutting edge below workpiece centerline (b). Both conditions result in poor performance.

on a stronger part of the insert, behind the nose.

4.5 Cutting Conditions

After deciding on the machine tool and cutting tool, the following main cutting conditions have to be considered:

- Cutting speed
- Depth of cut
- Feed rate

The choice of these cutting conditions will affect the productivity of the machining operation in general, and the following factors in particular: the life of the cutting tool; the surface finish of the workpiece; the heat generated in the cutting operation (which in turn affects the life of the tool and the surface integrity of the machined parts); and the power consumption.

Cutting Speed: Cutting speed refers to the relative surface speed between tool and work, expressed in surface feet per minute. Either the work, the tool, or both, can move during cutting. Because the machine tool is built to operate in revolutions per minute, some means must then be available for converting surface speeds into revolutions per minute (RPM). The common formula for conversion is:

$$\text{SFPM} = \frac{D \times \pi \times (\text{RPM})}{12}$$

where D is the diameter (in inches) of the workpiece or the rotating tool, π equals the constant 3.1416, and RPM is a function of the speed of the machine tool in revolutions per minute. For example:

If a lathe is set to run at 250 RPM and the diameter of the workpiece is 5 inches, then:

$$\text{SFPM} = \frac{3.1416 \times 5 \times 250}{12} = \frac{3927}{12}$$

Answer = 327.25 or 327 SFPM

The RPM can be calculated when another cutting speed is desired. For example, to use 350 SFPM with a workpiece which is 4 inches in diameter.

$$\text{RPM} = \frac{12 \times \text{SFPM}}{\pi \times D}$$

$$\text{RPM} = \frac{12 \times 350}{3.1416 \times 4} = \frac{4200}{12.57}$$

Answer = 334.13 or 334 RPM

All tool materials are meant to run at a certain SFM when machining various work materials. The SFPM range recommendations for tool and work materials are given in many reference publications.

Depth of Cut: The depth of cut relates to the depth the tool cutting edge engages the work. The depth of cut determines one linear dimension of the area of cut. For example: to reduce the outside diameter (OD) of a workpiece by .500 inches, the depth of cut would be .250 inches.

Feed Rate: The feed rate for lathe turning is the axial advance of the tool along the work for each revolution of the work expressed as inches per revolution (IPR). The feed is also expressed as a distance traveled in a single minute or IPM (inches per minute). The following formula is used to calculate the feed in IPM:

$$\text{IPM} = \text{IPR} \times \text{RPM}$$

Feed, speed, and depth of cut have a direct effect on productivity, tool life, and machine requirements. Therefore these elements must be carefully chosen for each operation. Whether the objective is rough cutting or finishing will have a great influence on the cutting conditions selected.

Roughing Cuts: When roughing, the goal is usually maximum stock removal in minimum time with minor consideration given to tool life and surface finish. There are several important points to keep in mind when rough cutting.

The first is to use a heavy feed because this makes the most efficient use of power and, with less tool contact, tends to create less chatter. There are some exceptions where a deeper cut is more advantageous than a heavy feed, especially where longer tool life is needed. Increasing the depth of cut will increase tool life over an increase in feed rate. But, as long as it is practical and chip formation is satisfactory, it is better to choose a heavy feed rate.

A heavy feed or deeper cut is usually preferable to higher speed, because the machine is less efficient at high speed. When machining common materials, the unit horsepower (HP) factor is reduced in the cut itself, as the cutting speed increases up to a certain critical value. But the machine inefficiencies will overcome any advantage when

machining heavy workpieces.

Even more important, tool life is greatly reduced at high cutting speeds unless coated carbide or other modern tool materials are used, and these also have practical speed limits. Tool life is decreased most at high speeds, although some decrease in tool life occurs when feed or depth of cut is increased. This stands to reason, because more material will be removed in less time. It becomes a choice then, between longer tool life or increased stock removal. Since productivity generally outweighs tool costs, the most practical cutting conditions are usually those which first, are most productive, and second, will achieve reasonable tool life.

Finishing Cuts: When taking finishing cuts, feed rate and depth of cut are of minor concern. The feed rate cannot exceed that which is necessary to achieve the required surface finish and the depth of cut will be light. However, the rule about speed will still apply. The speeds will generally be higher for finish cuts, but they must still be within the operating speed of the tool material.

Tool life is of greater concern for finish cuts. It is often better to strive for greater tool life at the expense of material removed per minute. If tool wear can be minimized, especially on a long cut, greater accuracy can be achieved, and matching cuts which result from tool changes, can be avoided.

One way to minimize tool wear during finishing cuts is to use the maximum feed rate that will still produce the required surface finish. The less time the tool spends on the cut, the less tool wear can occur. Another way to minimize tool wear during a long finishing cut is to reduce the speed slightly. Coolant, spray mist, or air flow, will also extend tool life because it reduces the heat of the tool.

4.6 Hard Turning

As the hardness of the workpiece increases, its machinability decreases accordingly and tool wear and fracture, as well as surface finish and integrity, can become a significant problem. There are several other mechanical processes and nonmechanical methods of removing material economically from hard or hardened metals. However, it is still possible to apply traditional cutting processes to hard metals and alloys by selecting an appropriate

tool material and machine tools with high stiffness and high speed spindles.

One common example is finish machining of heat-treated steel machine and automotive components using polycrystalline cubic boron nitride (PCBN) cutting tools. This process produces machined parts with good dimensional accuracy, surface finish, and surface integrity. It can compete successfully with grinding the same components, from both technical and economic aspects. According to some calculations grinding is over ten times more costly than hard turning.

Advanced cutting tool materials such as polycrystalline cubic boron nitride (PCBN) and ceramics (discussed in Chapter 1 - Cutting Tool Materials), have made the turning of hardened steel a cost effective alternative to grinding. Many machine shops have retired their cylindrical grinders in favor of less expensive and more versatile CNC lathes.

Compared to grinding, hard turning:

- permits faster metal removal rates, which means shorter cycle times.
- eliminates the need for coolant (dry vs. wet machining will be discussed later).
- shortens set up time and permits multiple operations to be performed in one chucking.

Today's sophisticated CNC lathes offer accuracy and surface finishes comparable to what grinders provide.

Hard turning requires much less energy than grinding, thermal and other damage to the workpiece is less likely to occur, cutting fluids may not be necessary and the machine tools are less

expensive. In addition, finishing the part while still chucked in the lathe eliminates the need for material handling and setting the part in the grinder. However, work holding devices for large and slender workpieces for hard turning can present problems, since the cutting forces are higher than in grinding.

Furthermore, tool wear and its control can be a significant problem as compared to the automatic dressing of grinding wheels. It is thus evident that the competitive position of hard turning versus grinding must be elevated individually for each application and in terms of product surface integrity, quality, and overall economics.

4.6.1 Dry vs. Wet Machining

Just two decades ago, cutting fluids accounted for less than 3 percent of the cost of most machining processes. Fluids were so cheap that few machine shops gave them much thought. Times have changed.

Today, cutting fluids account for up to 15 percent of a shop's production costs, and machine shop owners constantly worry about fluids.

Cutting fluids, especially those containing oil, have become a huge liability. Not only does the Environmental Protection Agency (EPA) regulate the disposal of such mixtures, but many states and localities also have classified them as hazardous wastes, and impose even stricter controls if they contain oil and certain alloys.

Because many high-speed machining operations and fluid nozzles create air-

borne mists, governmental bodies also limit the amount of cutting fluid mist allowed into the air. The EPA has proposed even stricter standards for controlling such airborne particulate, and the Occupational Safety and Health Administration (OSHA) is considering and advisory committee's recommendation to lower the permissible exposure limit to fluid mist.

The cost of maintenance, record keeping and compliance with current and proposed regulations is rapidly raising the price of cutting fluids. Consequently, many machine shops are considering eliminating the costs and headaches associated with cutting fluids altogether by cutting dry.

The decision to cut wet or dry must be made on a case-by-case basis. A lubricious fluid often will prove beneficial in low-speed jobs, hard-to-machine materials, difficult applications, and when surface finish requirements are demanding. A fluid with high cooling capacity can enhance performance in high-speed jobs, easy-to-machine materials, simple operations, and jobs prone to edge-buildup problems or having tight dimensional tolerances.

Many tough times though, the extra performance capabilities that a cutting fluid offers is not worth the extra expense incurred, and in a growing number of applications, cutting fluids are simply unnecessary or downright detrimental. Modern cutting tools can run hotter than their predecessors and sometimes compressed air can be used to carry hot chips away from the cutting zone.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

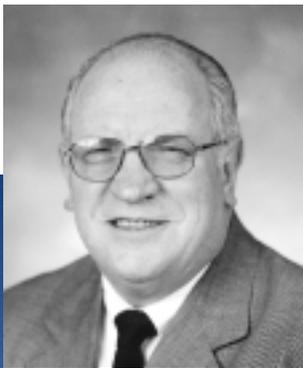
Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Finishing Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>
Prentice Hall: <http://www.prenhall.com>

Chapter 5

Turning Methods & Machines

5.1 Introduction

The basic engine lathe, which is one of the most widely used machine tools, is very versatile when used by a skilled machinist. However, it is not particularly efficient when many identical parts must be machined as rapidly as possible. As far back as 1850 there were efforts to develop variations of an engine lathe that could be operated by a relatively unskilled person for mass producing machined parts. The cutting tools were preset, or “set up” by a skilled machinist, and usually several cutting tools were in operation at the same time, reducing the time spent in machining each part. This is still the basic concept on which mass- production type lathes are based.

The turret lathe and automatic screw machine in their various forms have been developed and improved with the objectives of producing machined parts more rapidly and accurately at lower cost. On most machines of this type, the power available at the spindle has been greatly increased to take advantage of better cutting tool material. Mechanical power, in electrical, hydraulic, or pneumatic form, has replaced human muscle power for such functions as feeding tools, operating chucks or collets, and feeding bar stock in the machine.

5.2 Lathes and Lathe Components

Of the many standard and special types of turning machines that have been built, the most important, most versatile, and most widely recognized is the engine lathe. The standard engine lathe is not a high production machine, but it can be readily tooled up for many one-piece or short-run jobs. It is also possible to modify the basic machine for many higher production applications. The modern engine lathe provides a wide range of speeds and feeds which allow optimum settings for almost any operation. There have been advances in headstock design to provide greater strength and rigidity. This allows the use of high-horsepower motors so that heavy cuts with carbide tools are practical. To utilize this high power without losing accuracy, new lathes incorporate heavier beds, wider hardened ways, and deeper-sectioned carriages.

A schematic illustration of the components of an engine lathe is shown and described in Figure 5.1.

Headstock: The headstock is the powered end and is always at the operator’s left. This contains the speed changing gears and the revolving, driving spindle, to which any one of several types of work holders is attached. The center of the spindle is hollow so that long bars may be put through it for machining.

Tailstock: The tailstock is non-rotating but on hardened ways, it can be moved, to the left or right, to adjust to the length of the work. It can also be offset for cutting small-angle tapers.

Carriage: The carriage can be moved left or right either by handwheel or power feed. This provides the motion along the Z-axis. During this travel turning cuts are made.

Apron: The apron attached to the front of the carriage, holds most of the control levers. These include the levers which engage and reverse the feed lengthwise (Z-axis)

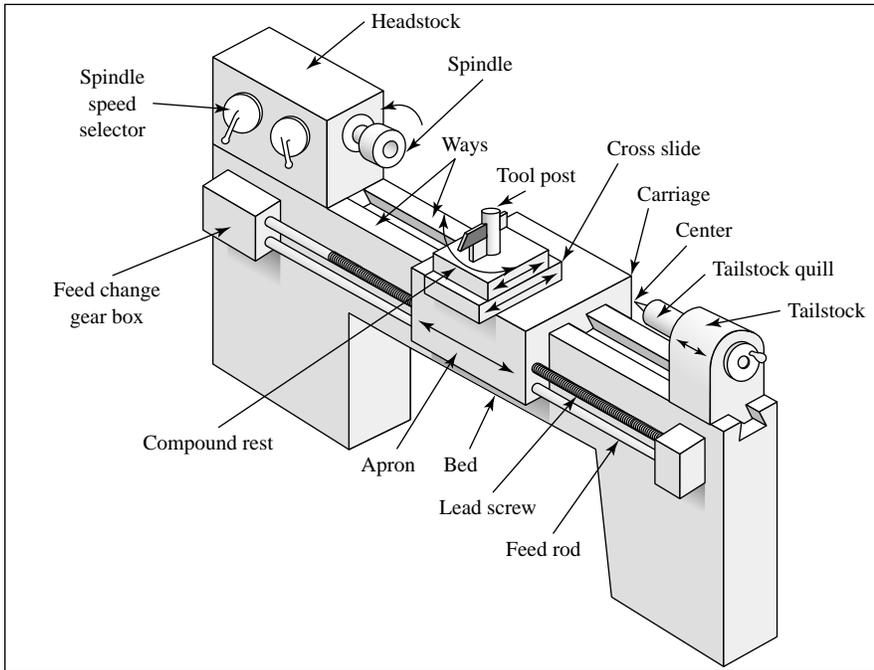


FIGURE 5.1: Schematic illustration of the components of a standard engine lathe.

or crosswise (X-axis) and the lever which engages the threading gears.

Cross Slide: The cross slide is mounted on the carriage and can be moved in and out (X-axis) perpendicular to the carriage motion. This is the part that moves when facing cuts are made with power feed, or at any time a cut must be made 'square' with the Z-axis. This, or the compound, is also used to set the depth of cut when turn-

ing. The cross slide can be moved by its handwheel or by power feed.

Compound Rest: The compound rest, or compound for short, is mounted on the carriage. It can be moved in and out by its handwheel for facing or for setting the depth of cut. It can also be rotated 360 degrees and fed by its handwheel at any angle. The compound does not have any power feed but it always moves longitudinally with the cross slide and the carriage.

Tool Post: The tool post is mounted on the compound rest. This can be any of several varieties but in its simplest form is

merely a slotted cylinder which can be moved left or right in the T-slot in the compound and clamped in place. It can also be rotated so as to present the cutter to the work at whatever angle is best for the job.

Bed: The bed of the lathe is its 'backbone'. It must be rigid enough to resist deflection in any direction under load. The bed is made of cast iron or a steel weldment, in a box or I-beam shape, and is supported on legs, a cabinet, or a bench.

Ways: The ways of the lathe are the flat or V-shaped surfaces on which the carriage and the tailstock are moved left and right. Each has its separate pair of ways, often one flat surface, for stability, and one V-way for guidance in a perfectly straight line. These ways are hardened and scraped or ground to close tolerances. The basic accuracy of movement of the carriage depends on the ways. A typical Toolroom Engine Lathe is shown in Figure 5.2.

Size: The size of a lathe is specified by two or three dimensions:

- The largest diameter workpiece which will clear the bed of the lathe. The center is the headstock spindle center.
- The largest diameter workpiece which will clear the cross slide is sometimes also specified.
- The longest workpiece which can be held on centers between the headstock and the tailstock.

A larger, more sophisticated lathe is shown in Figure 5.3.

A large 40" lathe with a steady rest is shown in Figure 5.4.

5.3 Turret Lathe

The standard engine lathe is versatile,

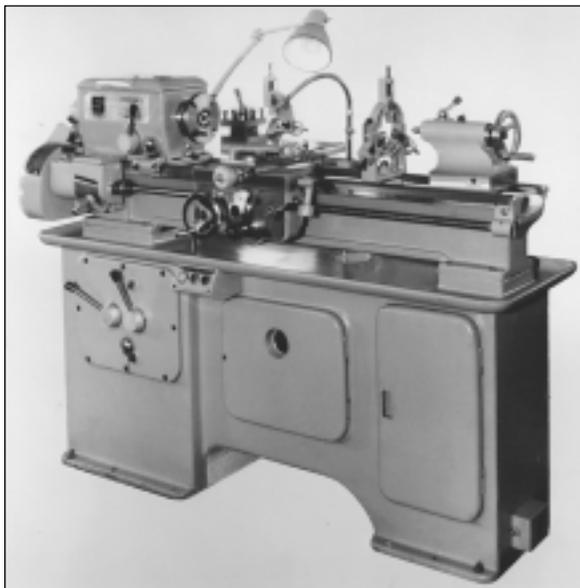


FIGURE 5.2: A typical toolroom engine lathe with face plate, square turret, follower, and steady rest. (Courtesy Summit Machine Tool Manufacturing Corp.)

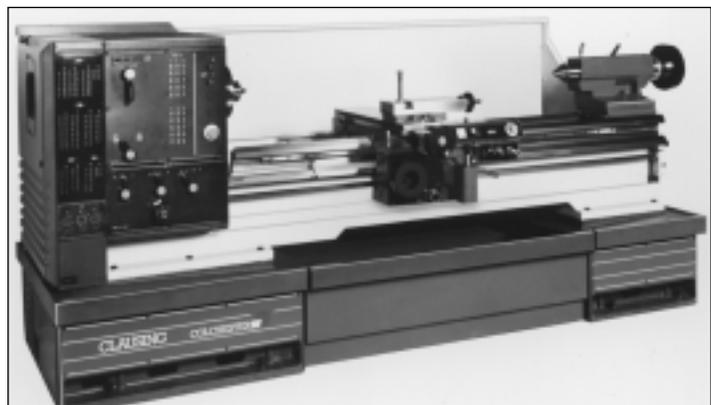


FIGURE 5.3: A more sophisticated 18-inch variable speed engine lathe permits optimal cutting speed selection. (Courtesy Clausing Industries, Inc.)

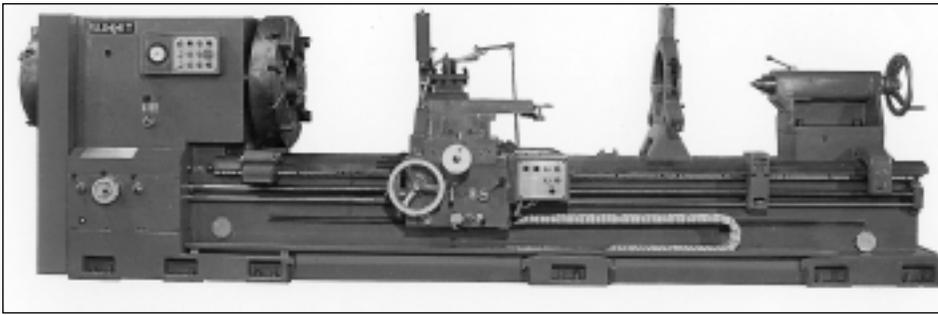


FIGURE 5.4: 40 inch lathe with steady rest is used to machine large cylindrical parts. (Courtesy Summit Machine Tool Manufacturing Corporation)

but it is not a high production machine. When production requirements are high, more automated turning machines must be used.

The turret lathe represents the first step from the engine lathe toward the high production turning machines. The turret lathe is similar to the engine lathe except that tool-holding turrets replace the tailstock and the tool post compound assembly. These machines possess special features that adapt them to production. The ‘skill of the worker’ is built into these machines, making it possible for inexperienced operators to reproduce identical parts. In contrast, the engine lathe requires a skilled operator and requires more time to produce parts that are dimensionally the same.

The principal characteristic of turret lathes is that the tools for consecutive operations are set up for use in the proper sequence. Although skill is required to set and adjust the tools properly, once they are correct, less skill is required to operate the turret lathe. Many parts can be produced before adjustments are necessary. These machines are normally used for small to medium sized production runs where the engine lathe is too slow but the additional production rate desired does not warrant a special machine.

A schematic illustration of the components of a turret lathe is shown in Figure 5.5.

Square and Hex Turrets: A square turret is mounted on the top of the cross slide and is capable of holding four tools. If several different tools are required, they are set up in sequence and can be quickly indexed and locked in correct working position. So that cuts can be duplicated, the slide is provided with

positive stops or feed trips. Likewise, the longitudinal position of the entire assembly may be controlled by positive stops on the left side of the apron. Cuts may be taken with square turret tools and with tools mounted on the hexagon turret simultaneously.

An outstanding feature is the turret in place of the tailstock. This turret mounted on either the sliding ram or the saddle, or on the back of the structure, carries anywhere from 4 to 18 tool stations. The tools are preset for the various operations. The tools are mounted in proper sequence on the various faces of the turret so that as the turret indexes between machining operations, the proper tools are engaged into position. For each tool there is a stop screw or electric/electronic transducer, which controls the distance the tool will feed and cut. When this distance is reached, an automatic trip lever stops further

movement of the tool by disengaging the drive clutch.

Like the engine lathe, the modern turret lathe provides fast spindle speeds, wide speed and feed ranges, high power, and great rigidity. The machine is operated in the high end of its speed range more than the engine lathe is, partly because the tools placed in the turret often work on small diameters on the workpiece, but also because the operator is more production conscious.

5.3.1 Horizontal Turret Lathes:

Horizontal turret lathes are made in two general designs and are known as the ram and saddle types. The ram-type turret lathe shown in Figure 5.6a has the turret mounted on a slide or ram which moves back and forth on a saddle clamped to the lathe bed. The saddle-type turret lathe shown in Figure 5.6b has the turret mounted directly on a saddle which moves back and forth with the turret.

5.3.2 Vertical Turret Lathes: A vertical turret lathe resembles a vertical boring mill, but it has the characteristic turret arrangement for holding the tools. It consists of a rotating chuck or table in the horizontal position with the turret mounted above on a cross rail. In addition, there is at least one side head provided with a square turret for holding tools. All tools mounted on the turret or

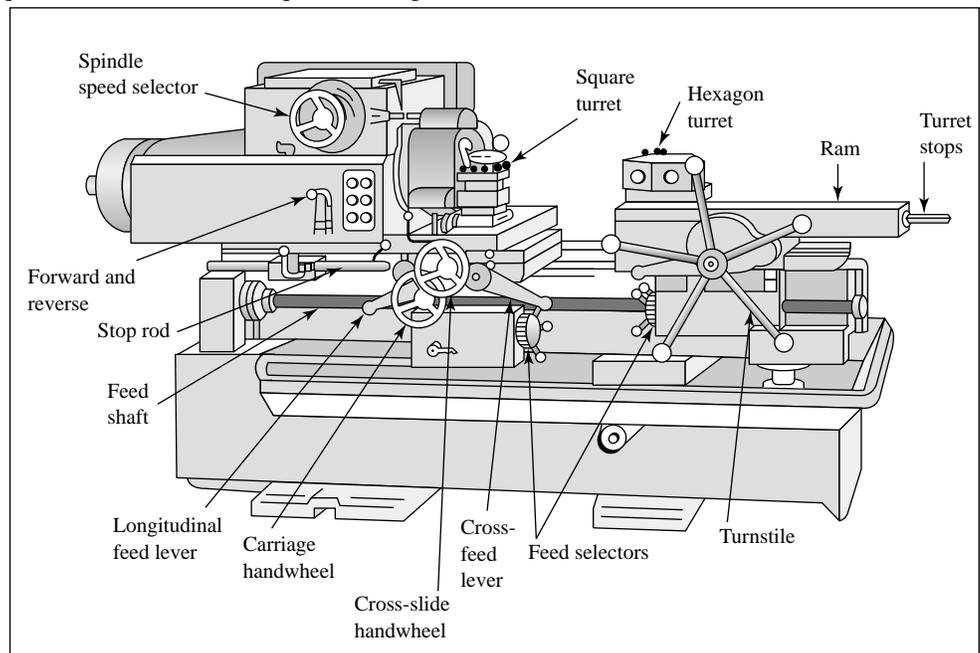


FIGURE 5.5: Schematic illustration of the components of a turret lathe.

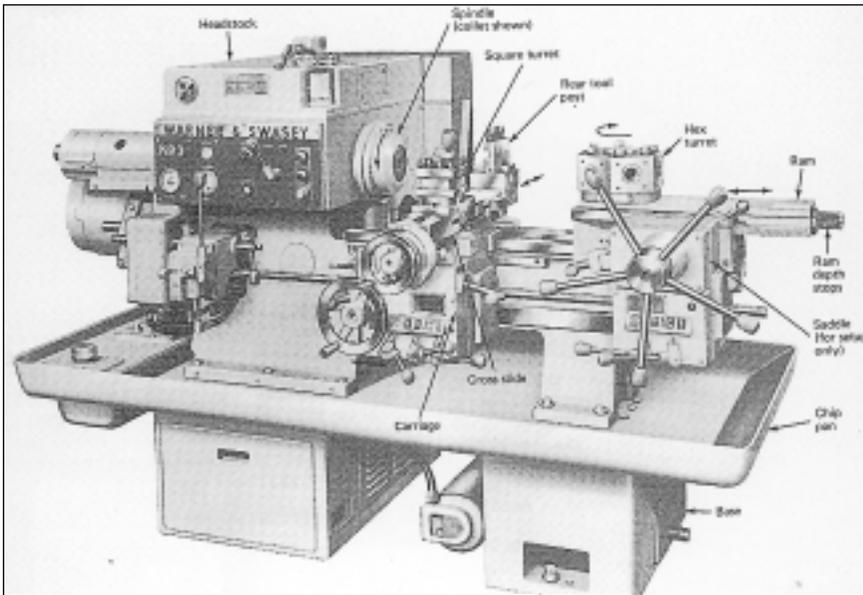


FIGURE 5.6a: Ram-type horizontal turret lathe has the turret mounted on a slide or ram. (Courtesy: National Acme Co., Div. DeVlieg-Bullard Inc.)

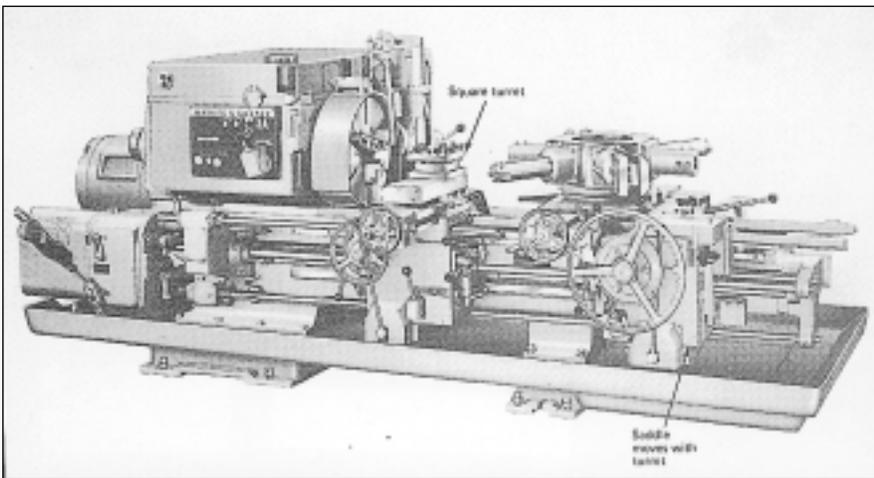
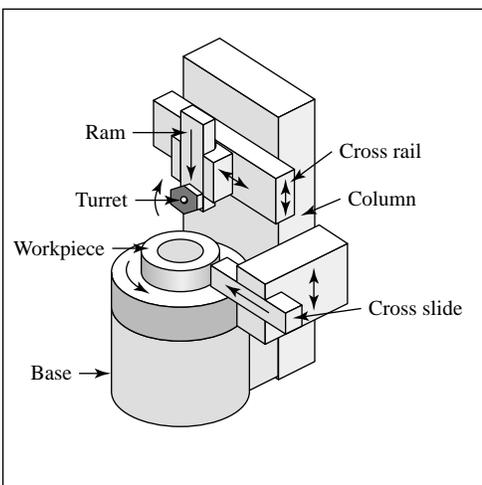


FIGURE 5.6b: Saddle-type turret lathe has the turret mounted directly on the saddle. (Courtesy National Acme Co., Div. DeVlieg-Bullard Inc.)



side head have their respective stops set so that the length of cuts can be the same in successive machining cycles. It is, in effect, the same as a turret lathe standing on the headstock end, and it has all the features necessary for the production of duplicate parts. This machine was developed to facilitate mounting, holding, and machining of large diameter heavy parts. Only chucking work is done on this kind of machine.

FIGURE 5.7: Vertical turning lathes are used for machining large-diameter and heavy parts.

A vertical turret lathe, shown in Figure 5.7, is provided with two cutter heads: the swiveling main turret head and the side head. The turret and side heads function in the same manner as the hexagonal and square turrets on a horizontal lathe. To provide for angle cuts both the ram and turret heads may be swiveled 30 degrees right or left of center.

The machine can be provided with a control that permits automatic operation of each head including rate and direction of feed, change in spindle feed, indexing of turret, starting, and stopping. Once a cycle of operations is pre-set and tools are properly adjusted, the operator need only load, unload, and start the machine. Production rate is increased over those manually operated machines, because they operate almost continuously and make changes from one operation to another without hesitation or fatigue. By reducing the handling time, and making the cycle automatic, an operator can attend more than one machine.

The turret lathe normally has a jawed chuck to hold the workpiece; however, a collet may be more suitable when producing parts from bar stock. A turning machine equipped with a collet and a turret is called a screw machine, but it is actually a special turret lathe. The special features of screw machines are aimed primarily at reducing idle time on the parts being machined, thereby increasing productivity.

In Figure 5.8 a vertical turning center is shown machining a heavy part.

5.3.3 Advantages of Turret Lathes:

The difference between the engine and turret lathes is that the turret lathe is adapted to quantity production work, whereas the engine lathe is used primarily for miscellaneous jobbing, toolroom, or single-operation work. The features of a turret lathe that make it a quantity production machine are:

- Tools may be set up in the turret in the proper sequence for the operation.
- Each station is provided with a feed stop or feed trip so that each cut of a tool is the same as its previous cut.
- Multiple cuts can be taken from the same station at the same time, such as two or more turning and/or boring cuts.
- Combined cuts can be made; tools on the cross slide can be used at the same time that tools on the turret are cutting.
- Rigidity in holding work and tools is

built into the machine to permit multiple and combined cuts.

- Turret lathes can also have attachments for taper turning, thread chasing and duplicating, and can be tape controlled.

5.4 Automated Equipment

There are turning machines which allow automatic chucking, indexing, feeding, spindle speed changes, and other work that has to be done by the operator on the engine lathe. These automatic lathes represent a refinement of the turret lathe, and they are particularly suitable for long run, mass production applications.

Automatic lathes may be made up as single-spindle or multiple-spindle machines. Generally, single-spindle machines provide for turning the workpiece, which is held in a collet or chucked on the headstock. Multiple-spindle automatic lathes usually provide means for indexing the workpiece to tools mounted on the various spindles. These tools might include drills, countersinks, boring bars, and other rotating cutters. Both single- and multiple-spindle automatics may be made up with vertical as well as horizontal spindle alignment.

As far as the machining processes on an automatic lathe are concerned, the fundamental considerations are the high speeds desired for good productivity, the economics of the cutting process, and the balancing of speeds on various phases of the operation to obtain the desired rate of wear on each cutting tool.

5.4.1 Single-Spindle Automatic Lathes

The majority of single-spindle automatic lathes are designed to machine workpieces that are located between two centers. Some, however, hold the workpiece in a chuck, collet, or specially designed fixture. Most have horizontal spindles. A conventional single-spindle automatic lathe has six major components: base, bed, and ways; headstock; work spindle; front tool slide; rear tool slide.

The feed rates of the tool slides are controlled by cams, hydraulics, or lead screws. Spindle speeds are changed to suit workpiece diameter/material requirements by means of change gears in the headstock.

A single-spindle automatic lathe is shown in Figure 5.9.

Tooling: Any of the several available workpiece holders that are suitable for the particular application may be used, including chucks, faceplate drives, collets, and specially designed fixtures.

Chucks, where used, should be power operated to avoid the time lost to manually actuate chucks.

Toolholders are normally designed with slots to locate, and clamps to hold individual cutting tools in their required locations. The assembled toolholders are, in turn, keyed and clamped in a specific location on the front and rear tool slides.

It is good practice to provide spare toolholders wherein a set of sharpened tools can be preset and clamped, ready to exchange for a set of dull tools. Setup time can also be saved by having spare toolholders preset with the tools required for the next part to be run.

DeVlieg tooling for a single- or multiple-spindle automatic lathe is shown in Figure 5.10.



FIGURE 5.8: A vertical turning center machining a heavy part. (Courtesy Giddings & Lewis, LLC)

Applications: Axle and transmission shafts, gear blanks, pump drives, and pinions are all particularly well suited for machining on single-spindle automatic lathes. In fact, almost any machinable metal part falling within its size capacity that can be chucked, fixtured, or run between centers is a potential candidate for this machine. Single-spindle automatic lathes perform turning, facing, chamfering, grooving, and forming operations, and are usually used for parts with moderate production rates.



FIGURE 5.9: Single-spindle automatic lathe (Courtesy: National Acme Co., Div. DeVlieg-Bullard Inc.)

5.4.2 Single-Spindle Automatic Screw Machines

Automatic screw machines are the present-day developments of earlier machines whose only function was the production of screws. Modern machines not only retain thread-cutting capabilities but are also capable of performing all turning operations. These machines produce a wide range of parts from bar stock fed through a hollow work spindle. Some machines are arranged to produce parts from coil stock.

Single-spindle automatic screw machines have horizontal hollow spindles aligned with stock feeding tubes. Most are cam controlled but camless versions, sometimes NC or CNC controlled, are more flexible and quickly set up, making them more suitable for shorter production runs. Machines are available in several sizes and have six major components: base, headstock, hollow work spindle, front tool slide, rear tool slide, and turret, as shown in Figure 5.11. A conventional single spindle automatic screw machine is shown in Figure 5.11

The feed rates and motion of tool slides are controlled by cams or hydraulics. Spindle speeds are changed to suit workpiece diameter/material by means of change gears in the machine base. Bar stock is fed automatically to a swing stop, or a turret stop, after each part is completed and cut off. The collet is automatically released during stock advances.

Tooling: Round, square, hex, and other standard-shape collets are available in sizes to suit commercial bar stock sizes. Specials are also made to suit.

Many special tools and toolholders are designed and made for certain applications, but a significant savings of time and money can be realized by the use of the standard tools and holders available. A large selection of standard tools are available from stock.

Applications: Single-spindle automatic screw machines are used to produce an extremely wide range of small parts including shafts, pins, knobs, screws, bolts, and so on, from any machinable metal. Flats and slots can be milled and cross holes drilled. It is normal for one operator to operate several machines, the number depending on the frequency required for reloading bar stock and adjusting or changing tools.

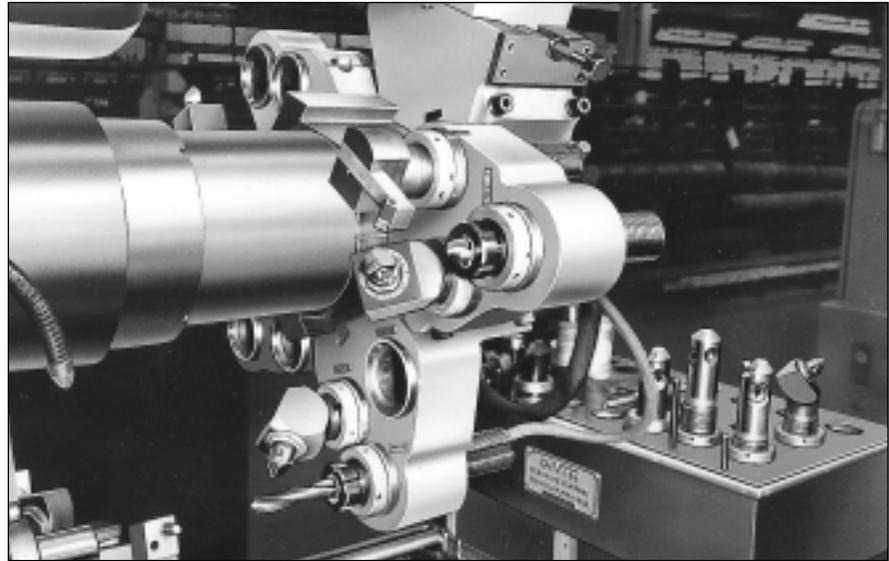


FIGURE 5.10: DeVlieg tooling for single- or multi-spindle automatic lathe (Courtesy: National Acme Co., Div. DeVlieg-Bullard Inc.)

5.4.3 Multiple-Spindle Automatic Bar and Chucking Machines

Conventional multiple-spindle automatic bar and chucking machines have two major advantages over single-spindle automatics - both of which reduce the time required to produce a part:

- The multiple-spindle machine performs work on each of its working stations concurrently; it is also possible to complete a different operation on a part at each position within the same time.
- The maximum time required to complete one piece is the time required for the longest cut, plus index time, and in certain instances the longest cut can be

broken up into increments. For example, a drilled hole that is the longest cut of a certain part may be completed in three or more positions.

Part sizes and complexity of design can be accommodated equally well on multi-spindle or single-spindle machines. Shorter changeover time favors single-spindle machines for short production runs, but the shorter machining time per piece of the multi-spindle machine makes it more economical for long runs.

A schematic diagram of a six-spindle automatic bar machine is shown in Figure 5.12

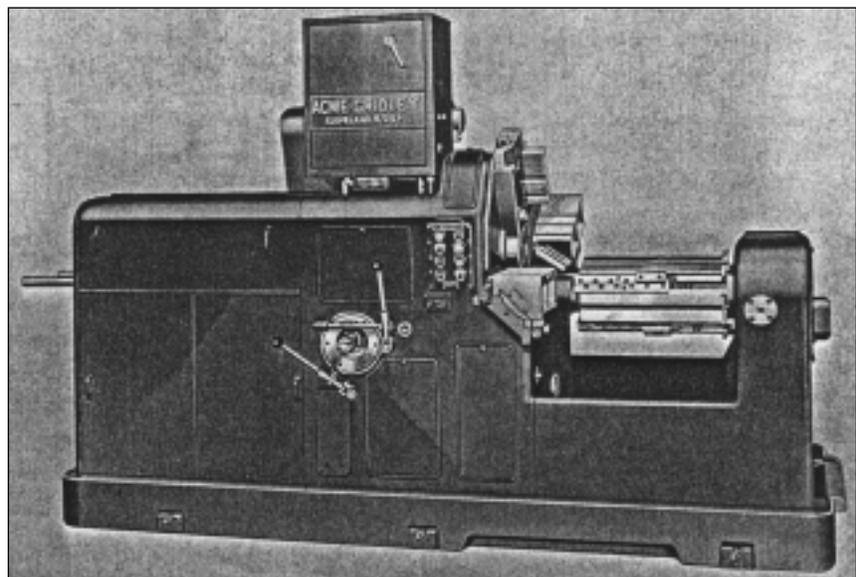


FIGURE 5.11: Conventional single-spindle automatic screw machine. (Courtesy: National Acme Co., Div. DeVlieg-Bullard Inc.)

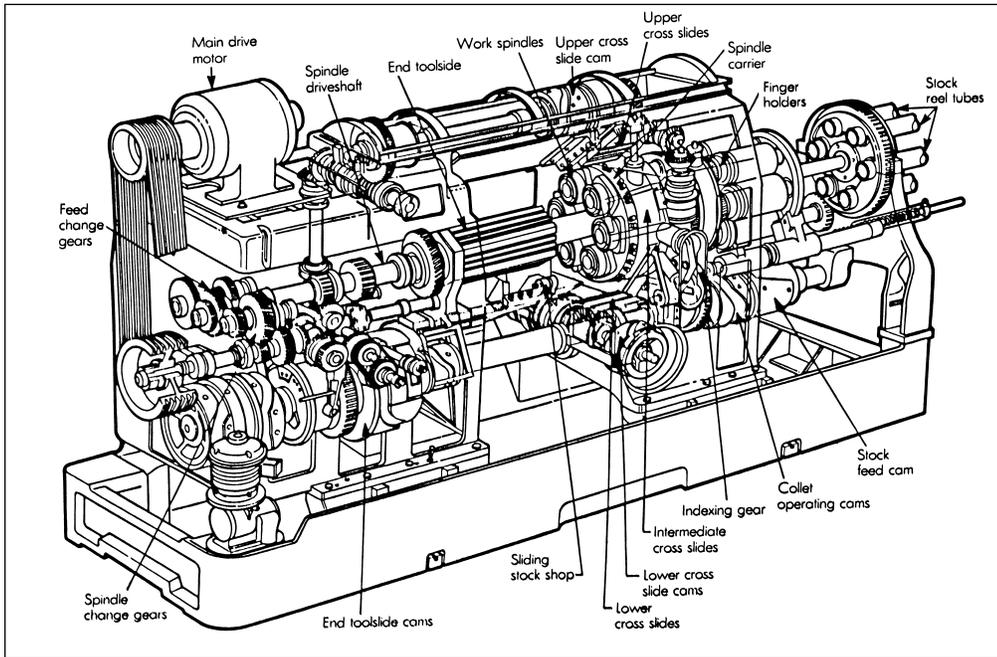


FIGURE 5.12: Schematic diagram of a six-spindle automatic bar machine (Courtesy National Acme Co., Div. DeVlieg-Bullard Inc.)

are more flexible in application. They do not, however, accept bar stock. Some other advantages are that they are convenient to load, operate, and adjust or change tooling.

The machine illustrated in Figure 5.13 has three major components: base and center column, carrier and work spindles, and machining heads. The machine is designed to permit each spindle to operate independently, having independent speeds and feeds. In effect, the machine illustrated can operate as seven individual machines all loaded and unloaded at a common station.

Machines with dual spindles and multiple-tool machining heads are available, permitting duplicate setups, or first and second

5.4.4 Multiple-Spindle Vertical Automatic Chucking Machines

Multiple-spindle vertical automatic

chucking machines are manufactured by several machine tool builders in several sizes and models ranging from 4 to 8 spindles. One maker supplies a 16-spindle machine that is, in reality double spindles for each position of an 8-spindle machine.

These machines use less floor space than an equivalent horizontal model and

chucking work to be performed (both ends). Double indexing is available and is used with dual-spindle setups. A multiple-spindle vertical automatic chucking machine is shown in Figure 5.13.

5.5 Computer Controlled Lathes

In the most advanced lathes, movement and control of the machine and its components are actuated by computer numerical controls (CNC). These lath-

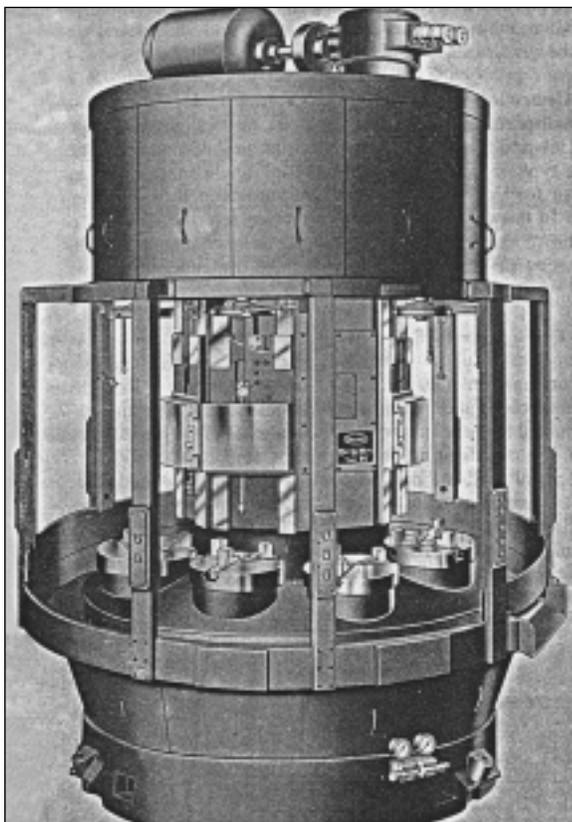


FIGURE 5.13: Multiple-spindle vertical automatic chucking machine. (Courtesy National Acme Co., Div. DeVlieg-Bullard Inc.)



FIGURE 5.14: Multi-station tool holder with four-plus-four tools (Courtesy: Dorian Tool)



FIGURE 5.15: A high production computer controlled Swiss type lathe (Courtesy: Hardinge, Inc.)

es are usually equipped with one or more turrets. Each turret is equipped with a variety of tools and performs several operations on different surfaces of the workpiece. A multi-station tool holder is shown in Figure 5.14.

These machines are highly automated, the operations are repetitive and maintain the desired accuracy. They are suitable for low to medium volumes of production. A high precision CNC lathe is shown in Figures 5.15.

More sophisticated machining systems, including boring and milling operations, will be discussed in a later chapter.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

Chapter 6

Grooving

& Threading

6.1 Introduction

Grooving and threading are both single point machining operations performed on lathes, automatic lathes, or machining centers. Outside diameter (OD) and face grooving operations are shown in Figures 6.1a and 6.1b respectively. Internal threading or tapping will be discussed in a later chapter.

6.2 Grooving or Recessing Operations

Grooving or recessing operations, sometimes also called necking operations, are often done on workpiece shoulders to ensure the correct fit for mating parts (Fig. 6.2a) When a thread is required to run the full length of the part to a shoulder, a groove is usually machined to allow full travel of the nut. (Fig. 6.2b) Grooving the workpiece prior to cylindrical grinding operations allows the grinding wheel to completely grind the workpiece without touching the shoulder.(Fig. 6.2c)



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>

Prentice Hall: <http://www.prenhall.com>



FIGURE 6.1a: Outside diameter (OD) grooving operation (Courtesy: Valenite Inc.)

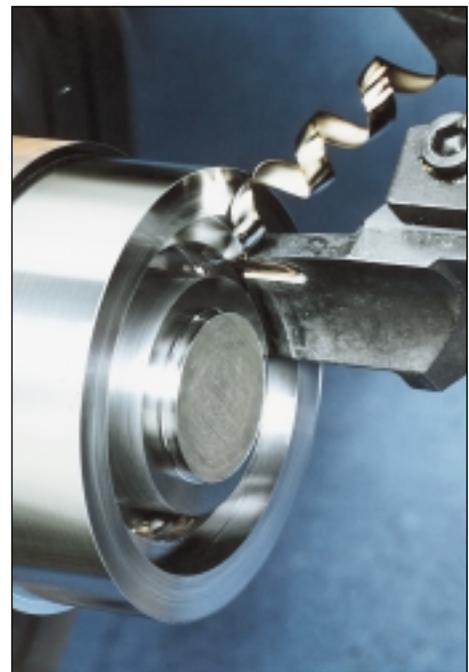


FIGURE 6.1b: Face Grooving operation (Courtesy: Valenite Inc.)

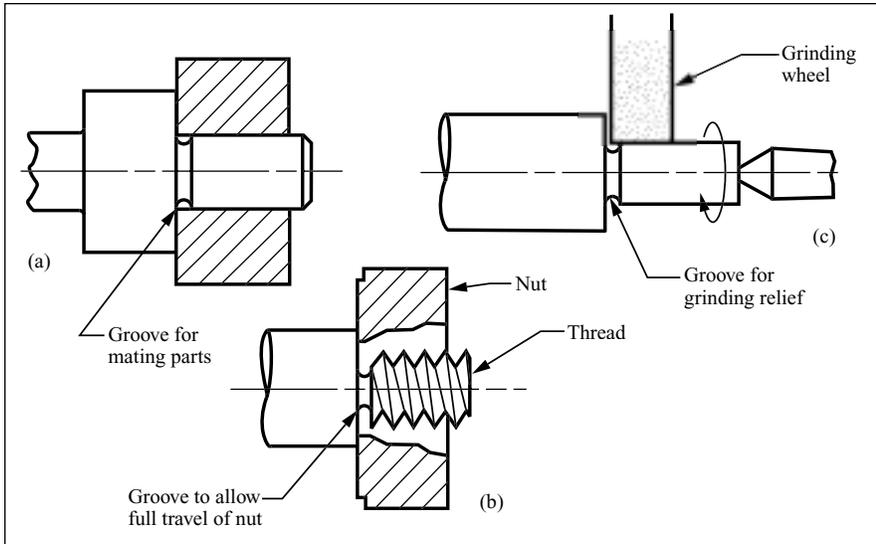


FIGURE 6.2: Grooving operations to provide clearance for a) mating parts; b) full travel of nut; c) grinding relief.

6.2.1 Face Grooving

With face grooving operations the tool is fed axially rather than radially towards the end surface of the workpiece. The tool must be adapted to the radial curve of the groove and the blade is therefore curved. When the machine spindle rotates in a counter-clockwise direction, a right-hand version of the tool is used and a left-hand version is used when the machine spindle rotates clockwise. A face grooving operation is shown in Figure 6.1b.

So that both insert and toolholder fit into the groove, both the outer and inner diameters of the groove must be considered. The diameter measured to the outside of the blade determines the limit for the smallest possible diameter which



FIGURE 6.3: Internal grooving holder with insert (Courtesy: Valenite Inc.)

can be machined, and the diameter measured to the inside of the blade determines the limit for the largest possible groove diameter.

6.2.2 Internal Grooving

The main problem with internal grooving is chip evacuation. There is a very high risk of chip jamming which can result in tool breakage, especially when machining small diameters. The chips have to be removed from the groove then change direction 90 degrees and pass the side of the toolholder to finally be removed from the hole. Introducing intermittent feed into the program is the best way to obtain short chips. An internal grooving holder with insert is shown in Figure 6.3.

Vibration is another common problem associated with internal grooving. Stability is related to the overhang, or how far into the workpiece the groove is to be machined. The risk of vibration is reduced by using the largest toolholder possible. The overhang should not exceed $2 - 2.5 \times$ the diameter. Internal grooving is a critical operation and it is important to choose a tool which optimizes chip evacuation with vibration-free machining.

Grooving tools are usually ground to the dimensions and shape required for a particular job. Most grooving tools are similar in appearance to the cutoff tool, except that the corners are carefully rounded because they reduce the possibility of cracks in the part, especially if

the part is to be heat treated.

6.3 Parting or Cut Off Operations

In parting operations the workpiece rotates while the tool carries out a radial feed movement. As with face turning, the tool is fed from the periphery of the workpiece towards the center and the cutting speed is reduced to zero - but here the similarities end. A typical parting operation is shown in Figure 6.4.

As the cutting tool progresses towards the center, another factor takes effect. As the diameter of the workpiece is reduced, the radial cutting force will cause the material to break before the insert has cut through it. This results in a pip or burr being formed in the center of the workpiece. This pip will always be there after parting, but its size can be reduced by choosing the correct insert geometry, feed rate, and support for the sagging workpiece.

In a parting operation, there is material on both sides of the insert. This means that the tools used are narrow and that the length of the toolholder increases with an increased diameter. Therefore, stability becomes a critical factor.

Since the size of the tool and toolholder must be optimized to meet requirements, only a small surface is present for drawing off heat, and there-



FIGURE 6.4: Parting or cut-off operation (Courtesy: Iscar Metals, Inc.)

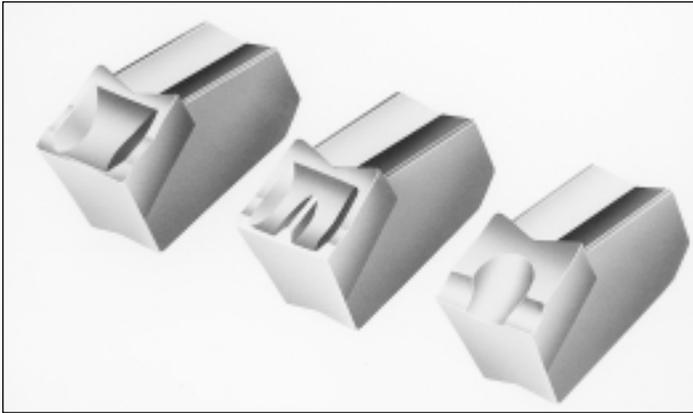


FIGURE 6.5: Chip control grooving and parting inserts (Courtesy: Iscar Metals, Inc.)

fore cutting fluid becomes important. Unfortunately, because of the space restrictions, the supply of cutting fluid is obstructed by the chips. Since chip evacuation is difficult and there is nothing against which to break the chips, the side surfaces can easily be damaged during the operation.

6.3.1 Insert Geometry

At the beginning of a cut the insert will work at a relatively high cutting speed, and must be able to resist plastic deformation. The speed reduces as the tool approaches the center, at which point it becomes zero.

Modern machines can be programmed so that the spindle speed is automatically increased towards the center, so that the cutting speed is kept constant. But the maximum spindle speed of the machine will be reached before the tool reaches the center, and this could result in insert edge build-up. Therefore a tough tool material will be needed to resist edge build-up as the tool gets closer to the center.

Advanced insert geometries are necessary for performing parting and grooving operations in a satisfactory way. A positive rake insert gives lower cutting forces and thus lower pressure on the workpiece, and this will reduce the size of the pip. However, a large positive rake angle means a weaker cutting edge.

The insert can have various lead angles. On straight or neutral inserts the lead angle is zero. This design provides a stronger cutting edge and a better surface finish, while maintaining closer tolerances with respect to perpendicular alignment. With an increased lead angle the axial cutting force increases,

cutting forces but this can cause vibration problems especially when small diameters are machined. In grooving operations a radial displacement of the insert results in an inaccurate groove depth.

6.3.2 Chip Control

With parting and grooving operations, the insert has machined surfaces on both sides of the feed direction. Therefore the chips must be formed in such a way that they are narrower than the groove, otherwise the surfaces can be damaged. In addition, the chips must be formed in such a way that they can be evacuated from the groove without disrupting the machining with long, unwieldy chip coils. Therefore the chips are formed in two directions: bent across their width and rolled together longitudinally to form a spiral spring-shaped chip. Figure 6.5 shows three chip control inserts.

In order to produce this ideal chip shape the insert is usually provided with a chip former as shown in Figure 6.5, which takes into account both the machining conditions and the workpiece material. It is shaped in such a way as to form a bank which the chips can climb against during machining. After a number of revolutions the chips will break automatically. The diameter of the spiral spring chips is influenced by the width of the insert, the height of the bank, the feed, and the workpiece material.

6.3.3 Tool Positioning

As with conventional turning, it is important that the cutting edge be positioned on the same level as the center line. In order to achieve satisfactory

causing the insert to deflect.

With large lead angles the deflection can be so strong that a rounding of the end surfaces occurs and results in a convex or a concave end surface. A reduced lead angle produces larger radial

cutting forces but this can cause vibration problems especially when small diameters are machined. In grooving operations a radial displacement of the insert results in an inaccurate groove depth.

results, a maximum deviation in positioning of only ± 0.004 inch from the center line is acceptable.

As the cutting edge deviates from the center line, the rake angle and the clearance angle will be changed. This change is due to the radius of the workpiece. A clearance angle that is too small may cause the cutting edge to rub against the workpiece. If the cutting edge is positioned too low, the tool will leave material in the center and a pip will be formed.

6.3.4 Operational Stability

With conventional external turning the tool overhang is not affected by the length of the workpiece. The size of the toolholder can be chosen so that it withstands the stresses which arise during the operation. However, with parting and grooving operations, consideration must be given to the depth of insertion and the width of the groove, which means that stability must often be compromised to meet specifications.

To obtain the best possible stability, the overhang should be as small as possible, so a holder for the shortest possible insertion depth should be chosen. Wider inserts can be used in order to improve the stability, but more material is wasted in the form of chips. This can be expensive with large batches and when machining expensive materials.

Vibration can also arise as a result of the deflection of the workpiece. The closer the chuck is to the parting position, the lower the effect of the stresses and the deflection of the workpiece will be. Therefore, if a workpiece has a tendency to vibrate, the machining should be done as close to the chuck as possible.

The risk of vibration must be kept to a minimum in order to obtain acceptable results in quality and tool life. In addition to choosing the best tool and most stable set up, the cutting data must be adapted to minimize the tendency of the tool and workpiece to vibrate.

6.3.5 Toolholder and Insert Selection

Modern parting and grooving cutting tools consist of a toolholder and an indexable insert developed specifically for a particular operation. The majority of inserts produced over the last decade were designed to work with the SELF-GRIP concept. This clamping method

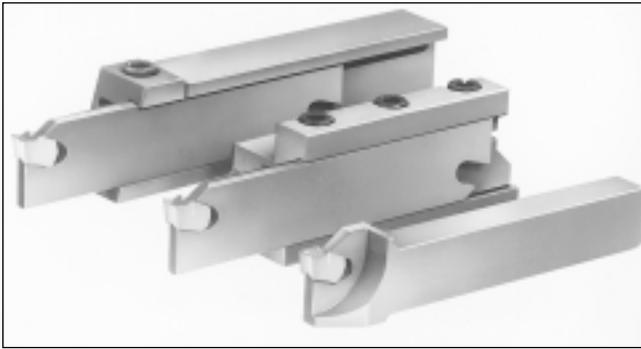


FIGURE 6.6: Self-grip tool holders use no external screw to hold the insert (Courtesy: Iscar Metals, Inc.)

incorporates no external screws or levers to hold the insert in place as shown in Figure 6.6. Instead, it relies on the rotation of the part and tool pressure to keep the insert seated in a wedge-style pocket. The inserts designed for this type of holder are usually single ended and their geometry permits unlimited depth of cut.

With double-ended inserts, also known as “dogbones”, the depth of cut is limited by the second cutting edge as shown in Figure 6.7. Dogbone inserts traditionally can only cut as deep as the overall length of the insert. Once the depth is reached, the trailing edge will begin to rub inside the groove that the tool is creating. In addition, dogbone inserts usually are secured by a screw-top clamp, which also limits the depth of cut as again shown in Figure 6.7.

Coatings for grooving and parting inserts vary from supplier to supplier. But titanium carbon nitride applied by the PVD process has practically become the industry standard for lower cutting speeds and tougher applications. And



FIGURE 6.7: Dogbone insert tool holders have limited depth of cut (Courtesy: Iscar Metals, Inc.)

when conditions are stable and higher cutting speeds are required, the choice narrows to titanium-aluminum-nitride coatings applied by the PVD process, or the newer medium-temperature CVD process. The reason is because TiAlN acts as a good heat barrier for the carbide substrate and

can handle elevated temperatures. Figure 6.8 shows a variety of grooving and parting tool holders with coated indexable inserts..

6.4 Grooving and Parting Recommendations

The ability to efficiently cut off workpieces and blanks in lathes has always been important in getting the job completed. Even in special purpose cutoff machines, a good parting tool is at the heart of the operation. Today’s modern indexable insert parting and grooving tools provide the same productivity levels as modern turning tools.

In parting operations, the objective is to separate one part of the workpiece from the other as efficiently and reliably as possible. In grooving operations, the principle is the same, although these operations are less sensitive because the grooves are usually not as deep. In grooving, the shape, accuracy and surface finish are the main requirements

that must be met.

Some important hints for applying grooving and parting tools:

- always use plenty of cutting fluid.
- set the center-height of the cutting edge accurately
- make sure toolholder/blade is accurately positioned at 90 degrees to the workpiece axis.
- use toolholder with the shortest possible length of insertion for the operation in question.
- select the largest shank/bar for the tool
- adapt the cutting speed to avoid vibrations
- reduce the feed rate for the final part when parting-off bar material/components.
- for axial grooving, make the first plunging cut at the largest diameter, farthest out on the face, to minimize the risk of chip jamming
- use the smallest possible lead angle for avoiding pips/burrs in parting-off
- when possible, use a toolholder with a strengthening radius between shank and blade.

6.5 Screw Threads and Threading

The screw thread dates back to 250 B.C, when it was invented by Archimedes. For centuries wooden screws, hand-made by skilled craftsmen, were used for wine presses and carpenters’ clamps throughout Europe and Asia. Precision in screw and thread manufacture did not come into being until the screwcutting lathe was invented by Henry Maudslay in 1797.



FIGURE 6.8: Various OD and ID grooving holders. (Courtesy: Iscar Metals, Inc.)

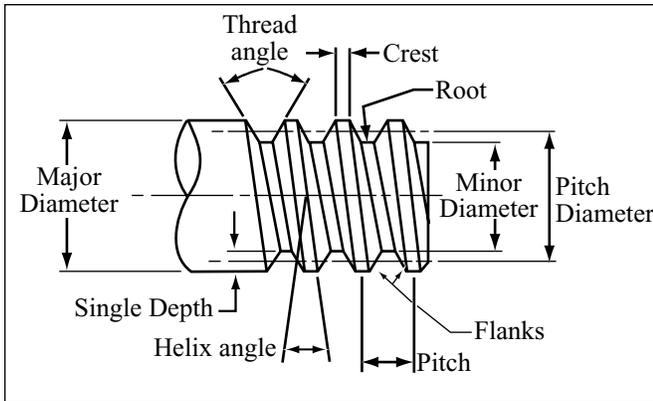


FIGURE 6.9: Screw Thread Nomenclature.

In the early 1800's, Maudslay also began to study the production of uniform and accurate screw threads. Until then no two screws were alike; manufacturers made as many threads per inch on bolts and nuts as suited their own needs. For example, one manufacturer made 10 threads per inch in 1/2 -in. -diameter threaded parts, whereas another made 12 threads, and so forth. During this period the need for thread standards became acute.

Despite many attempts at standardization, it was not until World War I that thread standards were developed. The thread profile was designated the **American National** thread form and was the principal type of thread manufactured in the United States until World War II.

During World War II the United States manufactured military equipment that used the American National thread form, which presented interchangeability problems with machinery made in Canada and Great Britain. Not until after World War II in 1948, did these countries agree upon a **Unified** thread form to provide interchangeability of threaded parts. The Unified thread form is essentially the same as the old American National, except that it has a rounded root and either a rounded or flat crest. The Unified thread form is mechanically interchangeable with the former American National threads of the same diameter and pitch. Today it is the principal thread form manufactured and used by the United States.

6.5.1 Screw Thread Nomenclature

Screw threads have many dimensions. It is important in modern manufacturing to have a working knowledge of screw thread terminology to identify and cal-

culate the dimensions correctly. Screw set nomenclature is shown in Figure 6.9.

The **major diameter** is the largest diameter of the screw thread. On an external thread it is the outside diameter; on an internal thread it is the diameter at the bottom or root of the thread.

The **minor diameter** is the smallest diameter of a screw thread. On an external thread, the minor diameter is at the bottom of the thread; on an internal thread the minor diameter is the diameter located at the crest.

The **pitch diameter** is an imaginary diameter that passes through the threads at the point where the widths of the groove and the thread are equal. The pitch diameter is the most important dimension on a screw thread; it is the basis from which all thread measurements are taken.

The **root** is the bottom surface connecting two sides of a thread. The **crest** is the top surface connecting two sides of a thread. **Pitch** is the linear distance from corresponding points on adjacent threads. The pitch is equal to 1 divided by the total number of threads per inch ($P=1/[\text{no.threads/in.}]$). A screw having a single lead with 16 threads per inch has a pitch equal to 1/16 in., commonly referred to as a "16-pitch thread".

The **lead** is the axial distance a threaded part advances in one complete rotation. On a **single lead** threaded part, the lead is equal to the pitch.

The **depth** is the distance, measured radially, between the crest and the root of a thread. This distance is often called the **depth of thread**.

The **flank** is the side of the thread. **Thread angle** is the angle between the flanks of the thread. For example, Unified and Metric screw threads have a thread angle of 60 degrees. **Helix** is the curved groove formed around a cylinder or inside a hole.

A **right-handed thread** is a screw thread that requires right-hand or clockwise rotation to tighten it. A **left-handed thread** is a screw thread that requires left-hand or counterclockwise rotation

to tighten it. **Thread fit** is the range of tightness or looseness between external and internal mating threads. **Thread series** are groups of diameter and pitch combinations that are distinguished from each other by the number of threads per inch applied to a specific diameter. The two common thread series used in industry are the **coarse** and **fine** series, specified as UNC and UNF.

6.5.2 Unified Thread Form

The Unified screw thread has a 60 degree thread angle with a rounded root and a crest that is **flat** or **rounded**. As mentioned earlier, this is the principal thread form used for screw thread fasteners used in the United States. The Unified screw thread system includes six main thread series:

1. Unified Coarse (UNC)
2. Unified Fine (UNF)
3. Unified Extra-Fine (UNEF)
4. Unified 8-Pitch (8 UN)
5. Unified 12-Pitch (12 UN)
6. Unified 16-Pitch (16 UN)

The **coarse-thread series** (UNC) is one of the more commonly used series on nuts, bolts, and screws. It is used when lower-tensile-strength materials (aluminum, cast iron, brass, plastics, etc.) require threaded parts. Coarse threads have a greater depth of thread and are required on these types of materials to prevent stripping the internal threads.

The **fine-thread series** (UNF) is used on higher-tensile-strength materials where coarse threads are not required. Because they have more threads per inch, they are also used where maximum length of engagement between the external and internal threads is needed.

The **extra-fine thread series** (UNEF) is used when even greater lengths of engagement are required in thinner materials. **Eight, 12 and 16-pitch** threads are used on larger-diameter threads for special applications. The 8-pitch is generally regarded as a coarse thread for larger diameters, 12 pitch is the fine series, and 16 is the extra-fine thread used on the larger-diameter threads.

The relationship between the **pitch diameter** or **major diameter** determines the helix angle of that thread. For example, a 12-pitch (12 UN) thread with a 1.250-in. major diameter will have a greater helix angle than a 12-



FIGURE 6.10: Grooving and threading tool holders (Courtesy: Valenite Inc.)

pitch thread with a 2.0-in. major diameter. Generally speaking, the lower the helix angle, the greater the tensile stress applied to the bolt for a given torque applied to the nut. The fastener with a lower helix angle will also resist vibration and loosening more effectively.

A grooving and threading holder is shown in Figure 6.10 and various grooving and threading inserts are shown in Figure 6.11

6.5.3 Acme Screw Threads

Acme screw threads are manufactured for assemblies that require the carrying of heavy loads. They are used for transmitting motion in all types of machine tools, jacks, large C-clamps, and vises. The Acme thread form has a 29 degree thread angle and a large flat at the crest and root (see Fig. 6.12).

Acme screw threads were designed to replace the Square thread, which is difficult to manufacture.

There are three classes of Acme threads (2G, 3G, and 4G), each having

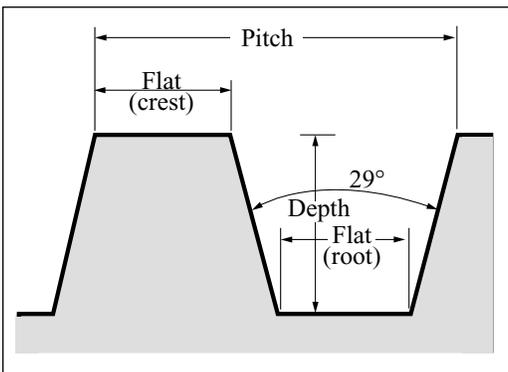


FIGURE 6.12: General purpose Acme screw thread



FIGURE 6.11: Various grooving and threading inserts (Courtesy: Valenite Inc.)

clearance on all diameters to provide for free movement.

Class 2G threads are used on most assemblies. Classes 3G and 4G are used when less backlash or looseness is permissible, such as on the lead screw of a lathe or the table screw of a milling machine.

6.5.4 Tapered Pipe Threads

Pipe threads, usually designated NPT (National Pipe Taper) are tapered threads used for sealing threaded joints such as water and air pipes. Most pipe threads have a slight taper (3/4-in./ft) and are cut using special pipe taps and dies. Pipe threads can also be machined using the taper attachment on an engine lathe.

6.6 Thread Turning

Development of threading tools has come a long way since the days of high speed tool-bits and tips ground to shape, which were then slowly fed along by the lathe lead screw. Most of today's threading is performed by indexable insert tools as part of a very rapid CNC process. What used to be a relatively difficult and time-consuming part of machining is now standard procedure as with any other operation. A typical part that requires a thread is routinely machined with fixed cycles of numerical control and a variety of other machine mechanisms and using tools which have the right thread shape. An ID and OD threading operation with coated indexable inserts is shown in Figure 6.13.

The principle of single point thread cutting is the feed movement of the tool in relation to

the workpiece rotation. The point generates the typical spiral groove that makes up the screw thread with a certain pitch. Basically, threading is a well-coordinated turning operation with a form-tool. During the feed passes, the tool is moved longitudinally along the workpiece and then withdrawn and moved back to the starting position for the next pass along the same thread groove.

The feed rate is a key factor that has to coincide with the pitch of the thread. The coordination is obtained by various means, depending on the type of machine; lead screw, cam or numerical control (usually handled as a sub-routine in CNC). The shape of the groove produced is determined by the shape of

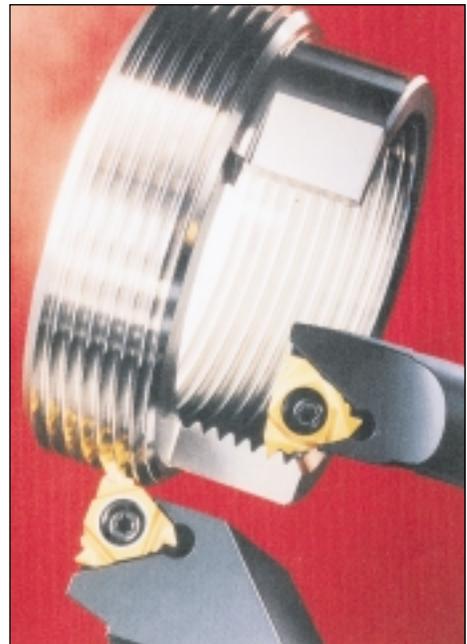


FIGURE 6.13: OD and ID threading Operation (Courtesy: Sandvik Coromant Corp.)

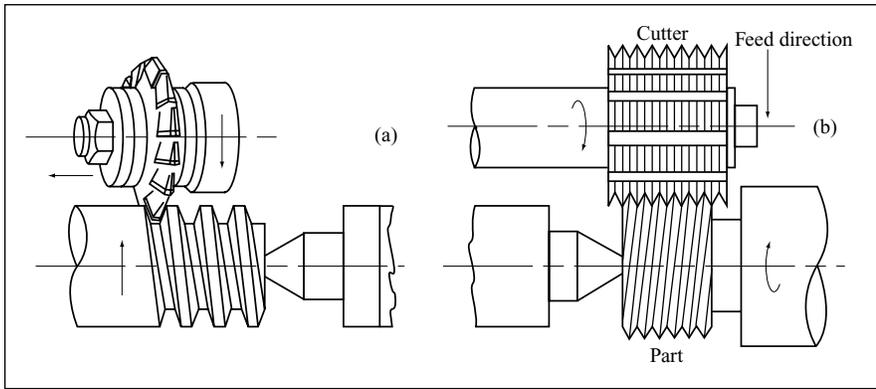


FIGURE 6.14: External thread milling with: a) single-row and b) multiple-row tooth cutter

the insert point, and the feed rate is considerably higher than for ordinary turning operations.

The relatively small 60 percent point angle of the tool makes the cutting edge susceptible to the forces and stresses of metal cutting. To counter this, a long established method has been to use the thread depth to determine the cutting depth, and to avoid machining in one pass. Instead, the depth is machined in several passes. The cutting tool opens up the thread groove by cutting deeper and deeper, usually by making 5 to 16 passes, depending on the thread pitch. As each pass is made, more and more material is removed per cut as a larger part of the edge is engaged. For this reason, the depth of cut is reduced successively as the passes are made.

It is best to have radial in-feeds which decrease successively as the passes are performed. The number of in-feed passes must be balanced to provide the edge with sufficient but not excessive cut into the workpiece. Too much cutting force with insufficient cutting depth leads to premature tool wear.

6.6.1 Left and Right-Hand Threads

The difference in direction between left and right-hand threads does not affect the thread profile; it does, however, have some effect on the choice and combination of tools. The method of cutting the thread depends on the workpiece design. Working towards the chuck is the most common method, even though working away from the chuck is in many cases also satisfactory.

The advantage in using right-hand tools for right-hand threads and left-hand tools for left-hand threads, is that the holder is designed to give maximum support to the insert. But under normal

cutting conditions this order is not critical. It is vital, however, that inserts always be used with holders of the same hand.

6.6.2 Toolholders and Insert Selection

Compared to conventional turning, the tool and machining parameters of threading are not so flexible. This is mainly because the feed is related to the pitch, the cutting depth is divided into passes, and the cutting speed is limited because of the pointed cutting edge.

Indexable inserts are available for external and internal threading. The inserts for internal threading are mirror images of the corresponding external inserts. Both external and internal inserts are available in right and left-hand versions. Since tolerances and cutting geometries differ between external and internal inserts, it is important that they should not be confused.

6.6.3 Coated Threading Inserts

The development within thread turning tools has been considerable during the past thirty years, since the introduction of the first flat inserts with a loose chip former clamped on top of the inserts. Today's modern inserts have done away with most of the possible problems that can arise with conventional threading inserts. This has made threading more closely resemble a turning operation.

The multi-purpose PVD

coated inserts allow a wider range of cutting speeds between the area characterized by built-up edge formation at lower speeds and plastic deformation at higher speeds. Threading involves many short cutting sequences and often relatively low cutting speeds throughout machining. Of major importance in threading is the ability of the cutting tool to keep the built-up edge tendency to an absolute minimum, or prevent it entirely, depending upon the workpiece material. A built-up edge will cause poor surface finish and eventually lead to edge breakdown and tool failure.

6.7 Thread Milling

Thread milling has been an established method of manufacturing accurate screw threads for many years. Long screws, such as lead screws on lathes and multiple start threads, are often manufactured by milling.

Milling a screw thread is done with either a single- or multiple-lead milling cutter. The rotating cutter is fed into the work to the required depth. The work is then rotated and fed longitudinally at a rate that will produce the proper lead on the part (Fig. 6.14). Any class of fit or thread form can be manufactured by the thread milling process.

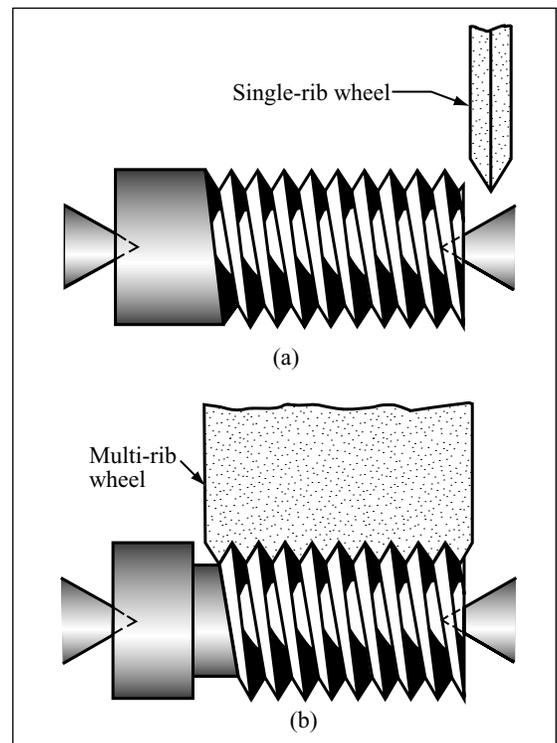


FIGURE 6.15: Shown are: a) single and b) multiple rib thread grinding wheels

6.8 Thread Grinding

Grinding a screw thread generally is done when the hardness of the material makes cutting a thread with a die or single point tool impractical. Grinding threads also results in greater accuracy and in superior surface finishes compared to what can be achieved with other thread-cutting operations. Taps, thread chasers, thread gages, and micrometer spindles all use ground threads.

Ground threads are produced by thread-grinding machines. A thread-grinding machine closely resembles a cylindrical grinder in appearance. It incorporates a precision lead screw to produce the correct pitch or lead on the threaded part. Thread-grinding machines also have a means of dressing or truing the cutting periphery of the grinding wheel so it will produce a precise thread form on the part. Grinding wheels used in producing ground

threads are single-or multiple-rib. (Fig. 6.15) Single-rib types are used for grinding longer threads and feed longitudinally for the required length of thread. The multiple-rib type of grinding wheel is generally used for forming short threads. This type of wheel is “plunged” into the workpiece to produce the thread.

Internal threading or tapping will be discussed in a later chapter as part of hole-making processes.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

Chapter 7

Shaping

& Planing

7.1 Introduction

Both the shaper and the planer are single point tools and cut only in straight lines. They both make the same types of cuts.

The shaper handles relatively small work. The planer handles work weighing up to several tons. The cutting stroke of the shaper is made by moving the tool bit attached to the ram. The cutting stroke of the planer is achieved by moving the work past a stationary tool bit.

The types of cuts which can be made with either machine are shown in Figure 7.1. Both the shaper and the planer usually cut only in one direction, so that the return stroke is lost time. However, the return stroke is made at up to twice the speed of the cutting stroke.

7.2 The Shaper

The shaper is a relatively simple machine. It is used fairly often in the toolroom or for machining one or two pieces for prototype work. Tooling is simple, and shapers do not always require operator attention while cutting. The horizontal shaper is the most common type, and its principal components are shown in Figure 7.2, and described as follows:

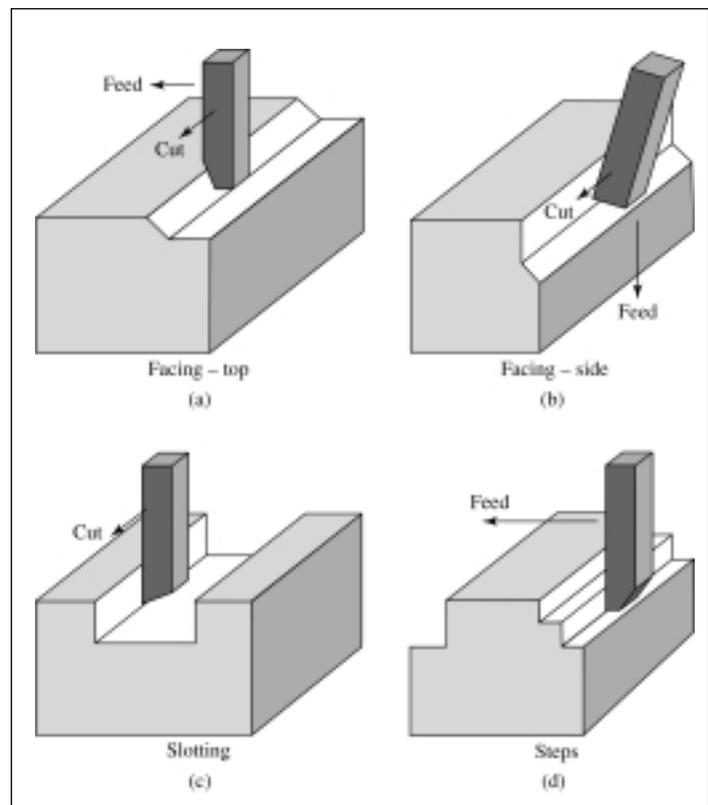


FIGURE 7.1: Typical cuts made by both shapers and planers.



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>

Prentice Hall: <http://www.prenhall.com>

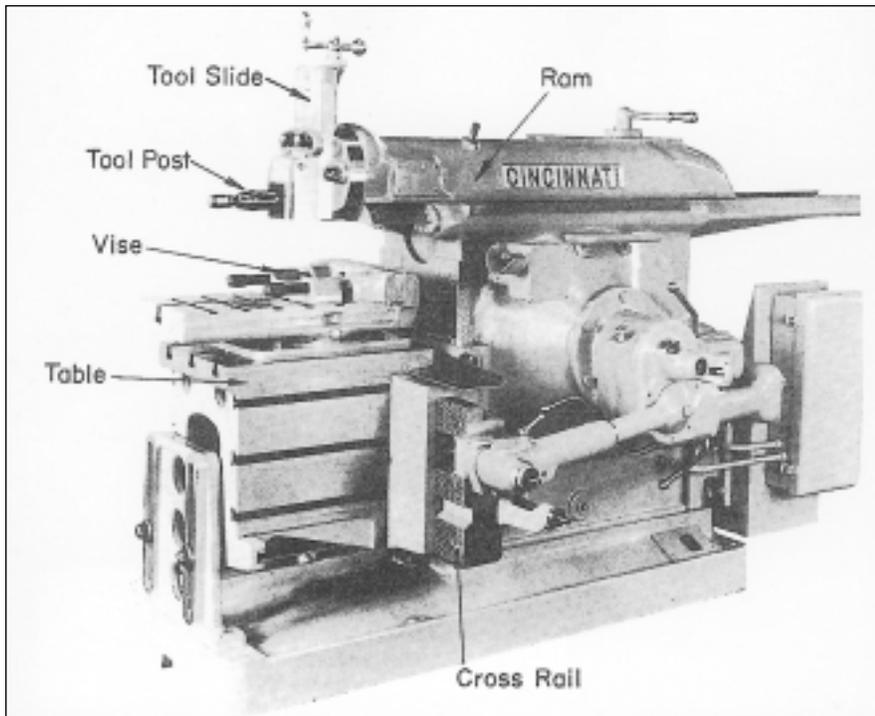


FIGURE 7.2: The horizontal shaper is the most common type; its principal components are shown here (Courtesy: Cincinnati Machine)

Ram: The ram slides back and forth in dovetail or square ways to transmit power to the cutter. The starting point and the length of the stroke can be adjusted.

Toolhead: The toolhead is fastened to the ram on a circular plate so that it can be rotated for making angular cuts. The toolhead can also be moved up or down by its hand crank for precise depth adjustments.

Attached to the toolhead is the tool-holding section. This has a tool post very similar to that used on the engine lathe. The block holding the tool post can be rotated a few degrees so that the cutter may be properly positioned in the cut.

Clapper Box: The clapper box is needed because the cutter drags over the work on the return stroke. The clapper box is hinged so that the cutting tool will not dig in. Often this clapper box is automatically raised by mechanical, air, or hydraulic action.

Table: The table is moved left and right, usually by hand, to position the work under the cutter when setting up. Then, either by hand or more often automatically, the table is moved sideways to feed the work under the cutter at the end or beginning of each stroke.

Saddle: The saddle moves up and down (Y axis), usually manually, to set

the rough position of the depth of cut. Final depth can be set by the hand crank on the tool head.

Column: The column supports the ram and the rails for the saddle. The mechanism for moving the ram and table is housed inside the column.

Toolholders: Toolholders are the same as the ones used on an engine lathe, though often larger in size. The cutter is sharpened with rake and clearance angles similar to lathe tools, though the angles are smaller because the work surface is usually flat. These cutters are fastened into the toolholder, just as in the lathe, but in a vertical plane.

Work Holding: Work holding is frequently done in a vise. The vise is specially designed for use in shapers and has long ways which allow the jaws to open up to 14 inches or more, therefore quite large work pieces can be held. The vise may also have a swivel base so that cuts may be made at an angle. Work which, due to size or shape, cannot be held in the vise, is clamped directly to the shaper table in much the same way as parts are secured on milling machine tables.

Shaper Size: The size of a shaper is the maximum length of stroke which it can take. Horizontal shapers are most often made with strokes from 16 to 24

inches long, though some smaller and larger sizes are available. These shapers use from 2 to 5 HP motors to drive the head and the automatic feed.

Shaper Width: The maximum width which can be cut depends on the available movement of the table. Most shapers have a width capacity equal to or greater than the length of the stroke. The maximum vertical height available is about 12 to 15 inches.

7.2.1 Drive Mechanisms

Shapers are available with either mechanical or hydraulic drive mechanisms. Figures 7.3a and 7.3b show diagrams of both shaper drive mechanisms.

Mechanical Drive

The less expensive shaper, the one most often purchased, uses a mechanical drive. This drive uses a crank mechanism (Fig. 7.3a). The bull gear is driven by a pinion which is connected to the motor shaft through a gear box with four, eight or more speeds available. The RPM of the bull gear becomes the strokes per minute (sometimes abbreviated SPM) of the shaper.

Cutting Speed: The cutting speed of the tool across the work will vary during the stroke as shown by the velocity diagram in Figure 7.3a. The maximum is at the center of the stroke. However, if the cutting speed chosen is somewhat on the slow side, the average speed may be used, and computations are greatly simplified.

Although the ratio varies somewhat, several shapers have a linkage using 220 degrees of the cycle for the cutting stroke and 140 degrees for the return stroke. This is close to a 3:2 ratio.

In setting up a mechanically operated shaper, the length of cut (in inches) is known, and the cutting speed (in feet per minute) is selected according to the kind of metal being cut. It is then necessary to compute the strokes per minute since that is how the shaper speed is controlled. Such calculations are beyond the scope of this text.

The stroke per minute available on a shaper will vary according to the size of the shaper. The larger shapers will have lower speeds. A 16 inch shaper may have speeds of 27 to 150 strokes per minute, while a 24 inch shaper will have 10 to 90 strokes per minute speeds available.

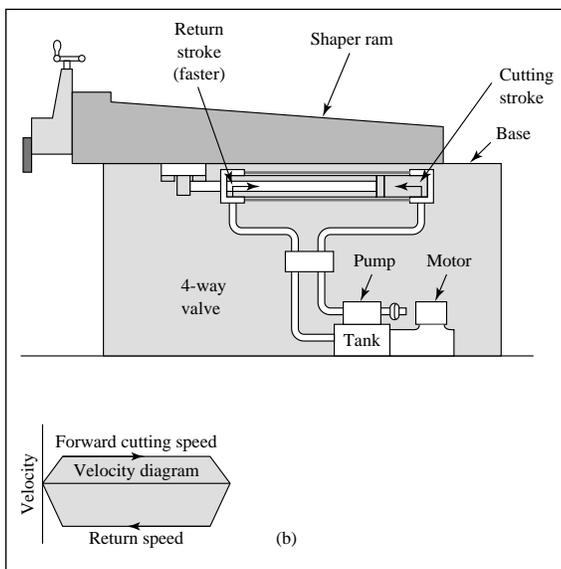
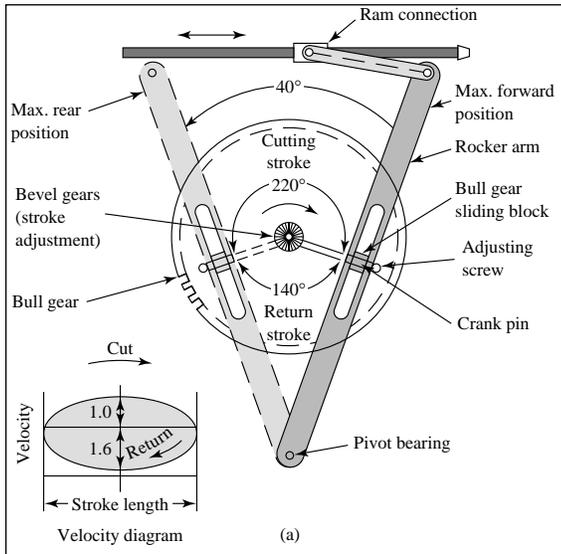


FIGURE 7.3: Shapers are available with either (a) mechanical drive mechanisms or (b) hydraulic drive mechanisms.

Cutting Feed: Feed per stroke on a shaper is comparable to the feed per revolution on a lathe. Coarse feeds for roughing range up to 0.100 inch per stroke (sometimes abbreviated as IPS), and finish cuts from 0.005 to 0.015 inch per stroke. Finish would also depend on the nose radius of the cutting tool.

Hydraulic Drive

The hydraulic shaper (Fig. 7.3b) has the same major parts as the mechanical one, however, the ram is driven by a hydraulic cylinder as shown in the simplified sketch. These shapers use 5 to 10 HP motors.

Cutting Speed and Feed: The cutting speed of the hydraulic shaper is infinitely variable by means of

hydraulic controls, as is the cross feed. The reverse stroke is made faster than the power stroke because of the smaller area in the return side of the cylinder, if a constant volume pump is used. Another method is to have the rate of fluid flow increased to speed up the return stroke.

Speed and feed on a hydraulic shaper are often controlled by simple dials. Speed is read directly in feet per minute and feed is read directly in decimal inches. The cutting speed remains nearly constant through the full stroke.

7.2.2 Vertical Shapers

The vertical shaper, sometimes called a slotter, has a vertical ram, with table and saddle similar to the horizontal shaper. If a rotary table is mounted on the regular table, a number of slots can be made at quite accurately spaced intervals. This machine can work either outside or inside a part, provided that the interior opening is larger than the tool

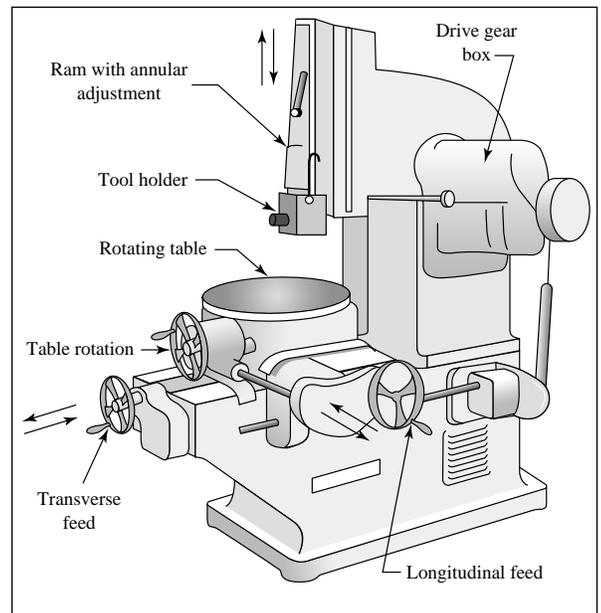


FIGURE 7.4: Schematic illustration of a vertical shaper, also called a slotter.

head. A schematic illustration of a vertical shaper is shown in Figure 7.4.

7.3 The Planer

A planer makes the same types of cuts as a shaper. However, it is a production-type machine for certain types of work. It can machine any flat or angular surface, including grooves and slots, in medium and large sized workpieces (see Fig. 7.1). Typical work would be machine beds and columns, marine diesel engine blocks, and bending plates for sheet metal work. These parts are usually large iron castings or

steel weldments and may weigh a few hundred pounds or several tons.

The most frequently used type of planer is the double-housing planer, shown in Figure 7.5, with the following components:

Frame: The frame is basically two heavy columns fastened together at the top with a large bracing section and fastened at the bottom to the machine bed. This creates a very strong, rigid structure which will handle heavy loads without deflection.

Crossrail: The crossrail is also a heavy box, or similar construction. It slides up and down on V- or flat ways, controlled by hand or by power-operated screws. These crossrails are so heavy that they are counterweighted,

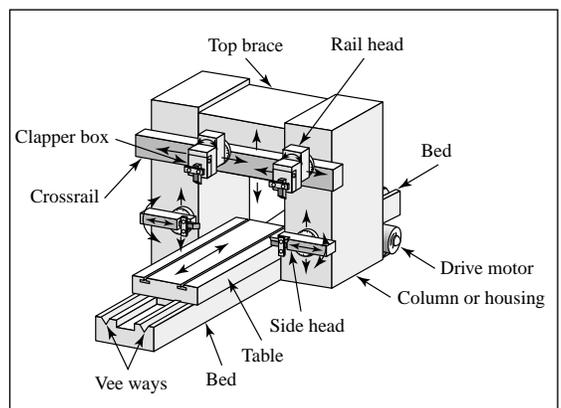


FIGURE 7.5: Schematic illustration of a double-housing planer.

with either cast iron weights or hydraulic cylinders, in order that they may be moved easily and positioned accurately. After being positioned, they are clamped in place.

Railheads: The two railheads can be moved left or right across the crossrail, each controlled by a separate lead screw, which can be turned by hand but usually by power feed. The railhead can be rotated, and vertically adjusted for depth of cut, the same as the shaper heads. They also have a clapper box (often with power lift) like the shaper.

Sideheads: The sideheads are independently moved up or down by hand or by power feed and can also be rotated and moved in or out for depth of cut.

Table: The table is a heavy casting which carries the work past the cutting heads. It runs on V- or flat ways. The table is driven either by a very long hydraulic cylinder or by a pinion gear driving a rack which is fastened under the center of the table. The motor driving the pinion gear is the reversible type with variable speed.

Bed: The bed of the planer must be a weldment or casting twice as long as the table. Thus a 12-ft table requires a 24-ft bed. The gearing of hydraulic cylinders for driving the table is housed under the bed.

Toolholders: Planers use high speed steel or carbide tipped cutting tools similar to those used on shapers. However, since planers make heavy cuts, their tools are much larger. Rake relief angles are similar to those used on lathes for cutting cast iron or steel, although relief angles are often only 3 to 5 degrees, because all cuts are on flat surfaces.

Work Holding: Holding the work while machining such heavy cuts at 60 to 100 feet per minute requires considerable force; therefore, the workpieces must be solidly fastened to the table. Because the reversal of direction occurs quite rapidly, the work must be especially well braced at the ends. The table

has T-slots, both lengthwise and across, in which heavy bolts and clamps may be used. Sometimes holes are drilled in the table so that large pins can be used to prevent the workpiece from going off the table when the machine reverses.

Planer Size: The size of planers is often referred to as a 30 inch planer or a 60 inch planer. This specifies the approximate width of the table which ranges from 30 to 72 inches. A more complete specification is:

Width of table x height under rail
x length of table
(For example: 48 inches x 48 inches
x 14 feet)

The width and height are usually, but not always, the same. Table length is often made to order and may be as short as 8 feet, or as long as 20 feet or more. The drive may be 15 HP on the smaller planers, and 100 HP or more on the larger models.

Mechanical and hydraulic power can be used for planers. Uniform cutting speed is attained throughout the cutting stroke. Acceleration and deceleration of the table takes place in a short distance of travel and does not influence the time to machine.

Double-housing Planers: Double-housing planers consist of a long heavy base on which the table reciprocates. The upright housing near the center on the side of the base, supports the cross-rail on which the tools are fed across the work. Figure 7.5 illustrates how the tools are supported both above and on the sides, and their adjustment for angle cuts. They are fed by power in either a vertical or a crosswise direction.

Open-sided Planers: Open-sided planers have the housing on one side only. The open side permits machining wider workpieces. Most planers have one flat and one double V-way, which allows for unequal bed and platen expansions. Adjustable dogs at the side of the bed control the stroke length of the platen.

Planers are often converted to planer-

mills, for more efficient machining.

7.3.1 Comparison of Shapers and Planers

Although both the planer and the shaper are able to machine flat surfaces, there is little overlapping in their application. They differ greatly in construction and in the method of operation. The planer is especially adapted to large work: the shaper can do only small work. On the planer the work is moved against a stationary tool: on the shaper the tool moves across the work, which is stationary. On the planer the tool is fed into the work; on the shaper the work is usually fed across the tool. The drive on the planer table is either by gears or by hydraulic means. The shaper ram also can be driven in this manner, but many times a quick-return link mechanism is used.

Most planers differ from shapers in that they approach more constant-velocity cutting speeds. Tools used in shaper and planer work are single point as used on a lathe, but are heavier in construction. The holder is designed to secure the tool bit near the centerline of the holder or the pivot point rather than at an angle as is customary with lathe toolholders.

Cutting tools for the planer operation are usually tipped with high-speed steel, cast alloy, or carbide inserts. High speed steel or cast alloys are commonly used in heavy roughing cuts and carbides for secondary roughing and finishing.

Cutting angles for tools depend on the tool used and the workpiece material. They are similar to angles used on other single-point tools, but the end clearance does not exceed 4 degrees. Cutting speeds are affected by the rigidity of the machine, how the work is held, tool, material, and the number of tools in operation. Worktables on planers and shapers are constructed with T-slots to hold and clamp parts that are to be machined.

Chapter 8

Drills & Drilling

Operations

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Finishing Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>

Prentice Hall: <http://www.prenhall.com>

8.1 Introduction

Drilling is the process most commonly associated with producing machined holes. Although many other processes contribute to the production of holes, including boring, reaming, broaching, and internal grinding, drilling accounts for the majority of holes produced in the machine shop. This is because drilling is a simple, quick, and economical method of hole production. The other methods are used principally for more accurate, smoother, larger holes. They are often used after a drill has already made the pilot hole.

Drilling is one of the most complex machining processes. The chief characteristic that distinguishes it from other machining operations is the combined cutting and extrusion of metal at the chisel edge in the center of the drill. The high thrust force caused by the feeding motion first extrudes metal under the chisel edge. Then it tends to shear under the action of a negative rake angle tool. Drilling of a single hole is shown in Figure 8.1 and high production drilling of a plate component is shown in Figure 8.2.



FIGURE 8.1: Drilling accounts for the majority of holes produced in industry today. (Courtesy Valenite Inc.)

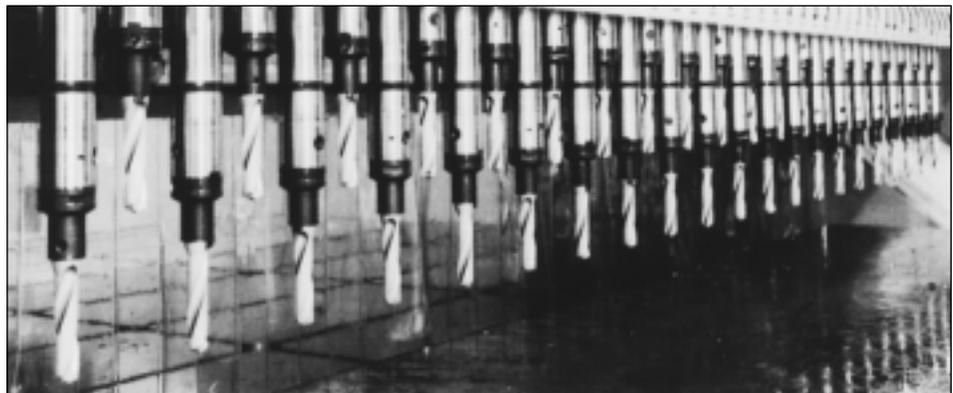


FIGURE 8.2: Holes can be drilled individually as shown in Figure 8.1, or many holes can be drilled at the same time as shown here. (Courtesy Sandvik Coromant Co.)

The cutting action along the lips of the drill is not unlike that in other machining processes. Due to variable rake angle and inclination, however, there are differences in the cutting action at various radii on the cutting edges. This is complicated by the constraint of the whole chip on the chip flow at any single point along the lip. Still, the metal removing action is true cutting, and the problems of variable geometry and constraint are present, but because it is such a small portion of the total drilling operation, it is not a distinguishing characteristic of the process. Many of the drills discussed in this chapter are shown in Figures 8.3.

The machine settings used in drilling reveal some important features of this hole producing operation. Depth of cut, a fundamental dimension in other cutting processes, corresponds most closely to the drill radius. The undeformed chip width is equivalent to the length of the drill lip, which depends on the point angle as well as the drill size. For a given set-up, the undeformed chip width is constant in drilling. The feed dimension specified for drilling is the feed per revolution of the spindle. A more fundamental quantity is the feed per lip. For the common two-flute drill, it is half the feed per revolution. The undeformed chip thickness differs from the feed per lip depending on the point angle.

The spindle speed is constant for any one operation, while the cutting speed varies all along the cutting edge. Cutting speed is normally computed for the outside diameter. At the center of

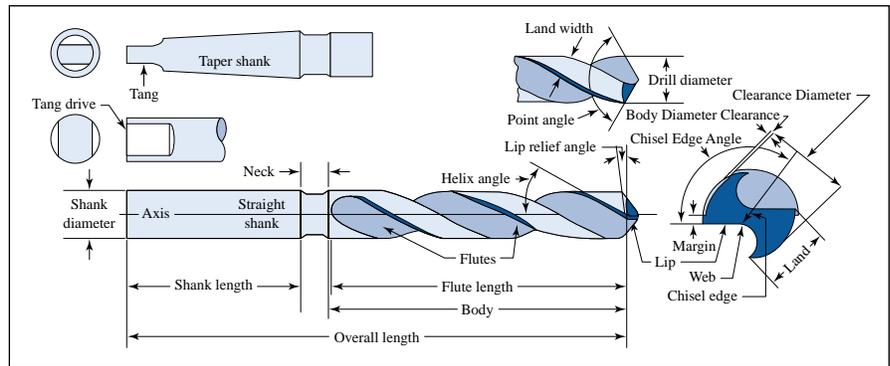


FIGURE 8.4: Nomenclature of a twist drill shown with taper and tang drives.

the chisel edge the cutting speed is zero; at any point on the lip it is proportional to the radius of that point. This variation in cutting speed along the cutting edges is an important characteristic of drilling.

Once the drill engages the workpiece, the contact is continuous until the drill breaks through the bottom of the part or is withdrawn from the hole. In this respect, drilling resembles turning and is unlike milling. Continuous cutting means that steady forces and temperatures may be expected shortly after contact between the drill and the workpiece.

8.2 Drill Nomenclature

The most important type of drill is the twist drill. The important nomenclature listed below and illustrated in Figure 8.4 applies specifically to these tools.

Drill: A drill is an end-cutting tool for producing holes. It has one or more cutting edges, and flutes to allow fluids to enter and chips to be ejected. The drill is composed of a shank, body, and point.

Shank: The shank is the part of the drill that is held and driven. It may be straight or tapered. Smaller diameter drills normally have straight shanks. Larger drills have shanks ground with a taper and a tang to insure accurate alignment and positive drive.

Tang: The tang is a flattened portion at the end of the shank that fits into a driving slot of the drill holder on the spindle of the machine.

Body: The body of the drill extends from the shank to the point, and contains the flutes. During sharpening, it is the body of the drill that is

partially ground away.

Point: The point is the cutting end of the drill.

Flutes: Flutes are grooves that are cut or formed in the body of the drill to allow fluids to reach the point and chips to reach the workpiece surface. Although straight flutes are used in some cases, they are normally helical.

Land: The land is the remainder of the outside of the drill body after the flutes are cut. The land is cut back somewhat from the outside drill diameter in order to provide clearance.

Margin: The margin is a short portion of the land not cut away for clearance. It preserves the full drill diameter.

Web: The web is the central portion of the drill body that connects the lands.

Chisel Edge: The edge ground on the tool point along the web is called the chisel edge. It connects the cutting lips.

Lips: The lips are the primary cutting edges of the drill. They extend from the chisel point to the periphery of the drill.

Axis: The axis of the drill is the centerline of the tool. It runs through the web and is perpendicular to the diameter.

Neck: Some drills are made with a relieved portion between the body and the shank. This is called the drill neck.

In addition to the above terms that define the various parts of the drill, there are a number of terms that apply to the dimensions of the drill, including the important drill angles. Among these terms are the following:

Length: Along with its outside diameter, the axial length of a drill is listed when the drill size is given. In addition, shank length, flute length, and neck length are often used. (see Fig. 8.4)

Body Diameter Clearance: The height of the step from the margin to the land is called the body diameter clearance.

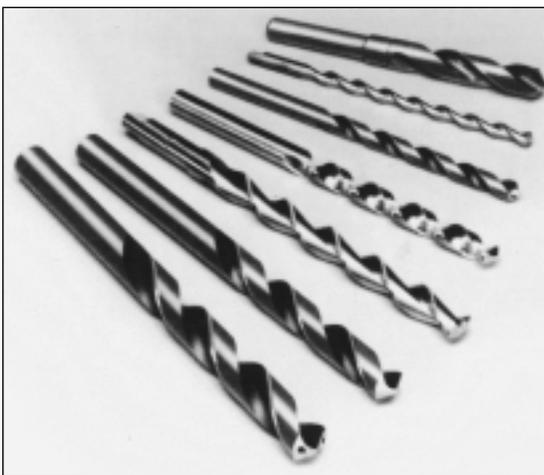


FIGURE 8.3: Many of the drills used in industry are shown here and described in this chapter. (Courtesy Cleveland Twist Drill Greenfield Industries)

Web Thickness: The web thickness is the smallest dimension across the web. It is measured at the point unless otherwise noted. Web thickness will often increase in going up the body away from the point, and it may have to be ground down during sharpening to reduce the size of the chisel edge. This process is called ‘web thinning’. Web thinning is shown in Figure 8.13.

Helix Angle: The angle that the leading edge of the land makes with the drill axis is called the helix angle. Drills with various helix angles are available for different operational requirements.

Point Angle: The included angle between the drill lips is called the point angle. It is varied for different work-piece materials.

Lip Relief Angle: Corresponding to the usual relief angles found on other tools is the lip relief angle. It is measured at the periphery.

Chisel Edge Angle: The chisel edge angle is the angle between the lip and the chisel edge, as seen from the end of the drill.

It is apparent from these partial lists of terms that many different drill geometries are possible.

8.3 Classes of Drills

There are different classes of drills for different types of operations. Workpiece materials may also influence the class of drill used, but it usually determines the point geometry rather than the general type of drill best suited

for the job. It has already been noted that the twist drill is the most important class. Within the general class of twist drills there are a number of drill types made for different kinds of operations. Many of the special drills discussed below are shown in Figure 8.5.

High Helix Drills: This drill has a high helix angle, which improves cutting efficiency but weakens the drill body. It is used for cutting softer metals and other low strength materials.

Low Helix Drills: A lower than normal helix angle is sometimes useful to prevent the tool from ‘running ahead’ or ‘grabbing’ when drilling brass and similar materials.

Heavy-duty Drills: Drills subject to severe stresses can be made stronger by such methods as increasing the web thickness.

Left Hand Drills: Standard twist drills can be made as left hand tools. These are used in multiple drill heads where the head design is simplified by allowing the spindle to rotate in different directions.

Straight Flute Drills: Straight flute drills are an extreme case of low helix drills. They are used for drilling brass and sheet metal.

Crankshaft Drills: Drills that are especially designed for crankshaft work have been found to be useful for machining deep holes in tough materials. They have a heavy web and helix angle that is somewhat higher than normal. The heavy web prompted the use

of a specially notched chisel edge that has proven useful on other jobs as well. The crankshaft drill is an example of a special drill that has found wider application than originally anticipated and has become standard.

Extension Drills: The extension drill has a long, tempered shank to allow drilling in surfaces that are normally inaccessible.

Extra-length Drills: For deep holes, the standard long drill may not suffice, and a longer bodied drill is required.

Step Drill: Two or more diameters may be ground on a twist drill to produce a hole with stepped diameters.

Subland Drill: The subland or multi-cut drill does the same job as the step drill. It has separate lands running the full body length for each diameter, whereas the step drill uses one land. A subland drill looks like two drills twisted together.

Solid Carbide Drills: For drilling small holes in light alloys and non-metallic materials, solid carbide rods may be ground to standard drill geometry. Light cuts without shock must be taken because carbide is quite brittle.

Carbide Tipped Drills: Carbide tips may be used on twist drills to make the edges more wear resistant at higher speeds. Smaller helix angles and thicker webs are often used to improve the rigidity of these drills, which helps to preserve the carbide. Carbide tipped drills are widely used for hard, abrasive non-metallic materials such as masonry.

Oil Hole Drills: Small holes through the lands, or small tubes in slots milled in the lands, can be used to force oil under pressure to the tool point. These drills are especially useful for drilling deep holes in tough materials.

Flat Drills: Flat bars may be ground with a conventional drill point at the end. This gives very large chip spaces, but no helix. Their major application is for drilling railroad track.

Three and Four Fluted Drills: There are drills with three or four flutes which resemble standard twist drills except that they have no chisel edge. They are used for enlarging holes that have been previously drilled or punched. These drills are used because they give better productivity, accuracy, and surface finish than a standard drill would provide on the same job.

Drill and Countersink: A combination drill and countersink is a useful tool

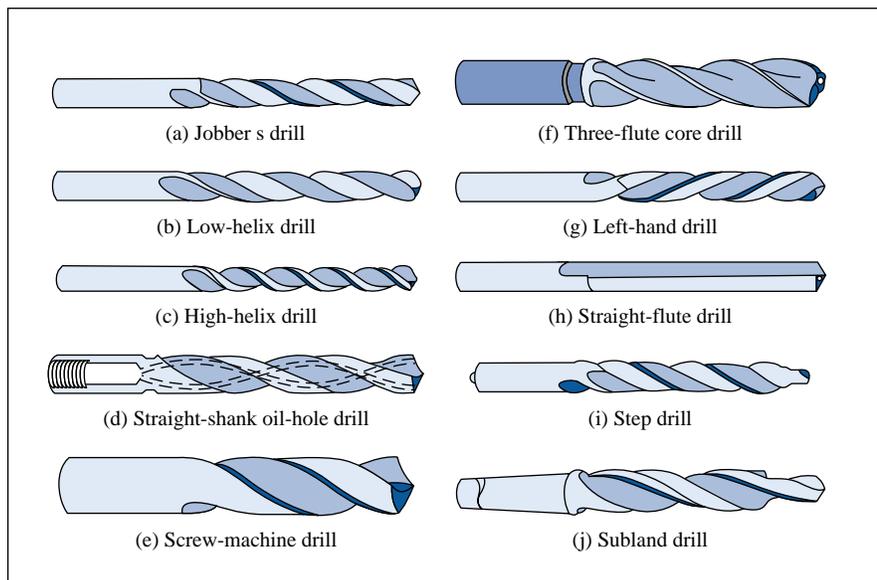


FIGURE 8.5: Special drills are used for some drilling operations.

for machining 'center holes' on bars to be turned or ground between centers. The end of this tool resembles a standard drill. The countersink starts a short distance back on the body.

A double-ended combination drill and countersink, also called a center drill, is shown in Figure 8.6.

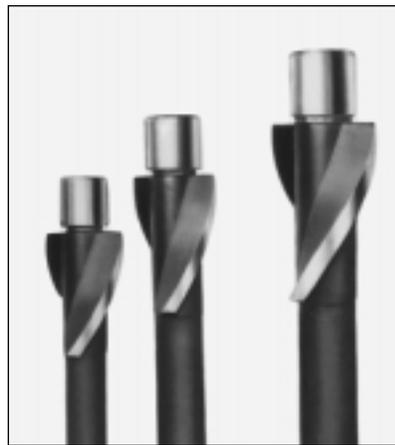
8.4 Related Drilling Operations

Several operations are related to drilling. In the following list, most of the operations follow drilling except for centering and spotfacing which precede drilling. A hole must be made first by drilling and then the hole is modified by one of the other operations. Some of these operations are described here and illustrated in Figure 8.7

Reaming: A reamer is used to enlarge a previously drilled hole, to provide a higher tolerance and to improve



FIGURE 8.6: A double-ended combination drill and countersink, also called a center drill. (Courtesy Morse Cutting Tools)



(a)



(b)

FIGURE 8.8: Counterboring tools (a) and countersinking operation (b) are shown here. (Courtesy The Weldon Tool Co.)

the surface finish of the hole.

Tapping: A tap is used to provide internal threads on a previously drilled hole.

Reaming and tapping are more involved and complicated than counterboring, countersinking, centering, and spot facing, and are therefore discussed in Chapter 11.

Counterboring: Counterboring produces a larger step in a hole to allow a bolt head to be seated

below the part surface.

Countersinking: Countersinking is similar to counterboring except that the step is angular to allow flat-head screws to be seated below the surface.

Counterboring tools are shown in Figure 8.8a, and a counter-sinking tool with two machined holes is shown in Figure 8.8b.

Centering: Center drilling is used for accurately locating a hole to be drilled afterwards.

Spotfacing: Spotfacing is used to provide a flat-machined surface on a part.

8.5 Operating Conditions

The varying conditions, under which drills are used, make it difficult to give set rules for speeds and feeds. Drill manufacturers and a variety of reference texts provide recommendations for proper speeds and feeds for drilling a variety of materials. General drilling speeds and feeds will be discussed here and some examples will be given.

Drilling Speed: Cutting speed may be referred to as the rate that a point on a circumference of a drill will travel in 1 minute. It is expressed in surface feet per minute (SFPM). Cutting speed is one of the most important factors that determine the life of a drill. If the cutting speed is too slow, the drill might chip or break. A cutting speed that is too fast rapidly dulls the cutting lips. Cutting speeds depend on the following seven variables:

- The type of material being drilled. The harder the material, the slower the cutting speed.
- The cutting tool material and diame-

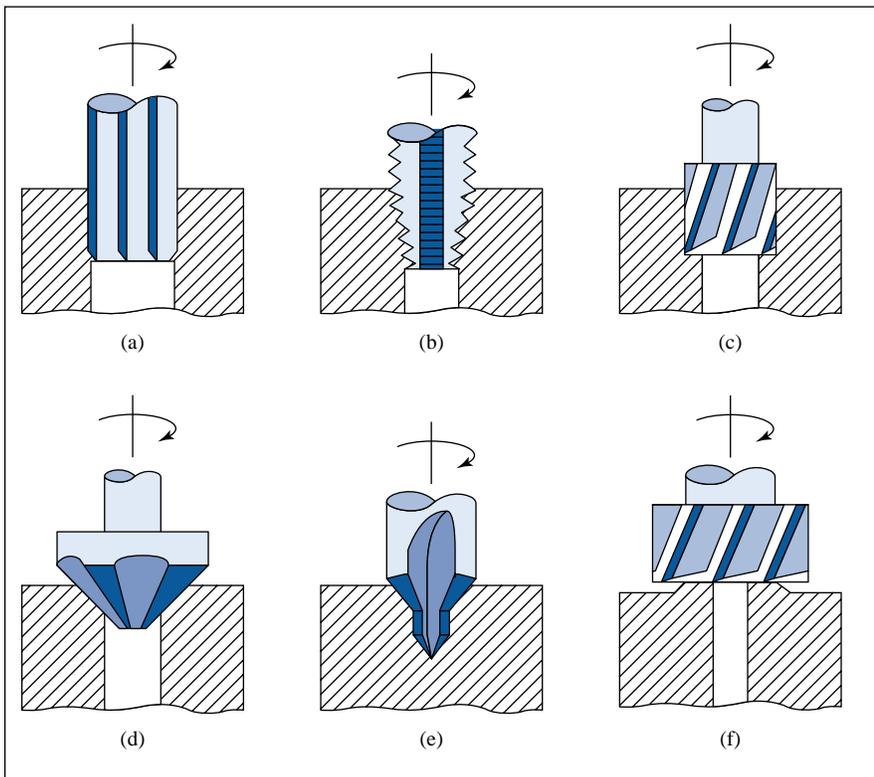


FIGURE 8.7: Related drilling operations: (a) reaming, (b) tapping, (c) counterboring, (d) countersinking, (e) centering, (f) spotfacing.

ter. The harder the cutting tool material, the faster it can machine the material. The larger the drill, the slower the drill must revolve.

- The types and use of cutting fluids allow an increase in cutting speed.
- The rigidity of the drill press.
- The rigidity of the drill (the shorter the drill, the better).
- The rigidity of the work setup.
- The quality of the hole to be drilled.

Each variable should be considered prior to drilling a hole. Each variable is important, but the work material and its cutting speed are the most important factors. To calculate the revolutions per minute (RPM) rate of a drill, the diameter of the drill and the cutting speed of the material must be considered.

The formula normally used to calculate cutting speed is as follows:

$$SFPM = (\text{Drill Circumference}) \times (\text{RPM})$$

Where:

SFPM = surface feet per minute, or the distance traveled by a point on the drill periphery in feet each minute.

Drill Circumference = the distance around the drill periphery in feet.

RPM = revolutions per minute

In the case of a drill, the circumference is:

$$\text{Drill Circumference} = \pi/12 \times (d) = .262 \times d$$

Where:

Drill Circumference = the distance around the drill periphery in feet.

π = is a constant of 3.1416

d = the drill diameter in inches.

By substituting for the drill circumference, the cutting speed can now be written as:

$$SFPM = .262 \times d \times \text{RPM}$$

This formula can be used to determine the cutting speed at the periphery of any rotating drill.

For example: Given a .25 inch drill, what is the cutting speed (SFPM) drilling cast iron at 5000 RPM?

$$SFPM = .262 \times d \times \text{RPM}$$

$$SFPM = .262 \times .25 \times 5000$$

$$\text{Answer} = 327.5 \text{ or } 327 \text{ SFPM}$$

RPM can be calculated as follows:

Given a .75 inch drill, what is the RPM drilling low carbon steel at 400 SFPM?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{400}{.262 \times .75} = \frac{400}{.1965}$$

$$\text{Answer} = 2035.62 \text{ or } 2036 \text{ RPM}$$

Drilling Feed: Once the cutting speed has been selected for a particular workpiece material and condition, the appropriate feed rate must be established. Drilling feed rates are selected to maximize productivity while maintaining chip control. Feed in drilling operations is expressed in inches per revolution, or IPR, which is the distance the drill moves in inches for each revolution of the drill. The feed may also be expressed as the distance traveled by the drill in a single minute, or IPM (inches per minute), which is the product of the RPM and IPR of the drill. It can be calculated as follows:

$$\text{IPM} = \text{IPR} \times \text{RPM}$$

Where:

IPM = inches per minute

IPR = inches per revolution

RPM = revolutions per minute.

For example: To maintain a .015 IPR feed rate on the .75 inch drill discussed above, what would the IPM feed rate be?

$$\text{IPM} = \text{IPR} \times \text{RPM}$$

$$\text{IPM} = .015 \times 2036$$

$$\text{Answer} = 30.54 \text{ or } 31 \text{ IPM}$$

The selection of drilling speed (SFPM) and drilling feed (IPR) for various materials to be machined often starts with recommendations in the

form of application tables from manufacturers or by consulting reference books.

8.5.1 Twist Drill Wear

Drills wear starts as soon as cutting begins and instead of progressing at a constant rate, the wear accelerates continuously. Wear starts at the sharp corners of the cutting edges and, at the same time, works its way along the cutting edges to the chisel edge and up the drill margins. As wear progresses, clearance is reduced. The resulting rubbing causes more heat, which in turn causes faster wear.

Wear lands behind the cutting edges are not the best indicators of wear, since they depend on the lip relief angle. The wear on the drill margins actually determines the degree of wear and is not nearly as obvious as wear lands. When the corners of the drill are rounded off, the drill has been damaged more than is readily apparent. Quite possibly the drill appeared to be working properly even while it was wearing. The margins could be worn in a taper as far back as an inch from the point. To restore the tool to new condition, the worn area must be removed. Because of the accelerating nature of wear, the number of holes per inch of drill can sometimes be doubled by reducing, by 25 percent, the number of holes drilled per grind.

8.5.2 Drill Point Grinding

It has been estimated that about 90 percent of drilling troubles are due to improper grinding of the drill point. Therefore, it is important that care be taken when resharpening drills. A good drill point will have: both lips at the same angle to the axis of the drill; both lips the same length; correct clearance angle; and correct thickness of web.

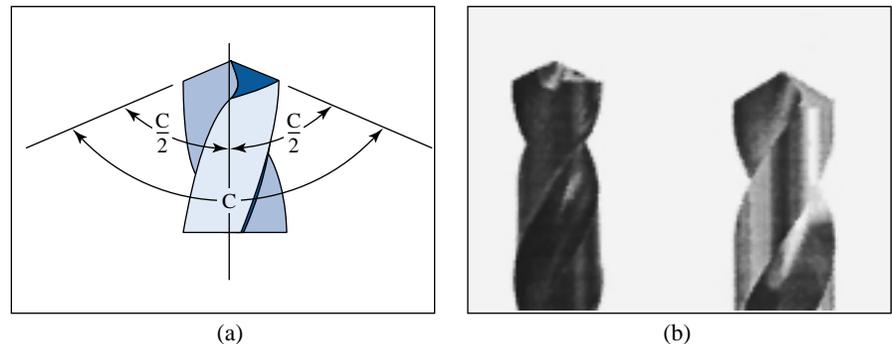


FIGURE 8.9: The included lip angle varies between 90 and 135 degrees (a); two drill points are shown in (b). (Courtesy Cleveland Twist Drill Greenfield Industries)

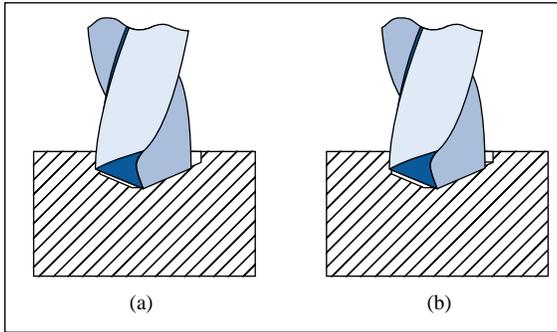


FIGURE 8.10: Drill with equal lip angle but unequal lip length (a), and drill with unequal lip angle and unequal lip length (b).

Lip Angle and Lip Length: When grinding the two cutting edges they should be equal in length and have the same angle with the axis of the drill as shown in Figure 8.9a. Figure 8.9b shows two ground drill points.

For drilling hard or alloy steels, angle C (Fig. 8.9a) should be 135 degrees. For soft materials and for general purposes, angle C should be 118 degrees. For aluminum, angle C should be 90 degrees.

If lips are not ground at the same angle with the axis, the drill will be subjected to an abnormal strain, because only one lip comes in contact with the work. This will result in unnecessary breakage and also cause the drill to dull quickly. A drill so sharpened will drill an oversized hole. When the point is ground with equal angles, but has lips of different lengths, a condition as shown in Figure 8.10a is produced.

A drill having cutting lips of different angles, and of unequal lengths, will be laboring under the severe conditions shown in Figure 8.10b.

Lip Clearance Angle: The clearance angle, or ‘backing-off’ of the point, is

the next important thing to consider. When drilling steel this angle A (Fig. 8.11a) should be from 6 to 9 degrees. For soft cast iron and other soft materials, angle A may be increased to 12 degrees (or even 15 degrees in some cases)

This clearance angle should increase gradually as the center of the drill is approached. The amount of clearance at the center of the drill determines the chisel point angle B (Fig. 8.11b).

The correct combination of clearance and chisel point angles should be as follows: When angle A is made to be 12 degrees for soft materials, angle B should be made approximately 135 degrees; when angle A is 6 to 9 degrees for harder materials, angle B should be 115 to 125 degrees.

While insufficient clearance at the center is the cause of drills splitting up the web, too much clearance at this point will cause the cutting edges to chip.

In order to maintain the necessary accuracy of point angles, lip lengths, lip clearance angle, and chisel edge angle,



FIGURE 8.12: Tool and cutter grinders, are used to properly sharpen drills and other cutting tools. (Courtesy K. O. Lee Co.)

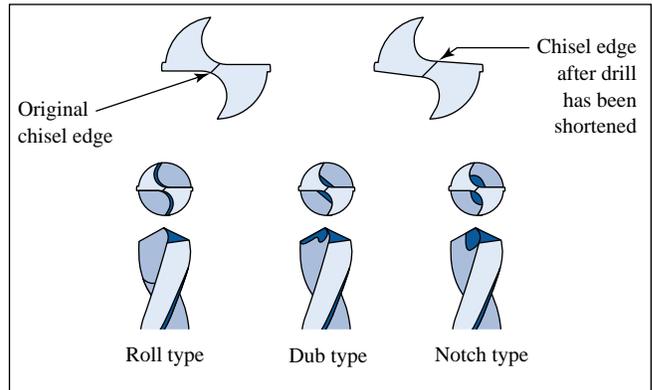


FIGURE 8.13: Web thinning restores proper web thickness after sharpening twist drills; three methods are shown.

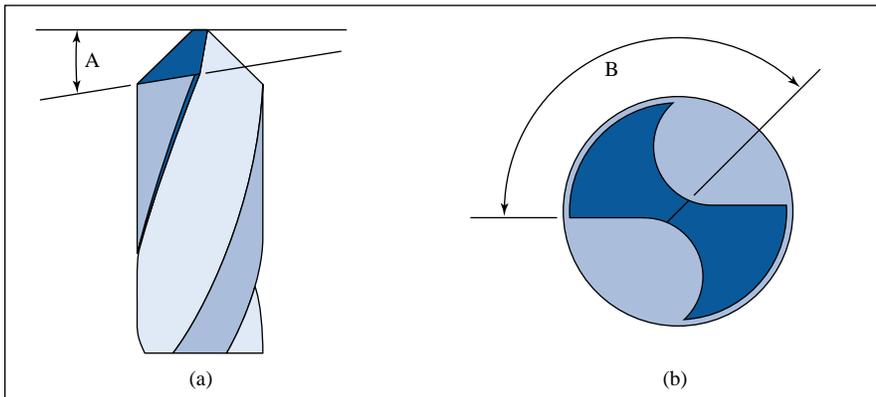


FIGURE 8.11: Drill lip clearance angle (a) and drill chisel point angle (b).

the use of machine point grinding is recommended. There are many commercial drill point grinders available today, which will make the accurate re-pointing of drills much easier. Tool and cutter grinders such as the one shown in Figure 8.12 are often used.

Twist Drill Web Thinning: The tapered web drill is the most common type manufactured. The web thickness increases as this type of drill is resharpened. This requires an operation called web thinning to restore the tool’s original web thickness. Without the web thinning process, more thrust would be required to drill, resulting in additional generated heat and reduced tool life. Figure 8.13 illustrates a standard drill before and after the web thinning process. Thinning is accomplished with a radiused wheel and should be done so the thinned section tapers gradually

from the point. This prevents a blunt wedge from being formed that would be detrimental to chip flow. Thinning can be done by hand, but since point centrality is important, thinning by machine is recommended.

8.6 Spade Drills

The tool generally consists of a cutting blade secured in a fluted holder (See Figure 8.14). Spade drills can machine much larger holes (up to 15 in. in diameter) than twist drills. Spade drills usually are not available in diameters smaller than 0.75 inch. The drilling depth capacity of spade drills, with length-to-diameter ratios over 100 to 1 possible, far exceeds that of twist drills. At the same time, because of their much greater feed capability, the penetration rates for spade drills exceed those of twist drills by 60 to 100 percent. However, hole finish generally suffers because of this. Compared to twist drills, spade drills are much more resistant to chatter under heavy feeds once they are fully engaged with the workpiece. Hole straightness is generally improved (with comparable size capability) by using a spade drill. However, these advantages can only be gained by using drilling machines of suitable capability and power.

The spade drill is also a very economical drill due to its diameter flexibility. A single holder will accommodate many blade diameters as shown in

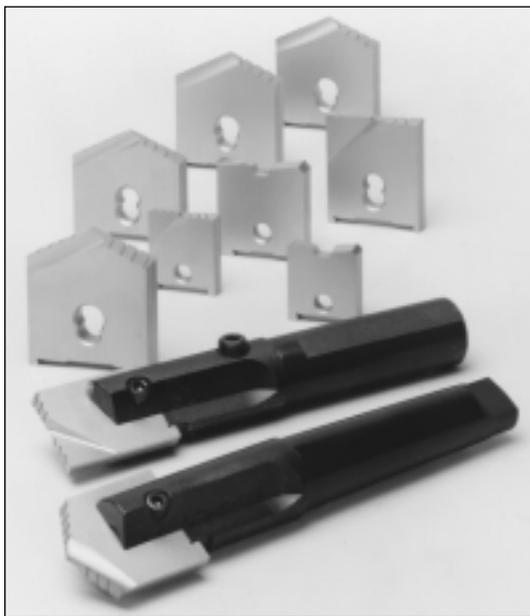


FIGURE 8.14: Spade drills with various cutting blades. (Courtesy Kennametal Inc.)

Figure 8.14. Therefore, when a diameter change is required, only the blade needs to be purchased which is far less expensive than buying an entire drill.

8.6.1 Spade Drill Blades

The design of spade drill blades varies with the manufacturer and the intended application. The most common design is shown in Figure 8.15. The locator length is ground to a precision dimension that, in conjunction with the ground thickness of the blade, precisely locates the blade in its holder. When the seating pads properly contact the holder, the holes in the blade and holder are aligned and the assembly can be secured with a screw.

The blade itself as shown in Figure 8.15, possesses all the cutting geometry necessary. The point angle is normally 130 degrees but may vary for special applications. In twist drill designs, the helix angle generally determines the cutting rake angle but since spade drills have no helix, the rake surface must be ground into the blade at the cutting edge angle that produces the proper web thickness. The cutting edge clearance angle is a constant type of relief, generally 6 to 8 degrees. After this clearance is ground, the chip breakers are ground, about 0.025 inch deep, in the cutting edge.

These chip breakers are necessary on spade drill blades and not optional as with twist drills. These notches make the chips narrow enough to flush around the holder. Depending on the feed rate, the grooves can also cause a rib to form in the chip. The rib stiffens the chip and causes it to fracture or break more easily which results in shorter, more easily removed chips. Margins on the blade act as bearing surfaces once the tool is in a bushing or in the hole being drilled. The width of the margins will vary from 1/16 to 3/16 inches, depending on the tool size. A slight back taper of 0.004 to 0.006 inch is normally provided and outside diameter clearance angles are generally 10 degrees.

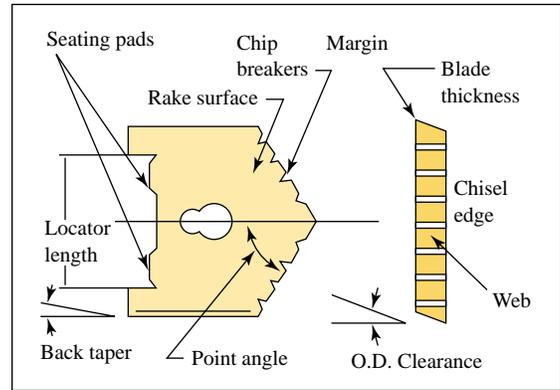


FIGURE 8.15: Spade drill cutting blades shows geometry specifications.

8.6.2 Spade Drill Blade Holders

The blade holder makes up the major part of the spade drill. The blade holder is made of heat-treated alloy steel and is designed to hold a variety of blades in a certain size range as shown in Figure 8.14. Two straight chip channels or flutes are provided for chip ejection.

The holder shank designs are available in straight, Morse taper, and various other designs to fit the machine spindles. The holders are generally supplied with internal coolant passages to ensure that coolant reaches the cutting edges and to aid chip ejection.

When hole position is extremely critical and requires the use of a starting bushing, holders with guide strips are available. These strips are ground to fit closely with the starting bushing to support the tool until it is fully engaged in the workpiece. The strips may also be ground to just below the drill diameter to support the tool in the hole when the set-up lacks rigidity.

8.6.3 Spade Drill Feeds and Speeds

The cutting speed for spade drills is generally 20 percent less than for twist drills. However, the spade drill feed capacity can be twice that of twist drills. The manufacturers of spade drills and other reference book publishers provide excellent recommendations for machining rates in a large variety of metals. These published rates should generally be observed. Spade drills work best under moderate speed and heavy feed. Feeding too lightly will result in either long, stringy chips or chips reduced almost to a powder. The drill cutting edges will chip and burn because of the absence of the thick, heat absorbing, C-shaped chips. Chips can possibly jam



FIGURE 8.16: Indexing drills were introduced using square inserts; three sizes are shown here. (Courtesy Kennametal Inc.)

and pack, which can break the tool or the workpiece. If the machine cannot supply the required thrust to maintain the proper feed without severe deflection, a change in tool or machine may be necessary.

8.7 Indexable Carbide Drills

Indexable drilling has become so efficient and cost effective that in many cases it is less expensive to drill the hole rather than to cast or forge it. Basically, the indexable drill is a two fluted, center cutting tool with indexable carbide

inserts. Indexable drills were introduced using square inserts (see Fig. 8.16). Shown in Figure 8.17a are indexable drills using the more popular Trigon Insert (see Fig. 8.17b). In most cases two inserts are used, but as size increases, more inserts are added with as many as eight inserts in very large tools. Figure 8.18 shows six inserts being used.

Indexable drills have the problem of zero cutting speed at the center even though speeds can exceed 1000 SFPM at the outermost inserts. Because speed generally replaces feed to some degree, thrust forces are usually 25 to 30 percent of those required by conventional tools of the same size. Indexable drills have a shank, body, and multi-edged point. The shank designs generally available are straight, tapered and number 50 V-flange.

The bodies have two flutes that are normally straight but may be helical. Because no margins are present to provide bearing support, the tools must rely on their inherent stiffness and on the balance in the cutting forces to maintain accurate hole size and straightness. Therefore, these tools are usually limited to length-to-diameter ratios of approximately 4 to 1.

The drill point is made of pocketed carbide inserts. These inserts are usually specially designed. The cutting rake can be negative, neutral, or positive,

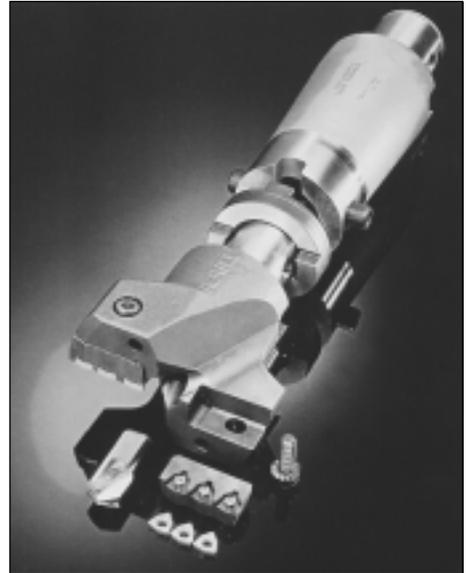


FIGURE 8.18: Indexable drill using six Trigon inserts for drilling large holes. (Courtesy Kennametal Inc.)

depending on holder and insert design. Coated and uncoated carbide grades are available for drilling a wide variety of work materials. Drills are sometimes combined with indexable or replaceable inserts to perform more than one operation, such as drilling, counterboring, and countersinking.

As shown in Figure 8.19a and Figure 8.19b, body mounted insert tooling can perform multiple operations. More examples will be shown and discussed in Chapter 10: Boring Operations and Machines.

The overall geometry of the cutting edges is important to the performance of indexable drills. As mentioned earlier, there are no supporting margins to keep these tools on line, so the forces required to move the cutting edges through the work material must be balanced to minimize tool deflection, particularly on starting, and to maintain hole size.

While they are principally designed for drilling, some indexable drills, as shown in Figure 8.20, can perform facing, and boring in lathe

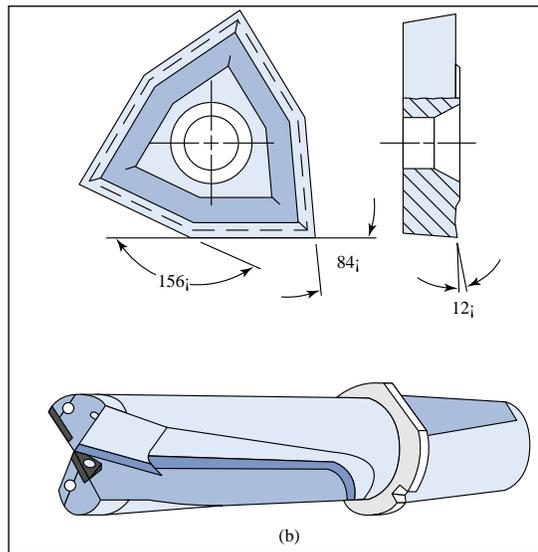
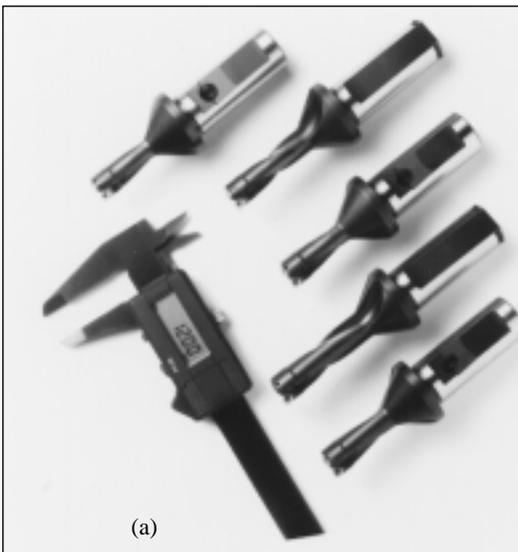


FIGURE 8.17: (a) Indexable drills using Trigon inserts. (b) A Trigon insert and holder. (Courtesy Komet of America, Inc.)

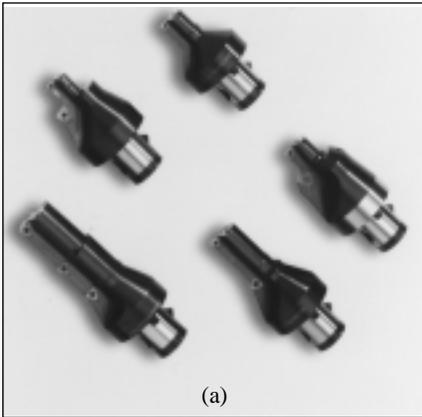
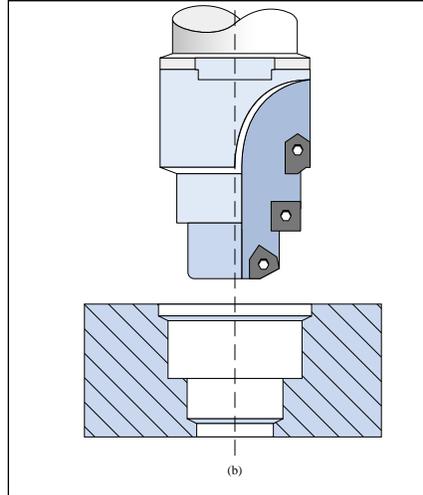


FIGURE 8.19: Body-mounted insert tooling can perform multiple operations. (Courtesy Komet of America, Inc.)



applications. How well these tools perform in these applications depends on their size, rigidity, and design.

8.7.1 Indexable Carbide Drill Operation

When used under the proper conditions, the performance of indexable drills is impressive. However, the manufacturer's recommendations must be carefully followed for successful applications.

Set-up accuracy and rigidity is most important to tool life and performance. Chatter will destroy turning or milling inserts just as it destroys turning or milling inserts. If the inserts fail when the tool is rotating in the hole at high speed, the holder and workpiece will be damaged. Even if lack of rigidity has only a minor effect on tool life, hole size and finish will be poor. The machine must be powerful, rigid and

capable of high speed. Radial drill presses do not generally meet the rigidity requirements. Heavier lathes, horizontal boring mills, and N/C machining centers are usually suitable.

When installing the tool in the machine, the same good practice followed for other drill types should be observed for indexable drills. The shanks must be clean and free from burrs to ensure good holding and to minimize runout. Runout in indexable drilling is dramatically amplified because of the high operating speeds and high penetration rates.

When indexing the inserts is necessary, make sure that the pockets are clean and undamaged. A small speck of dirt or chip, or a burr will cause stress in the carbide insert and result in a microscopic crack, which in turn, will lead to early insert failure.

8.7.2 Indexable Drill Feeds and Speeds

Indexable drills are very sensitive to machining rates and work materials. The feed and speed ranges for various materials, as recommended by some manufacturers of these tools, can be very broad and vague, but can be used as starting points in determining exact feed and speed rates. Choosing the correct feed and speed rates, as well as selecting the proper insert style and grade, requires some experimentation. Chip formation is a critical factor and must be correct.

In general, soft low carbon steel calls for high speed (650 SFPM or more), and low feed (0.004/0.006 IPR). Medium and high carbon steels, as well as cast iron, usually react best to lower speed and higher feed. The exact speed and feed settings must be consistent with machine and set-up conditions, hole size and finish requirements, and chip formation for the particular job.

8.8 Trepanning

In trepanning the cutting tool produces a hole by removing a disk shaped piece also called slug or core, usually from flat plates. A hole is produced without reducing all the material removed to chips, as is the case in drilling. The trepanning process can be used to make disks up to 6 in. in diameter from flat sheet or plate. A trepanning tool also called a "Rotabroach" with a core or slug is shown in Figure 8.21a and an end view of a Rotabroach is shown in Figure 8.21 b.

Trepanning can be done on lathes, drill presses, and milling machines, as well as other machines using single point or multi point tools. Figure 8.22 shows a Rotabroach cutter machining holes through both sides of a rectangular tube on a vertical milling machine.

Rotabroach drills provide greater tool life because they have more teeth than conventional drilling tools. Since more teeth are engaged in the workpiece, the material cut per hole is distributed over a greater number of cutting edges. Each cutting edge cuts less material for a given hole. This extends tool life significantly.

Conventional drills must contend with a dead center area that is prone to chip, thus reducing tool life. In the chisel-edge region of a conventional drill the cutting speed approaches zero. This

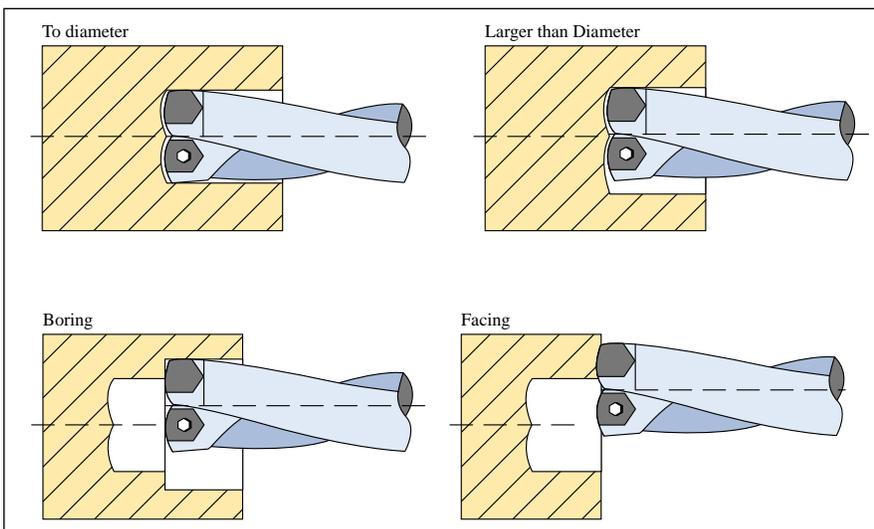


FIGURE 8.20: In addition to drilling, indexable drills can perform boring and facing operations.

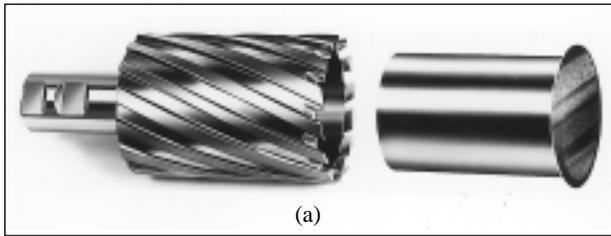


FIGURE 8.21: Trepanning tool also called Rotabroach with core or slug. (Courtesy Hougen Manufacturing, Inc.)

is quite different from the speed at the drill O.D. Likewise, thrust forces are high due to the point geometry. Rotabroach drills cut in the region from the slug O.D. to the drill O.D. Since only a small kerf is machined, cutting speeds are not so different across the face of a tooth. This feature extends tool life and provides uniform machin-

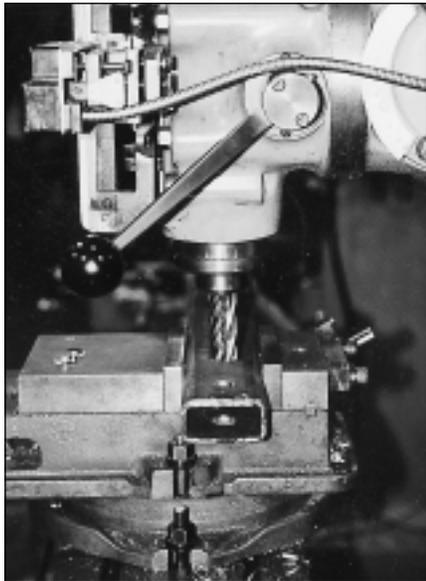


FIGURE 8.22: Rotabroach machining setup on a milling machine. (Courtesy Hougen Manufacturing, Inc.)

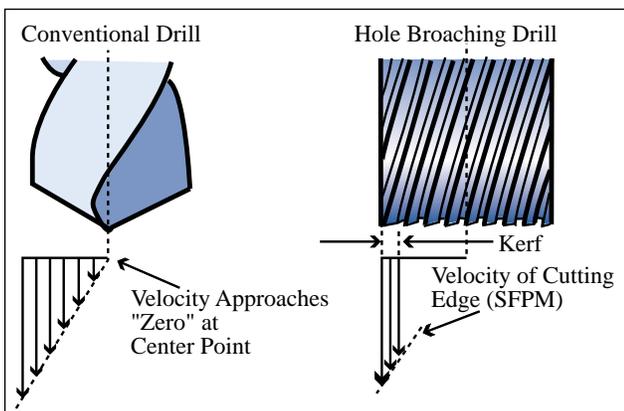
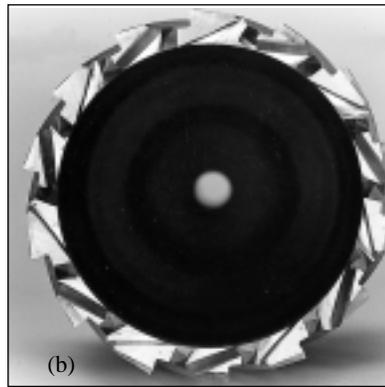


FIGURE 8.23: Drilling holes with conventional drill and hole broaching drill. Surface speed increases with distance from center.



larger number of cutting edges and chip evacuation, combine to reduce the chip load per tooth so this drill can cut at higher feed rates than trepanning tools and hole saws. Like the hole saw, a hole broaching tool has a fixed diameter. One size fits one hole.

8.8.2 Cutting Tool Material Selection

M2 High Speed Steel (HSS)

is the standard Rotabroach cutting tool material. M2 has the broadest application range and is the most economical tool material. It can be used on ferrous and non-ferrous materials and is generally recommended for cutting materials up to 275 BHN. M2 can be applied to harder materials, but tool life is dramatically decreased.

TiN coated M2 HSS Rotabroach drills are for higher speeds, more endurance, harder materials or freer cutting action to reduce power consumption. The TiN coating reduces friction and operates at cooler temperatures while presenting a harder cutting edge surface. Increased cutting speeds of 15 to 25 % are recommended to obtain the benefits of this surface treatment. The reduction in friction and resistance to edge build-up are key benefits. The ability to run at higher speeds at less power is helpful for applications where the machine tool is slightly underpowered and TiN coated tools are recommended for these applications. TiN coated tools are recommended for applications on materials to 325 BHN.

Carbide cutting tool materials are

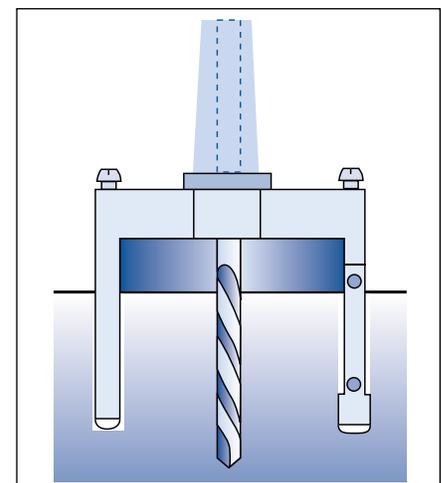


FIGURE 8.24: Traditional trepanning tool orbits around a center drill.

ability. Figure 8.23 shows drilling holes with conventional drills and hole broaching drills.

8.8.1 Trepanning Operations

Trepanning is a roughing operation. Finishing work requires a secondary operation using reamers or boring bars to get a specified size and finish. Of the many types of hole-making operations, it competes with indexable carbide cutters and spade drilling.

Several types of tools are used to trepan. The most basic is a single or double point cutter (Fig. 8.24). It orbits the spindle centerline cutting the periphery of the hole. Usually, a pilot drill centers the tool and drives the orbiting cutter like a compass inscribing a circle on paper. Single/double point trepanning tools are often adjustable within their working diameter. They are efficient and versatile, but do begin to have rigidity problems when cutting large holes - 6 1/2 inches in diameter is about the maximum.

A hole saw is another tool that trepans holes. It is metalcutting's version of the familiar doorknob hole cutter used in wood. Hole saws have more teeth and therefore cut faster than single, or double-point tools. Both hole

saws and single-point tools curl up a chip in the space, or gullet, between the teeth, and carry it with them in the cut.

Hole broaching tools are hybrid trepanners. (Fig. 8.21a and 8.21b) They combine spiral flutes like a drill with a broach-like progressive tool geometry that splits the chip so it exits the cut along the flutes. With this design, the

also available as a special option on Rotabroach drills. Carbide offers certain advantages over high-speed steel. Applications are limited and need to be discussed with a manufacturer's representative.

8.8.3 Rigidity and Hole Size Tolerance

Rotabroach drills were originally designed as roughing tools to compete with twist drills and provide similar hole tolerances. Many users have successfully applied Rotabroach drills in semi-finishing applications, reducing the number of passes from two or more to just one. A rigid machine tool and set-up are required to produce holes to these specifications. Tolerances will vary with the application and are impossible to pin point.

Spindle rigidity or "tightness" and workpiece rigidity are more crucial than with a twist drill. Even if a twist drill runs out slightly at first, the conical point tends to center itself before the O.D. of the tool engages the workpiece. The higher thrust of a twist drill also tends to "preload" the spindle and fixture. The trepanning cutter relies more on the rigidity of the system (workpiece, holder, and spindle). If excessive spindle runout or, worse yet, spindle play exists, the cutter may chatter on entry. At best this will cause a bell-mouthed hole with poor finish, but it can easily lead to drill breakage.

Hole tolerances are dependent on much more than the accuracy of any tool and its grind. The machine tool, workpiece, fixture, selection of speeds and feeds, projection and type of application also play an important part in determining overall results.

8.8.4 Chip Control

In material such as aluminum, tool steels and cast iron, proper selection of feeds and speeds usually causes the chips to break up and allows them to be flushed out of the cut by the cutting fluid. In many other materials, such as mild and alloy steels, the chips tend to be long and frequently wrap themselves around the drill to form a "bird's nest". In most manual operations this is an annoyance that is outweighed by the other benefits of the method. In automated operations, however, the build-up of chips around the drill cannot be tolerated. Besides the obvious problems that this can cause, the nest of chips impedes the flow of additional chips trying to escape from the flutes.

This in turn can cause the flutes to pack and may result in drill breakage.

There are several methods that can be used to break up the chips if this cannot be accomplished by adjusting the feeds and speeds. One method is to use an interrupted feed cycle. It is recommended that the drill not be retracted as with a "peck" cycle, because chips may become lodged under the cutting edges. Instead employ an extremely short dwell approximately every two revolutions. This will produce a chip that is usually short enough not to wrap around the tool. A programmed dwell may not be necessary since some hesitation is probably inherent between successive feed commands in an NC system

8.8.5 Advantages of Trepanning Tools

The twist drill has a center point, which is not really a point at all - it's the intersecting line where two cutting edge angles meet at the web of the drill. This point is the so-called "dead zone" of a twist drill.

It's called a dead zone because the surface speed of the cutting edges (a factor of revolutions per minute and diameter of the drill) approaches zero as the corresponding diameter nears zero. Slower surface speed reduces cutting efficiency and requires increased feed pressure for the cutting edges to bite into the material. In effect, the center of the drill does not cut - it pushes its way through the material. The amount of thrust required to overcome the resistance of the workpiece often causes the stock to deform or dimple around the hole, and creates a second problem - burrs or flaking around the hole's breakthrough side. As material at the bottom of the hole becomes thinner and thinner, if the feed is not eased off, the drill will push through, typically leaving two jagged remnants of stock attached.

Trepanning tools produce holes faster than more conventional tooling as shown in Figure 8.25. From left to right are shown

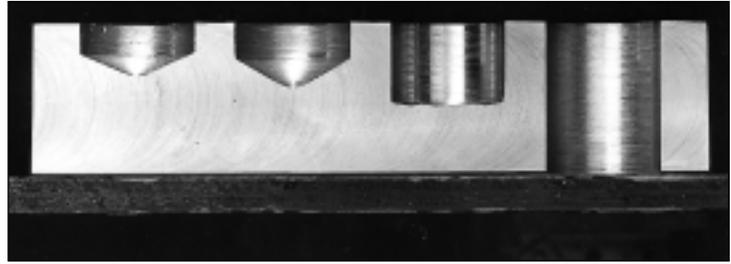


FIGURE 8.25: Trepanning produces holes faster than more conventional tooling. (Courtesy Hougen Manufacturing, Inc.)

a 1 1/2 inch hole drilled into a 2 inch thick 1018 steel plate with: a spade drill, with a twist drill, with an indexable carbide drill, and with a Rotabroach. With approximately 50% to 80% faster drilling time, the cost per hole can be substantially lower.

An indirect yet significant source of savings attributable to trepanning tooling is the solid slug it provides. Separating chips from coolant and oils is increasingly called for by scrap haulers. In one application, while significant gains in productivity were made with hole-broaching tools, the savings in going from chips to a solid slug was enough to justify the change in process.

In Figure 8.26 the workpiece is a tube holder for an industrial heat exchanger. When this workpiece is finished, better than 60 per cent of the plate has been reduced to scrap.

Sixty percent of this heat exchanger plate was converted into chips by the sheer number of holes drilled. Besides increasing production, trepanning tooling's solid core by-product increased scrap value from \$0.17 per pound of chips, to \$0.37 per pound for the core metal.

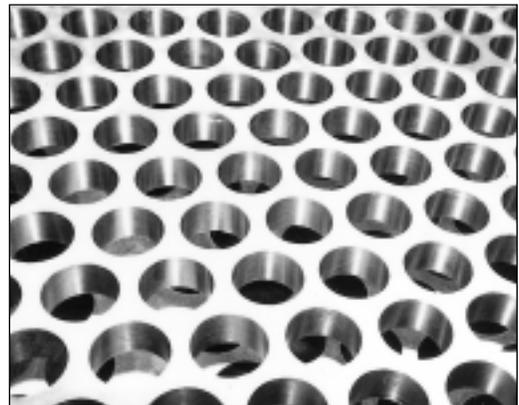


FIGURE 8.26: Sixty percent of this heat exchanger plate was converted to chips. (Courtesy Hougen Manufacturing, Inc.)

Chapter 9

Drilling Methods & Machines

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing

9.1 Introduction

One of the most important and essential tools in any metalworking shop is the drilling machine or drill press. Although the drilling machine is used primarily for drilling holes, it is often used for reaming, boring, tapping, counterboring, countersinking, and spotfacing.

All drilling machines operate on the same basic principle. The spindle turns the cutting tool, which is advanced either by hand or automatically into a workpiece that is mounted on the table or held in a drill press vise. Successful operation of any drilling machine requires a good knowledge of the machine, proper set-up of the work, correct speed and feed, and proper use of cutting fluids applied to the cutting tool and work.

9.2 Types of Drill Presses

Many types and sizes of drilling machines are used in manufacturing. They range in size from a simple bench mounted sensitive drill press to the large multiple-spindle machines able to drive many drills at the same time.

Figure 9.1 shows a schematic diagram of a standard vertical drill press as well as a schematic diagram of a turret-drilling machine. Described below are these and other types of drill presses such as sensitive and radial drills.



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>

Prentice Hall: <http://www.prenhall.com>

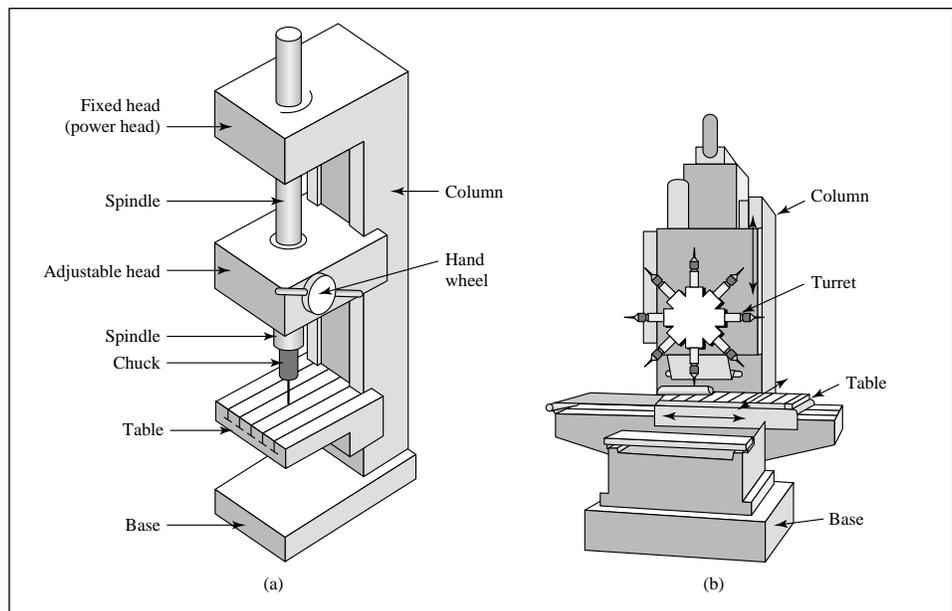


FIGURE 9.1: Schematic illustration of (a) vertical drill press, (b) CNC turret drilling machine.

9.2.1 Simple Drill Press

A simple drill press (Fig.9.2) may be floor mounted as shown, or have a shorter main post and be mounted on a bench. The motions of this machine are very simple. The table on a floor model can be raised or lowered and rotated around the machine column. The spindle rotates and can be raised and lowered, with a stroke of 4 to 8 inches. Stops can be set to limit and regulate the depth.

9.2.2 Sensitive Drill Press

The name 'sensitive' is used to indicate that the feed is hand operated and that the spindle and drilling head are counterbalanced so that the operator can 'feel' the pressure needed for efficient cutting. A table mounted sensitive drill press is shown in Figure 9.3.

The drill press has the same motions as the previous one plus a telescoping screw for raising and lowering the table and a sliding 'drill head'. These two features allow easier handling of parts of varying heights.

9.2.3 Radial Drill

For handling medium to very large size

castings, weldments, or forgings, radial drills are ideal. The length of the arm along which the spindle housing rides specifies their size. This arm can be from 3 to 12 feet long. The column that holds the arm may be from 10 to 30 inches in diameter. A radial drill is shown in Figure 9.4.

For very large work, the arm may be rotated 180 degrees and work placed on the shop floor. Speeds and feeds are dialed in by the machine operator and are the same as for other drill presses. Drilling is either hand or power feed.

9.3 Drilling Machine Components

Rigid and accurate construction of drilling machines is important to obtain proper results with the various cutting tools used. The sensitive drilling machine construction features are discussed in this section because its features are common to most other drilling machines.

Base: The base is the main supporting member of the machine. It is heavy gray iron or ductile iron casting with slots to support and hold work that is too large for the table

Column: The round column may be



FIGURE 9.3: A table mounted sensitive drill used for drilling small holes. (Courtesy Clausing Industries, Inc.)

made of gray cast iron or ductile iron for larger machines, or steel tubing for smaller bench drill presses. It supports the table and the head of the drilling machine. The outer surface is machined to function as a precision way of aligning the spindle with the table.

Table: The table can be adjusted up or down the column to the proper height. It can also be swiveled around the column to the desired working position. Most worktables have slots and holes for mounting vises and other work-holding accessories. Some tables are semi universal, meaning that they can be swiveled about the horizontal axis.

Head: The head houses the spindle, quill, pulleys, motor, and feed mechanism. The V-belt from the motor drives a pulley in the front part of the head, which in turn drives the spindle. The spindle turns the drill. Two head assemblies are shown in Figure 9.2 b and c. Speeds on a stepped V pulley drive are changed by changing the position of the V-belt (Fig. 9.2b) Speeds on a variable-speed drive mechanism are changed by a hand wheel on the head. (Fig. 9.2c) The spindle must be revolving when this is done.

Quill assembly: The spindle rotates within the Quill (Fig. 9.5) on bearings.

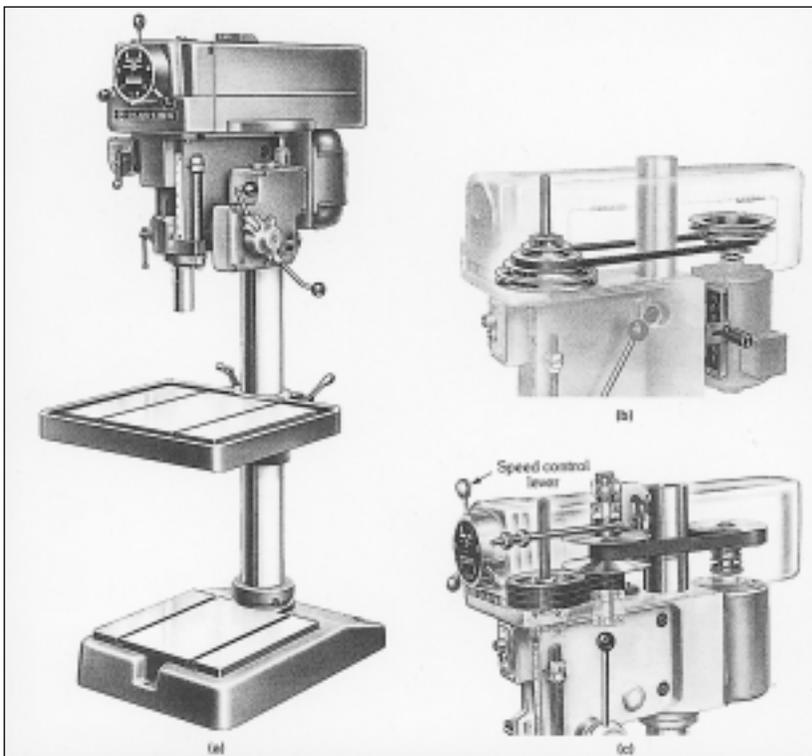


FIGURE 9.2: a) a sensitive drill press is used for drilling holes; b) speeds on a stepped V pulley drive are changed by hanging the position of the V belt; c) speeds on a variable-speed drive mechanism are changed by the handwheel on the head. (Courtesy Clausing Industries, Inc.)

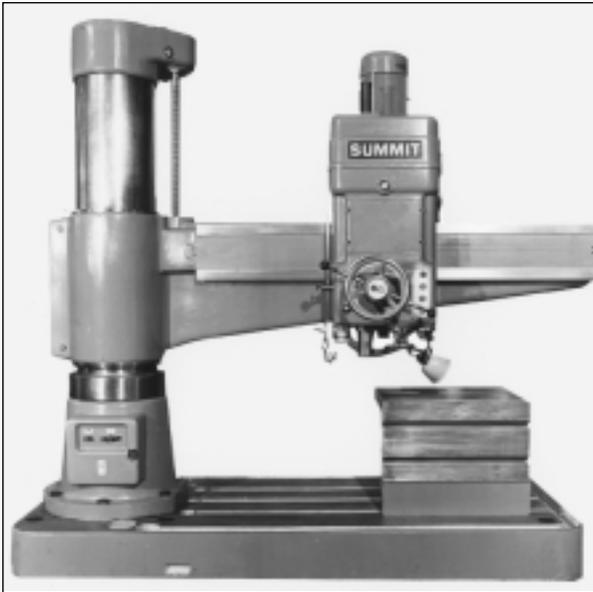


FIGURE 9.4: Radial drills are used to machine large castings, weldments or forgings. (Courtesy Summit Machine Tool Manufacturing Corp.)

The quill moves vertically by means of a rack and pinion. The quill assembly makes it possible to feed or withdraw the cutting tool from the work. Located on the lower end of the spindle is either a Morse tapered hole or a threaded stub where the drill chuck is mounted. For drilling larger holes, the drill chuck is removed and Morse tapered cutting tools are mounted.

Size Classification: The size (capacity) of a drilling machine is determined by all the following features:

- Twice the distance from the center of

the spindle to the inner face of the column

- The maximum length of quill travel
- The size of the Morse taper in the spindle
- The horsepower of the motor.

9.4 Drilling Systems

Drilling systems are usually automated and computer controlled. Speeds, feeds, and depth of cut are often pre-set. Such systems combine drilling operations with reaming, tapping, countersinking, etc. Figure 9.6 shows a 3-axis CNC drilling machine.

9.4.1 Multi Spindle Drilling

This type of drilling can be done on drill presses by using special attachments. The spindle locations are adjustable, and the number of spindles may be from two to eight. Drills, reamers, countersinks, etc., can be used in the spindles. The RPM and feed rate of all spindles in one drill head are the same, and the horsepower needed is the sum of the power for all cutting tools used. In this type of machine, a large number of holes may be drilled at one time. Several different diameters of drills may be used at the same time.

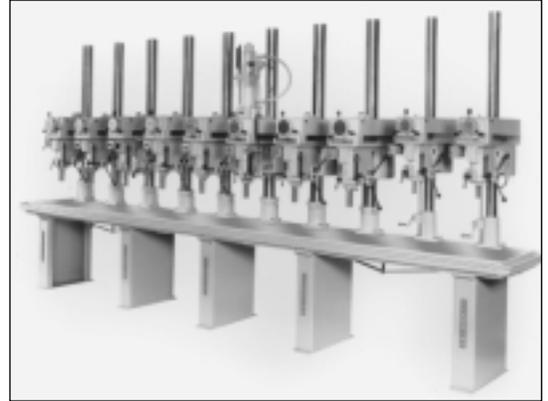


FIGURE 9.7: Gang drilling machines permit economical ways to perform several different operations. (Courtesy Clausing Industries, Inc.)

9.4.2 Gang Drilling

An economical way to perform several different operations on one piece is by gang drilling as shown in Figure 9.7. This might include drilling two or more sizes of holes, reaming, tapping, and countersinking. The work is held in a vise or special fixture and is easily moved along the steel table from one spindle to the next.

The drill presses usually run continuously so the operator merely lowers each spindle to its preset stop to perform the required machining operation.

9.4.3 Turret Drill

Turret drills (Fig. 9.1b) with either six or eight spindles enable the operator to use a wide variety of cutters and yet

move the workpiece only a few inches, according to the hole spacing. The turret can be rotated (indexed) in either direction, and then lowered, by hand or automatically, to make the cut.

Some turret drills have automatic, hydraulically controlled spindles. Speeds, feeds, and depths of cut can be preset for fast production. Figure 9.1b shows an automatic machine. These machines are also made with the entire operation computer controlled, (CNC turret

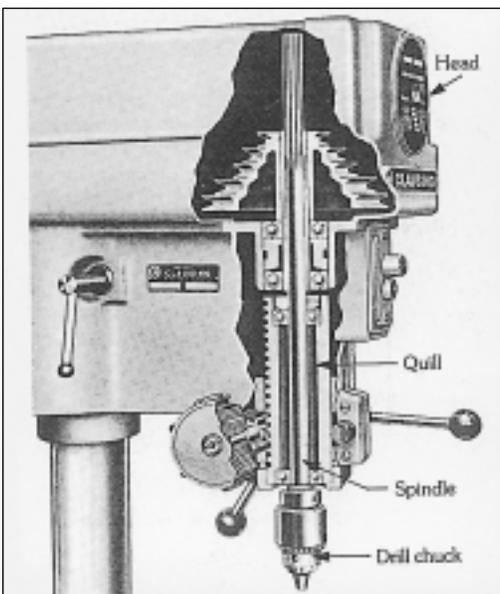


FIGURE 9.5: The spindle rotates within the quill. The quill moves vertically by means of a rack and pinion. (Courtesy: Clausing Industries, Inc.)

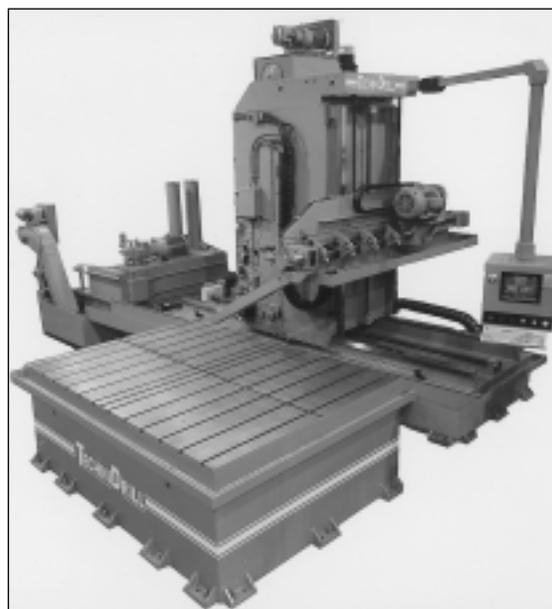


FIGURE 9.6: Shown is a 3-axis CNC drilling Machine (Courtesy: TechniDrill Systems, Inc.)

drill), so that the operator merely has to load and unload the parts. A numerically controlled turret drill is shown in Figure 9.8.

9.5 Operation Set-up

In drilling operations the three most common work holding methods are:

- Vises
- Angle Plate
- Drill Jigs

Vises: Vises are widely used for holding work of regular size and shape, such as flat, square, and rectangular pieces. Parallels are generally used to support the work and protect the vise from being drilled. Figure 9.9 shows a typical vise. Vises should be clamped to the table of the drill press to prevent them from spinning during operation. Angular vises tilt the workpiece and provide a means of drilling a hole at an angle without tilting the table. An angular vise is shown in Figure 9.10.

Angle Plates: An angle plate supports work on its edge. Angle plates

accurately align the work perpendicular to the table surface, and they generally have holes and slots to permit clamping to the table and holding of the workpiece.

Drill Jigs: A drill jig is a production tool used when a hole, or several holes, must be drilled in a large number of identical parts. Figure 9.11 shows

a diagram of a typical drill jig. The drill jig has several functions. First, it is a work holding device, clamping the work firmly. Second, it locates work in the correct position for drilling. The third function of the drill jig is to guide the drill straight into the work. This is accomplished by use of drill bushings.

9.5.1 Tool Holding Devices

Some cutting tools used in drilling can be held directly in the spindle hole of the machine. Others must be held with a drill chuck, collet, sleeve, socket, or one of the many tool-holding devices shown in Figure 9.12.

Drill Chucks: Cutting tools with straight shanks are generally held in a drill chuck. The most common drill chuck uses a key to lock the

cutting tool. Drill chucks, both with key and keyless, are shown in Figures 9.13.

Sleeves: Cutting tools with tapered shanks are available in many different sizes. When a cutting tool that has a smaller taper than the spindle taper is used, a sleeve must be fitted to the shank of the cutting tool.

Sockets: If the cutting tool has a tapered shank larger than the spindle taper, a socket is used to reduce it to the correct size. Figure 9.14 shows various size keyless drill chucks with one straight and two tapered shank mountings.

9.6 Deep-hole Drilling

The term ‘deep holes’ originally referred to hole depths of over 5 x the diameter. Today, deep-hole drilling is a collective name for methods for the

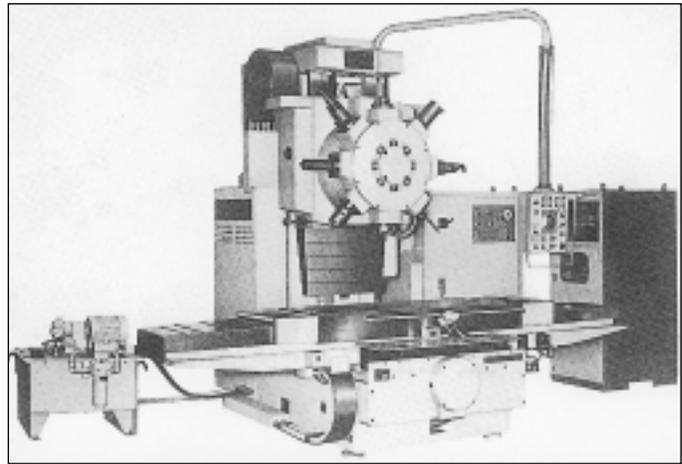


FIGURE 9.8: Numerically controlled turret drill automatically positions the worktable. (Courtesy: Kanematsu USA, Inc.)

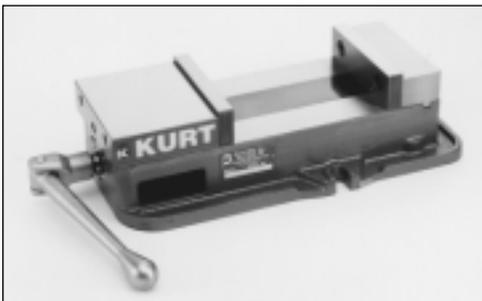


FIGURE 9.9: Vises should be clamped to the table of drill presses to prevent them from spinning. (Courtesy Kurt Manufacturing Co.)



FIGURE 9.10: The angle vise tilts the workpiece and provides a means of drilling a hole at an angle. (Courtesy: Palmgren Steel Products, Inc.)

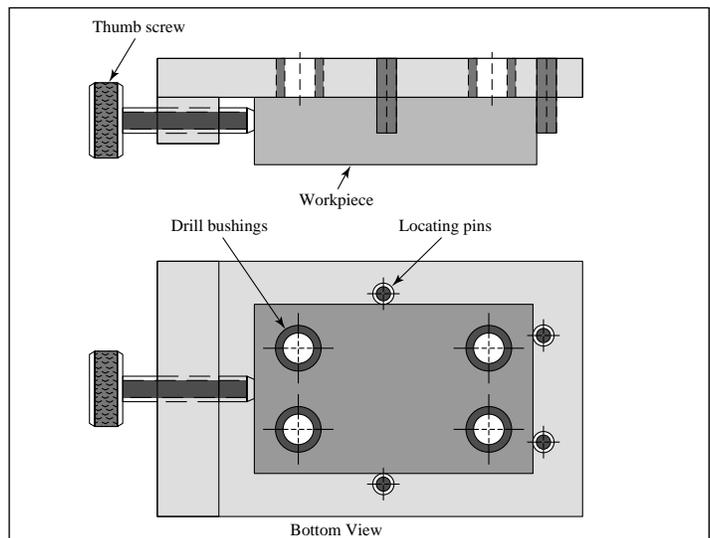


FIGURE 9.11: Drill jigs locate and clamp workpieces, and guide the drill through a drill bushing.



FIGURE 9.12: Various tool-holding devices such as chucks, collets, sleeves, and sockets are shown. (Courtesy Lyndex Corp.)

machining of both short and deep holes. Deep-hole drilling is the preferred method for drilling hole depths of more than 10 x the diameter, but because of the method's high metal-removal capac-



FIGURE 9.13: Key and keyless chucks are used to hold drills for holmaking operations. (Courtesy Bridgeport Machine, Inc.)



FIGURE 9.14: Various size drill chucks are shown. (Courtesy: Royal Products)

ity and precision, it is also competitive for small holes down to 2 x the diameter.

During drilling, it is important that the chips be broken and that they can be transported away without jamming and affecting the drilled surface. In deep-hole drilling, cutting fluid supply and chip transport have been provided for by the development of three different systems that permit trouble-free machining of hole depths of more than 100 x the diameter. The three systems are called: the Gun Drilling System, the

Ejector System (two-tube system) and the Single Tube System (STS).

Some of the tools used in deep-hole drilling are shown in Figure 9.15. Hyper Tool manufactured the gun drills, and Sandvik manufactured the indexable tools.

9.6.1 Gun Drilling Systems

The gun drill system uses the oldest principle for cutting fluid supply. The cutting fluid is supplied through a duct inside the drill and delivers coolant to the cutting edge, after which it removes the chips through a V-shaped chip flute along the outside of the drill. Due to the V-groove, the cross section of the tube occupies 3/4 of its circumference. Figure 9.16 shows a gun drilling system and its component parts.

Gun Drills

Gun drills belong to the pressurized coolant family of hole making tools.



FIGURE 9.15: Deep-hole drilling tools; the gun drills were manufactured by Hyper Tool and the indexable tools were manufactured by Sandvik. (Courtesy TechniDrill Systems, Inc.)

They are outstanding for fast, precision machining regardless of hole depth. As a rule, a gun drill can hold hole straightness within 0.001 inch per inch (IPI) of penetration, even when the tool is reasonably dull. For most jobs a gun drill can be used to cut from 500 to 1000 inches in alloy steel before re-sharpening is necessary. In aluminum, it might be 15,000 inches, while in cast iron it is usually around 2000 inches. Figure 9.17a shows a gun drilling tool and Figure 9.17b shows the gun drilling process.

Depending on the tool's diameter, a gun drill is seldom run at feed rates exceeding 0.003 inches per revolution (IPR). This is extremely light compared to twist drill feeds, which typically range from 0.005 IPR to 0.010 IPR. But gun drilling does use a relatively high

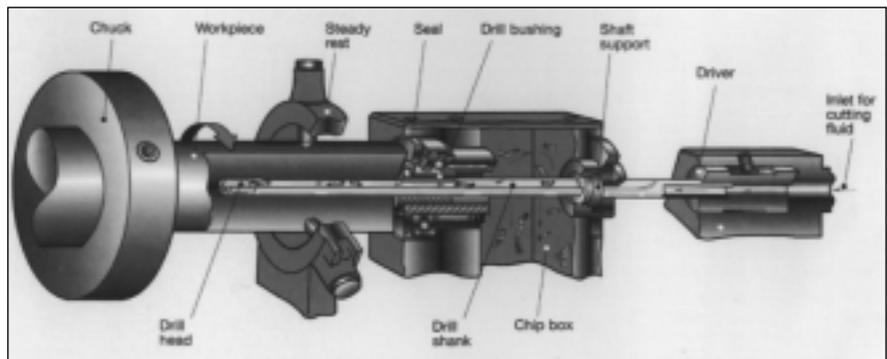


FIGURE 9.16: Schematic diagram of a gun drilling system with major components. (Courtesy Sandvik Coromant Co.)

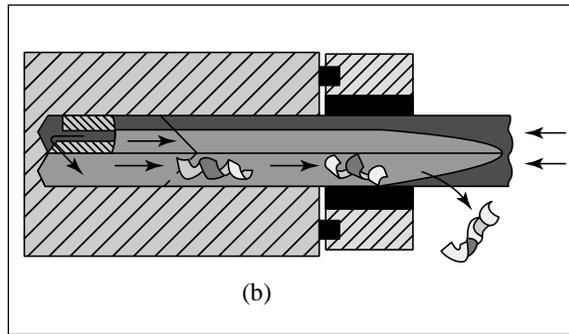
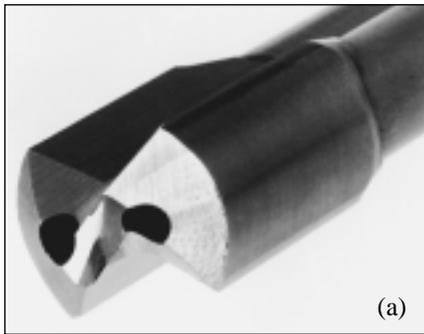


FIGURE 9.17: (a) Gun drilling head. (b) A drawing of a gun drilling process. (Courtesy Star Cutter Co.)

speed compared to high speed steel (HSS) twist drilling. This accounts for the high metal-removal rates associated with the process. In aluminum, speeds may be 600 surface feet per minute (SFPM), in steels from 400 SFPM to 450 SFPM.

Speeds and feeds for gun drilling are based on the workpiece material and shop floor conditions. Published charts only provide starting points. On-the-floor experimentation is critical to determine the right combination for maximum tool life.

Gun Drill Body

The body of a gun drill is typically constructed from 4120 aircraft quality steel tubing that is heat treated to between 35 to 40 Rc. A 4140 steel driver is brazed to one end of the tube and a carbide tool tip is brazed to the other end. Figure 9.18 shows five different tool tip geometries with various coolant hole placements.

There are two body styles for multiple flute tools; milled and crimped. The former is a thick wall tubular shaft with the flutes milled into the body. The latter is a thin wall tubular shaft that has

the flutes swaged into it. The number of flutes depends on the material being cut. When drilling in a material that breaks easily into small chips, such as cast iron, a two flute tool is the choice. On the other hand, for a material such as D2 tool steel, a single flute design is preferred. In this case, chips tend to be stringy and a single flute tool will minimize the chance of jamming as they are removed from the hole.

Figure 9.19 shows both a crimp style gun drill body with two flutes produced by swaging and a conventional milled style gun drill. The coolant holes in the crimped body have an irregular shape that permits carrying a much larger volume of coolant than comparable holes in a conventional equivalent diameter tool body. Also, the flutes that are formed are much deeper than milled tools because allowance does not have to be made for wall thickness between flute and coolant hole. These deeper flutes improve the chip removal efficiency of the tool.

Gun Drill Tip

A conventional gun drill has a hole in its carbide tip underneath the cutting

edge. Pressurized cutting fluid is pumped through the tool's body and out the hole (see Figure 9.18). The fluid serves a three-fold purpose: it lubricates and cools the cutting edge; it forces the chips back along the flute in the tool body; and it helps to stiffen the shank of the tool.

A new design has one hole in the top of the tool tip that effectively directs

fluid at the cutting edge. The other hole that is in the conventional location helps to provide the chip ejection function. Total flow of cutting fluid is doubled with this two-hole arrangement. More importantly, the design produces chips about half the size of a conventional gun drill of the same diameter using the same speed and feed rate, so that packing of chips along the tool's shank is avoided in most materials.

The most common tool tip material is C2 carbide, which is one of the harder grades and is generally associated with cast iron applications. Because excessive tool wear is a major problem when cutting steel, a hard grade such as C2 is recommended, even though C5 carbide is labeled as the steel machining grade in most text books. C5 carbide is a shock resistant grade, not a wear-resistant grade, so that it is not as suitable for a gun drill tool tip. C3 carbide is harder than C2, and is used for certain applications; however, greater care must be taken when re-sharpening this material because it is easier to heat check the cutting edge.

Recently, coatings such as titanium nitride are being applied to gun drill tips to extend tool life. Physical Vapor Deposition (PVD) is the only practical process for depositing coatings on precision tools such as gun drills, but the results have not been encouraging. Unlike coating high-speed steel tools, PVD coating of a carbide gun drill tip does not seem to form a good metallurgical bond. The coating wipes off during the metal cutting process. Using Chemical Vapor Deposition (CVD) will form a metallurgical bond between the coating and carbide substrate, but the high heat required by the process distorts the tool. Hopefully these problems will be resolved in the near future.



FIGURE 9.18: Shown are five different tool tip geometries with various coolant hole placements (Courtesy: Star Cutter Co.)

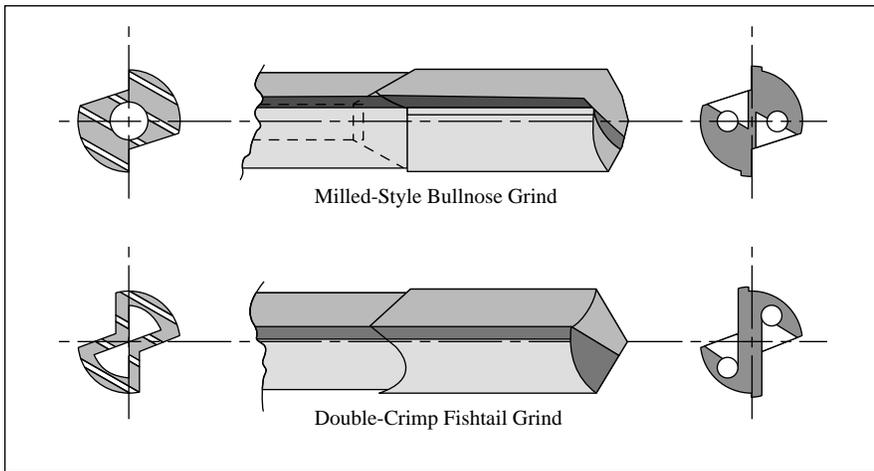


FIGURE 9.19: There are two body styles of multifluted gun drills: milled style and double-crimp style.

9.6.2 The Ejector System

The Ejector System consists of drill head, outer tube, inner tube, connector, collet and sealing sleeve. The drill head is screwed to the drill tube by means of a four-start square thread. The inner tube is longer than the outer tube. The drill tube and the inner tube are attached

to the connector by means of a collet and a sealing sleeve. The collet and sealing sleeve must be changed for different diameter ranges. Figure 9.20 shows the Ejector System and its components.

9.6.3 The Single Tube System (STS)

The Single Tube System is based on

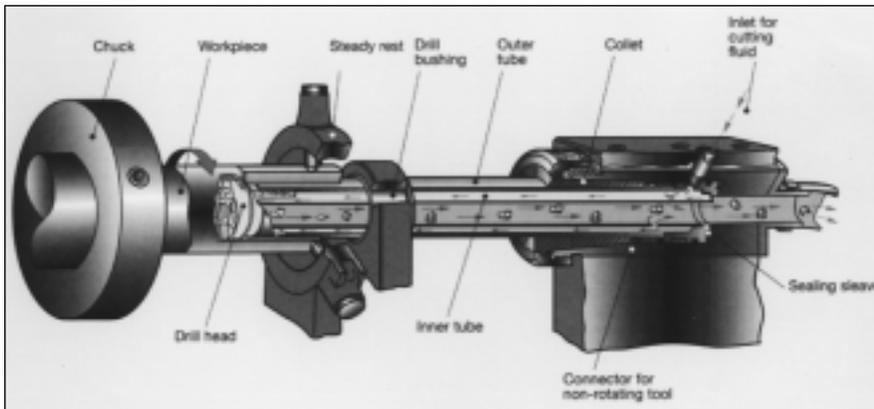


FIGURE 9.20: The ejector system and its major components. (Courtesy Sandvik Coromant Co.)

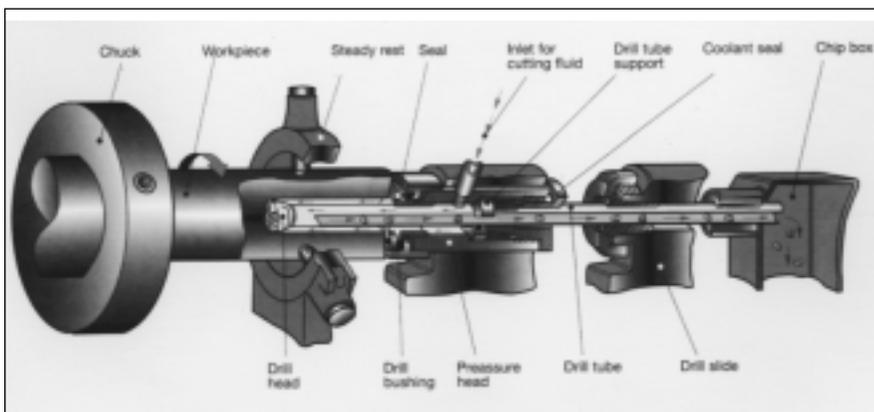


FIGURE 9.21: The single-tube system (STS) and its major components. (Courtesy Sandvik Coromant Co.)

external cutting fluid supply and internal chip transport. As a rule, the drill head is screwed onto the drill tube. The cutting fluid is supplied via the space between the drill tube and the drilled hole. The cutting fluid is then removed along with the chips through the drill tube. The velocity of the cutting fluid is so high that chip transport takes place through the tube without disturbances. Since chip evacuation is internal, no chip flute is required in the shank, so tip cross-section can be made completely round, which provides much higher rigidity than the gun drill system. Figure 9.21 shows the Single Tube System and its components.

9.6.4 Comparison of STS and Ejector Systems

Both the Single Tube System and the Ejector System have wide ranges of application, but there are times when one system is preferable to the other.

STS is preferable in materials with poor chip formation properties such as stainless steel, low carbon steel, and materials with an uneven structure, when chip breaking problems exist. STS is also more advantageous for long production runs, uniform and extremely long workpieces and for hole diameters greater than 7.875 inches.

The Ejector System requires no seal between the workpiece and the drill bushing. The system can therefore be adapted easily to existing machines and is preferable in NC lathes, turning centers, universal machines and machining centers. Since the cutting fluid is supplied between the outer and inner tubes, no space is required between the drill tube and the hole wall as in the case of STS drilling. The Ejector System is therefore often used for machining in workpieces where sealing problems can arise. The Ejector System can be used to advantage when it is possible to use a predrilled hole instead of a drill bushing for guidance, for example in machining centers.

9.6.5 Operational Requirements

Machining with high cutting speeds and high demands on surface finishes and tolerances requires a machine tool that is both very rigid and very powerful. It is possible to use conventional machines with sufficient power and rigidity.

Machine Requirements: The high

feed speeds that characterize deep hole drilling impose high demands on available power. In order to achieve good precision, the machine must be rigid and the spindle bearings free of play. Good chip breaking often requires high feed and the feed must be constant, otherwise the chip breaking may vary, leading to chip jamming. The best possible chip breaking can be



FIGURE 9.22: The length of a deep-hole drilling machine depends on the diameter and the length of the workpiece. (Courtesy Sandvik Coromant Co.)

obtained with infinitely adjustable feed. It is important that the machine be equipped with safety devices to protect the machine, the tool and the workpiece. The purpose of the safety device is to stop the machine automatically in the event of overloading. The machine spindle should not be able to start until the pressure of the cutting fluid has reached a preset minimum. The temperature and quantity of the cutting fluid should also reach a correct level before the machine starts.

Best are overload protections that are connected to the feed pressure. It is extremely important that the overload limits be set no more than 10 - 13 percent above the actual drill pressure for each drill diameter and feed. The feed will then be able to stop before the drill is damaged.

Machine Types: The design of deep hole drilling machines varies. The lengths of the machines are adapted to the special diameter ranges and lengths of the workpiece. A special very long machine is shown in Figure 9.22

Deep hole drilling machines are often designed to permit a choice between a rotating workpiece, a rotating tool or both rotating workpiece and rotating tool. In the machining of asymmetric workpieces, the machine works with a rotating drill and a non-rotating workpiece, since the workpiece cannot rotate at sufficient speed. In the machining of long, slender workpieces, a non-rotating drill is fed into a rotating workpiece. When the hole must meet high straight-

ness requirements, both the drill and the workpiece rotate. The direction of rotation of the drill is then opposite to that of the workpiece.

The Single Tube System is difficult to adapt to standard machines, while Ejector drilling and, in some cases, gun drilling, can be done relatively simply in conventional machines. The largest extra costs are then for the cutting fluid system, chip removal arrangement, fil-

ter tank and pump. Figure 9.23 shows a special gun-drilling machine to drill six camshafts simultaneously. This machine includes auto loading and unloading of parts.

Chip Breaking: Of primary importance in drilling operations is transporting the chips away from the cutting edges of the drill. Excessively long and large chips can get stuck in the chip ducts. A suitable chip is as long as it is wide. However, the chips should not be broken harder than necessary, since chip breaking is power consuming and the heat that is generated increases wear on the cutting edges. Chips with a length 3 - 4 times their width can be acceptable, provided that they can pass through the chip duct and drill tube without difficulties. Chip formation is affected by the work material, chip breaker geometry, cutting speed, feed and choice of cutting fluid.

Coolant System: The purposes of the coolant in a drilling system are:

- Support and lubrication of the pads
- Improvement of the tool life
- Dissipation of heat
- Flushing of chips

The coolant system has to provide an adequate supply of clean coolant to the tool at the correct pressure and temperature.

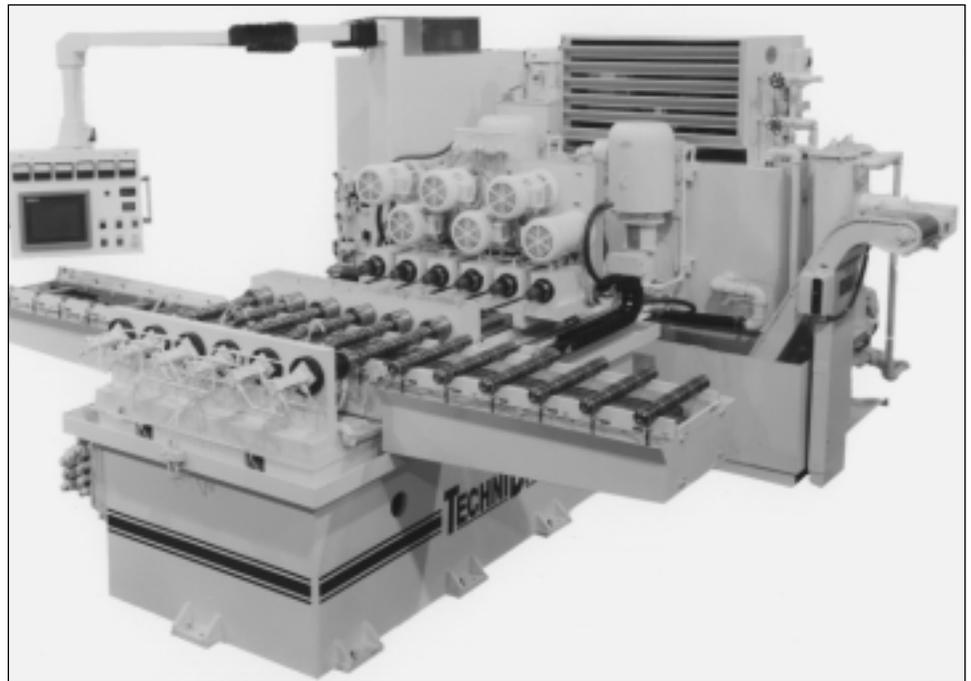


FIGURE 9.23: Special gun drilling machine is shown drilling six camshafts simultaneously. This machine includes automatic loading and unloading of parts. (Courtesy TechniDrill Systems, Inc.)

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>

Prentice Hall: <http://www.prenhall.com>

Chapter 10

Boring Operations

& Machines

10.1 Introduction

Boring, also called Internal Turning, is used to increase the inside diameter of a hole. The original hole is made with a drill, or it may be a cored hole in a casting. Boring achieves three things:

Sizing: Boring brings the hole to the proper size and finish. A drill or reamer can only be used if the desired size is 'standard' or if special tools are ground. The boring tool can work to any diameter and it will give the required finish by adjusting speed, feed and nose radius. Precision holes can be bored using micro adjustable boring bars (Fig. 10.1)

Straightness: Boring will straighten the original drilled or cast hole. Drills, especially the longer ones, may wander off-center and cut at a slight angle because of eccentric forces on the drill, occasional hard spots in the material, or uneven sharpening of the drill (see Fig. 8.10). Cored holes in castings are almost never completely straight. The boring tool being moved straight along the ways with the carriage feed will correct these errors.

Concentricity: Boring will make the hole concentric with the outside diameter within the limits of the accuracy of the chuck or holding device. For best concentricity, the turning of the outside diameter and the boring of the inside diameter is done in one set-up, that is, without moving the work between operations. The basics discussed in Chapters 4 and 5, the Turning Chapters, also apply to boring. However, with boring there are a number of limitations that must be taken into account in order to reach a high stock removal rate combined with satisfactory accuracy, surface finish and tool life. Therefore, in this chapter the limitations that distinguish internal turning from external turning will be discussed in greater detail. A typical boring operation is shown in Figure 10.2.



FIGURE 10.1: Adjustable boring bar for precision holes. (Courtesy: National Acme Co. Div. DeVlieg-Bullard, Inc.)

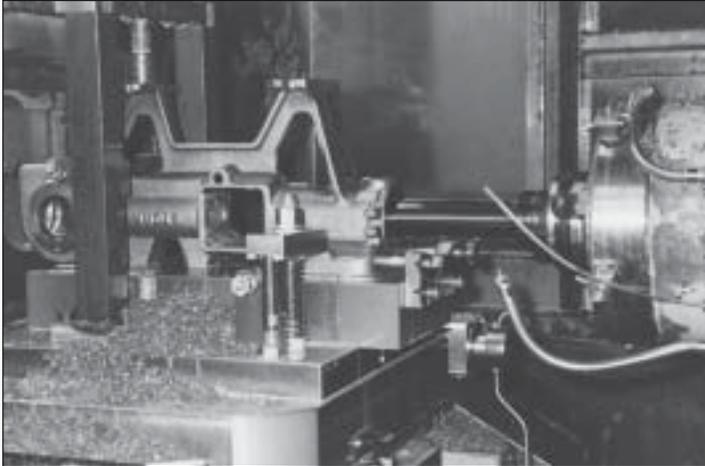


FIGURE 10.2: Typical horizontal boring operation. (Courtesy Sandvik Coromant Co.)

10.2 Boring Operations

Most of the turning operations that occur with external turning are also to be found in boring. With external turning, the length of the workpiece does not affect the tool overhang and the size of the tool holder can be chosen so that it withstands the forces and stresses that arise during the operation. However, with internal turning, or boring, the choice of tool is very much restricted by the work piece's hole diameter and length.

A general rule, which applies to all machining, is to minimize the tool overhang in order to obtain the best possible stability and thereby accuracy. With boring the depth of the hole determines the overhang. The stability is increased when a larger tool diameter is used, but even then the possibilities are limited since the space allowed by the diameter of the hole in the workpiece must be taken into consideration for chip evacuation and radial movements.

The limitations with regards to stability in boring mean that extra care must be taken with production planning and preparation. By understanding how cutting forces are affected by the tool geometry and the cutting data chosen, and also understanding how various types of boring bars and tool clamping will affect the stability, deflection and vibration can be kept to a minimum.

10.3 Cutting Forces

On engagement, the tangential force and the radial cutting force will attempt to push the tool away from the

workpiece, which results in the deflections.

The tangential force will try to force the tool downwards and away from the centerline. Due to the curving of the internal hole diameter the clearance

angle will also be reduced. Therefore with small diameter holes it is particularly important that the clearance angle of the insert be sufficient in order to avoid contact between the tool and the wall of the hole.

The radial deflection will reduce the cutting depth. In addition to the diametrical accuracy being affected, the chip thickness will change with the varying size of the cutting forces. This causes vibration, which is transferred from the cutting edge to the tool holder. The stability of the tool and clamping will be the factor that determines the magnitude of the vibration and whether it is amplified or dampened.

Insert Geometry: The geometry of the insert has a decisive influence on the cutting process. A positive insert has a positive rake angle. The insert's edge angle and clearance angle together will equal less than 90 degrees. A positive rake angle means a lower tangential cutting force. However, a positive rake angle is obtained at the cost of the clearance angle or the edge angle. If the clearance angle is small there is a risk of abrasion between the tool and workpiece and the friction can give rise to vibration. In those

cases where the rake angle is large and the edge angle is small, a sharper cutting edge is

obtained. The sharp cutting edge penetrates the material more easily but it is also more easily changed or damaged by edge or other uneven wear.

Edge wear means that the geometry of the insert is changed, resulting in a reduction in the clearance angle. Therefore, with finish machining it is the required surface finish of the workpiece that determines when the insert must be changed. Generally, the edge wear should be between .004 and .012 inches for finishing and between .012 and .040 inches for rough machining.

Lead Angle: The lead angle affects the axial and radial directions of the cutting forces. A small lead angle produces a large axial cutting force component while a large lead angle results in a larger cutting force in the radial direction. The axial cutting force has a minimal negative effect on the operation since the force is directed along the boring bar. To avoid vibrations, it is consequently advantageous to choose a small lead angle but, since the lead angle also affects other factors such as the chip thickness and the direction of the chip flow, a compromise often has to be made.

The main disadvantage of a small lead angle is that the cutting forces are distributed over a shorter section of the cutting edge than with a large lead angle. Furthermore, the cutting edge is exposed to abrupt loading and unloading when the edge enters and leaves the workpiece. Since boring is done in most cases, in a pre-machined hole and is designated as light machining, small lead angles generally do not cause a problem. Lead angles of 15 degrees or less are normally recommended. However, at a lead angle of 15 degrees the radial cutting force will be virtually double that of the cutting force with a 0 degree lead angle. A

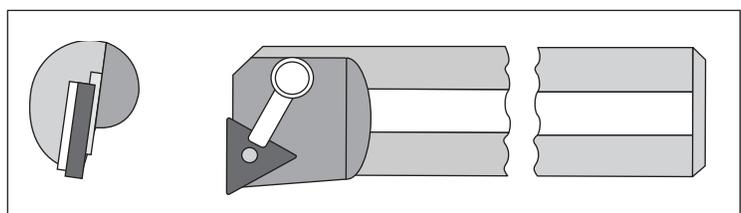


FIGURE 10.3: Typical indexable insert boring bar with 0 deg lead angle.

typical indexable insert boring bar with a 0 degree lead angle is shown in Figure 10.3.

Nose Radius: The nose radius of the insert also affects the distribution of cutting forces. The greater the nose radius, the greater the radial and tangential cutting forces, and the emergence of vibration. However, this is not the case with radial cutting forces. The deflection of the tool in a radial direction is instead affected by the relationship between the cutting depth and the size of the nose radius. If the cutting depth is smaller than the nose radius, the radial cutting forces will increase with increased cutting depth. If the cutting depth is equal to or greater than the size of the nose radius, the radial deflection will be determined by the lead angle. Therefore, it's a good idea to choose a nose radius which is somewhat smaller than the cutting depth. In this way the radial cutting forces can be kept to a minimum, while utilizing the advantages of the largest possible nose radius, leading to a stronger cutting edge, better surface finish and more even pressure on the cutting edge.

10.4 Chip Breaking and Evacuation

Obtaining relatively short, spiral shaped chips is the goal in internal turning. These are easy to evacuate and do not place such large stresses on the cutting edge when chip breaking occurs. Hard breaking of the chips, i.e. when short chips are obtained, demands power and can increase vibration in the boring bar. However, this is preferred over having long chips, which can make chip evacuation more difficult. Chip breaking is affected by a number of factors such as the insert geometry, nose radius, lead angle, cutting depth, feed and cutting speed. Generally, reduced feed and/or increased cutting speed results in longer chips. The shape of the chip breaker affects the radius of the chip, where any built-up edge or crater wear can also act as chip breaker. The direction in which the chips flow and the way that they turn in the spiral, is affected by the lead angle or the combination of cutting depth and nose radius.

The parameters that affect chip control also affect the direction and size of the cutting force. Therefore, it is necessary to choose a grade and

insert geometry that, together with the selected machining parameters, fulfill the requirements for good chip control. At the same time, the machine, boring bar and tool clamping must provide sufficient stability in order to resist the cutting forces that arise.

During boring operations the chip flow can be critical, particularly when deep holes are being machined. The centrifugal force presses the chips outwards. With boring, this means that the chips remain in the workpiece. The remaining chips could get pressed into the machined surface or get jammed and damage the tool. Therefore, as with internal turning, tools with an internal cutting fluid supply are recommended. The chips will then be flushed out of the hole effectively. Compressed air can be used instead of cutting fluid and with trough holes; the chips can be blown through the spindle and collected in a container.

10.5 Boring Rigidity

Part geometries can have external turning operations as well as internal operations. Internal single point turning is referred to as boring, and can be utilized for either a roughing or finishing operation. Single point boring tools consist of a round shaft with one insert pocket designed to reach into a part hole or cavity to remove internal stock in one or several machine passes. Figure 10.4 shows various sizes and styles of boring bars.

The key to productivity in boring operations is the tool's rigidity. Boring bars are often required to

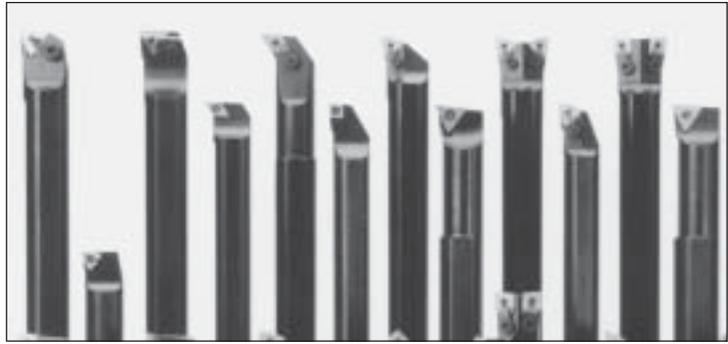


FIGURE 10.4: Various sizes and styles of boring bars. (Courtesy Dorian Tool)



FIGURE 10.5: Boring bars are often required to reach long distances into parts to remove stock. (Courtesy Sandvik Coromant Co.)

reach long distances into parts to remove stock (see Fig. 10.5). Hence, the rigidity of the machining operation is compromised because the diameter of the tool is restricted by the hole size and the need for added clearance to evacuate chips. The practical overhang limits for steel boring bars is four times their shank diameter. When the tool overhang exceeds this limit, the metal removal rate of the boring operation is compromised significantly due to lack of rigidity and the increased possibility of vibration.

Boring Bar Deflection: The size of the boring bar's deflection is dependent on the bar material, the diameter, the overhang and size of the radial and tangential cutting forces. Boring bar deflection can be calculated, but such calculations are beyond the scope of this book.

Increasing the diameter of the tool to create an increased moment of inertia can counteract this deflection. Choosing a boring bar made of a material that has a higher coefficient of elasticity can also counteract deflection. Since steel has a lower coefficient of elasticity than cemented carbide. Cemented carbide boring bars are better for large overhangs.

Compensating for Deflection: Even with the best tool clamping,

some vibration tendency will occur in boring. Radial deflection affects the machined diameter. Tangential deflection means that the insert tip is moved in a downward direction away from the centerline. In both cases the size and direction of the cutting forces are affected by changes in the relationship between the chip thicknesses and insert geometry.

If the exact size of the deflection of the insert tip is known in advance, then the problem can be avoided. By positioning the insert tip distance above the centerline, the insert under the effect of the tangential force, will take up the correct position during machining. In the same way, setting the machine at a cutting depth that is greater than the desired cutting depth compensates for the radial deflection. When cutting begins, the radial cutting force reduces the cutting depth.

Even if the approximate deflection can be calculated, the practical outcome will be somewhat different because the clamping is never absolutely rigid and because it is impossible to calculate the cutting force exactly.

Boring Bar Clamping: The slightest amount of mobility in the fixed end of the boring bar will lead to deflection of the tool. The best stability is obtained with a holder that completely encases the bar. This type of holder is available in two styles: a rigid (Fig. 10.6a) or flange mounted bar, or a divided block (Fig. 10.6b) that clamps when tightened. With a rigidly mounted bar, the bar is either preshrunk into the holder and/or welded in. With flange mounting, a flange with a through hole is normally used. The flange is usually glued onto the

shank of the bar at a distance that gives the required overhang. The bar is then fed into the holder and clamped by means of a screw connection or by being held in the turret.

Less efficient are those tool-clamping methods where the screw clamps onto the bar. This form generally results in vibration and is not recommended. Above all, this method must not be used for the clamping of cemented carbide bars. Cemented carbide is more brittle than steel and cracks will occur as a result of vibration, which in turn may result in breakage.

10.6 Boring Bars

Boring bars are made in a wide variety of styles as shown in Figure 10.4. Single-point boring bars (Fig. 10.7) are easily ground but difficult to adjust when they are used in turret and automatic lathes and machining centers, unless they are held in an adjustable holder (Fig. 10.8).

More expensive boring bars are provided with easily adjustable inserts. These bars are made in standard sizes, with a range of 1/4 to 1/2 inch on the diameter. A fine adjustment is included in increments of 0.001 inch or in some cases 0.0001 inch. They are standard up to about 6 inches in diameter. A boring bar with adjustments is shown in Figure 10.9. A different style of adjustable boring bar with two indexable inserts is shown in Figure 10.10.

Standard boring bars with interchangeable heads to permit various internal operations such as turning, profiling, grooving, and threading are shown in Figure 10.11.

Many times it may be economical to order special bars with two or more preset diameters, set at the proper distance apart. These spe-



FIGURE 10.7: Single-point boring bar. (Courtesy Morse Cutting Tools)



FIGURE 10.8: Adjustable boring head for single-point boring tools. (Courtesy Kennametal Inc.)

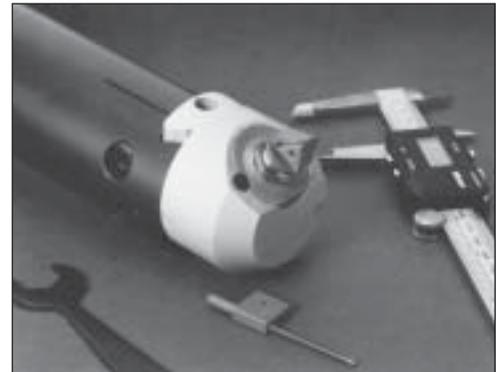


FIGURE 10.9: Adjustable boring bar with fine-tuning adjustment. (Courtesy Valenite Inc.)



FIGURE 10.10: Adjustable boring bar with two indexable inserts. (Courtesy Kennametal Inc.)

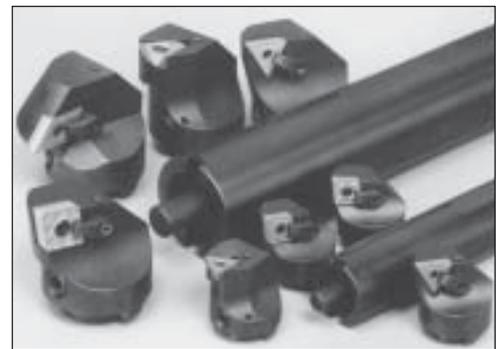


FIGURE 10.11: Standard boring bar with interchangeable heads for various internal operations such as turning, profiling, grooving, and threading. (Courtesy Valenite Inc.)

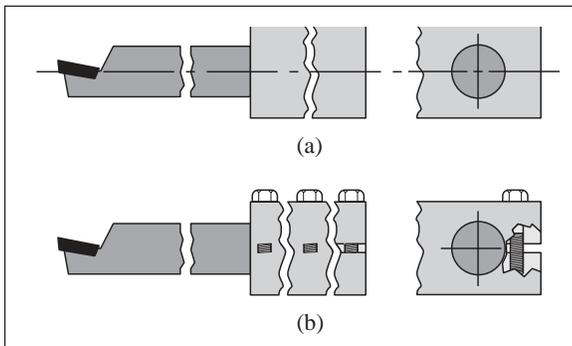


FIGURE 10.6: Two proper boring bar clamping methods.



FIGURE 10.12: Special multi-operation boring bar. (Courtesy: National Acme Co. Div. Devlieg-Bullard, Inc.)

cial bars cost more and are generally only used when large quantities make their use economical. Sometimes this may be the only way to hold the required tolerances and concentricity. Such a special boring bar is shown in Figure 10.12.



FIGURE 10.13: A twin-cutter adjustable boring head with indexable Trigon inserts. (Courtesy Komet of America, Inc.)



FIGURE 10.14: Various indexable replaceable cartridges used in special boring heads. (Courtesy Valenite Inc.)

Other special boring bars, sometimes called boring heads, are designed with replaceable cartridges. A twin cutter adjustable boring tool is shown in Figure 10.13. Various replaceable cartridges for special boring heads are shown in Figure 10.14.

Boring Bar Types: Boring bars are available in steel, solid carbide, and carbide-reinforced steel. The capacity to resist deflection increases as the coefficient of elasticity increases. Since the elasticity coefficient of carbide is three times larger than that of steel, carbide bars are preferred for large overhangs. The disadvantage of carbide is its poor ability to withstand tensile stresses. For carbide-reinforced bars, the carbide sleeves are pre-stressed to prevent tensile stresses.

Boring bars can be equipped with ducts for internal cooling, which is preferred for internal turning. An internal coolant supply provides efficient cooling of the cutting edge, plus better chip breaking and chip evacuation. In this way a longer tool life is obtained and quality problems, which often arise due to chip jamming, are avoided.

Boring Bar Choice: When planning production, it is very important to minimize cutting forces and to create conditions where the greatest possible stability is achieved so that the tool can withstand the stresses that always arise. The length and diameter of the boring bar will be of great significance to the stability of the tool. Since the appearance of the workpiece is the decisive factor when selecting the minimum overhang and maximum tool diameter that can be used, it is important to choose the tool, tool clamping and cutting data which minimize, as much as possible, the cutting forces which arise during the operation. The following recommendations should be followed in order to obtain the best possible stability:

- Choose the largest possible bar diameter, but at the same time ensure that there is enough room for chip evacuation.

- Choose the smallest possible overhang but, at the same time, ensure that the length of the bar allows the recommended clamping lengths to be achieved.
- A 0 degree lead angle should be used. The lead angle should, under no circumstances be more than 15 degrees.
- The indexable inserts should be positive rake that results in lower cutting forces.
- The carbide grade should be tougher than for external turning in order to withstand the stresses to which the insert is exposed when chip jamming and vibration occur.
- Choose a nose radius that is smaller than the cutting depth.

Modern boring bars are designed to take into account the demands that must apply because the operation is performed internally and the dimensions of the tool are determined by the hole depth and the hole diameter. With a positive rake insert geometry, less material deformation and low cutting forces are obtained. The tool should offer good stability to resist the cutting forces that arise and also to reduce deflection and vibration as much as possible. Due to space requirements, satisfactory chip control and good accessibility are also properties of greater importance than with external turning.

10.7 Boring Machines

Boring operations can be performed on other than boring machines, such as lathes, milling machines, and machin-

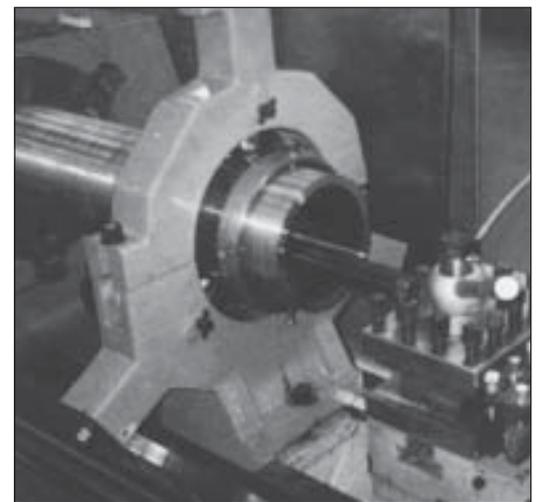


FIGURE 10.15: A typical boring operation performed on a lathe; a steady rest is being used to provide support for the part being machined. (Courtesy Sandvik Coromant Co.)

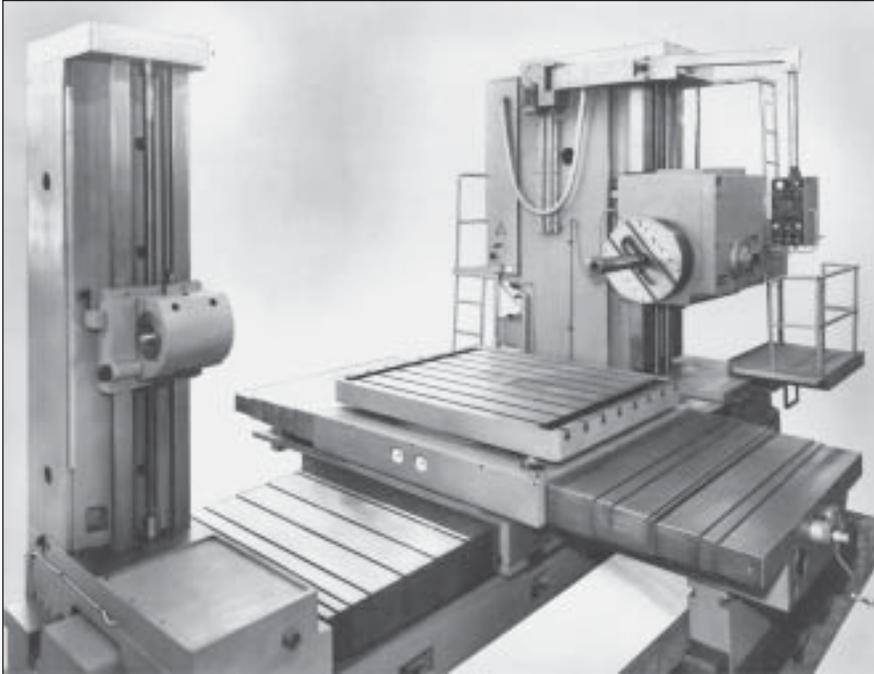


FIGURE 10.16: Table-type horizontal boring machine (HBM) (Courtesy Summit Machine Tool Manufacturing Corp.)

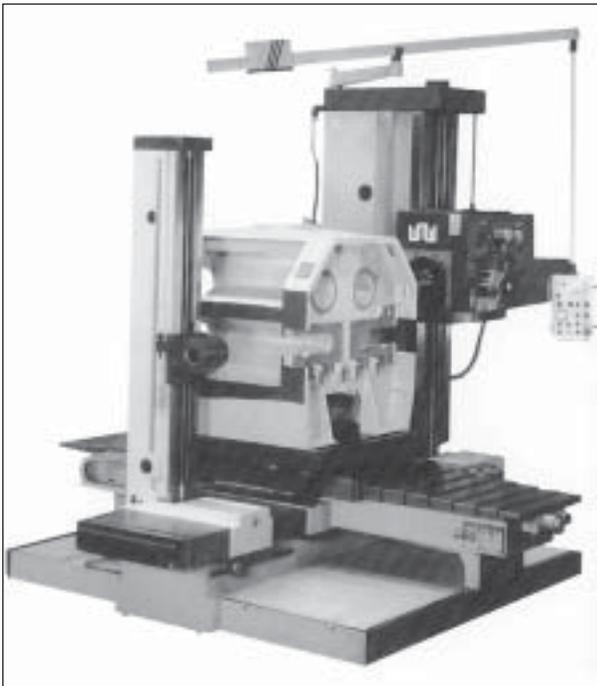


FIGURE 10.17: Large part being machined on a table-type horizontal boring machine. (Courtesy WMW Machinery Co., Inc.)

ing centers. A typical boring operation performed on a lathe is shown in Figure 10.15. A steady rest is being used to provide support for the part being machined.

Boring machines, like most other machine tools, can be classified as horizontal or vertical:

10.7.1 Horizontal Boring Machines (HBM)

The HBM is made to handle medium to very large-sized parts, but these parts are usually somewhat rectangular in shape, though they may be asymmetrical or irregular. The available cutting tools only limit the size of cut, the rigidity of the spindle, and the available horsepower. There are two types of Horizontal Boring Machines:

Table-type Horizontal Boring Machines (HBM)

The table-type HBM shown in Figure 10.16 is built on the same principles as the horizontal-spindle milling machines.

The base and column are fastened together, and the column does not move. The tables are heavy, ribbed castings which may hold loads up to 20,000 pounds. Figure 10.17 shows a large part being machined on a table-type horizontal boring machine.

Size of HBM: The basic size of

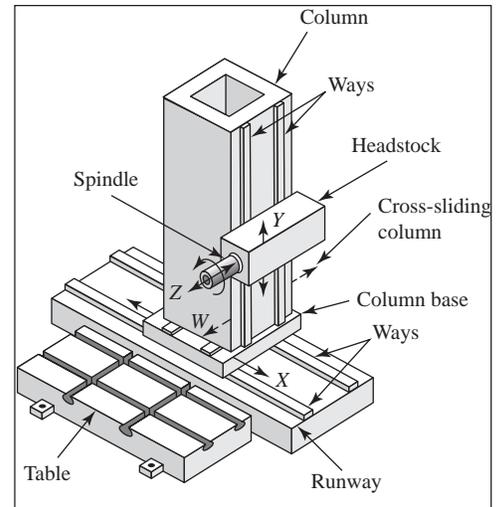


FIGURE 10.18: Principal parts of a floor-type horizontal boring machine (HBM).

an HBM is the diameter of the spindle. Table-type machines usually have spindles from 3 to 6 inches diameter. The larger sizes will transmit more power and, equally important, the spindle will not sag or deflect as much when using a heavy cutting tool while extended. The size is further specified by the size of the table. Although each machine has a 'standard' size table, special sizes may be ordered. The principal parts of the horizontal boring machine are shown in Figure 10.18.

Work Holding: Work holding is with clamps, bolts, or fixtures, the same as with other machines. Rotary tables allow machining of all four faces of a rectangular part or various angle cuts on any shape of part. Rotary tables up to 72 inches square or round are used for large work. If large, rather flat work is to be machined, an angle plate is used. The workpiece is bolted or clamped onto the angle plate so that the 'flat' face is toward the spindle. Figure 10.19 shows a five-axis ram-style machining center. Parts can be clamped to the table and numerically (NC or CNC) positioned to perform a boring operation.

Cutting Tools: Cutting tools are held in the rotating spindle by a tapered hole and a drawbar. To speed up the process of tool changing, either or both of two things are done:

- The drawbar (which pulls the tapered tool holder tightly into the spindle

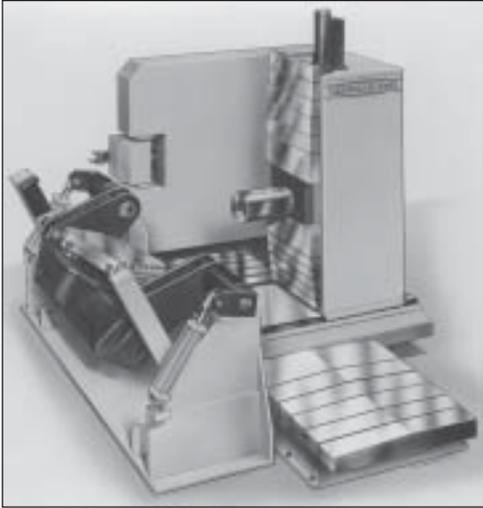


FIGURE 10.19: Five-axis ram-style machining center. (Courtesy Giddings and Lewis, LLC)

hole) can be power operated. Thus, the holder is pulled tight or ejected very quickly.

- Quick-change tooling is used. A basic holder is secured in the spindle. It has a taper into which tools may be secured by a quarter to half turn of the locking collar. Thus, the operator can change preset tools in 10 to 30 seconds.

Tool holders and quick-change tool holders in particular will be discussed in the milling chapters.

Speeds and Feeds: Speeds and feeds cover a wide range because of the wide variety of cutters that may be used on the HBM. Speeds from 15 to 1500 RPM and feed rates from

0.1 to 40 IPM are commonly used.

Floor Type Horizontal Boring Machine (HBM)

The floor type HBM (Fig. 10.18) is used for especially tall or long workpieces. The 'standard' 72-inch runway can be made almost any length required for special jobs. Lengths of 20 feet are in use today. The height of the column, which is usually 60 to 72 inches, can be made to order up to twice this height if the work requires it. Figure 10.20 shows a large floor-type horizontal boring machine.

HBM Table: The table is separate from the boring machine though it is, of course, fastened to the floor. It may be bolted to the runway.

The entire column and column base move left and right (the X axis) along special ways on the runway (Fig. 10.18). The runway must be carefully aligned and leveled when it is first installed, and then checked at intervals as the machine is used.

HBM Headstock: The headstock can be moved accurately up and down the column (the Y axis). The 6 to 10 inch diameter spindle rotates to do the machining. It is moved in

and out (the Z axis) up to 48 inches for boring cut, drilling, setting the depth of milling cuts, etc. As in the table-type HBM, the spindle diameter and table size specify the machine size.

Cutting Tools: Cutting tools are the same as those used on the table-type machine. Work holding is also the same, and angle plates are frequently used.

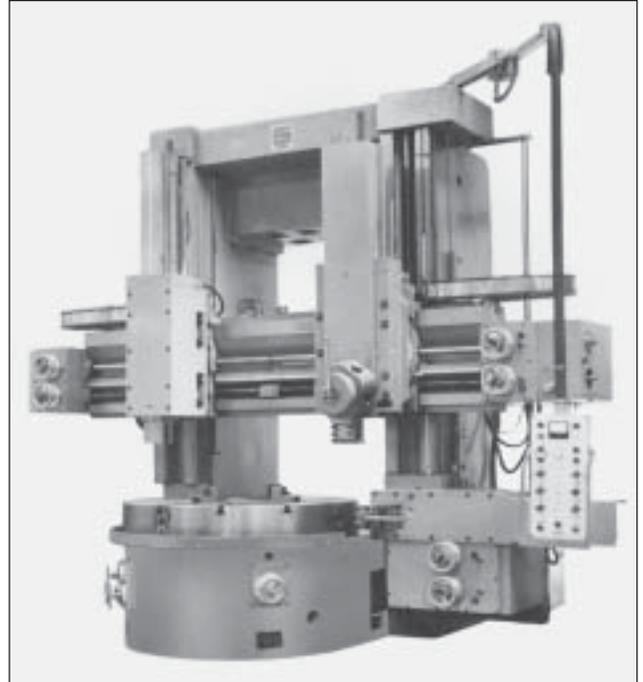


FIGURE 10.21: Vertical boring machine (VBM). (Courtesy Summit Machine Tool Manufacturing Corp.)

10.7.2 Vertical Boring Machines (VBM)

A general description of a vertical boring machine would be that it is a lathe turned on end with the headstock resting on the floor. This machine is needed because even the largest engine lathes cannot handle work much over 24 inches in diameter. A vertical boring machine is shown in Figure 10.21.

Today's VBMs are often listed as turning and boring machines. If facing is added to that name, it pretty well describes the principal uses of this machine. Just like any lathe, these machines can make only round cuts plus facing and contouring cuts.

Figure 10.22 shows the general construction and the motions available on the VBM. The construction is the same as that of the double-housing planer, except that a round table has been substituted for the long reciprocating table, and the toolholders are different since the VBM does not need clapper boxes.

The size of a vertical boring machine is the diameter of the revolving worktable. The double-housing VBM is most often made with table diameters from 48 inches to 144 inches. Larger machines

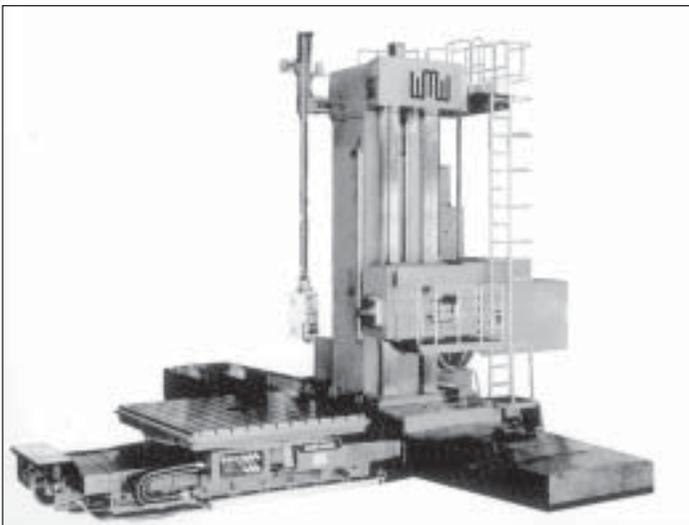


FIGURE 10.20: Large floor-type horizontal boring machine. (Courtesy WMW Machinery Co., Inc.)

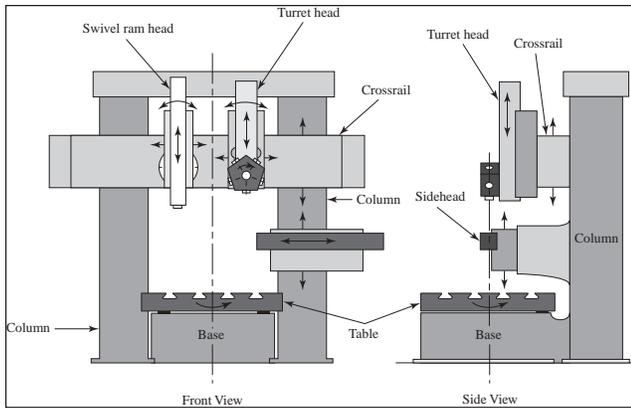


FIGURE 10.22: General construction, components and motions of a vertical boring machine (VBM).

have been made for special work. A rather large VBM is shown in Figure 10.23.

10.7.3 Jig Borers

Jig borers are vertical boring machines with high precision bearings. They are available in various sizes and used mainly in tool rooms for machining jigs and fixtures. More versatile numerically controlled machines are now replacing many jig borers.

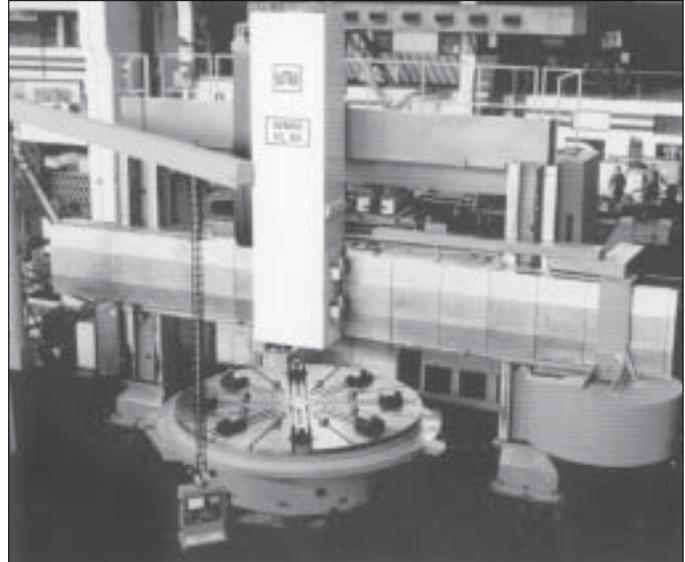


FIGURE 10.23: Large Vertical Boring Machine (Courtesy: WMW Machinery Co., Inc.)

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University
Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers
Former President
International Executive Board
Society of Carbide & Tool Engineers
Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

CHAPTER 11

Reaming and Tapping

11.1 Introduction

Twist drills do not make accurately sized or good finish holes; a reamer of some type is often used to cut the final size and finish. A reamer will not make the original hole; it will only enlarge a previously drilled or bored hole. It will cut to within +0.0005 inch of tool size and give finishes to 32 micro inches (u in).

Reamers are usually made of HSS although solid carbide and carbide tipped reamers are made in many sizes and styles. Regular chucking reamers are made in number and letter sizes, in fractional inch sizes, and in millimeter sizes. They can be purchased ground to any desired diameter.

Screw threads are used for a variety of purposes and applications in the machine tool industry. They are used to hold or fasten parts together (*screws, bolts, and nuts*), and to transmit motion (*the lead screw moves the carriage on an engine lathe*). Screw threads are also used to control or provide accurate movement (*the spindle on a micrometer*), and to provide a mechanical advantage (*a screw jack raises heavy loads*).

When defining a screw thread, one must consider separate definitions for an external thread (screw or bolt) and an internal thread (nut).

An external thread is a cylindrical piece of material that has a uniform helical groove cut or formed around it. An internal thread is defined as a piece of material that has a helical groove around the interior of a cylindrical hole. This chapter will discuss internal threads and tapping, the operation that produces such threads.

11.2 Reaming

Reaming has been defined as a machining process that uses a multi-edged fluted cutting tool to smooth, enlarge, or accurately size an existing hole. Reaming is performed using the same types of machines as drilling.

A reamer is a rotary cutting tool with one or more cutting elements used for enlarging to size and contour a previously formed hole. Its principal support during the cutting action is obtained from the workpiece. A typical reaming operation is shown in Figure 11.1.

11.2.1 Reamer Nomenclature

The basic construction and nomenclature of reamers is shown in Figure

11.2. This shows the most frequently used style for holes up to 1 inch, called a chucking reamer.

Solid reamers do almost all their cutting with the 45 degree chamfered

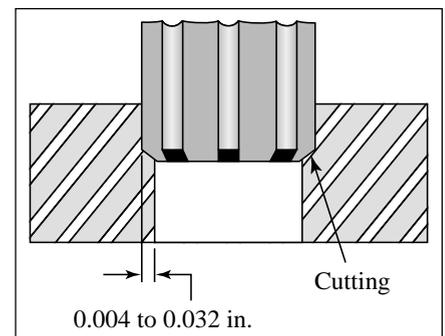


FIGURE 11.1: A typical reaming operation removes 0.004 to 0.032 in. of stock.

front end. The flutes guide the reamer and slightly improve the finish. Therefore, reamers should not be used for

mit passage of chips, and allow cutting fluid to reach the cutting edges.

Flute Length: Flute length is the

length of the flutes not including the cutter sweep.

Land: The land is the section of the reamer between adjacent flutes.

Margin: The margin is the unrelieved part of the periphery of the land adjacent to the cutting edge.

Neck: The neck is a section of reduced diameter connecting

shank to body, or connecting other portions of the reamer.

Overall Length: The overall length is the extreme length of the complete reamer from end to end, but not including external centers or expansion screws.

Shank: The shank is the portion of the reamer by which it is held and driven.

Straight Shank: A straight shank is a cylindrical shank.

Taper Shank: A taper shank is a shank made to fit a specified (conical) taper socket.

11.2.2 Types of Reamers

Reamers are made with three shapes of flutes and all are standard.

Straight Flute: Straight flute reamers are satisfactory for most work and the least expensive, but should not be used if a keyway or other interruption is in the hole.

Right-hand Spiral: Right-hand spiral fluted reamers give freer cutting action and tend to lift the chips out of the hole. They should not be used on copper or soft aluminum because these reamers tend to pull down into the hole.

Left-hand Spiral: Left-hand spiral fluted reamers require slightly more pressure to feed but give a smooth cut and can be used on soft, gummy mate-

rials, since they tend to be pushed out of the hole as they advance. It is not wise to use these in blind holes, because they push the chips down into the hole.

All reamers are used to produce smooth and accurate holes. Some are turned by hand, and others use machine power. The method used to identify left hand and right hand reamers is shown in Figure 11.3.

Machine Reamers

Machine reamers are used on both

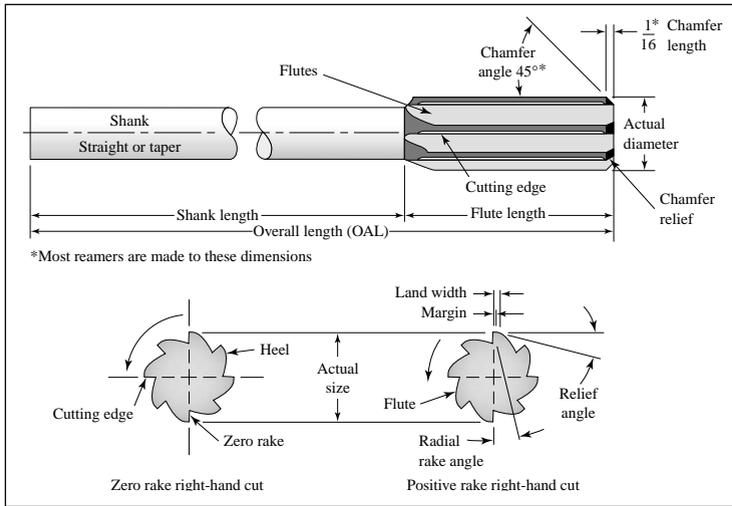


FIGURE 11.2: Construction and nomenclature of a straight-fluted machining reamer.

heavy stock removal.

Axis: The axis is the imaginary straight line which forms the longitudinal centerline of a reamer, usually established by rotating the reamer between centers.

Back Taper: The back taper is a slight decrease in diameter, from front to back in the flute length of reamers.

Body: The body is: 1) The fluted full diameter portion of a reamer, inclusive of the chamfer, starting taper and bevel. 2) The principal supporting member for a set of reamer blades, usually including the shank.

Chamfer: The chamfer is the angular cutting portion at the entering end of a reamer.

Chamfer Length: The chamfer length is the length of the chamfer measured parallel to the axis at the cutting edge.

Chamfer Relief Angle: The chamfer relief angle is the axial relief angle at the outer corner of the chamfer. It is measured by projection into a plane tangent to the periphery at the outer corner of the chamfer.

Clearance: Clearance is the space created by the relief behind the cutting edge or margin of a reamer.

Cutting Edge: The cutting edge is the leading edge of the land in the direction of rotation for cutting.

Flutes: The flutes are longitudinal channels formed in the body of the reamer to provide cutting edges, per-

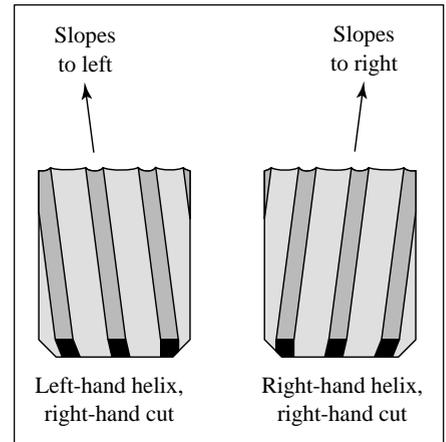


FIGURE 11.3: Method of identifying left-hand and right-hand reamers.

drilling machines and lathes for roughing and finishing operations. Machine reamers are available with tapered or straight shanks, and with straight or helical flutes. Tapered shank reamers (see Fig. 11.4) fit directly into the spindle, and the straight shank reamer, generally called the chucking reamer, fits into a drill chuck.

Rose Reamers: Rose reamers are machine reamers that cut only on a 45-degree chamfer (bevel) located on the end. The body of the rose reamer tapers slightly (about 0.001 inch per inch of length) to prevent binding during operation. This reamer does not cut a smooth hole and is generally used to bring a hole to a few thousands

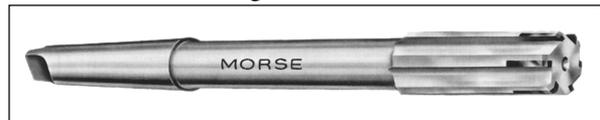


FIGURE 11.4: Carbide-tipped straight-fluted tapered-shank reamer. (Courtesy: Morse Cutting Tools)

undersize. Because the rose reamer machines a hole 0.001 to 0.005 inches under a nominal size, a hand reamer is used to finish the hole to size. All

hand reamers have a square shank and cannot be used and operated with machine power.

Fluted Reamers: Fluted reamers are machine reamers used to finish drilled holes. This type of reamer removes smaller portions of metal compared to the rose reamer. Fluted reamers have more cutting edges than rose reamers and therefore cut a smoother hole. Fluted reamers cut on the chamfered end as well as the sides. They are also available in solid carbide or have carbide inserts for cutting teeth.

Shell Reamers: Shell reamers (Fig. 11.5) are made in two parts: the reamer head and the arbor. In use, the reamer head is mounted on the arbor. The reamer head is available with either a rose or flute type, with straight or helical flutes. The arbor is available with either straight or tapered shank. The shell reamer is considered economical, because only the reamer is replaced when it becomes worn or damaged.

Hand Reamers

Hand reamers are finishing reamers distinguished by the square on their shanks (see Fig. 11.6). They are turned

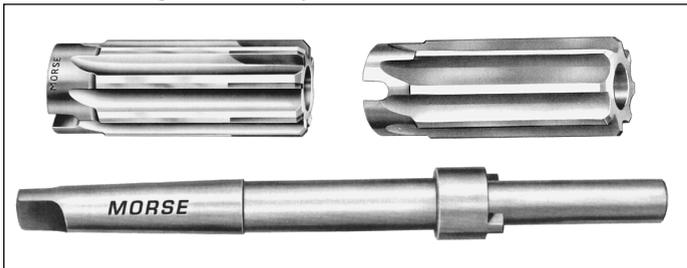


FIGURE 11.5: Shell reamer arbor with two reamer heads, one HSS and the other carbide tipped. (Courtesy: Morse Cutting Tools)

by hand with a tap wrench that fits over this square (see Fig. 11.7) this type of reamer cuts only on the outer cutting edges. The end of the hand reamer is tapered slightly to permit easy alignment in the drilled hole. The

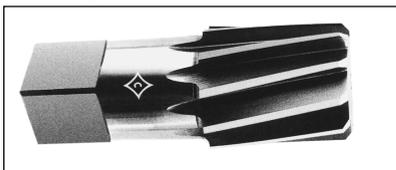


FIGURE 11.6: Left-hand-helix hand reamer, square-shanked hand reamers cannot be power driven. (Courtesy: Cleveland Twist Drill Greenfield Industries)

length of taper is usually equal to the reamer's diameter.

Hand reamers must never be turned by machine power, and must be started true and straight. They should never remove more than 0.001 to 0.005 inches of material. Hand reamers are available from 1/8 to over 2 inches in diameter and are generally made of carbon steel or high-speed steel.

Taper Hand Reamers: Taper hand reamers are hand reamers made to ream all standard size tapers. They are made for both roughing and finishing tapered holes. Similar to the straight hand reamer, this taper should be used carefully, and never with machine power.

Adjustable Reamers: (Fig. 11.8a) Adjustable reamers are used to produce any size hole within the range of the reamer. Their size is adjusted by sliding the cutting blades to and from the shank. The two adjusting nuts located at each end of the blades move these blades. Adjustable hand reamers are available in sizes from 1/4 to over 3 inches diameters. Each reamer has approximately 1/64-inch adjustment above and below its nominal diameter.

Expansion Hand Reamers: (Fig. 11.8b) Expansion hand reamers are like the adjustable reamers, but have a limited range of approximately 0.010 inch adjustment. Expansion reamers have an adjusting screw at the end of the reamer. When turned, this adjusting screw forces a tapered plug inside the body of the reamer, expanding its diameter. Expansion reamers are also

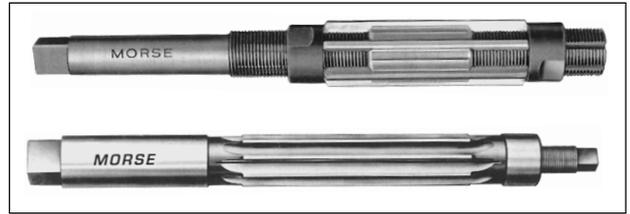


FIGURE 11.8: (a) Adjustable hand reamer. (b) A square-shanked expansion reamer. (Courtesy: Morse Cutting Tools)

available as machine reamers.

Care of Reamers: Because reamers are precision finishing tools, they should be used with care;

- * Reamers should be stored in separate containers or spaced in the tooling cabinet to prevent damage to the cutting edges.

- * Cutting fluids must always be used during reaming operations, except with cast iron.

- * A reamer must never be turned backward or the cutting edges will be dulled.

- * Any burrs or nicks on the cutting edges must be removed with an oilstone to prevent cutting oversize holes.

11.2.3 Operating Conditions

In reaming speed and feed are important; stock removal and alignment must be considered in order to produce chatter free holes.

Reaming Speeds: Speeds for machine reaming may vary considerably depending in part on the material to be reamed, type of machine, and required finish and accuracy. In general most machine reaming is done at about 2/3 the speed used for drilling the same material.

Reaming Feeds: Feeds for reaming are usually much higher than those used for drilling, often running 200 to 300 percent of drill feeds. Too low a feed may result in excessive reamer wear. At all times it is necessary that the feed be high enough to permit the reamer to cut rather than to rub or burnish. Too high a feed may tend to reduce the accuracy of the hole and may also lower the quality of the finish. The basic idea is to use as high a

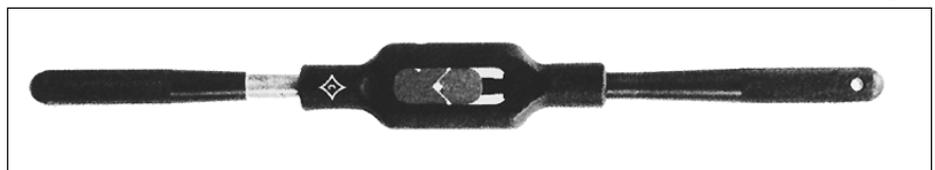


FIGURE 11.7: Tap wrenches are also used to hold hand reamers to finish drilled holes. (Courtesy: Cleveland Twist Drill Greenfield Industries)

feed as possible and still produce the required finish and accuracy.

Stock to be Removed: For the same reason, insufficient stock for reaming may result in a burnishing rather than a cutting action. It is difficult to generalize on this phase as it is tied in closely with type of material, feed, finish required, depth of hole, and chip capacity of the reamer. For machine reaming, .010 inch on a 1/4-inch hole, .015 inch on a 1/2 inch hole, up to .025 inch on a 1-1/2 inch hole seems a good starting point. For hand reaming, stock allowances are much smaller, partly because of the difficulty in forcing the reamer through greater stock. A common allowance is .001 inch to .003 inch.

Alignment: In the ideal reaming job, the spindle, reamer, bushing, and hole to be machined are all in perfect alignment. Any variation from this tends to increase reamer wear and detracts from the accuracy of the hole. Tapered, oversize, or bell-mouthed holes should call for a check of alignment. Sometimes the bad effects of misalignment can be reduced through the use of floating or adjustable holders. Quite often if the user will grind a slight back taper on the reamer it will also be of help in overcoming the effects of misalignment.

Chatter: The presence of chatter while reaming has a very bad effect on reamer life and on the finish in the hole. Chatter may be the result of one of several causes, some of which are listed:

- * Excessive speed.
- * Too much clearance on reamer.
- * Lack of rigidity in jig or machine.
- * Insecure holding of work.
- * Excessive overhand of reamer or spindle.
- * Too light a feed.

Correcting the cause can materially increase both reamer life and the quality of the reamed holes.

In reaming the emphasis is usually on finish, and a coolant is normally chosen for this purpose rather than for cooling.

11.2.4 Reaming Operations

Reaming operations can be performed on lathes, drills, and machining centers.

Lathe Reaming: Reaming on a lathe can only be done by holding the

reamer in the tail stock position either in a drill chuck for straight shank reamers, or directly in the tail stock quill for tapered shank reamers (see Fig. 11.4). Work to be reamed can either be held in a chuck or mounted onto the faceplate. In case of a turret lathe, the reamer can only be used in the hex turret.

Sometimes reamers are held in 'floating' holders in the tailstock. These holders allow the reamer to center itself on the previously drilled hole. Deep holes (over three times the diameter of the drill) tend to 'run out'. The reamer will not correct this condition and the hole must be bored if alignment is important.

Drill Press Reaming: Reaming on a drill press also requires the reamer to be held in the spindle with a drill chuck for straight shank machining reamers, or directly in the spindle for tapered shank reamers (see Fig. 11.4). The work to be reamed is usually held in a vise and centered on the drill table.

Reaming on a lathe is performed by rotating the work with a stationary reamer, while reaming on a drill press is performed with a rotating reamer and a stationary workpiece. 'Floating' heads can be used on drill presses as well as lathes.

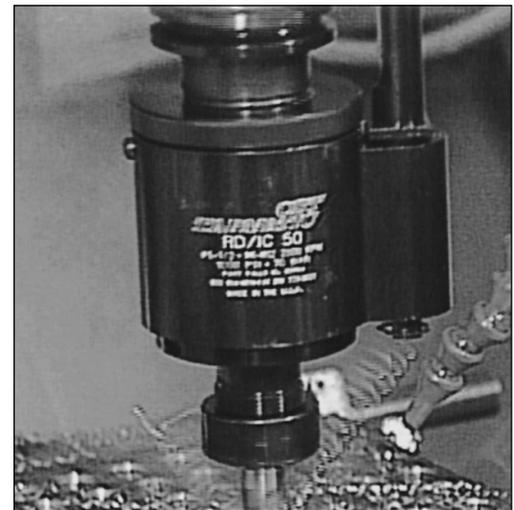


FIGURE 11.9: A typical automated tapping operation with self-reversing unit. (Courtesy: Tapmatic Corp.)

Machining Center Reaming: Reaming on a machining center is common. Reamers are usually held in the hex turret or in an automatic tool magazine. Set-ups are usually more complicated while speeds and feeds are preprogrammed.

11.3 Tapping

Tapping has been defined as: A process for producing internal threads using a tool (tap) that has teeth on its periphery to cut threads in a predrilled hole. A combined rotary and axial relative motion between tap and workpiece forms threads. A typical

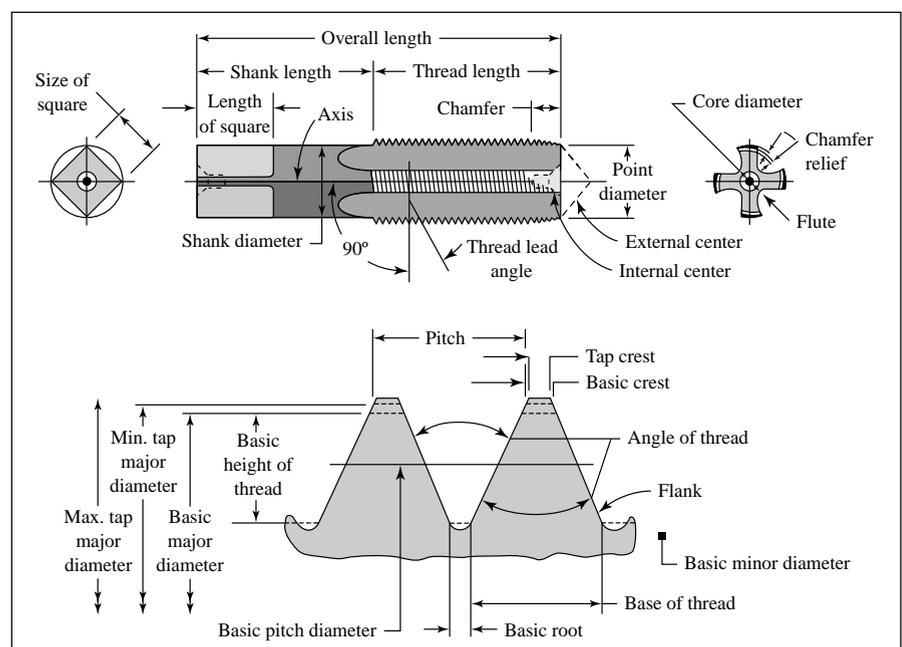


FIGURE 11.10: Tap and thread nomenclature.

automated tapping operation is shown in Figure 11.9.

11.3.1 Tap Nomenclature

Screw threads have many dimensions. It is important in modern manufacturing to have a working knowledge of screw thread terminology. A 'right-hand thread' is a screw thread that requires right-hand or clockwise rotation to tighten it. A 'left-hand thread' is a screw thread that requires left-hand or counterclockwise rotation to tighten it. 'Thread fit' is the range of tightness or looseness between external and internal mating threads. 'Thread series' are groups of diameter and pitch combinations that are distinguished from each other by the number of threads per inch applied to a specific diameter. The two common thread series used in industry are the coarse and fine series, specified as UNC and UNF. Tap nomenclature is shown in Figure 11.10.

Chamfer: Chamfer is the tapering of the threads at the front end of each land of a chaser, tap, or die by cutting away and relieving the crest of the first few teeth to distribute the cutting action over several teeth.

Crest: Crest is the surface of the thread which joins the flanks of the thread and is farthest from the cylinder or cone from which the thread projects.

Flank: Flank is the part of a helical thread surface which connects the crest and the root, and which is theoretically a straight line in an axial plane section.

Flute: Flute is the longitudinal channel formed in a tap to create cutting edges on the thread profile and to provide chip spaces and cutting fluid passage.

Hook Angle: The hook angle is the angle of inclination of a concave face, usually specified either as 'chordal hook' or 'tangential hook'.

Land: The land is one of the threaded sections between the flutes of a tap.

Lead of Thread: The lead of thread is the distance a screw thread advances axially in one complete turn. On a single start tap the lead and pitch are identical. On a multiple start tap the lead is the multiple of the pitch.

Major Diameter: This is the diameter of the major cylinder or cone, at a given position on the axis that bounds the crests of an external thread or the roots of an internal thread.

Minor Diameter: Minor diameter is the diameter of the minor cylinder or cone, at a given position on the axis that bounds the roots of an external thread or the crests of an internal thread.

Pitch Diameter: Pitch diameter is the diameter of an imaginary cylinder or cone, at a given point on the axis of such a diameter and location of its axis, that its surface would pass through the thread in a manner such as to make the thread ridge and the thread groove equal and, as such, is located equidistant between the sharp major and minor cylinders or cones of a given thread form. On a theoretically perfect thread, these widths are equal to one half of the basic pitch (measured parallel to the axis).

Spiral Point: A spiral point is the angular fluting in the cutting face of the land at the chamfered end. It is formed at an angle with respect to the tap axis of opposite hand to that of

rotation. Its length is usually greater than the chamfer length and its angle with respect to the tap axis is usually made great enough to direct the chips ahead of the tap. The tap may or may not have longitudinal flutes.

Square: Square is the four driving flats parallel to the axis on a tap shank forming a square or square with round corners.

11.3.2 Types of Taps

Taps are manufactured in many sizes, styles and types. Figure 11.11 shows some of the taps discussed below.

Hand Taps: Today the hand tap is used both by hand and in machines of all types. This is the basic tap design: four straight flutes, in taper, plug, or bottoming types. The small, numbered machine screw sizes are standard in two and three flutes depending on the size.

If soft and stringy metals are being tapped, or if horizontal holes are being made, either two- or three-flute taps can be used in the larger sizes. The flute spaces are larger, but the taps are weaker. The two-flute especially has a very small cross section.

The chips formed by these taps cannot get out; thus, they accumulate in the flute spaces. This causes added friction and is a major cause of broken taps.

Spiral Point Tap: The spiral point or 'gun' tap (Fig. 11.12a) is made the same as the standard hand tap (see Fig. 11.10) except at the point. A slash is ground in each flute at the point of the tap. This accomplishes several things:

- * The gun tap has fewer flutes (usually three), and they are shallower. This means a stronger tap.

- * The chips are forced out ahead of the tap instead of accumulating in the flutes, as they will with a plug tap.

- * Because of these two factors, the spiral point tap can often be run faster than the hand tap, and tap breakage is greatly reduced.

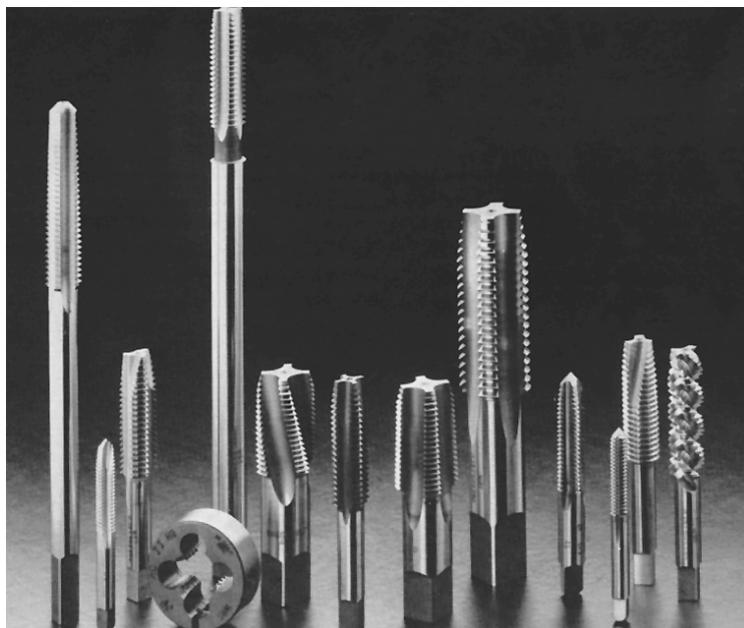


FIGURE 11.11: Some of the many styles and shapes of taps. (Courtesy: Greenfield Industries)

The gun tap has, in many cases, replaced the 'standard' style in industry, especially for open-ended trough holes in mild steel and aluminum. Both regular and spiral-point taps are made in all sizes including metric.

Spiral Flute Tap: The spiral flute-bottoming tap (Fig. 11.12b) is made in regular and fast spirals, that is, with small or large helix angle. They are sometimes called 'helical-fluted' taps. The use of these taps has been increasing since they pull the chip up out of the hole and produce good threads in soft metals (such as aluminum, zinc, and copper), yet also work well in Monel metal, stainless steel and cast

Taps are used for threading a wide range of materials both ferrous and non-ferrous. All pipe taps are supplied with 2-1/2 to 3-1/2 thread chamfer. The nominal size of a pipe tap is that of the pipe fitting to be tapped, not the actual size of the tap.

Ground Thread Pipe Taps are standard in American Standard Pipe Form (NPT) and American Standard Dryseal Pipe Form (NPTF). NPT threads require the use of a 'sealer' like Teflon tape or pipe compound. Dryseal taps are used to tap fittings that will give a pressure tight joint without the use of a 'sealer'. Figure 11.13 shows straight and spiral and spiral fluted pipe taps as well as a 'T' handle tap wrench.



FIGURE 11.12: (a) Spiral-point taps have replaced 'standard' taps in many cases. (b) A spiral-fluted bottoming tap. (Courtesy: Morse Cutting Tools)

steel. They are made in all sizes up to 1-1/2 inches and in metric sizes up to 12 mm.

While the 'standard' taps will efficiently do most work, if a great deal of aluminum, brass, cast iron, or stainless steel is being tapped, the manufacturer can supply 'standard' specials that will do a better job.

Pipe Taps: General Purpose Pipe

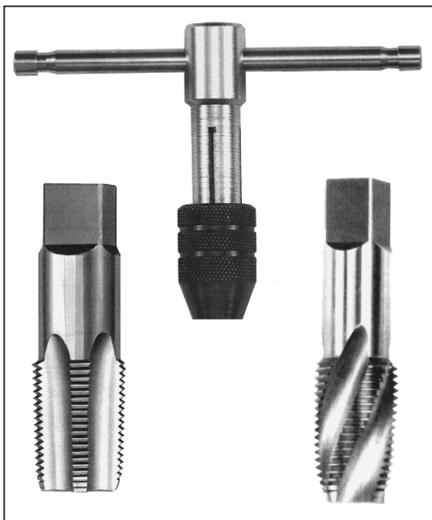


FIGURE 11.13: Straight and spiral-fluted pipe taps and a T-handle tap wrench. (Courtesy: Morse Cutting Tools)

Fluteless Taps: Fluteless taps (Fig. 11.14) do not look like taps, except for the spiral 'threads'. These taps are not round. They are shaped so that they 'cold form' the metal out of the wall of the

hole into the thread form with no chips. The fluteless tap was originally designed for use in aluminum, brass, and zinc alloys. However, it is being successfully used in mild steel and some stainless steels. Thus, it is worth checking for use where BHN is under 180. They are available in most sizes, including metric threads.

These taps are very strong and can often be run up to twice as fast as other styles, however, the size of the hole drilled before tapping must be no larger than the pitch diameter of the thread. The cold-formed thread often has a better finish and is stronger than a cut thread. A cutting oil must be used, and the two ends of the hole should be countersunk because the tap raises the metal at all ends.

Collapsing Taps: Collapsing taps (Fig. 11.15) collapse to a smaller diameter at the end of the cut. Thus, when used on lathes of any kind, they can be pulled back rapidly. They are made in sizes from about 1 inch up, in both machine and pipe threads. They use three to six separate 'chasers' which must be ground as a set. The tap holder and special dies make this assembly moderately expensive, but it is

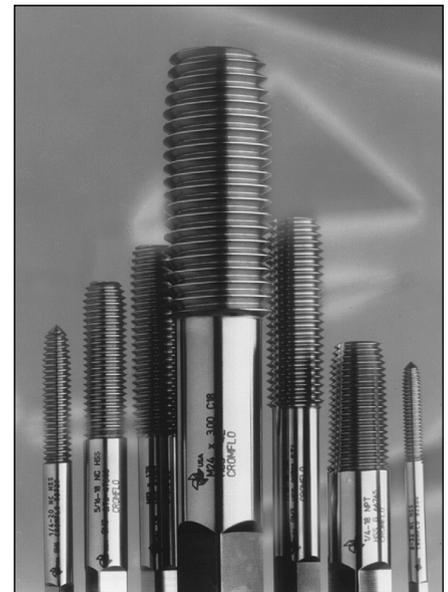


FIGURE 11.14: Fluteless taps are used to 'cold form' threads. (Courtesy: The Weldon Tool Co.)

economical for medium and high production work.

11.3.3 Operating Options

Some threads, both external and internal, can be cut with a single-point tool as previously shown. However, most frequently a die or tap of some type is used because it is faster and generally more accurate.

Taps are made in many styles, but a few styles do 90 percent of the work. Figure 11.10 shows the general terms used to describe taps. The cutting end of the tap is made in three different tapers.

The 'taper tap' is not often used today. Occasionally, it is used first as a starter if the metal is difficult to tap. The end is tapered about 5 degrees per side, which makes eight partial



FIGURE 11.15: Collapsing tap assemblies are more expensive, but economical for medium- and high-production runs. (Courtesy: Greenfield Industries)

threads.

The 'plug tap' is the style used probably 90 percent of the time. With the proper geometry of the cutting edge and a good lubricant, a plug tap will do most of the work needed. The end is tapered 8 degrees per side, which makes four or five incomplete threads.

The 'bottoming tap' (see Fig.

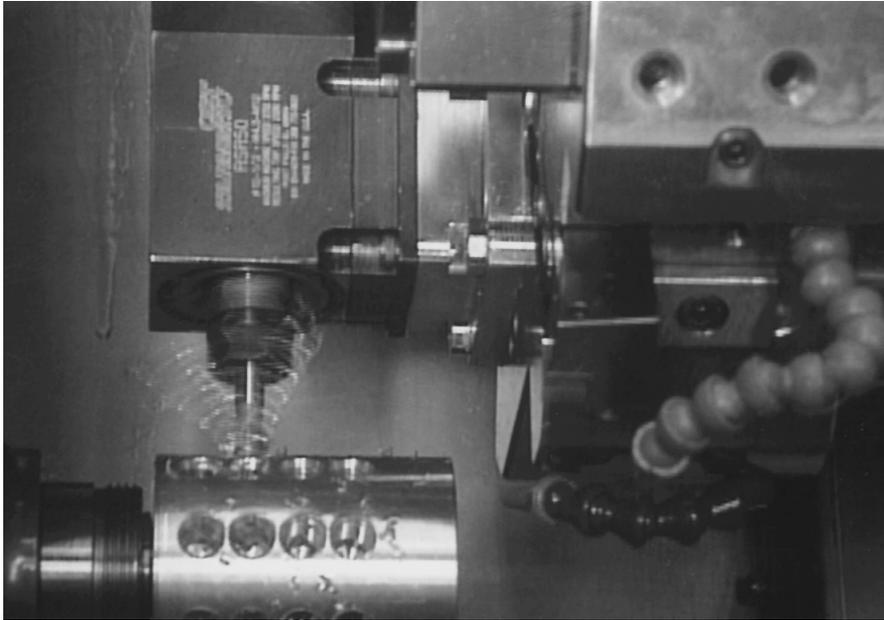


FIGURE 11.16: An automated multihole tapping operation on a round part. (Courtesy: Tapmatic Corp.)

11.12b) is used only for blind holes where the thread must go close to the bottom of the hole. It has only 1-1/2 to 3 incomplete threads. If the hole can be drilled deeper, a bottoming tap may not be needed. The plug tap must be used first, followed by the bottoming tap.

All three types of end tapers are made from identical taps. Size, length, and all measurements except the end taper are the same.

Material used for taps is usually high-speed steel in the M1, M2, M7, and sometimes the M40 series cobalt high-speed steels. A few taps are made of solid tungsten carbide.

Most taps today have ground threads. The grinding is done after hardening and makes much more accurate cutting tools. 'Cut thread' taps are available at a somewhat lower cost in some styles and sizes.

11.3.4 Tapping Operations

Just like reaming operations, tap-

ping can be performed on lathes, drills, and machining centers a multi hole tapping operation on a round part is shown in Figure 11.16.

Tap Drills: It is quite obvious that the taps shown here cannot cut their own opening. Thus, a

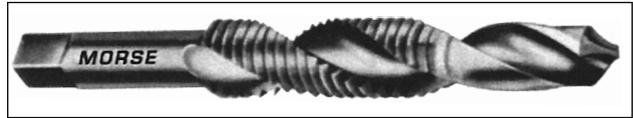


FIGURE 11.17: Combination drill and tap tools are used for one-pass drilling and tapping (Courtesy: Morse Cutting Tools)

tap is shown in Figure 11.17 and used to drill and tap in one pass.

The deeper the hole is threaded, the longer it takes to drill and tap and the more likely it is that the tap will break. Yet if there are too few threads holding the bolt, the threads will strip. Somewhere in between there is a depth of thread engagement that is the minimum that will hold enough so that the bolt will break before the threads let go. This is called the optimum depth.

Tap drilling must be deep enough in blind holes to allow for the two to five tapered threads on the tap plus chip clearance, plus the drill point.

Toolholders: Toolholders for hand tapping are called 'tap wrenches'. They are the same for taps and for reamers (see Fig. 11.7 and Fig. 11.13), because most taps have a square shank. Tap wrenches are adjustable and can be used on several sizes of taps.

When taps are used in drill presses or machining centers, a special head with a reversing, slip-type clutch is used. These tapping heads (Fig. 11.18) can be set so that if a hard spot is met in the metal, the clutch slips and the tap will not break. They are con-



FIGURE 11.18: Various special tap heads with reversing, slip-type clutches are used in drill presses and machining centers. (Courtesy Tapmatic Corp.)



FIGURE 11.19: Thread ‘chasing,’ or the manufacturing of outside threads, is performed with dies and self-opening die stocks. (Courtesy: Greenfield Industries)

structed so that when the hand-feed lever or the automatic numerical control machine cycle starts upward, the rotation reverses (and often goes faster) to bring the tap safely out of the hole.

Workholding: Workholding for tapping is the same as for any drill press or lathe work: clamps, vises, fixtures, etc. as needed. It is necessary to locate the tap centrally and straight in the hole. This is difficult in hand tapping but relatively easy in

machine tapping.

Numerical control is especially efficient, as it will locate over a hole, regardless of when it was drilled, if it was drilled from the same tape and on the same setup.

Single point threading was discussed in Chapter 6. Thread ‘chasing’ or the manufacturing of outside threads is also performed with dies and self-opening die stocks. Figure 11.19 shows a number of die heads and die chasers used in the manufacturing of threads.

Lubrication: The cutting edges on both taps and dies are buried in the material, so lubrication is quite necessary. For aluminum, light lard oil is used; other metals require a sulfur-based oil, sometimes chlorinated also. Figure 11.20 shows a tapping operation with an automated fluid dispensing system for machining centers. The ‘Automiser’ unit shown here dispenses

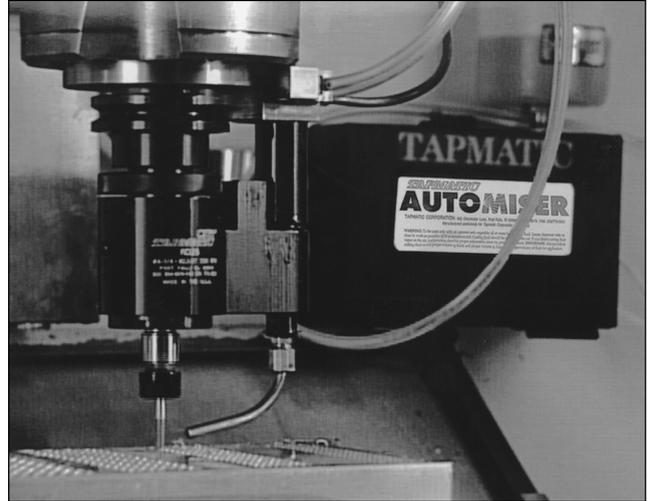


FIGURE 11.20: Multihold tapping operation with automatic coolant/lubrication system. (Courtesy: Tapmatic Corp.)

a lubricant/coolant through the tapping head automatically, while the head is in the machine spindle.

Copper alloys are stained by sulfur, so mineral oils or soluble oil must be used. Cast iron is often threaded without any lubricant.

There are several synthetic tapping fluids on the market today. They are somewhat more expensive but may save their cost in better threads and fewer broken tips.

CHAPTER 12

Milling Cutters and Operations

Metal Removal
Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining
Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes
Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining
Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes
Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University
Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers
Former President
International Executive Board
Society of Carbide & Tool Engineers
Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

12.1 Introduction

The two basic cutting tool types used in the metal working industry are of the single point and multi-point design, although they may differ in appearance and in their methods of application. Fundamentally, they are similar in that the action of metal cutting is the same regardless of the type of operation. By grouping a number of single point tools in a circular holder, the familiar milling cutter is created.

Milling is a process of generating machined surfaces by progressively removing a predetermined amount of material or stock from the workpiece which is advanced at a relatively slow rate of movement or feed to a milling cutter rotating at a comparatively high speed. The characteristic feature of the milling process is that each milling cutter tooth removes its share of the stock in the form of small individual chips. A typical face milling operation is shown in Figure 12.1.



FIGURE 12.1: A typical milling operation; the on-edge insert design is being used. (Courtesy Ingersoll Cutting Tool Co.)

12.2 Types of Milling Cutters

The variety of milling cutters available for all types of milling machines helps make milling a very versatile machining process. Cutters are made in a large range of sizes and of several different cutting tool materials. Milling cutters are made from High Speed Steel (HSS), others are carbide tipped and many are replaceable or indexable inserts. The three basic milling operations are shown in Figure 12.2. Peripheral and end milling cutters will be discussed below. Face

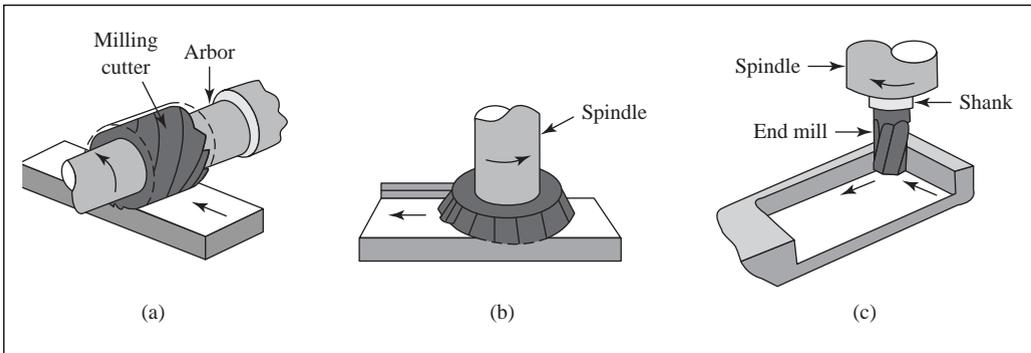


FIGURE 12.2: The three basic milling operations: (a) milling, (b) face milling, (c) end milling



FIGURE 12.3: High-speed steel (HSS) shell end milling cutter. (Courtesy Morse Cutting Tools)

milling cutters are usually indexable and will be discussed later in this chapter.

A high speed steel (HSS) shell end milling cutter is shown in Figure 12.3 and other common HSS cutters are shown in Figure 12.4 and briefly described below:

12.2.1 Periphery Milling Cutters

Periphery milling cutters are usually arbor mounted to perform various operations.

Light Duty Plain Mill: This cutter is a general purpose cutter for peripheral milling operations. Narrow cutters have straight teeth, while wide ones have helical teeth (Fig. 12.4c).

Heavy Duty Plain Mill: A heavy duty plain mill is similar to the light duty mill except that it is used for higher rates of metal removal. To aid it in this function, the teeth are more widely spaced and the helix angle

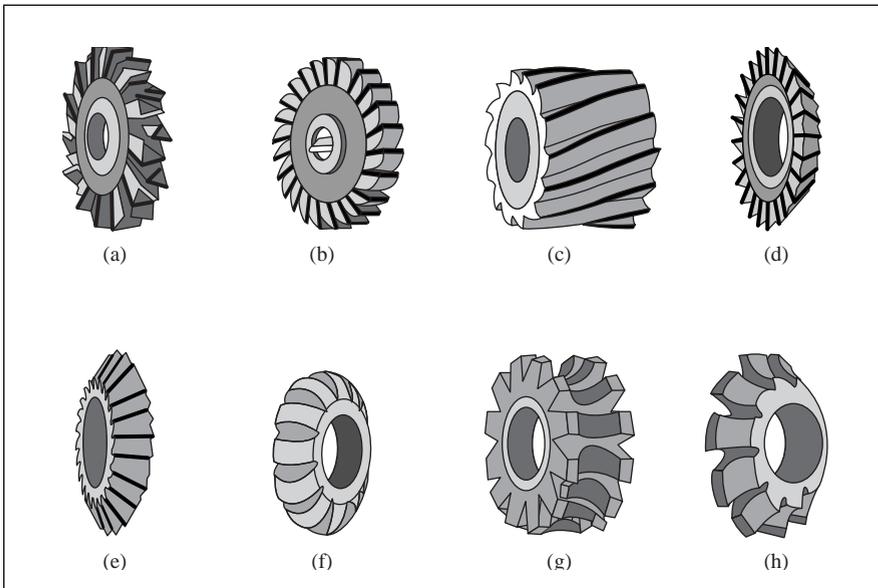


FIGURE 12.4: Common HSS milling cutters: (a) staggered-tooth cutter, (b) side milling cutter, (c) plain milling cutter, (d) single-angle milling cutter, (e) double-angle milling cutter, (f) convex milling cutter, (g) concave milling cutter, (h) corner rounded milling cutter.

is increased to about 45 degrees.

Side Milling Cutter:

The side milling cutter has a cutting edge on the sides as well as on the periphery. This allows the cutter to mill slots (Fig. 12.4b).

Half-Side Milling Cutter:

This tool is the same as the one previously described except that cutting edges are provided on a single side. It is used for

milling shoulders. Two cutters of this type are often mounted on a single arbor for straddle milling.

Stagger-tooth Side Mill: This cutter is the same as the side milling cutter except that the teeth are staggered so that every other tooth cuts on a given side of the slot. This allows deep, heavy-duty cuts to be taken (12.4a).

Angle Cutters: On angle cutters, the peripheral cutting edges lie on a cone rather than on a cylinder. A single or double angle may be provided (Fig. 12.4d and Fig. 12.4e).

Shell End Mill: The shell end mill has peripheral cutting edges plus face cutting edges on one end. It has a hole through it for a bolt to secure it to the spindle (Fig. 12.3).

Form Mill: A form mill is a peripheral cutter whose edge is shaped to produce a special configuration on the surface. One example of his class of tool is the gear tooth cutter. The exact contour of the cutting edge of a form mill is reproduced on the surface of the workpiece (Fig.12.4f, Fig.12.4g, and Fig.12.4h).

12.2.2 End Milling Cutters

End mills can be used on vertical and horizontal milling machines for a variety of facing, slotting, and profiling operations. Solid end mills are made from high speed steel or sintered carbide. Other types, such as shell end mills and fly cutters, consist of cutting tools that are bolted or otherwise fastened to adapters.

Solid End Mills: Solid end mills have two, three, four, or more flutes and cutting edges on the end and the periphery. Two flute end mills can be fed directly along their longitudinal axis into solid material because the cutting faces on the end meet. Three



FIGURE 12.5a: Various single- and double-ended HSS end mills. (Courtesy The Weldon Tool Co.)



FIGURE 12.5b: Various tapered HSS end mills. (Courtesy The Weldon Tool Co.)



FIGURE 12.6: (a) Ball-nose end-milling cutters are available in diameter ranging from 1/32 to 2 1/2 inches. (Courtesy The Weldon Tool Co.) (b) HSS dovetail cutters can be used on both vertical and horizontal milling machines. (Courtesy Morse Cutting Tools)



(a)

(b)

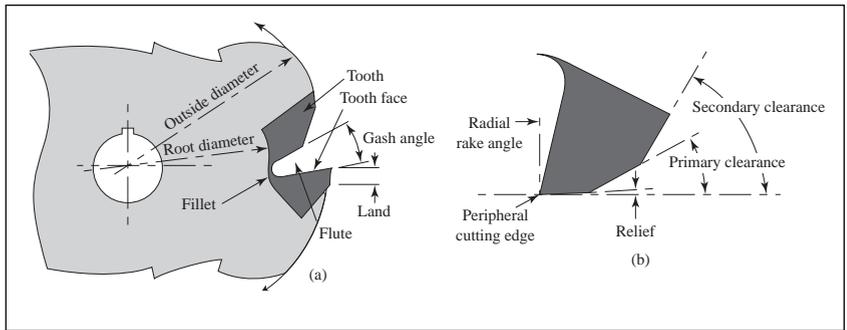


FIGURE 12.7: Milling cutter configuration: (a) plain milling cutter nomenclature; (b) plain milling cutter tooth geometry.

and four fluted cutters with one end cutting edge that extends past the center of the cutter can also be fed directly into solid material.

Solid end mills are double or single ended, with straight or tapered shanks. The end mill can be of the stub type, with short cutting flutes, or of the extra long type for reaching into deep cavities. On end mills designed for effective cutting of aluminum, the helix angle is increased for improved shearing action and chip removal, and the flutes may be polished. Various single and double-ended end mills are shown in Figure 12.5a. Various tapered end mills are shown in Figure 12.5b.

Special End Mills: Ball end mills (Fig. 12.6a) are available in diameters ranging from 1/32 to 2 1/2 inches in single and double ended types. Single purpose end mills such as Woodruff key-seat cutters, corner rounding cutters, and dovetail cutters (Fig.12.6b) are used on both vertical and horizontal milling machines. They are usually made of high speed steel and may have straight or tapered shanks.

12.3 Milling Cutter Nomenclature

As far as metal cutting action is concerned, the pertinent angles on the tooth are those that define the configuration of the cutting edge, the

orientation of the tooth face, and the relief to prevent rubbing on the land.

The terms defined below and illustrated in Figures 12.7a and 12.7b are important and fundamental to milling cutter configuration.

Outside Diameter: The outside diameter of a milling cutter is the diameter of a circle passing through the peripheral cutting edges. It is the dimension used in conjunction with the spindle speed to find the cutting speed (SFPM).

Root Diameter: This diameter is measured on a circle passing through the bottom of the fillets of the teeth.

Tooth: The tooth is the part of the cutter starting at the body and ending with the peripheral cutting edge. Replaceable teeth are also called inserts.

Tooth Face: The tooth face is the surface of the tooth between the fillet and the cutting edge, where the chip slides during its formation.

Land: The area behind the cutting edge on the tooth that is relieved to avoid interference is called the land.

Flute: The flute is the space provided for chip flow between the teeth.

Gash Angle: The gash angle is measured between the tooth face and the back of the tooth immediately ahead.

Fillet: The fillet is the radius at the bottom of the flute, provided to allow chip flow and chip curling.

The terms defined above apply primarily to milling cutters, particularly to plain milling cutters. In defining the configuration of the teeth on the cutter, the following terms are important.

Peripheral Cutting Edge: The cutting edge aligned principally in the direction of the cutter axis is called the peripheral cutting edge. In peripheral milling, it is this edge that removes the metal.



FIGURE 12.8: A variety of indexable milling cutters. (Courtesy Ingersoll Cutting Tool Co.)



(a)



(b)

FIGURE 12.10: (a) Face milling cutter with wedge clamped inserts. (Courtesy Iscar Metals, Inc.) (b) Face milling cutters with indexable inserts and wedge clamped milling cartridges. (Courtesy Greenleaf Corp.)

Face Cutting Edge: The face cutting edge is the metal removing edge aligned primarily in a radial direction. In side milling and face milling, this edge actually forms the new surface, although the peripheral cutting edge may still be removing most of the metal. It corresponds to the end cutting edge on single point tools.

Relief Angle: This angle is measured between the land and a tangent to the cutting edge at the periphery.

Clearance Angle: The clearance angle is provided to make room for chips, thus forming the flute. Normally two clearance angles are provided to maintain the strength of the tooth and still provide sufficient chip space.

Radial Rake Angle: The radial rake angle is the angle between the tooth face and a cutter radius, measured in a plane normal to the cutter

axis.

Axial Rake Angle: The axial rake angle is measured between the peripheral cutting edge and the axis of the cutter, when looking radially at the point of intersection.

Blade Setting Angle: When a slot is provided in the cutter body for a blade, the angle between the base of the slot and the cutter axis is called the blade setting angle.

12.4 Indexable Milling Cutters

The three basic types of milling operations were introduced earlier. Figure 12.8 shows a variety of indexable milling cutters used in all three of the basic types of milling operations (Fig. 12.2).

There are a variety of clamping systems for indexable inserts in milling cutter bodies. The examples shown cover the most popular methods now in use:

is used to support the insert, the wedge must absorb all of the force generated during the cut. This is why wedge clamping on the cutting face of the insert is preferred, since this method transfers the loads generated by the cut through the insert and into the cutter body. Both of the wedges clamping methods are shown in Figure 12.9.

The wedge clamp system however, has two distinct disadvantages. First, the wedge covers almost half of the insert cutting face, thus obstructing normal chip flow while producing premature cutter body wear, and secondly, high clamping forces causing clamping element and cutter body deformation can and often will result. The excessive clamping forces can cause enough cutter body distortion that in some cases when loading inserts into a milling body, the last insert slot will have narrowed to a point where the last insert will not fit into the body. When this occurs, several of the other inserts already loaded in the milling cutter are removed and reset. Wedge clamping can be used to clamp individual inserts (Fig. 12.10a) or indexable and replaceable milling cutter cartridges as shown in Figure 25.10b.

12.4.2 Screw Clamping

This method of clamping is used in conjunction with an insert that has a pressed countersink or counterbore. A torque screw is often used to eccentrically mount and force the insert against the insert pocket walls. This clamping action is a result of either offsetting the centerline of the screw toward the back walls of the insert

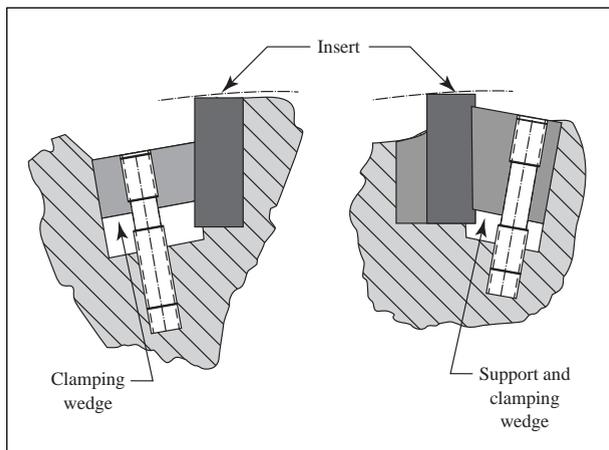


FIGURE 12.9: Two methods of wedge clamping indexable milling cutter inserts.

12.4.1 Wedge Clamping

Milling inserts have been clamped using wedges for many years in the cutting tool industry. This principle is generally applied in one of the following ways: either the wedge is designed and oriented to support the insert as it is clamped, or the wedge clamps on the cutting face of the insert, forcing the insert against the milling body. When the wedge

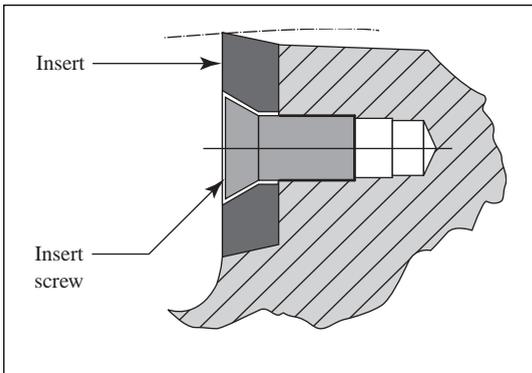


FIGURE 12.11: Screw clamping method for indexable inserts.

pocket, or by drilling and tapping the mounting hole at a slight angle, thereby bending the screw to attain the same type of clamping action.

The Screw clamping method for indexable inserts is shown in Figure 12.11.

Screw clamping is excellent for small diameter end mills where space is at a premium. It also provides an open unhampered path for chips to flow free of wedges or any other obstructive hardware. Screw clamping produces lower clamping forces than those attained with the wedge clamping system. However, when the cutting edge temperature rises significantly, the insert frequently expands and

causes an undesirable retightening effect, increasing the torque required to unlock the insert screw. The screw clamping method can be used on indexable ball milling cutters (Fig. 12.12a) or on indexable insert slotting and face milling cutters as shown in Figure 12.12b.

12.5 Milling Cutter Geometry

There are three industry standard milling cutter geometries: double negative, double positive, and positive/negative. Each cutter geometry type has certain advantages and disadvantages that must be considered when selecting the right milling cutter for the job. Positive rake and negative rake milling

cutter geometries are shown in Figure 12.13.

Double Negative Geometry: A double negative milling cutter uses only negative inserts held in a negative pocket. This provides cutting edge strength for roughing and severe interrupted cuts. When choosing a cutter geometry it is important to remember that a negative insert tends to push the cutter away, exerting considerable force against the workpiece. This could be a problem when machining flimsy or lightly held workpieces, or when using light machines. However, this tendency to push the work down, or push the cutter away from the workpiece may be beneficial in some cases because the force tends to 'load' the system, which often reduces chatter.

Double Positive Geometry: Double positive cutters use positive inserts held in positive pockets. This is to provide the proper clearance for cutting. Double positive cutter geometry provides for low force cutting, but the inserts contact the workpiece at their weakest point, the cutting edge. In positive rake milling, the cutting forces tend to lift the workpiece or pull the cutter into the work. The greatest advantage of double posi-

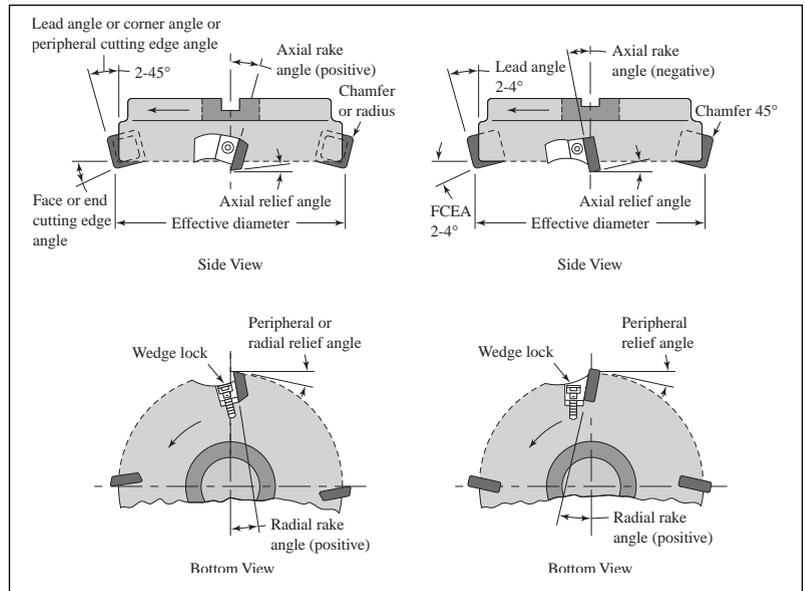


FIGURE 12.13: Positive-rake and negative-rake face milling cutter nomenclature.



(a)



(b)

FIGURE 12.12: (a) Indexable-insert ball-nosed milling cutters using the screw clamping method. (Courtesy Ingersoll Cutting Tool Co.) (b) Slotting cutters and face milling with screw-on-type indexable inserts. (Courtesy Duramet Corp.)

tive milling is free cutting. Less force is exerted against the workpiece, so less power is required. This can be especially helpful with machining materials that tend to work harden.

Positive / Negative Geometry: Positive/negative cutter geometry combines positive inserts held in negative pockets. This provides a positive axial rake and a negative radial rake and as with double positive inserts, this provides the proper clearance for cutting. In the case of positive/negative cutters, the workpiece is contacted away from the cutting edge in the radial direction and on the cutting edge in the axial direction. The positive/negative cutter can be considered a low force cutter because it uses a free cutting positive insert. On the other hand, the positive/negative cutter provides contact away from the cutting edge in the radial direction, the feed direction of a face mill.

In positive/negative milling, some of the advantages of both positive and negative milling are available. Positive/negative milling combines the free cutting or shearing away of the chip of a positive cutter with some of the edge strength of a negative cutter.

Lead Angle: The lead angle (Fig. 12.14) is the angle between the insert and the axis of the cutter. Several factors must be considered to determine which lead angle is best for a specific operation. First, the lead angle must be small enough to cover the depth of cut. The greater the lead angle, the less the depth of cut that can be taken for a given size insert. In addition, the part being machined may require a small lead angle in order to clear a portion or form a certain shape on the part. As the lead angle increases, the forces change toward the direction of the workpiece. This could cause deflections when machining thin

sections of the part.

The lead angle also determines the thickness of the chip. The greater the lead angle for the same feed rate or chip load per tooth, the thinner the chip becomes. As in single point tooling, the depth of cut is distributed over a longer surface of contact. Therefore, lead angle cutters are recommended when maximum material removal is the objective. Thinning the chip allows the feed rate to be increased or maximized.

Lead angles can range from zero to 85 degrees. The most common lead angles available on standard cutters are 0, 15, 30 and 45 degrees. Lead angles larger than 45 degrees are usually considered special, and are used for very shallow cuts for fine finishing, or for cutting very hard work materials.

Milling cutters with large lead angles also have greater heat dissipating capacity. Extremely high temperatures are generated at the insert cutting edge while the insert is in the cut. Carbide, as well as other tool materials, often softens when heated, and when a cutting edge is softened it will wear away more easily. However, if more of the tool can be employed in the cut, as in the case of larger lead angles, the tool's heat dissipating capacity will be improved which, in turn, improves tool life. In addition, as lead angle is increased, axial force is increased and radial force is reduced, an important factor in controlling chatter.

The use of large

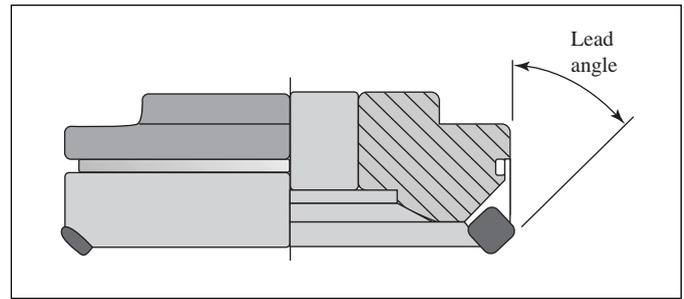


FIGURE 12.14: Drawing of a positive lead angle on an indexable-insert face milling cutter.

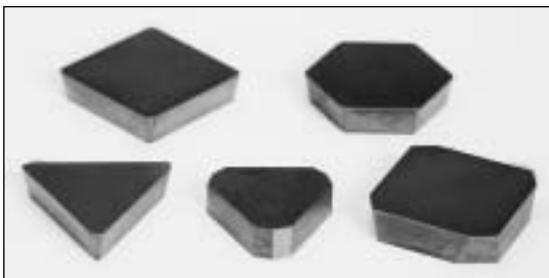
lead angle cutters is especially beneficial when machining materials with scaly or work hardened surfaces. With a large lead angle, the surface is spread over a larger area of the cutting edge. This reduces the detrimental effect on the inserts, extending tool life. Large lead angles will also reduce burring and breakout at the workpiece edge.

The most obvious limitation on lead angle cutters is part configuration. If a square shoulder must be machined on a part, a zero degree lead angle is required. It is impossible to produce a zero degree lead angle milling cutter with square inserts because of the need to provide face clearance. Often a near square shoulder is permissible. In this case a three degree lead angle cutter may be used.

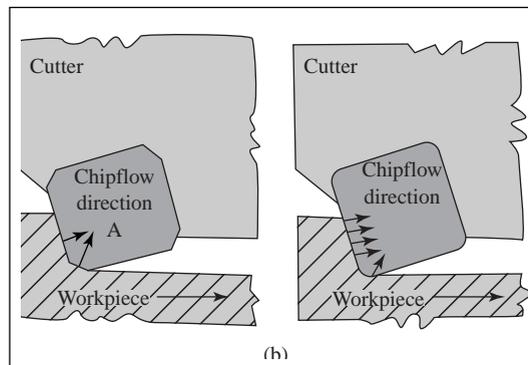
12.5.1 Milling Insert Corner Geometry

Indexable insert shape and size were discussed in Chapter 2. Selecting the proper corner geometry is probably the most complex element of insert selection. A wide variety of corner styles are available. The corner style chosen will have a major effect on surface finish and insert cost. Figure 12.15a shows various sizes and shapes of indexable milling cutter inserts.

Nose Radius: An insert with a nose radius is generally less expensive than a similar insert with any other corner geometry. A nose radius is also the strongest possible corner geometry because it has no sharp corners where two flats come together, as in the case of a chamfered corner. For these two



(a)



(b)

FIGURE 12.15: (a) Various sizes and shapes of indexable milling cutter inserts. (Courtesy American National Carbide Co.) (b) indexable milling cutter insert chip flow directions are shown.

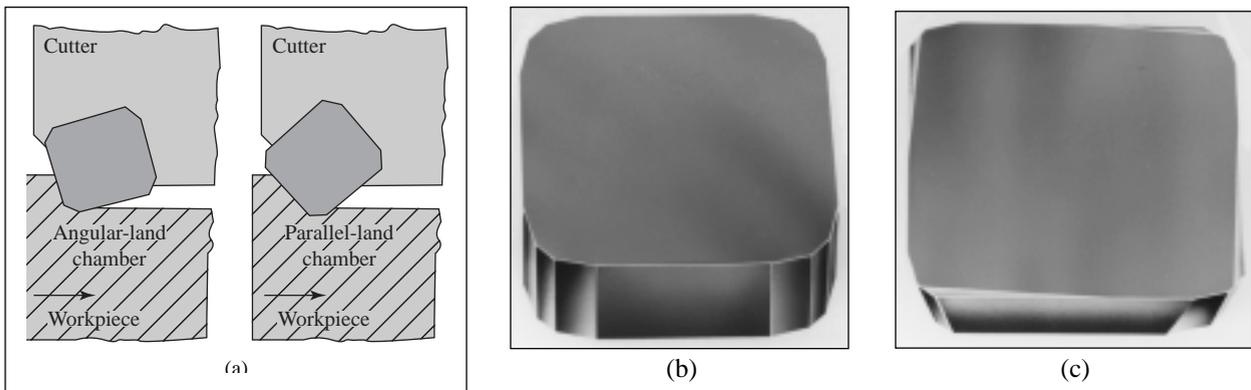


FIGURE 12.16: (a) indexable milling cutter inserts with angular-land chamfer and parallel-land chamfer. (b and c) Two precision ground indexable milling cutter inserts. (Courtesy Iscar Metals, Inc.)

reasons alone, a nose radius insert should be the first choice for any application where it can be used.

Inserts with nose radii can offer tool life improvement when they are used in 0 to 15 degree lead angle cutters, as shown in Figure 12.15b. When a chamfer is used, as in the left drawing, the section of the chip formed above and below point A, will converge at point A, generating a large amount of heat at that point, which will promote faster than normal tool wear. When a radius insert is used, as shown in the right drawing, the chip is still compressed, but the heat is spread more evenly along the cutting edge, resulting in longer tool life.

The major disadvantage of an insert with a nose radius is that the surface finish it produces is generally not as good as other common corner geometries. For this reason, inserts with nose radii are generally limited to roughing applications and applications where a sweep wiper insert is used for the surface. A sweep wiper is an insert with a very wide flat edge or a very large radiused edge that appears to be flat. There is usually only one wiper blade used in a cutter and this blade gets its name from its sweeping action that blends the workpiece surface to a very smooth finish.

Inserts with nose radii are not available on many double positive and positive/negative cutters because the clearance required under the nose radius is different from that needed under the edge. This clearance difference would require expensive grinding procedures that would more than offset the other advantages of nose radius inserts.

Chamfer: There are two basic ways in which inserts with a corner chamfer

can be applied. Depending both on the chamfer angle and the lead angle of the cutter body in which the insert is used, the land of the chamfer will be either parallel or angular (tilted) to the direction of feed, as shown in Figure 12.16a.

Inserts that are applied with the chamfer angular to the direction of feed normally have only a single chamfer. These inserts are generally not as strong and the cost is usually higher than inserts that have a large nose radius. Angular-land chamfer inserts are frequently used for general purpose machining with double negative cutters.

Inserts designed to be used with the chamfer parallel to the direction of feed may have a single chamfer, a double chamfer, or a double chamfer and corner break. The larger lands are referred to as primary facets and the smaller lands as secondary facets. The cost of chamfers, in relation to other types of corner geometries, depends upon the number of facets. A single facet insert is the least expensive, while multiple facet inserts cost more because of the additional grinding expense. Figure 12.16b shows two precision ground indexable milling cutter inserts. A face milling cutter with six square precision ground indexable milling cutter inserts was shown in Figure 12.10a.

The greatest advantage of using inserts with the land parallel to the direction of feed is that, when used correctly, they generate an excellent surface finish. When the land width is greater than the advance per revolution, one insert forms the surface. This means that an excellent surface finish

normally will be produced regardless of the insert face runout. Parallel-land inserts also make excellent roughing and general purpose inserts for positive/negative and double positive cutters. When a parallel land chamfer insert is used for roughing, the land width should be as small as possible to reduce friction.

Sweep Wipers: Sweep wipers are unique in both appearance and application. These inserts have only one or two very long wiping lands. A single sweep wiper is used in a cutter body filled with other inserts (usually roughing inserts) and is set approximately 0.003 to 0.005 inches higher than the other inserts, so that the sweep wiper alone forms the finished surface.

The finish obtained with a sweep wiper is even better than the excellent finish attained with a parallel land chamfer insert. In addition, since the edge of the sweep wiper insert is exceptionally long, a greater advance per revolution may be used. The sweep wiper also offers the same easy set-up as the parallel-land insert.

Sweep wiper inserts are available with both flat and crowned wiping surfaces. The crowned cutting edge is ground to a very large radius, usually from three to ten inches. The crowned cutting edges eliminate the possibility of saw-tooth profiles being produced on the machined surface because the land is not exactly parallel to the direction of feed, a condition normally caused by spindle tilt. On the other hand, sweep wipers with flat cutting edges produce a somewhat better finish if the land is perfectly aligned with the direction of feed.

12.6 Basic Milling Operations

Before any milling job is attempted, several decisions must be made. In addition to selecting the best means of holding the work and the most appropriate cutters to be used, the cutting speed and feed rate must be established to provide good balance between rapid metal removal and long tool life.

Proper determination of a cutting speed and feed rate can be made only when the following six factors are known:

- Type of material to be machined
- Rigidity of the set-up
- Physical strength of the cutter
- Cutting tool material
- Power available at the spindle
- Type of finish desired

Several of these factors affect cutting speed only, and some affect both cutting speed and the feed rate. The tables in reference handbooks provide approximate figures that can be used as starting points. After the cutting speed is chosen, the spindle speed must be computed and the machine adjusted.

Cutting Speed: Cutting speed is defined as the distance in feet that is traveled by a point on the cutter periphery in one minute. Since a cutter's periphery is its circumference:

$$\text{Circumference} = \text{Pi} \times d$$

in case of a cutter, the circumference is:

$$\begin{aligned} \text{Cutter circumference} &= \text{Pi}/12 \times d \\ &= .262 \times d \end{aligned}$$

Since cutting speed is expressed in surface feet per minute (SFPM)

$$\text{SFPM} = \text{Cutter circumference} \times \text{RPM}$$

by substituting for the cutter circumference, the cutting speed can be expressed as:

$$\text{SFPM} = .262 \times d \times \text{RPM}$$

The concept of cutting speed (SFPM) was introduced in Chapter 4 (Turning Tools and Operations) and explained again in Chapter 8 (Drills and Drilling Operations). It has again been reviewed here without giving additional examples. However, since milling is a multi-point operation, feed needs to be explained in more detail than in previous chapters.

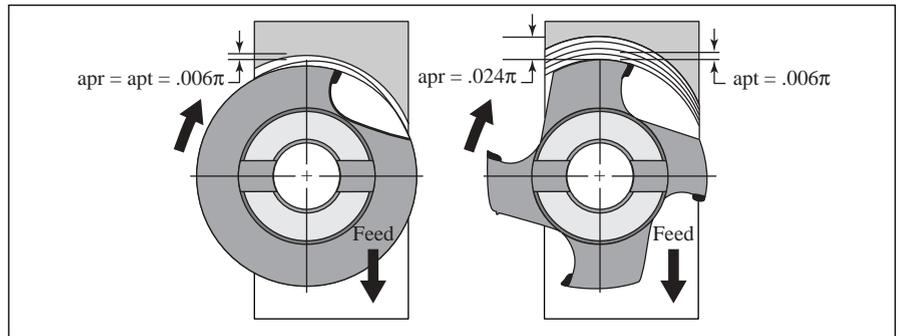


FIGURE 12.17: Drawing of a milling cutter showing the difference between advance per revolution (*apr*) and advance per tooth (*apt*).

Feed Rate: Once the cutting speed is established for a particular workpiece material, the appropriate feed rate must be selected. Feed rate is defined in metal cutting as the linear distance the tool moves at a constant rate relative to the workpiece in a specified amount of time. Feed rate is normally measured in units of inches per minute or IPM. In turning and drilling operations the feed rate is expressed in IPR or inches per revolution.

When establishing the feed rates for milling cutters, the goal is to attain the fastest feed per insert possible, to achieve an optimum level of productivity and tool life, consistent with efficient manufacturing practices. The ultimate feed rate is a function of the cutting edge strength and the rigidity of the workpiece, machine and fixturing. To calculate the appropriate feed rate for a specific milling application, the RPM, number of effective inserts (N) and feed per insert in inches (IPT or **apt**) should be supplied.

The milling cutter shown in Figure 12.17 on the left (one insert cutter) will advance .006 inches at the cutter centerline every time it rotates one full revolution. In this case, the cutter is said to have a feed per insert or an IPT (inches per tooth), **apt** (advance per tooth) and an **apr** (advance per revolution) of .006 inches. The same style of cutter with 4 inserts is shown in the right hand drawing. However, to maintain an equal load on each insert, the milling cutter will now advance .024 inches at the centerline every time it rotates one full revolution. The milling cutter on the right is said to have an IPT and **apt** of .006 inches, but an **apr** (advance per revolution) of .024 inches (.006 inch for each insert).

These concepts are used to deter-

mine the actual feed rate of a milling cutter in IPM (inches per minute) using one of the following formulas:

$$\begin{aligned} \text{IPM} &= (\text{IPT}) \times (\text{N}) \times (\text{RPM}) \\ &\text{or} \\ \text{IPM} &= (\text{apt}) \times (\text{N}) \times (\text{RPM}) \end{aligned}$$

where:

- IPM = inches per minute
- N = number of effective inserts
- IPT = inches per tooth
- apt** = advance per tooth
- RPM = revolutions per minute

For Example: When milling automotive gray cast iron using a 4 inch diameter face mill with 8 inserts at 400 SFPM and 30.5 IPM, what **apr** and **apt** would this be?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{400}{.262 \times 4} = 382$$

$$\text{apr} = \frac{\text{IPM}}{\text{RPM}} = \frac{30.5}{382} = .080 \text{ in.}$$

$$\text{apt} = \frac{\text{apr}}{\text{N}} = \frac{.080}{8} = .010 \text{ in.}$$

Answer: = .080 in. **apr**
= .010 in. **apt**

When milling a 300M steel landing gear with a 6 inch diameter 45 degree lead face mill (containing 10 inserts) at 380 SFPM and a .006 inch advance per tooth, what feed rate should be run in IPM?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{380}{.262 \times 6} = 242$$

$$\text{IPM} = \text{apt} \times \text{N} \times \text{RPM} = .006 \times 10 \times 242 = 14.5$$

Answer: = 14.5 IPM

The following basic list of formulas can be used to determine IPM, RPM, **apt**, **apr**, or N depending on what

information is supplied for a specific milling application:

- IPM = inches per minute
- N = number of effective inserts
- apr = inches of cutter advance every revolution
- apt = inches of cutter advance for each effective insert every revolution
- RPM = revolutions per minute

Find	Given	Using
IPM	apr, RPM	$IPM = apr \times RPM$
IPM	RPM, N, apt	$IPM = apt \times N \times RPM$
apr	IPM, RPM	$apr = IPM/RPM$
RPM	IPM, apr	$RPM = IPM/apr$
RPM	IPM, N, apt	$RPM = \frac{IPM}{N \times apt}$
N	IPM, RPM, apt	$N = \frac{IPM}{RPM \times apt}$
apt	IPM, N, RPM	$apt = \frac{IPM}{RPM \times N}$

Note: In the formulas shown above IPT can be substituted for apt and IPR can be substituted for apr.

Horsepower Requirements: In metal cutting, the horsepower consumed is directly proportional to the volume (Q) of material machined per unit of time (cubic inches / minute). Metals have distinct unit power factors that indicate the average amount of horsepower required to remove one cubic inch of material in a minute. The power factor (k*) can be used either to determine the machine size in terms of horsepower required to make a specific machining pass or the feed rate that can be attained once a depth and width of cut are established on a particular part feature. To determine the metal removal rate (Q) use the following:

$$Q = D.O.C. \times W.O.C \times IPM$$

where:

- D.O.C. = depth of cut in inches
- W.O.C. = width of cut in inches
- IPM = feed rate, in inches/minute

The average spindle horsepower required for machining metal workpieces is as follows:

$$HP = Q \times k^*$$

where:

- HP = horsepower required at the machine spindle
- Q = the metal removal rate in cubic inches/minute
- k* = the unit power factor in HP/cubic inch/minute

***k factors are available from reference books**

For example: What feed should be selected to mill a 2 inch wide by .25 inch depth of cut on aircraft aluminum, utilizing all the available horsepower on a 20 HP machine using a 3 inch diameter face mill?

$$HP = Q \times k^*$$

$$k^* = .25 \text{ H.P./in.}^3/\text{min. for aluminum}$$

The maximum possible metal removal rate (Q), for a 20 H.P. machine running an aluminum part is:

$$Q = \frac{HP}{k} = \frac{20}{.25} = 80 \text{ in}^3/\text{min.}$$

Answer: Q = 80 in.³/min.

To remove 80 in.³/min., what feed rate will be needed?

$$Q = (D.O.C.) \times (W.O.C.) \times IPM$$

$$IPM = \frac{Q}{(D.O.C.) \times (W.O.C.)} = \frac{80}{.25 \times 2} = 160$$

Answer: = 160 IPM

12.6.1 Direction of Milling Feed

The application of the milling tool in terms of its machining direction is critical to the performance and tool life

of the entire operation. The two options in milling direction are described as either conventional or climb milling. Conventional and climb milling also affects chip formation and tool life as explained below. Figure 12.18 shows drawings of both conventional and climb milling.

Conventional Milling: The term often associated with this milling technique is ‘up-cut’ milling. The cutter rotates against the direction of feed as the workpiece advances toward it from the side where the teeth are moving upward. The separating forces produced between cutter and workpiece oppose the motion of the work. The thickness of the chip at the beginning of the cut is at a minimum, gradually increasing in thickness to a maximum at the end of the cut.

Climb Milling: The term often associated with this milling technique is ‘down-cut’ milling. The cutter rotates in the direction of the feed and the workpiece, therefore advances towards the cutter from the side where the teeth are moving downward. As the cutter teeth begin to cut, forces of considerable intensity are produced which favor the motion of the workpiece and tend to pull the work under the cutter. The chip is at a maximum thickness at the beginning of the cut, reducing to a minimum at the exit. Generally climb milling is recommended wherever possible. With climb milling a better finish is produced and longer cutter life is obtained. As each tooth enters the work, it immediately takes a cut and is not dulled while building up pressure to dig into the work.

Advantages and Disadvantages: If the workpiece has a highly abrasive surface, conventional milling will usu-

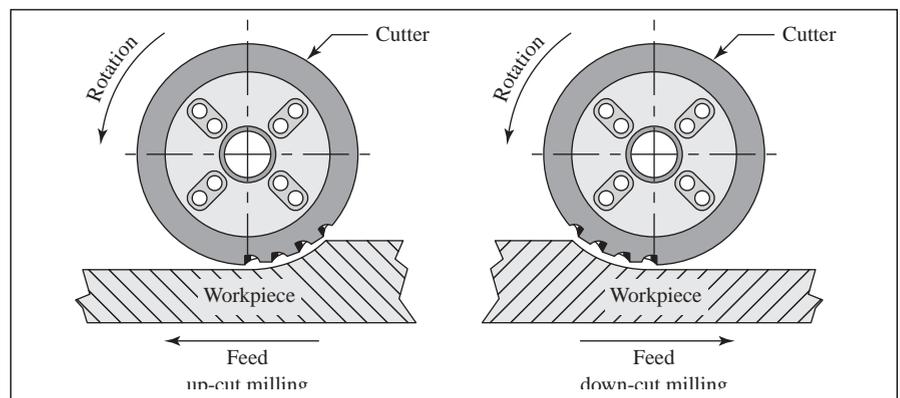


FIGURE 12.18: Conventional or up-milling as compared to climb or down-milling.

ally produce better cutter life since the cutting edge engages the work below the abrasive surface. Conventional milling also protects the edge by chipping off the surface ahead of the cutting edge.

Limitations on the use of climb milling are mainly affected by the condition of the machine and the rigidity with which the work is clamped and supported. Since there is a tendency

for the cutter to climb up on the work, the milling machine arbor and arbor support must be rigid enough to overcome this tendency. The feed must be uniform and if the machine does not have a backlash eliminator drive, the table gibs should be tightened to prevent the workpiece from being pulled into the cutter. Most present-day machines are built rigidly enough. Older machines can usually be tightened to

permit use of climb milling.

The downward pressure caused by climb milling has an inherent advantage in that it tends to hold the work and fixture against the table, and the table against the ways. In conventional milling, the reverse is true and the workpiece tends to be lifted from the table.

CHAPTER 13

Milling Methods and Machines

Metal Removal
Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining
Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes
Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining
Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes
Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

13.1 Introduction

Modern milling machines look much the same as they did 25 years ago. However, they now must cut super alloys, titanium, and high tensile steels to closer tolerances and at faster rates than previously. To handle these requirements, the new milling machines provide higher horsepower, greater stiffness, and wider speed and feed ranges than before. In addition, more accurate lead screws, closer alignment, numerical control (NC) and computer numerical control (CNC) all result in faster work with better finishes and greater accuracy than ever before attained. A modern CNC vertical milling machine is shown in Figure 13.1

13.2 Types of Milling Machines

The many types of milling machines used in manufacturing have been

grouped into three general classes.

- Column and Knee Machines
- Bed Type Milling Machines
- Special Purpose Machines

The common subtypes are also identified and discussed.

13.2.1 Column and Knee Machines

Column and knee milling machines are made in both vertical and horizontal types. The schematic diagrams (Fig. 13.2a and Fig.13.2b) show both types of machines. Versatility is a major feature of knee and column milling machines. On a basic machine of this type, the table, saddle, and knee can be moved. Many accessories such as universal vises, rotary tables, and dividing heads, further increase the versatility of this type of machine.

Regardless of whether the machine is of the vertical or horizontal type, several components on all column and knee milling machines are similar, except for size and minor variations because of manufacturer's preference. These similarities are described in terms of general shape, geometric relationship to the rest of the machine, function, and the material from which the components are made. Figure 13.3a shows a standard vertical milling machine and Figure 13.3b shows a 3 axis CNC vertical milling machine

A universal Horizontal Column and Knee milling machine is shown in Figure 13.4.

Column: The column, which is usually combined with the base as a single casting, is cast gray iron or ductile iron. The column houses the spindle, bearings, and the necessary gears, clutches, shafts, pumps, and shifting mechanisms for transmitting



FIGURE 13.1: Modern CNC vertical milling machine (Courtesy Ingersoll Milling Machine Co.)

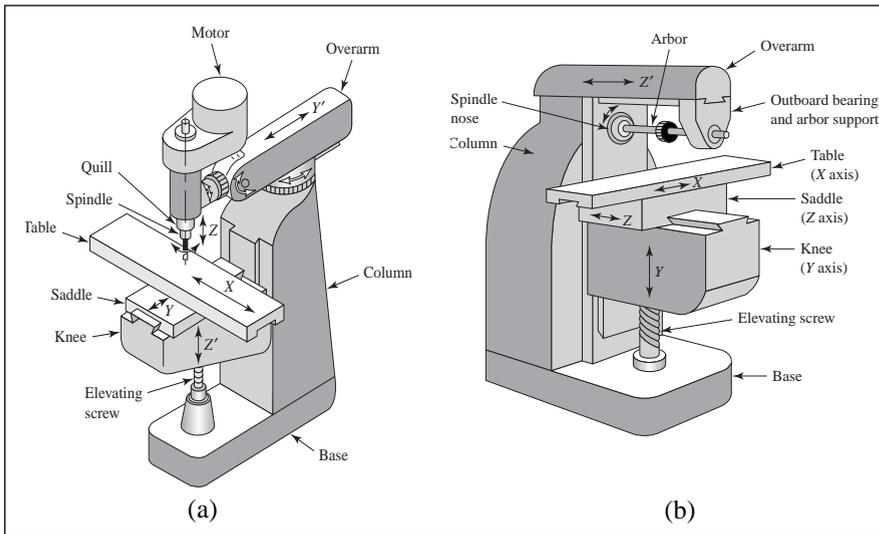


FIGURE 13.2: Schematic illustration of the motions and components (a) of a vertical-spindle column and knee type milling machine and (b) of a horizontal-spindle column and knee type of machine.

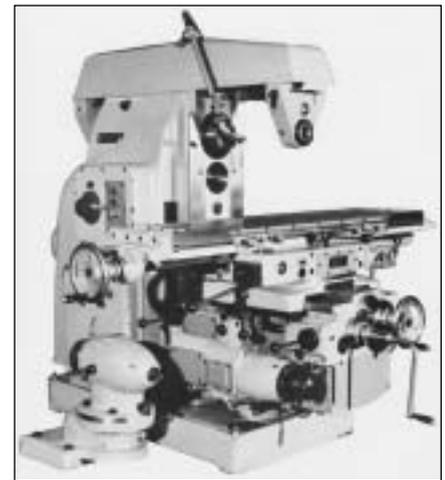


FIGURE 13.4: A universal horizontal-spindle column and knee type milling machine. (Courtesy Summit Machine Tool Manufacturing Corp.)

Spindles are hollow so that a drawbar can be used to hold arbors securely in place.

The front of the spindle is machined to accept standard arbors. The two keys that fit into corresponding slots in the arbor do the actual driving of the arbor. The internal taper, which is accurately ground so that it is concentric with the spindle, locates the arbor.

Knee: The knee is a casting that is moved up or down the slide on the front of the column by the elevating screw. Two dovetail or square slides are machined at 90 degrees to each other. The vertical slide mates with the slide on the front of the column, and the horizontal slide carries the saddle. It contains the necessary gears, screws, and other mechanisms to provide power feeds in all directions. The operator can select various feed rates with the controls mounted on the knee.

Saddle: The saddle for a plain milling machine is a casting with two slides machined at an exact 90 degree angle to each other. The lower slide fits the slide on the top of the knee, and the upper slide accepts the slide on the bottom of the table. The surfaces of the slides that make contact with the knee and the table are parallel to each other. Locks for both the cross slide and table is fitted to the saddle, along with the nuts that engage with the cross feed and table feed screws.

On a universal milling machine the saddle is made in two pieces and is more complex because it must allow the table to swivel through a limited arc. The lower part has a dovetail slide



FIGURE 13.3: (a) Stand vertical-spindle column and knee type milling machine. (Courtesy Bridgeport Machinery, Inc.) (b) Three-axis CNC vertical-spindle column and knee type milling machine. (Courtesy Chevalier Machinery, Inc.)

power from the electric motor to the spindle at the selected speed. The gears usually run in oil and are made of carburized alloy steel for long life. Some of the necessary controls are usually mounted on the side of the column.

The base is usually hollow, and in many cases serves as a sump for the cutting fluid. A pump and filtration system can be installed in the base. The hole in the center of the base houses the support for the screw that raises and lowers the knee.

The machined vertical slide on the front of the column may be of the square or dovetail type. The knee moves up and down on this slide. The slide must be machined at a 90 degree

angle to the face of the column in both the lateral and vertical planes. The tolerances are very close and are usually expressed in minutes or seconds of arc. The large hole in the face of the column casting is for the spindle. The hole is very accurately bored perpendicular to the front slide in two planes and parallel to the upper slide.

Spindle: On a horizontal milling machine, the spindle is one of the most critical parts. It is usually machined from an alloy steel forging and is heat treated to resist wear, vibration, thrust, and bending loads. The spindle is usually supported by a combination of ball and straight roller bearings, or by tapered roller bearings that absorb both radial loads and end thrust loads.



FIGURE 13.5: CNC bed type milling machine. (Courtesy WMW Machinery Co., Inc.)

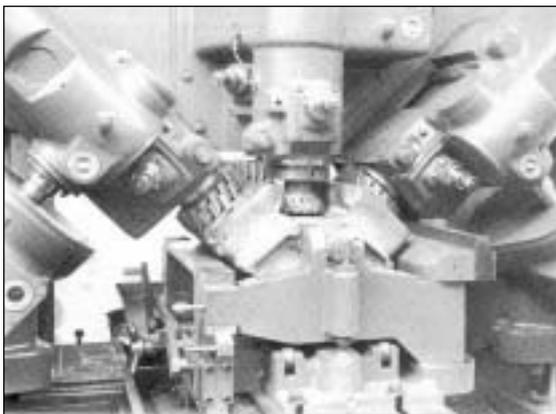


FIGURE 13.6: Machining operation showing a bed-type milling machine with three heads (Courtesy: Cincinnati Machine)

that fits the top of the knee and a circular slide above it is graduated in degrees for a small portion of its periphery. The upper portion of the saddle consists of a circular face that fits against the lower circular slide, a central pivot point, and a dovetail slide that accepts the table. Locking bolts moving in a circular T-slot are provided so that the two parts of the saddle can be locked in any position.

Table: Milling machine tables vary greatly in size, but generally they have the same physical characteristics. The bottom of the table has a dovetail slide that fits in the slide on top of the saddle. It also has bearings at each end to carry the table feed screw. The top of the table is machined parallel with the slide on the bottom and has several full length T-slots for mounting vises or other work holding fixtures.

A dial graduated in thousandths of an inch is provided to allow for accurate table movement and placement. The table feed screw usually has an

Acme thread.

Milling machines with vertical spindles (see Fig. 13.3a), are available in a large variety of types and sizes. The head, which houses the spindle, motor, and feed controls, is fully universal and can be placed at a compound angle to the surface of the table. The ram, to which the head is attached, can be moved forward and back and locked in any position. A turret on top of the column allows the head and ram assembly to swing laterally, increasing the reach of the head of the machine.

Some ram-type milling machines can be used for both vertical and horizontal milling. On ram-type vertical mills that have the motor in the column, power is transmitted to the spindle by gears and splined shafts. Some heavy duty vertical mills have a spindle and head assembly that can be moved only vertically by either a power feed mechanism or manually. These are generally known as over arm-type vertical milling machines.

13.2.2 Bed-Type Milling Machines

High production calls for heavy cuts, and the rigidity of a knee and column type of milling machine may not be sufficient to take the high forces. A bed-type

milling machine is often ideal for this kind of work. In this machine the table is supported directly on a heavy bed, while the column is placed behind the bed. A CNC bed-type milling machine is shown in Figure 13.5, and an operational set-up with three spindles is shown in Figure 13.6.

There are several advantages of the bed-type machine, particularly for production runs. Hydraulic table feeds are possible; the hydraulic components being housed in the bed casting. This allows very high feed forces; variable feed rates during any given cut, and automatic table cycling. The spindle may be raised or lowered by a cam and template arrangement to produce special contours. The basically heavier construction allows more power to be supplied to the spindle, which gives higher productivity through faster metal removal. Duplex bed-type milling machines have two columns and spindles for milling two surfaces on a part simultaneously. A large 5-axis CNC bed-type milling machine is shown in Figure 13.7.

The chief disadvantage of a bed-type milling machine compared to one of the knee and column type is that it is less versatile for machining small parts. Its advantages lie in its higher productivity, its adaptability to large sized machines, and its ease of modification to special applications.

13.2.3 Special Purpose Milling Machines

As industrial products have become



FIGURE 13.7: Large 5-axis CNC bed type milling machine. (Courtesy Ingersoll Milling Machine Co.)

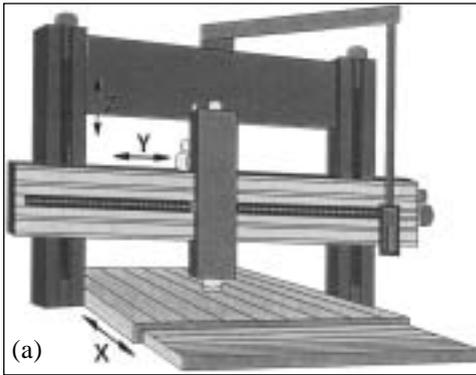


FIGURE 13.8: (a) Illustration of a planer-type milling machine. (Courtesy Kennametal Inc.) (b) Part being machined on a planer-type milling machine. (Courtesy Ingersoll Milling Machine Co.)



the cutter, duplicating the shape of the template. A profile milling operation is shown, using a ball-nosed milling cutter, in Figure 13.9a. A 5 axis, multi-spindle profile milling machine is shown in Figure 13.9b.

Die sinking and other processes involving the machining of cavities can be done on three-dimensional profilers. An accurate pattern of the cavity is made

more complex, new and unusual variations of the more common milling machines have been developed. The objectives are to accommodate larger work, make many duplicate parts, locate holes and surfaces precisely, or to do other unusual machining jobs.

Planer-Type Milling Machines:

The general arrangement of these types of machines is similar to that for planers (Chapter 7), except that in place of individual tool bits, milling heads are installed. The table of the machine carries the work past the rotating cutter heads, which are individually powered and can be run at different speeds if necessary. As many as four cutter heads can be used, with two mounted on the cross rail and two on the vertical pillars. An illustration of a planer type milling machine is shown in Figure 13.8a. A part being machined on a

planer-type milling machine is shown in Figure 13.8b.

Planer-type machines are used mostly for machining parts like the bedways for large machine tools, and other long workpieces that require accurate flat and angular surfaces or grooves.

Profile Milling Machines:

Two dimensional profiling can be done by using a template, or with a numerically controlled vertical milling machine. Some profilers have several spindles, and a number of duplicate parts can be produced in each cycle. Hydraulic-type profilers have a stylus that is brought into contact with the template to start the operation. The operator then moves the stylus along the template, causing hydraulic fluid under pressure to flow to the proper actuating cylinders. The table moves the work past

of wood, plaster, or soft metal. The stylus follows the contour of the pattern guiding the cutter as it machines out the cavity. Numerically controlled milling machines are also used for this type of work.

13.3 Computer Controlled Machining Systems

Several of the standard machines discussed in previous chapters of this text are capable of performing multiple operations. A lathe for example is capable of turning, facing, drilling, threading, etc. A drilling machine is capable of drilling, reaming, counter-sinking, tapping, and so on.

However, when increased production rates require the purchase of additional machining capability, it is almost always more economical and feasible to purchase multifunctional ma-

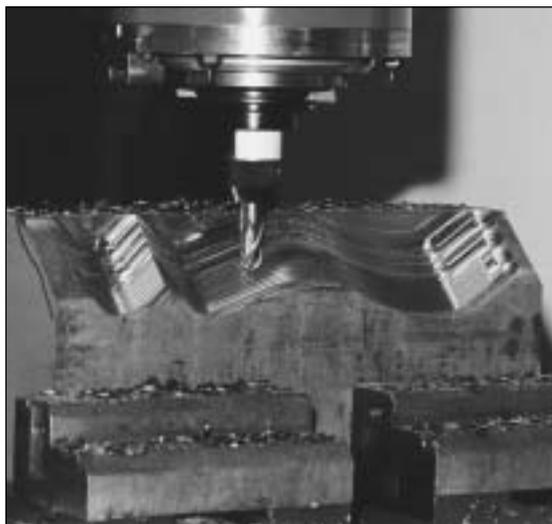


FIGURE 13.9a: Profile milling operation is shown, using a ball-nosed end milling cutter. (Courtesy Ingersoll Milling Machine Co.)

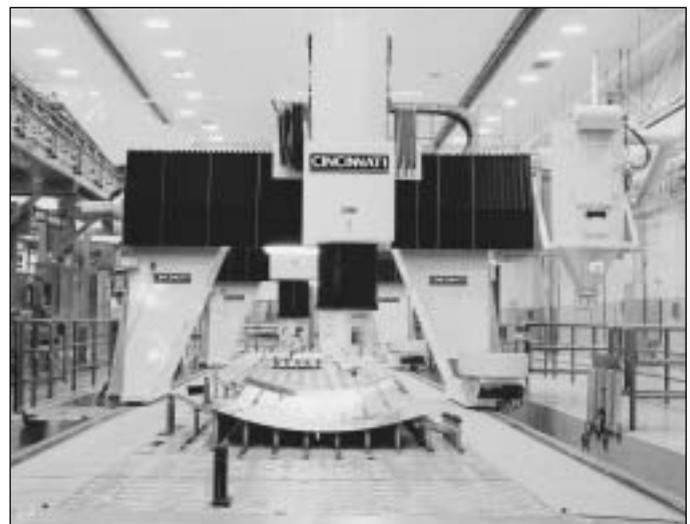
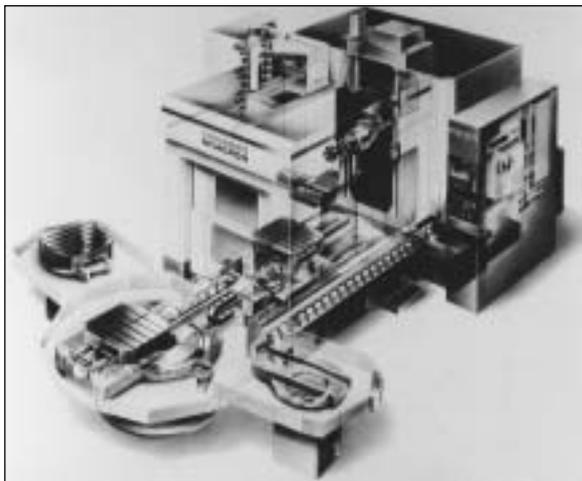


FIGURE 13.9b: A large 5-axis multispindle profile milling machine. (Courtesy Cincinnati Machine)



FIGURE 13.10: Preset machining center tools are stored in a toolholder magazine. (Courtesy Kennametal Inc.)

FIGURE 13.11: Horizontal machining center with pallet shuttling system. (Courtesy Cincinnati Machine)



chines capable of quick changes, simultaneous machining, and automatic processing.

The term “Automation” was coined in the 1950’s and applied to devices that automatically load and unload parts. The

term is now accepted to cover, in addition to loading and unloading, functions such as processing, measuring workpiece size, adjusting a machine to maintain size, and repeating of workpiece cycles.

Computer Controlled Manufacturing Systems discussed here will be Machining Centers and Flexible Machining Systems.

13.3.1 Machining Centers

Machining centers are designed and built to provide for flexible manufacturing. They can be used to machine just a few parts or large production runs. Programming can be relatively simple and the use of ‘canned’ cycles provides a great deal of versatility. An NC machining center by definition is able to perform milling, drilling, and boring cuts and has either indexing

turret tool holders or provides for automatic tool change. A tool holder magazine is shown in Figure 13.10

Machining centers are built in either horizontal or vertical configuration.

The relative merits of each will be discussed briefly.

Horizontal Machines: Horizontal machines tend to be advantageous for heavy box-shaped parts, such as gear housings, which have many features that need to be machined on the side faces. The horizontal machine easily supports heavy workpieces of this type. If a rotary indexing worktable is added, four sides of the workpiece can be machined without re-fixturing.

Pallet systems used to shuttle pieces in and out of the workstation tend to be easier to design for horizontal machines, where everything in front of the main column is open and accessible. A horizontal machining center with a pallet shuttling system is shown in Figure 13.11.

Vertical Machines: Vertical machining centers are often preferred for flat parts that must have through holes. Fixtures for these parts are more easily designed and built for a vertical spindle. Also, the thrust of the cut developed in drilling or in milling pockets can be absorbed directly by the bed of the machine. The motions of a 3 and 5-axis CNC vertical machining center are shown in Figure 13.12.

The vertical machine is preferred where three-axis work is done on a single face as in mold and die work. The weight of the head of a vertical machine as it extends away from the column, particularly on large machines, can be a factor in maintaining accuracy, as there may be some tendency for it to drop and lose accuracy and cause chatter. A vertical machining center is shown in Figure 13.13 and a machining center with an adjustable head, tool change magazine and a pallet system is shown in Figure 13.14

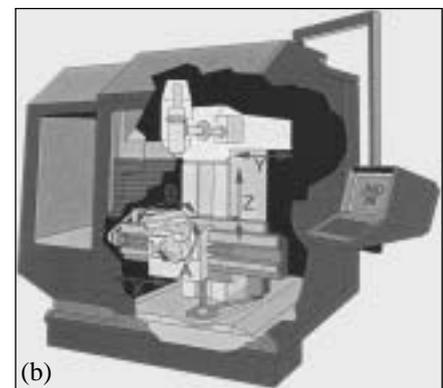
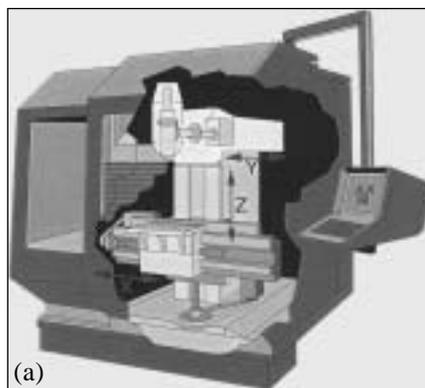


FIGURE 13.12: The motions of (a) a 3-axis CNC vertical machining center and (b) a 5-axis CNC vertical machining center. (Courtesy Kennametal Inc.)

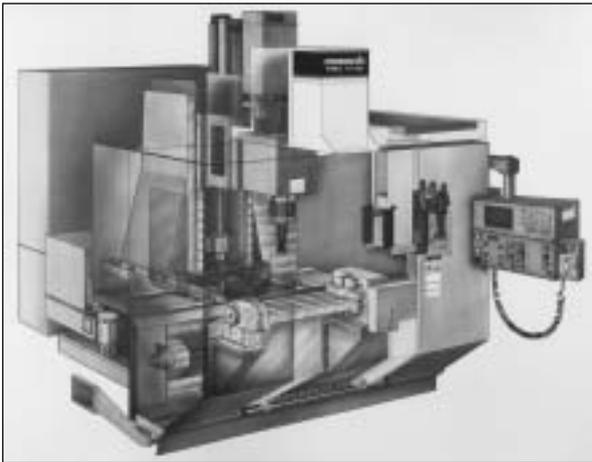


FIGURE 13.13: Transparent rendition of a vertical machining center. (Courtesy Monarch Machine Tool)



FIGURE 13.14: Machining center with adjustable head, tool magazine and pallet system (Courtesy Cincinnati Machine)

13.3.2 Flexible Machining Systems

Flexible machining systems employ one or more machining centers, usually along with other equipment, to produce medium-volume workpieces. A workpiece-handling system is required, and a central computer typically controls the entire arrangement. A flexible machining system is shown in Figure 13.15.

Material Handling: Parts are moved from storage and between machine elements by means of one of several different types of systems. The material-handling system selected must be capable of routing any part to any machine in any order and also to provide a bank of parts ahead of each machine to realize maximum productivity. Parts are normally loaded and unloaded manually.

The various types of material-handling systems used include: automated guided vehicles, towline systems, roller conveyer systems, overhead conveyer

systems, monorails, cranes and robots. An FMS material-handling system is shown in Figure 13.16.

Control Systems: The computer controls of flexible machining systems have three functional levels

Master Control – The master control monitors and controls the entire system, including routing workpieces to appropriate machines, scheduling work, and monitoring machine functions.

Direct Numerical Control – A DNC computer distributes appropriate programs to individual CNC machines and supervises and monitors their operations.

Element Control – The third and lowest level of control is computer control of the machining

cycles of individual machines.

13.4 Milling Machine Attachments and Accessories

Many accessories have been developed for milling machines. Some are specialized and can be used for only a few operations. Others, such as vises, arbors, and collets, are used in almost all milling operations.

13.4.1 Special Milling Heads

Several types of special heads have been developed for use on horizontal or vertical milling machines. These accessories increase the versatility of the machine. For example, a vertical head can be attached to a conventional horizontal column and knee milling ma-



FIGURE 13.15: Flexible machining system. (Courtesy Sandvik Coromant Co.)



FIGURE 13.16: FMS material-handling system. (Courtesy Giddings + Lewis LLC)

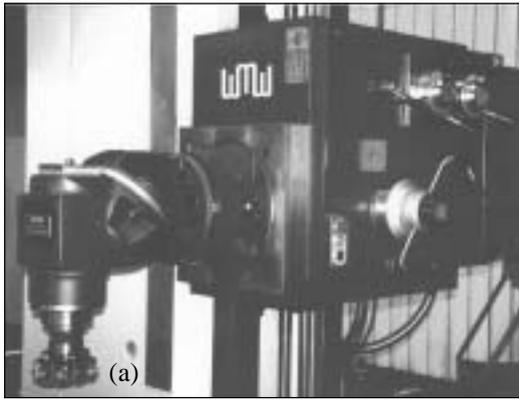


FIGURE 13.17: (a) Universal milling head attachment. (b) A universal milling head attachment being used on a horizontal milling machine. (Courtesy WMW Machinery Co., Inc.)



chine, greatly increasing its usefulness, especially in small shops with a limited number of machines.

Vertical Heads: Vertical heads are generally attached to the face of the column or to the overarm of a horizontal milling machine. The head is a semi-universal type, which pivots only on the axis parallel to the centerline of the spindle, or it is fully universal. Fully universal heads can be set to cut compound angles. Both types of heads are powered by the spindle of the milling machine and accept standard arbors and collets. A universal milling head attachment is shown in Figure 13.17a. Figure 13.17b shows a universal milling head attachment being used on a horizontal milling machine.

Rack-Milling Attachment: The rack-milling attachment bolts to the spindle housing of the milling machine. Its spindle is at a right angle to the main spindle of the machine. Both spur and helical racks can be milled with this attachment, and it can also be used to mill worms. Some rack-milling attachments have an outboard support for the spindle, which makes it possible to take heavier cuts.

Slotting Attachment: This attachment that is bolted to the column of a horizontal milling machine can be swiveled 90 degrees in either direction from the vertical position. It is used primarily in toolmaking and prototype work for cutting keyways, internal splines, and square or rectangular cavities. The crank that actuates the reciprocating slide is driven directly by the spindle, and the stroke is adjustable.

High Speed Attachment: When spindle speeds beyond the operating

range of the machine are necessary, high speed attachments can be placed on both horizontal and vertical milling machines. A gear train is generally used to step up the speed as much as 6 to 1, which allows more efficient use of small cutters.

13.4.2 Vises and Fixtures

In all milling operations, the work is held by fixtures, vises, or clamping arrangements. In most cases the work is held stationary in relation to the table while it is being machined, but work held in indexing heads and rotary tables can be moved in two planes while machining operations are in progress.

Plain Vise: Plain milling vises (Fig. 13.18a) are actuated by an Acme threaded screw, and the movable jaw moves on either a dovetail or rectangular slide. The vises are usually cast of high grade gray cast iron or ductile iron and can be heat treated. Steel keys are attached in slots machined into the bottom of the vise, parallel with and perpendicular to, the fixed jaw to allow accurate placement on the milling table. The jaw inserts are usually heat treated alloy steel and are attached by cap screws. The jaw width and maxi-

mum opening classify vises of this type. Cam-operated plain milling vises are widely used in production work because of the savings in time and effort and the uniform clamping pressure that can be achieved.

Swivel-Base Vise: A swivel-base vise is more convenient to use than the plain vise, although it is somewhat less rigid in construction. The base, which is graduated in degrees, is slotted for keys that align it with the T-slots in the table. The upper part of the vise is held to the base by T-bolts that engage a circular T-slot. The swivel-base vise, when used on a milling machine with a semi-universal head, makes it possible to mill compound angles on a workpiece.

Universal Vise: A universal vise (Fig. 13.18b) is used mostly in toolroom diemaking, and prototype work. The base of the vise is graduated in degrees and held to the table by T-bolts. The intermediate part of the vise has a horizontal pivot upon which the vise itself can rotate 90 degrees. Because there are several joints and piv-

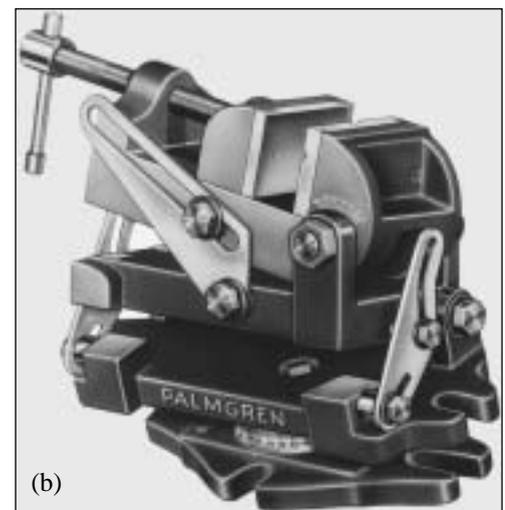


FIGURE 13.18: (a) A plain vise. (b) A universal vise, which is generally used in a toolroom diemaking and prototype work. (Courtesy Palmgren Steel Products, Inc.)

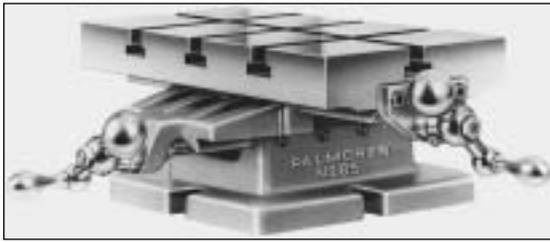


FIGURE 13.19: Adjustable-angle tables can be tilted in one or more directions to machine irregularly shaped parts. (Courtesy Palmgren Steel Products, Inc.)

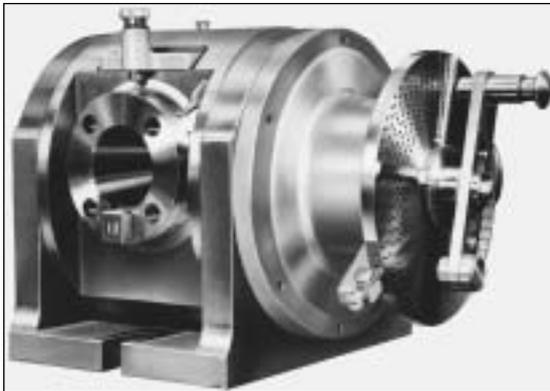


FIGURE 13.20: Indexing heads, also called dividing heads, can be used on vertical and horizontal milling machines. (Courtesy Cincinnati Machine)

ots in the vise assembly, the universal vise is usually the least rigid of the various types of milling machine vises.

Angle Plates: Several types of angle plates can be used to hold work or work holding fixtures for milling (Fig. 13.19). Plain angle plates are available in T-slotted or blank form and are usually strong iron castings. Adjustable angle plates may tilt in one direction only or have a swivel base. They are very useful for milling workpieces that are irregular in shape

and cannot be held easily in a vise.

Holding fixtures, that are a combination of a simple angle plate and a collet, are sometimes used to hold round or hexagonal work for milling. The collet-holding fixture may be manually or air operated. Both fixtures can be bolted to the milling table in the vertical or horizontal position or attached to an adjustable angle plate for holding workpieces at simple or compound angles to the table or other reference surface.

Indexing Heads: The indexing head, also known as the dividing head (Fig. 13.20), can be used on vertical and horizontal milling machines to space the cuts for such operations as making splines, gears, worm wheels, and many other parts requiring accurate division. It can also be geared to the table screw for helical milling operations such as cutting flutes in twist drills and making helical gears.

Indexing heads are of the plain or universal type. Plain heads cannot be tilted; universal heads can be tilted to the vertical or any intermediate position. The spindle of the indexing head can be fitted with a chuck, or with other work holding devices, including collets or a center. A cross-sectional view of an indexing head is shown in Figure 13.21.

Most indexing heads have a worm



(a)



(b)

FIGURE 13.22: (a) Rotary tables can be used on both vertical and horizontal milling machines. (b) A rotary table, that can also be tilted. (Courtesy Palmgren Steel Products, Inc.)

and wheel reduction ratio of 40:1, requiring 40 turns of the hand crank to make the spindle revolve once. When the necessary index plates are available, all divisions up to and including 50 can be achieved by plain indexing. For some numbers above 50, differential indexing is necessary.

In recent years, programmable precision indexers have become fairly common in shops doing work that requires accurate spacing of complex hole patterns on surfaces. The indexer may be mounted with the axis of the chuck vertical or horizontal, and in some cases the chuck may be replaced with a specially made holding fixture or a faceplate. If necessary, a tailstock may be used to support the end of the workpiece. The controller is capable of storing a series of programs, each of which may incorporate as many as 100 operational steps or positions.

Rotary Table: Rotary tables (Fig. 13.22) are available in a wide range of sizes and can be used on both vertical and horizontal milling machines. Most can also be clamped with the face at a 90 degree angle to the surface of the milling machine table. The face of the rotary table has four or more T-slots and an accurately bored hole in the center, which is concentric with the axis about which the table rotates.

The base of the rotary table, which

FIGURE 13.21: Cross-sectional view of indexing head. (Courtesy Cincinnati Machine)

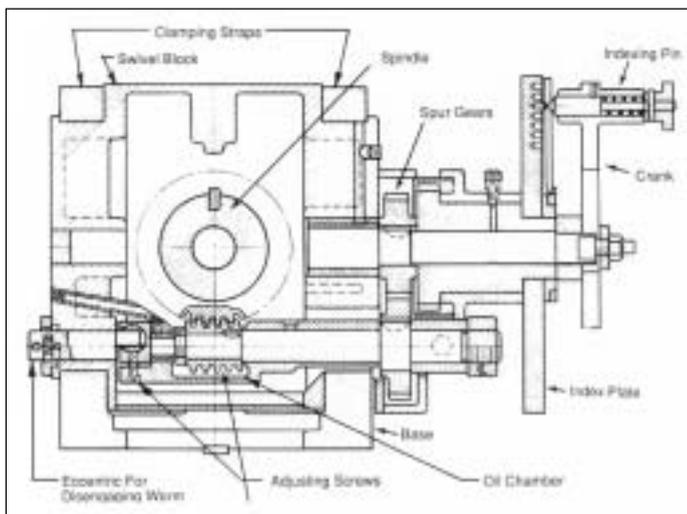
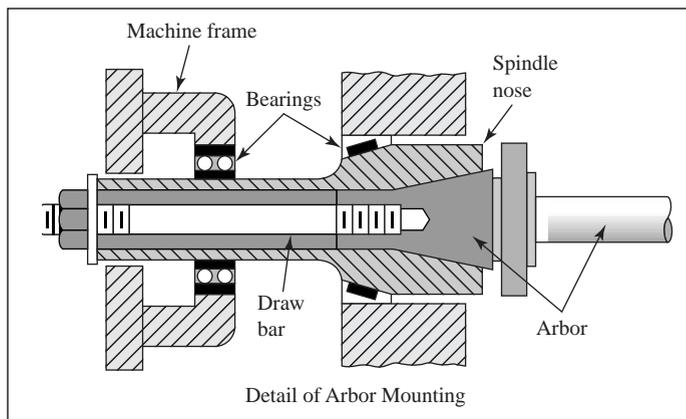


FIGURE 13.23:
Schematic drawing of an arbor mounting with drawbolt used in horizontal milling machines.



houses the worm drive mechanism, is graduated in degrees, and the handwheel can be graduated in increments as small as 5 minutes or 1/12 of 1 degree. On some rotary tables an index plate may be attached to the base.

Rotary tables can also be geared to the table feed screw. When set up in this manner, the rotary table can be used to make plate cams and to generate a number of other irregular shapes.

13.4.3 Arbors, Collets, and Tool Holders

Several basic types of arbors and collets are used to hold milling cutters and to transmit power from the spindle to the cutter. Regardless of type, they are usually precisely made of alloy steel and heat treated for wear resistance and strength.

Arbors: Arbors for horizontal milling machines (Fig. 13.23) are available in three basic types: style A, style B, and style C. A draw bolt, that goes through the spindle of the machine, screws into the small end of the taper and draws the arbor tightly into the tapered hole in the milling machine spindle. Power is transmitted from the spindle to the arbor by two short keys that engage with the slots on the flange

of the arbor.

Style A arbors consist of the tapered portion that fits the spindle, the shaft on which the cutter or cutters fit, the spacers, and the nut. The shaft has a keyway along its entire length. The outboard end of the arbor has a pilot that fits into a bronze bushing in the outboard support of the milling machine overarm (Fig. 13.2b). One or more cutters can be mounted on the arbor, either adjacent to each other or separated by spacers and shims. Style A arbors are used primarily for light and medium duty milling jobs.

Style B arbors are used for heavy milling operations, especially where it is necessary to provide support close to a milling cutter, such as in a straddle milling operation. One or more bearing sleeves may be placed on the arbor as near to the cutters as possible. An outboard bearing support is used for each bearing sleeve on the arbor.

Style C arbors are used to hold and drive shell end mills (Fig. 25.25b) and some types of face milling cutters and require no outboard support. In some cases, they can also be fitted with adapters for mounting other types of cutters.

Collets: On some vertical milling machines the spindle is bored to accept



(a)



(b)

FIGURE 13.25: (a) End milling cutter toolholders. (b) Shell end milling cutter toolholders. (Courtesy Lyndex Corp.)

a collet that has a partly straight and partly tapered shank. The collet is secured by a drawbar that is screwed into a tapped hole in the back of the collet and tightened from the top of the spindle. Some milling machine manufacturers offer collet arrangements that do not need a drawbar. Collets of this type can be closed with a lever-operated cam or with a large locking nut. Figure 13.24a shows various size collets and Figure 13.24b shows a number of tool holders including collets and a set-up fixture.

Toolholders: Standard tool holders are available for end mills and shell mills as shown in Figures 13.25a and 13.25b respectively. For some operations that require the use of tools with non-standard shank sizes, chucks can be used to hold the tool. These chucks are available with Morse taper or straight shanks. Either type can be used in milling machines when the proper adapters or collets are available.

Offset boring heads (Fig. 13.26) are



(a)



(b)

FIGURE 13.24: (a) Collets of various sizes. (Courtesy Lyndex Corp.) (b) Various toolholders with collets and a setup fixture. (Courtesy Valenite Inc.)



FIGURE 13.26: Offset boring heads are often used in milling machines for boring, facing, and chamfering operations. (Courtesy Bridgeport Machine, Inc.)

often used in vertical milling machines for boring, facing, chamfering, and outside diameter turning operations. They are available with straight, Morse taper, or standard milling machine taper shanks and usually have three mounting holes for boring bars. Two of the holes are usually parallel with the centerline of the tool, and one is perpendicular to the centerline. Some boring heads have two adjusting mechanisms, and the movable slide can be adjusted accurately in increments of 0.0001 inch.

Flycutters can be used for facing operations. The tools in cutters of this type are adjusted so that both a roughing and finishing cut are taken in one pass.

13.5 Types of Milling Operations

Milling cutters are used either individually or in combinations to machine various surfaces as described below and shown in Figure 13.27.

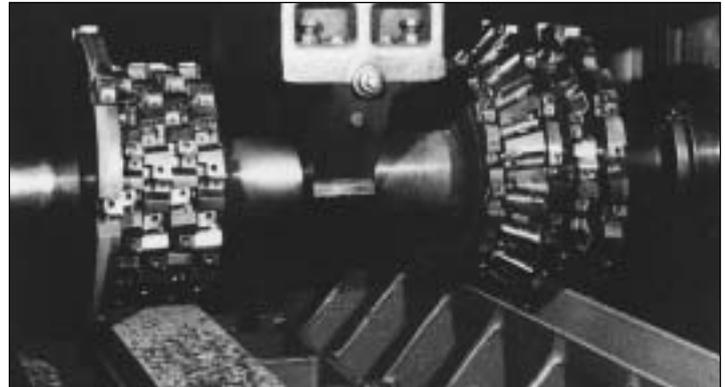
Plain Milling: Plain milling is the process of milling a surface that is parallel to the axis of the cutter and basically flat. It is done on plain or universal horizontal milling machines with cutters of varying widths that have teeth only on the periphery.

Side Milling: For side milling, a



FIGURE 13.28: Conventional or up-milling as compared to climb or down-milling.

FIGURE 13.29: In gang milling operations, horizontal, vertical, or angular surfaces are machined in one pass. (Courtesy Sandvik Coromant Co.)



cutter that has teeth on the periphery, and on one or both sides, is used. When a single cutter is being used, the teeth on both the periphery and sides may be cutting. The machined surfaces are usually either perpendicular or parallel to the spindle. Angle cutters can be used to produce surfaces that are at an angle to the spindle for such operations as making external dovetails or flutes in reamers.

Straddle Milling: In a typical straddle milling set-up (Fig. 13.27) two-side milling cutters are used. A straddle milling operation is shown in Figure 13.28. The cutters are half-side or plain side milling cutters, and have straight or helical teeth. Stagger-tooth side milling cutters can also be used.

The cutters cut on the inner sides only, or on the inner sides and the periphery. If the straddle milling operation involves side and peripheral

cuts, the diameter of the two cutters must be exactly the same. When cutters with helical teeth are used, the helix angles must be opposite.

Since straddle milled surfaces must be parallel to each other and are usually held to close tolerances in terms of width, the condition and size of the collars and shims that separate the cutters are important. The arbor must also turn as true as possible to avoid cutting the workpiece undersize.

Usually, a combination of collars and steel shims can be assembled to provide the correct spacing between cutters. For some production operations, a special collar can be made from alloy or medium-carbon steel, heat treated, and surface ground to length. The faces must be perpendicular to the bore and parallel to each other. The cutters should be keyed to the arbor, and the outboard bearing supports must be placed as close to the cutters as possible.

Gang Milling: In gang milling, three or more cutters are mounted on the arbor, and several horizontal, vertical, or angular surfaces are machined in one pass (Fig. 13.29). When making a gang milling set-up, several different types of cutters can be used, depending on the job to be done. Cutters used for producing vertical or angular surfaces must be of the side-

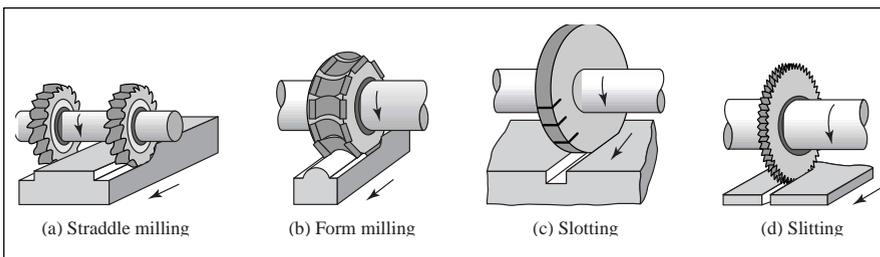


FIGURE 13.27: Drawing of a milling cutter showing the difference between advance per revolution (*apr*) and advance per tooth (*apt*).



(a)

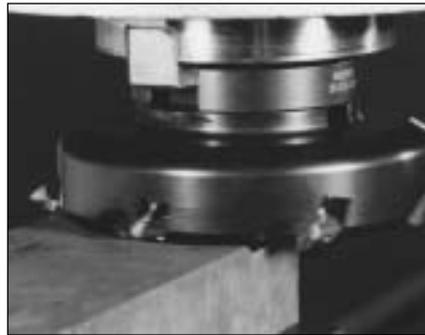
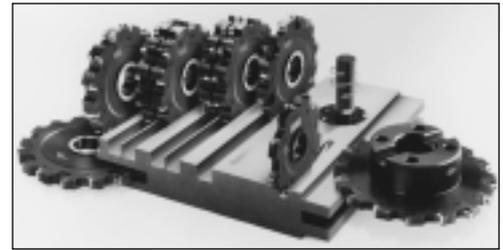
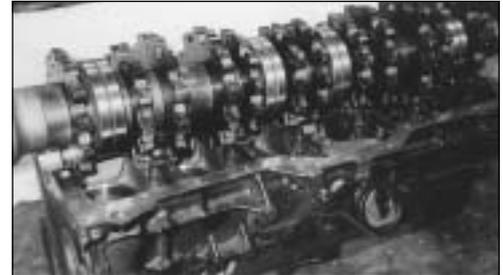


FIGURE 13.31: Face milling operation. (Courtesy Valenite Inc.)



(a)



(b)

FIGURE 13.33: Milling operations: (a) side and face milling cutters used to machine various slots, (b) various indexable milling cutters used to machine an automotive engine block. (Courtesy Sandvik Coromant Co.)



(b)

FIGURE 13.30: (a) Slotting operation used in the manufacturing of rotors. (Courtesy Iscar Metals, Inc.) (b) Slitting operation to separate cast automotive parts. (Courtesy Sandvik Coromant Co.)

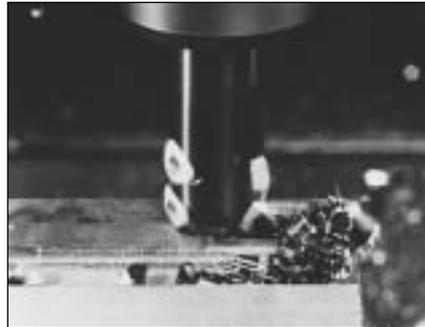


FIGURE 13.32: End milling operation. (Courtesy Valenite Inc.)

cutting type; plain milling cutters of the proper width can be used for horizontal surfaces. In some cases face mills with the teeth facing inward can be used at one or both ends of the gang milling set-up.

When only one wide plain helical milling cutter is used as part of a gang milling set-up, the side thrust caused by that cutter should be directed toward the spindle of the machine. If possible, interlocking cutters with opposite helix angles should be used to eliminate side thrust and reduce the possibility of chatter.

Because of the time and effort involved in setting up the milling machine for gang milling, the process is used mainly for production work. Since all or almost all of the workpiece is being machined at one time, power and rigidity are very desirable features in the machine being used. Every effort should be made to control vibration, including the use of support bars that are bolted to both the knee and the outboard bearing support.

Form Milling: The number of parallel surfaces and angular relationships

that can be machined by peripheral milling is limited almost only by cutter design. Form cutters are expensive, but often there is no other satisfactory means of producing complex contours as shown in Figure 13.27.

Slotting and Slitting Operations: Milling cutters of either the plain or side-cutting type are used for slotting and slitting operations (Fig.13.30a and 13.30b). Slotting and Slitting are usually done on horizontal milling machines, but can also be done on vertical mills by using the proper adaptors and accessories. Metal slitting cutters of various diameters and widths are also used to cut slots. A number of identical cutters can also be mounted on the same arbor for cutting fins. When the thickness of the fins must be held to

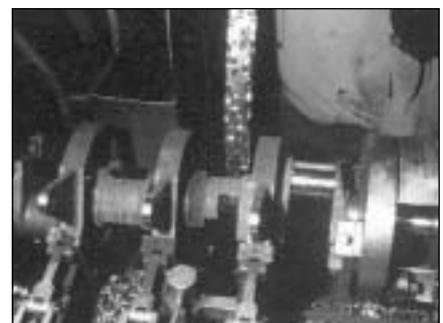
close tolerances, spacers are usually machined and surface ground to provide the necessary accuracy.

Face Milling: Face milling (Fig. 13.31) can be done on vertical and horizontal milling machines. It produces a flat surface that is perpendicular to the spindle on which the cutter is mounted. The cutter ranges in size and complexity from a simple, single-tool flycutter to an inserted-tooth cutter with many cutting edges. Large face mills are usually mounted rigidly to the nose of the spindle. They are very effective for removing large amounts of metal, and the workpiece must be securely held on the milling table.

End Milling: (Fig. 13.32) End milling is probably the most versatile milling operation. Many types of end mills can be used on both vertical and



(a)



(b)

FIGURE 13.34: Two turn milling operations. (Courtesy Sandvik Coromant Co.)

horizontal milling machines. End mills are available in sizes ranging from 1/32 to 6 inches (for shell end mills) and in almost any shape needed.

Milling cutters can be used individually or in pairs to machine various slots (Fig. 13.33a). Many milling cutters can also be mounted onto an arbor to perform in a gang milling operation of automotive engine blocks as shown in Figure 25.33b

13.6 Turn Milling

Turn milling consists of a number of

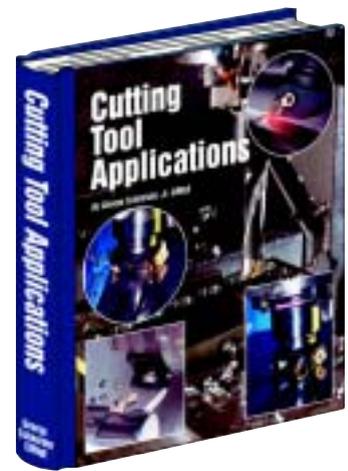
different machining methods where a milling cutter machines a rotating workpiece. These methods are primarily used for machining various eccentrically shaped parts; planes, tapered and cylindrical surfaces; grooves and inside holes.

Turn milling requires a machine tool with certain functions and a number of axes. Machining centers, turning centers, specially adapted lathes, milling machines, boring mills and special-purpose machines are used. When other operations of turning and milling

are combined in the machines, single set-up machining leads to advantages of fast through-put times and flexibility of production. Two turn milling operations are shown in Figure 13.34a and Figure 13.34b.

Advantages associated with turn milling are: capability of machining large and unbalanced parts which cannot be rotated at high speeds; complex surface shapes, eccentric parts and components with additional elements that protrude; log, unstable shafts or thin-walled parts.

CHAPTER 14



Broaches and Broaching

Metal Removal
Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining
Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes
Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining
Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes
Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

14.1 Introduction

The broaching operation is similar to shaping with multiple teeth and is used to machine internal and external surfaces such as holes of circular, square, or irregular shapes, keyways, and teeth of internal gears. A broach is a long multitooth cutting tool with successively deeper cuts. Each tooth removes a predetermined amount of material in a predetermined location. The total depth of material removed in one path is the sum of the depth of cut of each tooth. Broaching is an important pro-

duction process and can produce parts with very good surface finish and dimensional accuracy. Broaching competes favorably with other processes such as boring, milling, shaping and reaming. Although broaches tend to be expensive, the cost is justified because of their use for high production runs. A two station broaching operation is shown in Figure 14.1.

14.2 Broaching

Tooling is the heart of any broaching process. The broaching tool is based on a concept unique to the process -

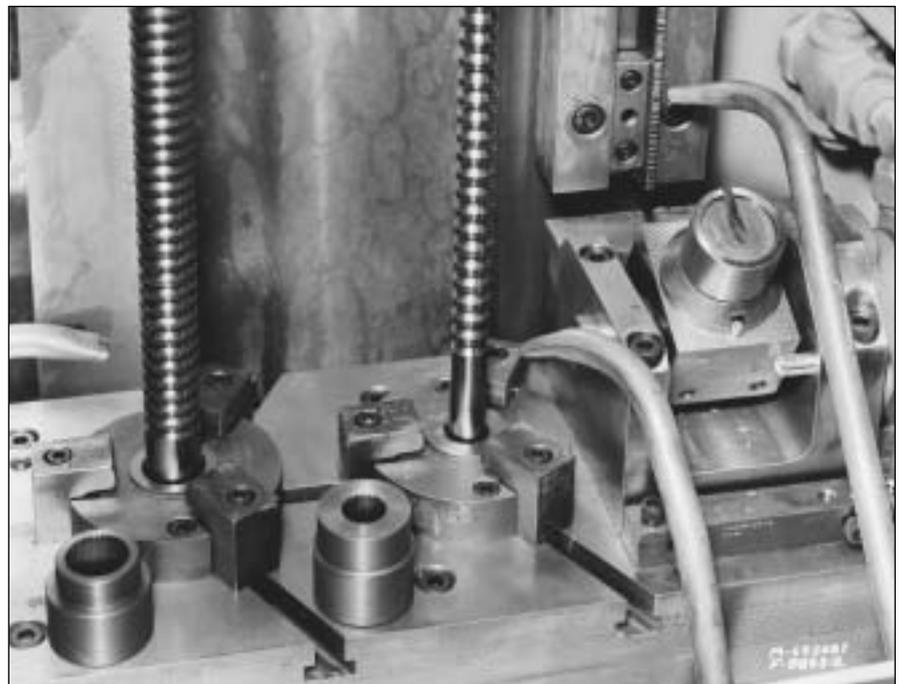


FIGURE 14.1: Typical broaching operation of an internal spline. (Courtesy Detroit Broach & Machine Co.)



FIGURE 14.2: A large broach is shown (Courtesy Detroit Broach & Machine Co.)



FIGURE 14.3: A couple of small broached parts are shown. (Courtesy Detroit Broach & Machine Co.)

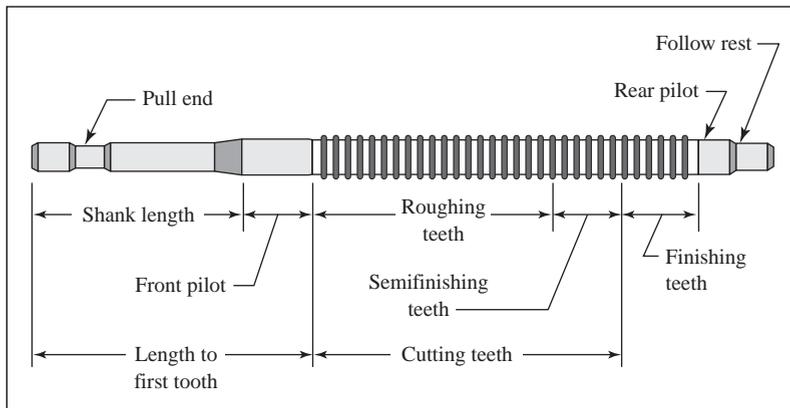


FIGURE 14.4: Principal parts of a round internal-pull broach.

rough, semi-finish, and finish cutting teeth combined in one tool or string of tools. A broach tool frequently can finish machine a rough surface in a single stroke. A large broach is shown in Figure 14.2

For exterior surface broaching, the broach tool may be pulled or pushed across a workpiece surface, or the surface may move across the tool. Internal broaching requires a starting hole or opening in the workpiece so the broaching tool can be inserted. The tool or the workpiece is then pushed or pulled to force the tool through the starter hole. Almost any irregular cross-section can be broached as long as all surfaces of the section remain parallel to the direction of broach travel. A couple of small broached parts are shown in Figure 14.3

14.2.1 Broaching Tools

A broach is like a single point tool with many 'points' each of which cuts like a flat-ended shaper tool, although some broaches have teeth set diagonally, called sheer cutting. The principal parts of an internal broach are shown in Figure 14.4

14.3 Broach Nomenclature

Front Pilot: When an internal pull broach is used, the pull end and front pilot are passed through the starting hole. Then the pull end is locked to the pull head of the broaching machine. The front pilot assures correct axial alignment of the tool with the starting hole, and serves as a check on the starting hole size.

Length: The length of a broach tool or string of tools, is determined by the amount of stock to be removed, and limited by the machine stroke.

Rear Pilot: The rear pilot maintains tool alignment as the final finish teeth pass through the workpiece hole. On round tools the diameter of the rear pilot is slightly less than the diameter of the finish teeth.

Broach tooth nomenclature and terminology are shown in Figure 14.5a.

Cutting Teeth: Broach teeth are usually divided into three separate sections along the length of the tool: the roughing teeth, semi-finishing teeth, and finishing teeth (Fig. 14.4). The first roughing tooth is proportionately the smallest tooth on the tool. The subsequent teeth progressively increase in size up to and including the first

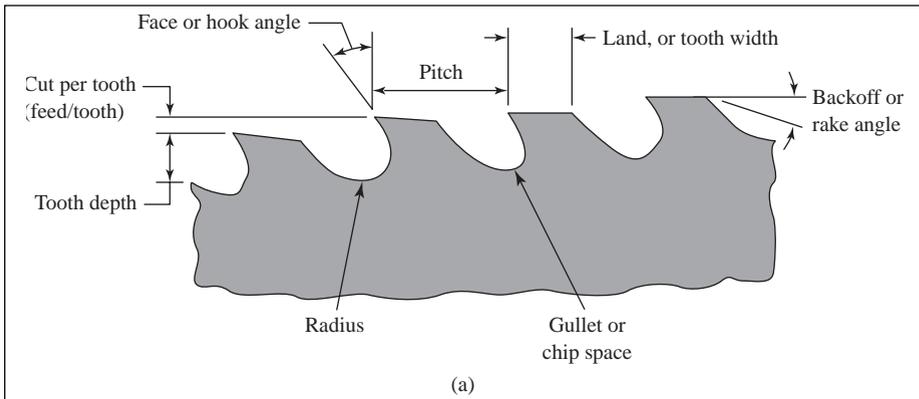
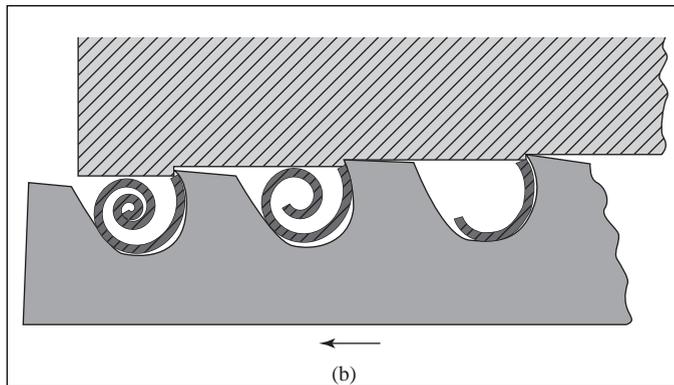


FIGURE 14.5: (a) Broach tooth nomenclature and terminology. (b) Illustration of how a chip fills the gullet during a broaching operation.



finishing tooth. The difference in height between each tooth, or the tooth rise, is usually greater along the roughing section and less along the semi-finishing section. All finishing teeth are the same size. The face is ground with a hook or face angle that is determined by the workpiece material. For instance, soft steel workpieces usually require greater hook angles; hard or brittle steel pieces require smaller hook angles.

Tooth Land: The land supports the cutting edge against stresses. A slight clearance or backoff angle is ground onto the lands to reduce friction. On roughing and semi-finishing teeth, the entire land is relieved with a backoff angle. On finishing teeth, part of the land immediately behind the cutting edge is often left straight, so that repeated sharpening (by grinding the face of the tooth) will not alter the tooth size.

Tooth Pitch: The distance between teeth, or pitch, is determined by the length of cut and influenced by type of workpiece material. A relatively large pitch may be required for roughing teeth to accommodate a greater chip load. Tooth pitch may be smaller on semi-finishing teeth to reduce the overall length of the broach tool. Pitch is calculated so that preferably, two or

more teeth cut simultaneously. This prevents the tool from drifting or chattering.

Tooth Gullet: The depth of the tooth gullet is related to the tooth rise, pitch, and workpiece material. The tooth root radius is usually designed so that chips curl tightly within themselves, occupying as little space as possible. (Fig. 14.5b)

When designing broaches, attention must also be given to chip load, chipbreakers, shear angles and side relief.

Chip Load: As each tooth enters the workpiece, it cuts a fixed thickness of material. The fixed chip length and thickness produced by broaching create a chip load that is determined by the design of the broach tool and the pre-determined feed rate.

This chip load feed rate cannot be altered by the machine operator as it can in most other machining operations. The entire chip produced by a complete pass of each broach tooth must be freely contained within the preceding tooth gullet (Fig. 14.5b). The size of the tooth gullet is a function of the chip load and the type of chips produced. However the form that each chip takes depends on the workpiece material and hook. Brittle materials produce flakes. Ductile or

malleable materials produce spiral chips.

Chipbreakers: Notches, called chipbreakers, are used on broach tools to eliminate chip packing and to facilitate chip removal. The chipbreakers are ground into the roughing and semi-finishing teeth of the broach, parallel to the tool axis. Chipbreakers on alternate teeth are staggered so that one set of chipbreakers is followed by a cutting edge. The finishing teeth complete the job. Chipbreakers are vital on round broaching tools. Without the chipbreakers, the tools would machine ring-shaped chips that would wedge into the tooth gullets and eventually cause the tool to break.

Shear Angle: Broach designers may place broach teeth at a shear angle to improve surface finish and reduce tool chatter. When two adjacent surfaces are cut simultaneously, the shear angle is an important factor in moving chips away from the intersecting corner to prevent crowding of chips in the intersection of the cutting teeth.

Another method of placing teeth at a shear angle on broaches is by using a herringbone pattern. An advantage of this design is that it eliminates the tendency for parts to move sideways in the workholding fixtures during broaching.

Side Relief: When broaching slots, the tool becomes enclosed by the slot during cutting and must carry the chips produced through the entire length of the workpiece. Sides of the broach teeth will rub the sides of the slot and cause rapid tool wear unless clearance is provided. Grinding a single relief angle on both sides of each tooth does this. Thus only a small portion of the tooth near the cutting edge, called the side land, is allowed to rub against the slot. The same approach is used for one sided corner cuts and spline broaches.

14.4 Types of Broaches

Two major types of broaches are the push broach and the pull broach. A second division is internal and external broaches.

Push and Pull Broaches: A push broach must be relatively short since it is a column in compression and will buckle and break under too heavy a load. Push broaches are often used with a simple arbor press if quantities of work are low. For medium to high

FIGURE 14.6: Various broach configurations, both round and flat types.

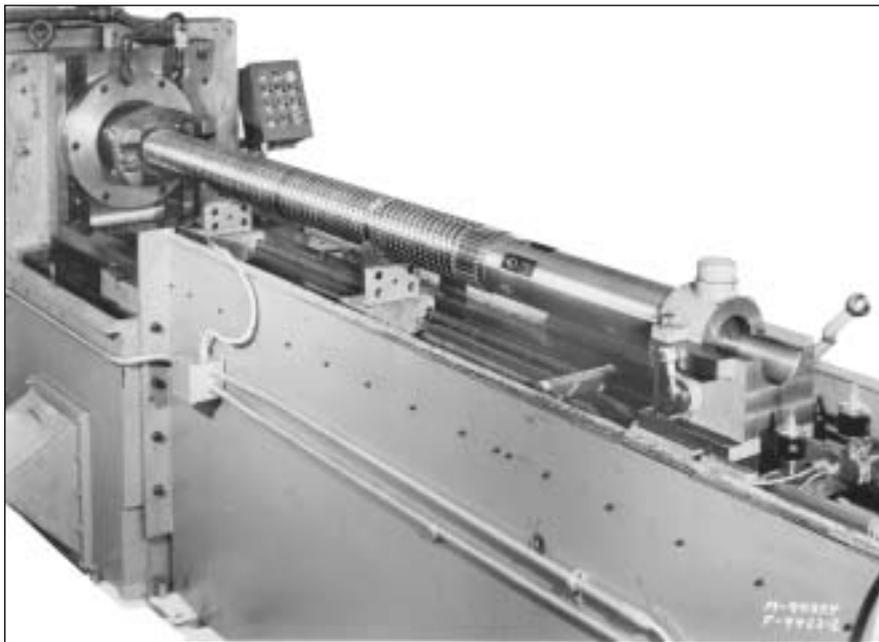
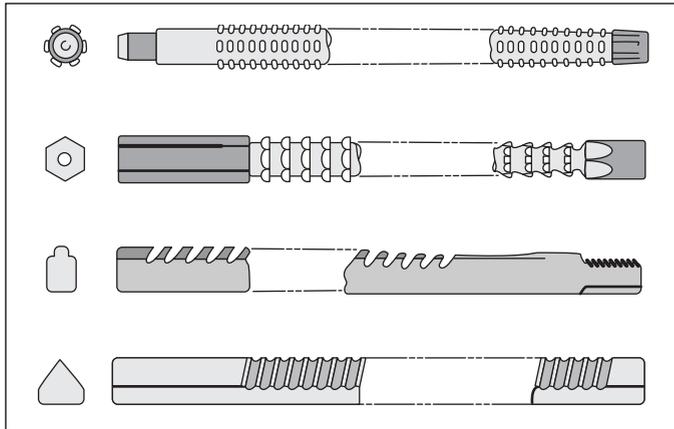


FIGURE 14.7: A large spline broaching operation using a horizontal broaching machine. (Courtesy US Broach & Machine Co.)

volume production they are used in broaching machines.

Pull broaches (Fig. 14.4) are pulled either up, down, or horizontally through or across the workpiece, always by a machine. Flat or nearly flat broaches may be pull type, or the broach may be rigidly mounted, with the workpiece then pulled across the broaching teeth. Automobile cylinder blocks and heads are often faced flat by this method. Figure 14.6 shows various broach configurations both round and flat types.

Figure 14.1 shows a vertical spline broaching operation; Figure 14.7 shows a large spline broaching operation using a horizontal broaching machine.

14.4.1 Internal Broaches

Internal broaches are either pulled or

pushed through a starter hole. The machines can range from fully automated multi stationed verticals, to horizontal pull types, to simple presses. Figure 14.8 shows a variety of forms that can be produced by internal broaches.

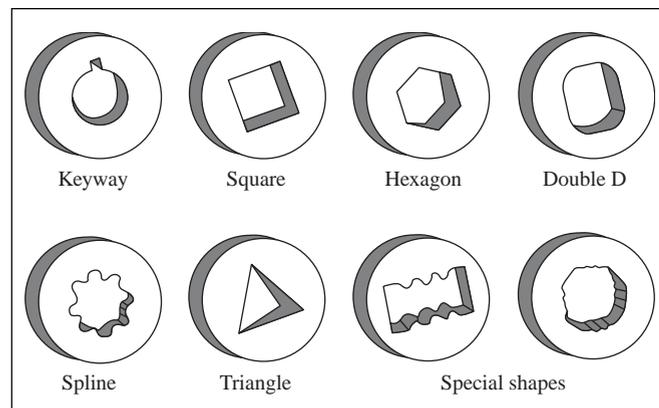


FIGURE 14.8: A variety of forms that can be produced with an internal broach.

Keyway Broach: Almost all keyways in machine tools and parts are cut by a keyway broach - a narrow, flat bar with cutting teeth spaced along one surface. Both external and internal keyways can be cut with these broaches. Internal keyways usually require a slotted bushing or horn to fit the hole, with the keyway broach pulled through the horn, guided by the slot.

If a number of parts, all of the same diameter and keyway size, are to be machined, an internal keyway broach can be designed to fit into the hole to support the cutting teeth. Only the cutting teeth extend beyond the hole diameter to cut the keyway. Bushings or horns are not required.

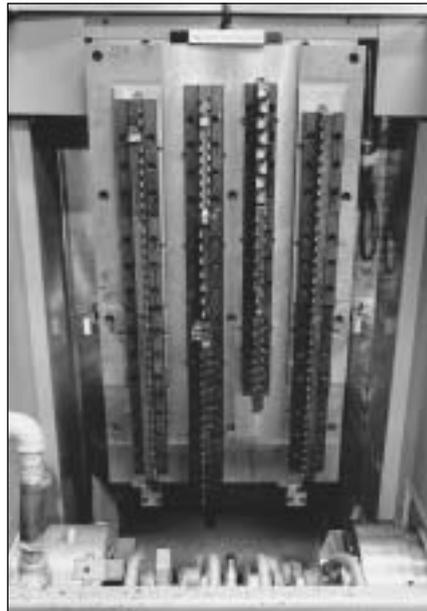
Burnishers: Burnishers are broaching tools designed to polish rather than cut a hole. The total change in diameter produced by a burnishing operation may be no more than 0.0005 to 0.001 inch. Burnishing tools, used when surface finish and accuracy are critical, are relatively short and are generally designed as push broaches.

Burnishing buttons sometimes are included behind the finishing tooth section of a conventional broaching tool. The burnishing section may be added as a special attachment or easily replaced shell. These replacement shells are commonly used to reduce tooling costs when high wear or tool breakage is expected. They are also used to improve surface finish.

Shell Broaches: Shell broaches can be used on the roughing, semi-finishing and finishing sections of a broach tool. The principal advantage of a shell broach is that worn sections can be removed and re-sharpened or replaced, at far less cost than a conventional single piece tool. When shells are used for the finishing teeth of long broaches, the teeth of the



(a)



(b)

FIGURE 14.9: (a) A surface broach assembly. (Courtesy Detroit Broach & Machine Co.) (b) A surface broach assembly with indexable carbide inserts. (Courtesy Ingersoll Cutting Tools)

shell can be ground to far greater accuracy than those of a long conventional broach tool and the tool can continue to be used by replacing the shell. Shell broaches are similar to shell milling cutters that were discussed in Chapter 12.

14.4.2 Surface Broaches

The broaches used to remove material from an external surface are commonly known as surface broaches. Such broaches are passed over the workpiece surface to be cut, or the workpiece passes over the tool on horizontal, vertical, or chain machines to produce flat or contoured surfaces.

While some surface broaches are of solid construction, most are of built-up design, with sections, inserts, or indexable tool bits that are assembled end-to-end in a broach holder or sub holder. The holder fits on the machine slide and provides rigid alignment and support. A surface broach assembly is shown in Figure 14.9a.

Sectional Broaches: Sectional broaches are used to broach unusual or difficult shapes, often in a single pass. The sectional broach may be round or flat, internal or external. The principle behind this tool is similar to that of the shell broach, but straight sections of teeth are bolted along the long axis of the broach rather than being mounted on an arbor. A complex broaching tool

can be built up from a group of fairly simple tooth sections to produce a cut of considerable complexity.

Carbide Broach Inserts: Broaching tools with brazed carbide broach inserts are frequently used to machine cast iron parts. Present practice, such as machining automotive engine blocks, has moved heavily to the use of indexable inserts (Fig. 14.9b) and this has drastically cut tooling costs in many applications.

Slab Broaches: Slab broaches, simple tools for producing flat surfaces, come closest to being truly general purpose broaches. A single slab broach can be used to produce flat surfaces having different widths and depths on any workpiece by making minor adjustments to the broach, fixture and/or machine.

Slot Broaches: Slot broaches are for cutting slots, but are not as general purpose in function as slab broaches. Adjustments can easily be made to produce different slot depths, but slot widths are a function of the broach width. When sufficient production volume is required however, slot broaches are often faster and more economical than milling cutters. In broaching, two or more slots can often be cut simultaneously.

14.5 Types of Broaching Machines

The type of broach cutting tool re-

quired for a given job is the single most important factor in determining the type of broaching machine to be used. Second in importance is the production requirement. Taken together, these factors usually determine the specific type of machine for the job.

The type of broach tool (internal or surface) immediately narrows down the kinds of machines that could be used. The number of pieces required per hour, or over the entire production run, will further narrow the field.

For internal broaching, the length of a broach in relation to its diameter may determine whether it must be pulled rather than pushed through the workpiece, for a broach tool is stronger in tension than in compression. This in turn, helps determine the type of machine for the job.

The type of drive, hydraulic or electromechanical, is another important factor in machine selection. So are convertibility and automation. Some machine designs allow for conversion from internal to surface work. Some designs are fully automated; others are limited in scope and operate only with close operator supervision.

14.5.1 Vertical Broaching Machines

About 60 percent of the total numbers of broaching machines in existence are vertical, almost equally divided between vertical internals and vertical surface or combination machines. Vertical broaching machines, used in every major area of metalworking, are almost all hydraulically driven. Figure 14.1 shows a vertical broaching operation.

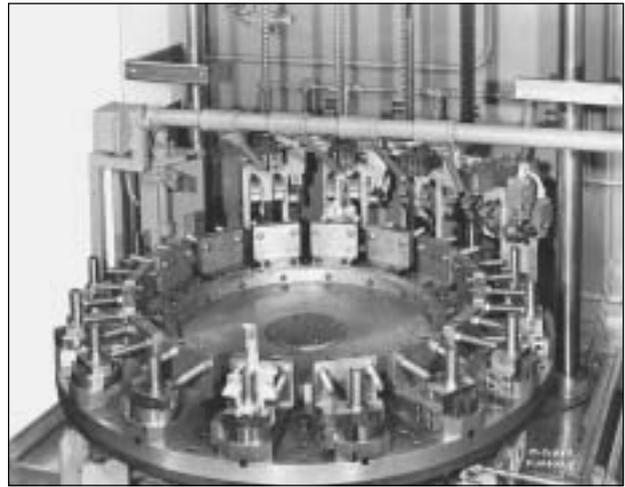
One of the essential features that promoted their development however, is beginning to turn into a limitation. Cutting strokes now in use often exceed existing factory ceiling clearances. When machines reach heights of 20 feet or more, expensive pits must be dug for the machine, so that the operator can work at the factory floor level. A large vertical broaching machine is shown in Figure 14.10a.

Vertical internal broaching machines are table-up, pull-up, pull-down, or push-down, depending on their mode of operation.

Vertical Table-up: Today table-up machines are demanded to meet the cell concept (flexible) manufacturing, where short runs of specialized compo-



(a)



(b)

FIGURE 14.10: (a) A large vertical broaching machine. (b) Special multi-station vertical broaching machine fixture. (Courtesy US Broach & Machine Co.)

nents are required. Upon completion of short runs (1 - 2 years) the machines can be re-tooled and moved to another area of the plant without the problem of what to do with pits in shop floors. With this type of machine the part sits on a table that moves up while the broach is stationary. Stroke lengths from 30 to 90 inches and capacities from 5 to 30 tons are the limits for this machine.

Vertical Internal Pull-up: The pull-up type, in which the workpiece is placed below the worktable, was the first to be introduced. Its principal use is in broaching round and irregular shaped holes. Pull-up machines are now furnished with pulling capacities of 6 to 50 tons, strokes up to 72 inches, and broaching speeds of 30 FPM. Larger machines are available; some have electro-mechanical drives for greater broaching speed and higher productivity.

Vertical Internal Pull-down: The more sophisticated pull-down machines, in which the work is placed on top of the table, were developed later than the pull-up type. These pull-down machines are capable of holding internal shapes to closer tolerances by means of locating fixtures on top of the work table. Machines come with pulling capacities of 2 to 75 tons, 30 to 110 inch strokes, and speeds of up to 80 FPM.

Vertical Internal Push-down: Vertical push down machines are often nothing more than general-purpose hydraulic presses with special fixtures. They are available with capacities of 2 to 25 tons, strokes up to 36 inches, and speeds as high as 40 FPM. In some cases, universal machines have been designed which combine as many as three different broaching operations, such as push, pull, and surface, simply through the addition of special fixtures.

A special multi-station vertical broaching machine fixture is shown in Figure 14.10b.

A vertical broaching machine with loading and unloading conveyers is shown in Figure 14.11

14.5.2 Horizontal Broaching Machines

The favorite configuration for broaching machines seems now to have come full circle. The original gear or screw driven machines were designed as horizontal units. Gradually, the vertical machines evolved as it became apparent that floor space could be much more efficiently used with vertical units. Now the horizontal machine, both hydraulically and mechanically driven, is again finding increasing favor among users because of its very long strokes and the limitation that ceiling height places on vertical machines. About 40 percent of all broaching machines are now horizontal. For some types of work such as

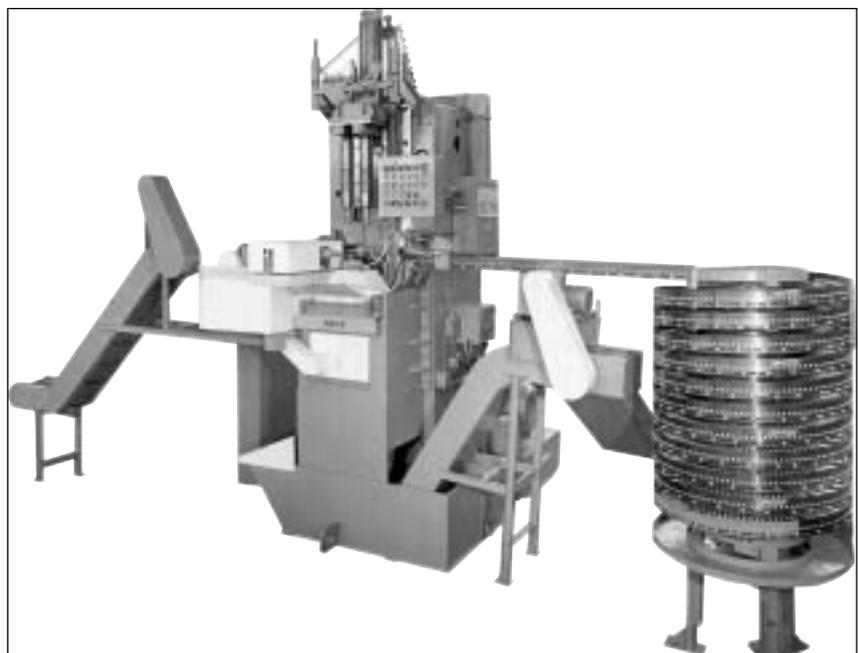
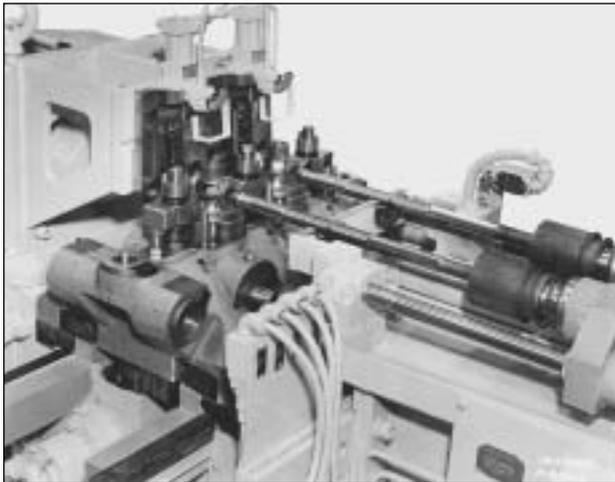
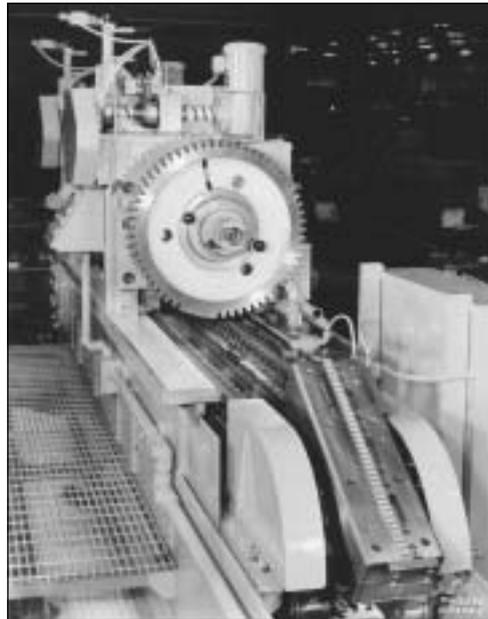


FIGURE 14.11: Vertical broaching machine with loading and unloading conveyers. (Courtesy Detroit Broach & Machine Co.)



(a)

FIGURE 14.12: (a) Two-station internal horizontal broaching machine. (b) Gear broaching operation. (Courtesy Detroit Broach & Machine Co.)



(b)

roughing and finishing automotive engine blocks, they are used exclusively. A two station internal horizontal broaching machine is shown in Figure 14.12a

Horizontal Internal Broaching Machines: By far the greatest amount of horizontal internal broaching is done on hydraulic pull type machines for which configurations have become somewhat standardized over the years. Fully one third of the broaching machines in existence are this type, and of these nearly one fourth are over twenty years old. They find their heaviest application in the production of general industrial equipment but can be found in nearly every type of industry.

Hydraulically driven horizontal internal machines are built with pulling capacities ranging from 2 1/2 to 75

tons, the former representing machines only about 8 feet long the latter machines over 35 feet long. Strokes up to 120 inches are available, with cutting speeds generally limited to less than 40 FPM.

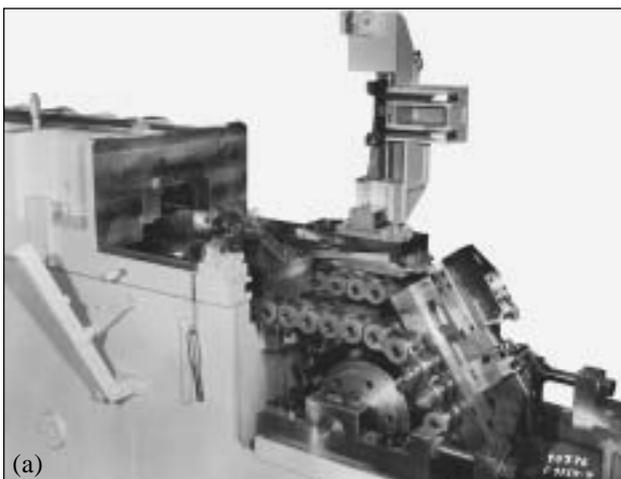
Horizontal Surface Broaching Machines: This type accounts for only about 10 percent of existing broaching machines, but this is not indicative of the percentage of the total investment they represent or of the volume of work they produce. Horizontal surface broaching machines belong in a class by themselves in terms of size and productivity. Only the large continuous horizontal units can match or exceed them in productivity. Horizontal surface units are manufactured in both hydraulically and electro-mechanically driven models, with the latter now

becoming dominant.

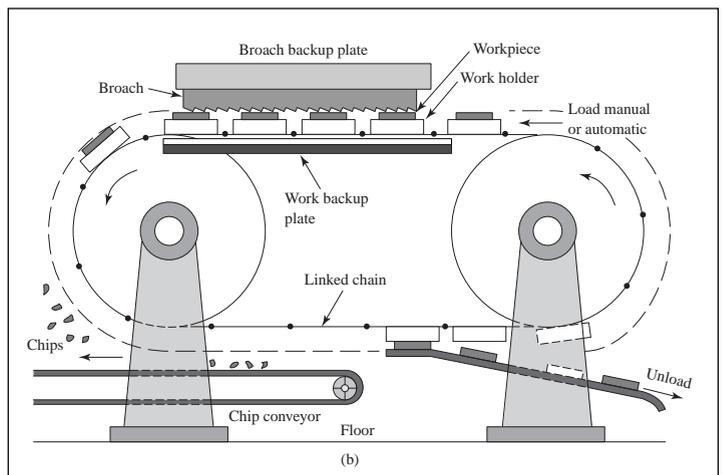
A gear broaching operation is shown in Figure 14.12b.

The older hydraulically driven horizontal surface machines now are produced with capacities up to 40 tons, strokes up to 180 inches, and normal cutting speeds of 100 FPM. These machines, a major factor in the automotive industry for many years, turn out a great variety of cast iron parts. They use standard carbide cutting tools and have some of the highest cutting speeds used in broaching.

But electro-mechanically driven horizontal surface machines are taking over at an ever-increasing rate for some applications, despite their generally higher cost. Because of their smooth ram motion and the resultant improvements in surface finish and

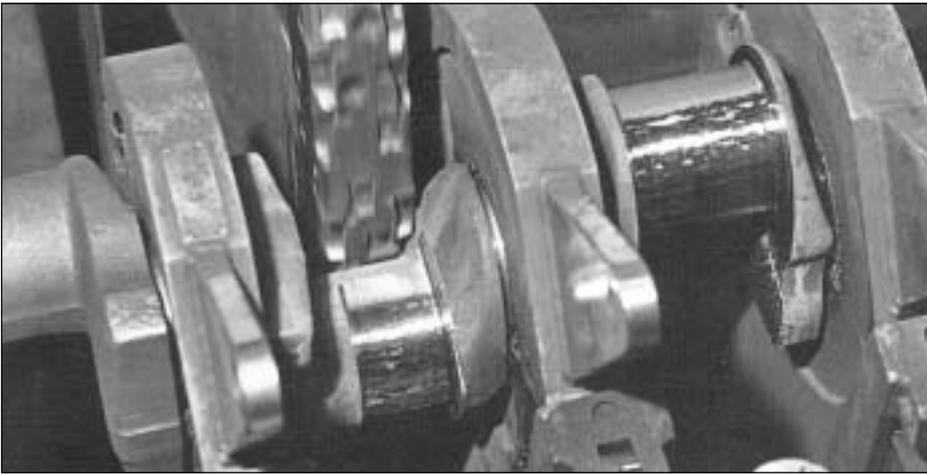


(a)



(b)

FIGURE 14.13: (a) Continuous chain broaching operation. (Courtesy US Broach & Machine Co.) (b) Schematic illustration of a continuous chain broaching machine.



(a)

(b)

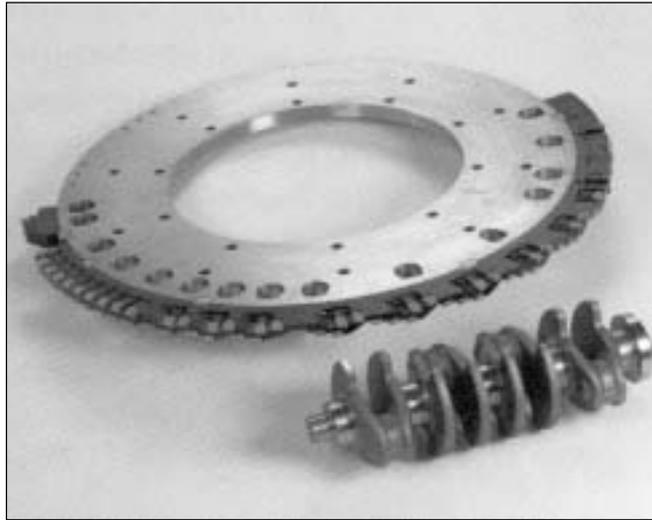


FIGURE 14.14: (a) Turn-broaching operation of a crankshaft. (b) Turn-broach cutter assembly. (Courtesy Sandvik Coromant Co.)

part tolerances, these machines have become the largest class of horizontal surface broaching units built. They are available with pulling capacities in excess of 100 tons, strokes up to 30 feet, and cutting speeds, in some instances, of over 300 FPM.

14.5.3 Chain Broaching Machines

These have been the most popular type of machine produced for high-production surface broaching. The key to the productivity of a continuous horizontal broaching machine is elimination of the return stroke by mounting the workpieces, or the tools, on a continuous chain (Fig. 14.13a)

Most frequently, the tools remain stationary, mounted in a tunnel in the top half of the machine, and the chain mounted workpieces pass underneath them. A schematic of a chain-broaching machine is shown in Figure 14.13b.

14.6 Turn-Broaching

Turn-broaching is an efficient method for machining steel and nodular cast iron crankshafts. Special turn-broaching machines are available for linear, circular and spiral operating methods. The peripheral type cutter assemblies are built in segments as shown in Figure 14.14b.

The turn-broaching systems basically use similar standardized components for roughing and finishing. The type of machine determines the tool design: linear, circular or spiral. The number of segments and roughing inserts in the tool depend on the stock removal rate required. The finishing segments are fitted with inserts in adjustable cartridges that can be set to close tolerances. The segment for roughing has fixed insert pockets. A turn-broaching operation of a crankshaft is shown in Figure 14.14a.

Tool segments are computer designed and manufactured for each machine to suit the required form and tolerance of each crankshaft. The number of inserts and positions of each segment are designed to give low cutting forces. The roughing segments have hardened, fixed insert seats and big chip pockets. Inserts are tangentially mounted and locked in position by a center screw. A turn-broach cutter assembly is shown in Figure 14.14b.

Long tool life results due to the short engagement of the individual cutting edges. High machine utilization is obtained because the finishing cutters need only be changed once per shift and the roughing cutters about once every third shift.

CHAPTER 15

Saws and Sawing

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University
Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers
Former President
International Executive Board
Society of Carbide & Tool Engineers
Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

15.1 Introduction

Once sawing was considered a secondary machining process and saws were used mostly for cutting bar stock in preparation for other machining operations. In recent years, the development of new types of saws and better blade materials have made metal sawing a much more effective, versatile and economical process. In many cases bandsaws are now being used as the primary means of shaping certain types of metal parts.

When the proper sawing machines and blades are used, sawing is one of the most economical means of cutting metal. The saw cut (kerf) is narrow, and relatively few chips are produced in making a cut. When a bandsaw is used for cutting the contours of complex shapes, only a small portion of the metal is removed in the form of chips. Therefore, the power used in removing large amounts of waste metal is at a minimum.

15.2 Sawing

Sawing is a process where a narrow slit is cut into the workpiece by a tool consisting of a series of narrowly spaced teeth called a saw blade. Sawing is normally used to separate work parts into two or more pieces or to cut off an unwanted section of a part. These processes are often called cut-off operations and since many manufacturing projects require cut-off operations at some point of the production sequence, sawing is an important manufacturing process.

Sawing is basically a simple process. As the blade moves past the work, each tooth takes a cut. Depending on the thickness or diameter of the work, the number of teeth cutting at one time varies from 2 to 10 or more. Saws may be of the continuous cutting (band or rotary) or reciprocating type. A typical sawing operation is shown in Figure 15.1.

The cutting speeds and characteristics of the materials must be understood before the proper blades and operating conditions can be selected. Saws are an effective and efficient

category of machine tools found in almost every type of machine shop.

15.3 Saw Blades

All saw blades have certain common characteristics and terminology. Some of these terms are shown in Figure 15.2, and others are explained below.

Rake Angles: Rake angles are 0 degrees or neutral rake on most saw blades. Some have a positive rake angle as shown in Figure 15.2a.

Width: The width of a saw blade is its total width including the teeth.

Set: The set of a saw blade means the offsetting of some teeth so that the

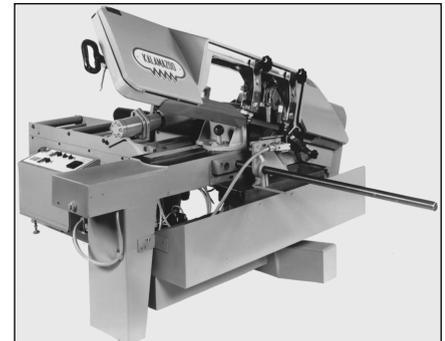


FIGURE 15.1: Typical sawing operation. (Courtesy: Clausing Industries, Inc.)

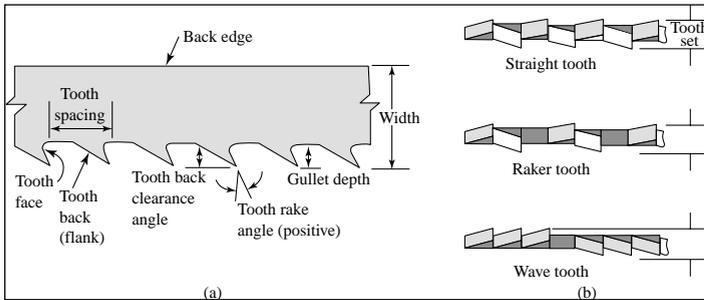


FIGURE 15.2: Saw blade characteristics and terminology.

back of the blade clears the cut. The 'raker' set is most frequently used and is furnished with all hacksaws and band saws unless otherwise specified. (See Fig. 15.2b)

Kerf: The kerf is the width of the cut made by the saw blade or the material cut away. The thickness of the blade is called the gage.

Pitch: The pitch of a saw blade is the distance between the tops of two adjacent teeth. This is specified in teeth per inch.

15.3.1 Saw Blade Material

Saw blades are made from various materials as explained below:

Carbon Steel: General utility for small lot, low speed work. The least expensive blade, these may have a hard 'back' for greater wear.

High Speed Steel: This costs two to three times as much as carbon steel, but it is much longer wearing and is a necessity for the 'difficult-to-machine' metals.

High Speed Edge: This is a carbon steel blade, which has a narrow strip with HSS teeth welded on. This is a tough blade, intermediately priced, and widely used for most materials.

Tungsten Carbide Tipped Blades: Available in a few sizes. Used only on large, very rigid sawing machines for high production sawing of difficult materials.

15.3.2 Saw Blade Selection

The process of choosing the best bandsaw blade for a particular job must start with an evaluation of the material to be cut. Such factors as hardness, machinability, cross-sectional shape and area must be considered.

After the material to be cut has been properly identified, the selector on the machine can be used to help select the

proper blade and cutting speed. Tables and selectors are helpful, but the operator often must make choices that affect the three variables present in every sawing operation: cutting rate, tool life, and accuracy. Generally, increasing any one variable results in a decrease in one or both of the others. For example, an increase in cutting rate always reduces tool life and may affect accuracy,

improper heat treatment is time consuming and potentially dangerous.

15.3.3 Saw Blade Welding

Practically all vertical metal-cutting band-saws have an attachment for electrically butt-welding blades. It is usually set on the column of the machine at the operator's left and consist of a blade cutter, a small grinding wheel, and the butt welding machine. The blade welding attachment can be used for making saw bands from bulk saw-blade stock or for welding bands that have been cut and inserted into a hole in a workpiece that is to be band-sawed internally.

The importance of making good welds in saw blades couldn't be over-emphasized. Breakage caused by poor welding, improper joint finishing, or

improper heat treatment is time consuming and potentially dangerous.

Butt Welder: The resistance-type butt welders found on almost all vertical bandsaws operate by causing electrical current to flow through the ends of the bandsaw blade while pressure is being applied. The high resistance where the blade ends meet causes the metal to become white-hot momentarily, and the blade ends fuse. Provision is made for annealing (softening) the welded joint. As the operator presses the anneal button for a very short time, current flows through the completed joint until the joint heats to a dull red. The joint then anneals as it cools slowly.

15.4 Sawing Equipment

In most sawing operations, the work is held stationary and the saw blade is moved relative to it. As shown in Figure 15.3, there are three basic types of sawing operations, according to the saw blade motion involved:

15.4.1 Hacksawing

Hacksawing involves a linear reciprocating motion of the saw against the workpiece. This method of sawing is often used in cut-off operations. Cutting only takes place on the forward stroke of the saw blade. Due to this intermittent cutting action, hacksawing is less efficient than other sawing methods. Hacksawing can be

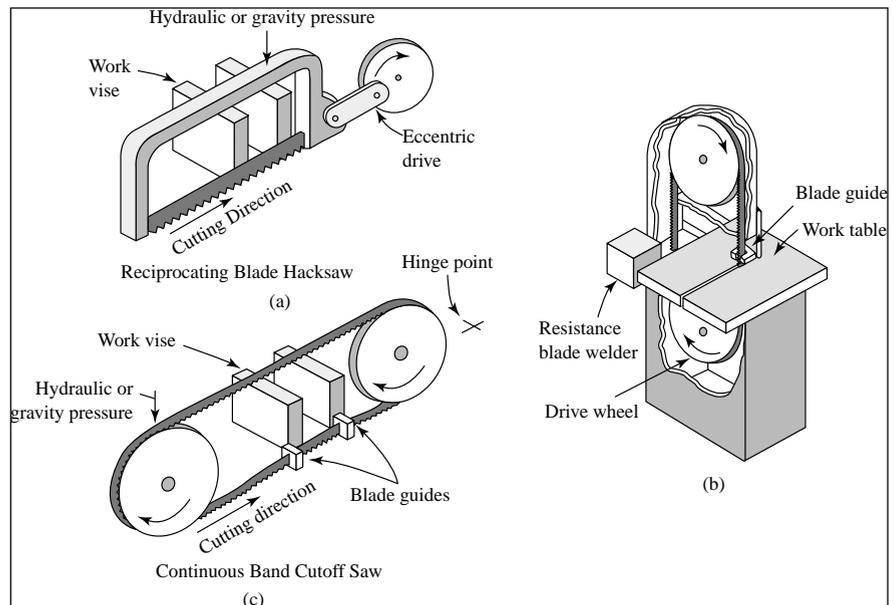


FIGURE 15.3: Three basic types of sawing operations: (a) hacksawing, (b) vertical bandsawing, (c) horizontal bandsawing.

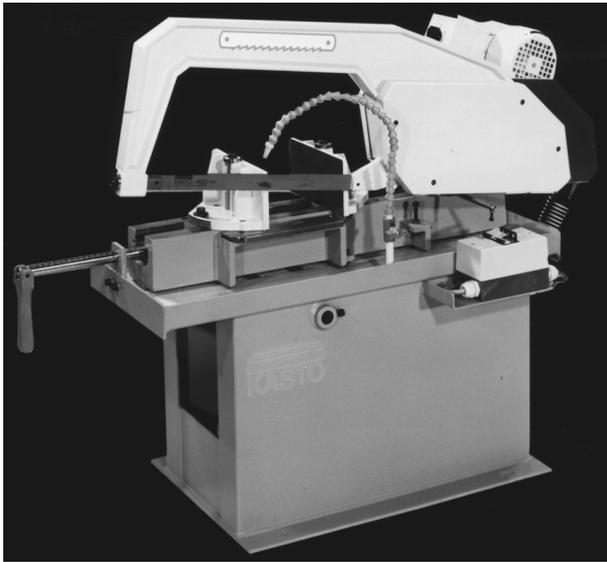


FIGURE 15.4a: Semiautomatic power hacksaw.

done manually or with a power hacksaw. A power hacksaw provides a drive mechanism to operate the saw blade at a desired speed and feed rate. (Fig. 15.3a)

Power Hacksaw: The power hacksaw is the original and least expensive saw for the work. As shown in Figure 15.4a, these saws work the same as a hand hacksaw: They cut on the forward stroke and then lift slightly so that the blade does not drag on the return stroke.

The size of a power hacksaw is the cross section of the largest piece of stock that it can cut. Typical sizes are 6 x 6 inches to 24 x 24 inches. The motors used will vary from 1 to 10 horsepower.

The speed of these saws is in strokes per minute. This may be from 30 strokes per minute for large cuts with heavy saws on difficult materials, up to 165 strokes per minute on carbon

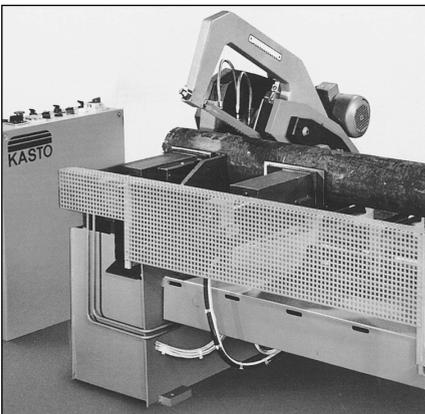


FIGURE 15.4b: Automatic power hacksaw used in high-production sawing. (Courtesy: Kasto-Racine, Inc.)

steels and nonferrous materials. The hacksaw usually has four to six different speeds available.

Feed may be a positive advance per stroke or may be gaged by a friction or pressure drive. The smaller power hacksaws feed about 0.006 inches per stroke and the larger ones 0.012 to 0.030 inches per stroke. Feed pressures will be 450 to 750 pounds on the blades. Work is held in a built-in vise, which

may be hand or power operated.

Automatic power hacksaws (Fig. 15.4b) will feed the stock a preset length, clamp the vise, cut off, and raise the saw for the next cut, all with preset gages and limit switches. These will cut accurate lengths to within 0.010 inches or less. They are, of course, expensive and so they would be used only if a large amount of work is to be done.

15.4.2 Bandsawing

Bandsawing involves a linear continuous motion, using a bandsaw blade made in the form of an endless loop. The band saw provides a pulley-like drive mechanism to continuously move and guide the bandsaw blade past the work. Bandsaws are classified as vertical or horizontal. This designation refers to the direction of saw blade motion during cutting. Vertical bandsaws are used for cut-off and other operations such as contouring and slotting. Horizontal bandsaws are normally used for cut-off operations as alternatives to power hacksaws. (Fig. 15.3b and Fig. 15.3c)

Vertical Bandsaws: All vertical bandsaws, regardless of whether they are light, mediums, or heavy-duty machines, are made up of certain basic components. Although these major parts of the machine may be made by different methods, depending on the manufacturer, their function is essentially the same. A typical vertical bandsaw is shown in Figure 15.5a.

Vertical bandsaws are available in sizes and configurations ranging from

light-duty hand-fed machines to heavy-duty machines with power feed tables. The light-duty machines usually have two wheels and are driven through a variable speed belt drive, V belts and step pulleys, or some other type of speed change mechanism. Blades ranging from 3/16 inch to 5/8 inch in width can be used on light-duty machines.

Table Types: The table of the vertical metal cutting bandsaw is usually made of cast iron and fitted with a tilting mechanism so that simple or



FIGURE 15.5a: Typical vertical bandsaw. (Courtesy: Clausing Industries, Inc.)

compound angle cuts can be made. On fixed-table machines, the table does not move with the work, but can be tilted 45 degrees to the right and 10 degrees to the left on most machines. The work can be fed and guided manually, or a weight operated feed mechanism can be used to supply the feed pressure.

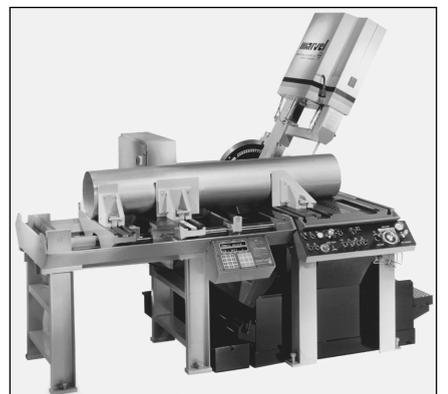


FIGURE 15.5b: High-production automated vertical bandsaw table machine. (Courtesy: Armstrong-Blum Mfg. Co.)

Vertical bandsaws with power tables are generally heavy-duty machines. The feed pressure is provided by the mechanism that moves the table; the operator can vary the feed rate.

There is usually enough power available to make effective use of high-speed steel or tungsten carbide saw blades rather than the high carbon steel blades used on light-duty machines. Coolant systems are also widely used on power table machines, thus allowing higher cutting speeds and higher feed rates along with longer blade life. Many types of fixtures can be used on power table machines, particularly when they are used for repetitive operations. A high production automated table machine is shown in Figure 15.5b.

Accessories: Most bandsaws that do not have a coolant system have an air pump that directs a stream of air at the point where the blade is cutting the workpiece. This removes the chips, letting the operator see the layout lines clearly, and provides some cooling.

If the machine has a fluid coolant system, the tank and pump are usually located in the base. A separate switch controls the pump. Coolant systems are usually found on medium and heavy duty vertical bandsaws.

Blade welding attachments, which are a specialized form of electric butt-welding machines, are a standard accessory on almost all bandsaws. The blade welder usually consists of cast copper or bronze blade clamps, a grinder, a saw thickness gage, and the necessary switches and operating levers.

Weight operated feed devices can be used on bandsaws not fitted with power feed attachments. This reduces operator fatigue and generally results

in more uniform feed rates and longer blade life.

Other attachments such as fixtures for cutting arcs and circles, ripping fences, and miters, are used extensively on bandsaws. Special fixtures for holding specific types of workpieces are often designed for use in mass production applications.

Horizontal Bandsaws: Because horizontal bandsaws are used primarily for cutting bar stock and structural shapes, they are also known as cut-off saws. The band-type cut-off saw is widely used because it is easy to set up and takes a narrow saw cut, thus requiring less power to operate and wasting less material. The cutting action is continuous and rapid. The blade is supported close to either side of the material being cut, so the cut is accurate if the machine is properly adjusted and the blade is in good condition. A typical horizontal bandsaw is shown in Figure 15.6a.

Horizontal bandsaws range in capacity from small, fractional horsepower machines, (Fig. 15.6a), to large heavy-duty industrial saws, as shown in Figure 15.6b.

The saw guides are an important factor in accurate cut-off operations. The saw blade has to twist as it leaves the idler pulley and the guides make the blade travel perpendicular to the material being cut. Tungsten carbide inserts help minimize wear. Figure 15.7 shows a more advanced horizon-

tal band saw with an automated table stock feeding system.

Controls and Accessories: On light duty saws, the controls are simple, consisting mainly of an off-on switch, a means for changing blade speed, and possibly a control for feed pressure. On the larger machines a control panel is usually mounted on the saw head. It consists of the necessary switches, valves, and instruments that indicate blade speed in feet per minute, feed rate in inches per minute, and other factors, such as blade tension. Some machines used for production work are capable of fully automatic operation and can be preset to cut a given number of pieces of work. A counter is usually part of the instrumentation on semiautomatic and automatic machines.

There are coolant systems on almost all medium and heavy duty horizontal bandsaws. The coolant extends blade life and allows higher cutting speeds and metal removal rates. The operator controls the rate of coolant flow. Solid lubricants such as wax or grease can also be used. Wax in stick form is usually applied manually to the blade on light-duty machines.

15.4.3 Comparison of Hacksaws and Band Saws

The decision as to which type of cut-off saw to buy is often influenced by custom or habit. However, there are definite factors that can be considered.

Cost: A hacksaw is much less expensive, often about half the cost of a band saw of equal size and power.

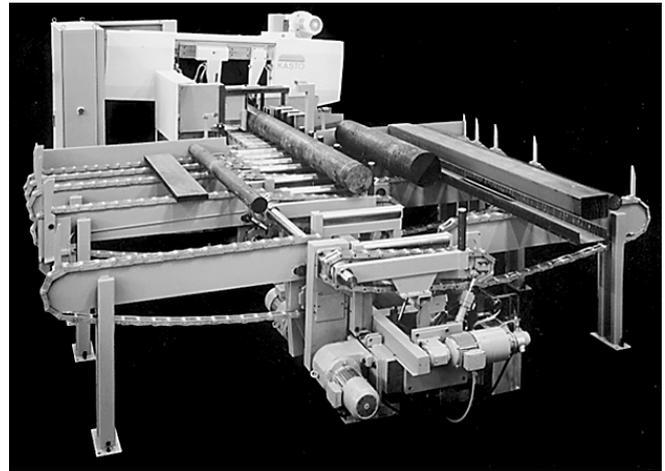


FIGURE 15.7: Horizontal Band Saw with automated table stock feeding system (Courtesy: Kasto-Racine, Inc.)



FIGURE 15.6a: Typical horizontal bandsaw. (Courtesy: Clausing Industries, Inc.)

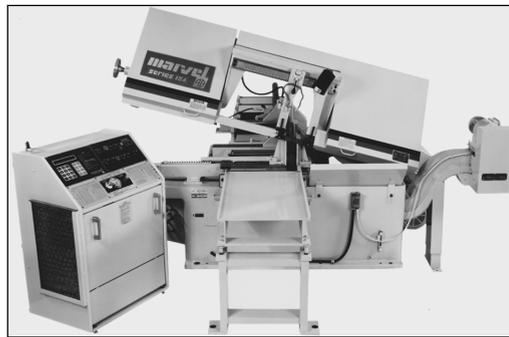


FIGURE 15.6b: Large, heavy-duty industrial horizontal bandsaw. (Courtesy: Armstrong-Blum Mfg. Co.)

Saw blades: The hacksaw blades may cost one-half to one-quarter the cost of a band-saw blade. However, the hacksaw will become dull in one-half to one-quarter the number of cuts that the band saw will make.

The hacksaw blade is almost unbreakable and is somewhat less likely to have its teeth stripped off by hard spots in the material being cut.

Kerf: The band-saw blade is thinner than the hacksaw blade, especially for the larger sizes. Thus less metal is wasted in the cut. However, this 'saving' is often lost because of the 2 to 6 inch long 'stub end', which is thrown into the scrap, bin when the bar of stock is used up.

Speed: The band saw will cut off stock up to twice as fast as the hacksaw. However, it does take more care and more time to change blades, adjust saw guides, and regulate feeds. Thus, the plain hacksaw can be used by less experienced operators.

15.5 Band Sawing Operations

The types of work described here accounts for most of the band sawing operations used in metalworking.

15.5.1 Cut-off Sawing

Although cut-off sawing can be done on any type of vertical or horizontal bandsaw, the majority of cut-off

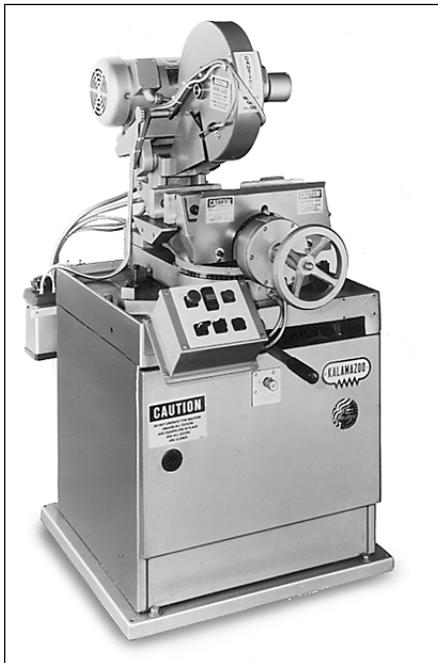


FIGURE 15.8: Semi-automatic Circular Saw (Courtesy: Clausing Industries, Inc.)

sawing is done on powerful horizontal machines. A variety of work-holding devices and fixtures can be used to hold tubing, angle iron, and other shapes.

Blade selection is important in terms of economy and the finish on the material being cut. The precision tooth type blade is used extensively with the recommended pitch ranging from 10 teeth per inch for sections up to 3/8 in. thickness to 4 teeth per inch for material over 3 in. thick. Manufacturers' manuals should be consulted when heavy cuts are being attempted. The claw tooth type of blade is used when cutting some tough steels because the tooth penetrates the surface of the work more easily.

Stock feeders are often used on cut-off machines, along with an indexing mechanism that allows the operator to automatically repeat cuts of pre-selected lengths. Almost all cut-off operations are done with a liquid coolant delivered to the saw cut by a pump.

15.5.2 Contour Sawing

Contour sawing, both internal and external, is one of the most versatile operations that can be done with a bandsaw. It may range from simple shapes cut on a fractional horsepower machine to complex internal cuts made with tilting table machines. Blade selection is important when cutting complex contours, especially when small radii or corners are involved. Select the widest blade that will allow turns of the proper radius.

For internal work, a hole must be drilled so that the blade can be passed through it and re-welded. For plain contouring, the hole is drilled perpendicular to the face of the workpiece. When the internal shape has corners, holes must be drilled at the corners so that the blade can be turned and the cut started in another direction.

15.5.3. Friction Sawing

Friction sawing is a unique process.

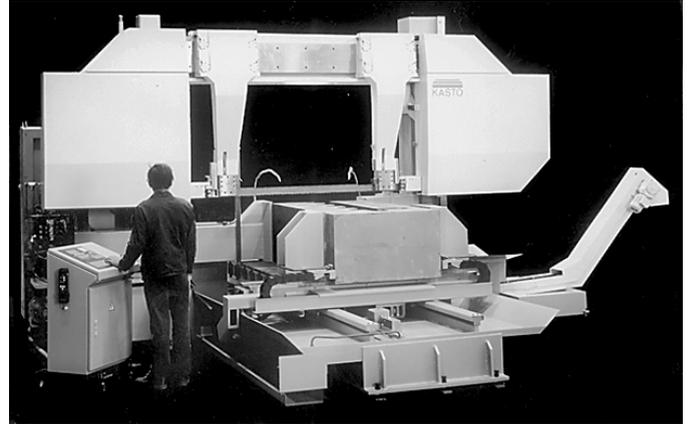


FIGURE 15.9: Automated Band Saw with computer-controlled functions (Courtesy: Kasto-Racine, Inc.)

A bandsaw blade with dull teeth traveling at very high speed, 6000 to 15000 SFPM (surface feet per minute), is used to cut both hard and soft ferrous metals. Friction sawing works particularly well on metals that have poor heat conductivity because the heat-affected zone remains very small. It is the fastest method of cutting ferrous metals less than 1 in. thick.



FIGURE 15.10: Typical cold saw. (Courtesy: Clausing Industries, Inc.)

As the blade contacts the work, the metal at the point of contact immediately becomes white hot and is carried out by the teeth. The blade itself remains relatively cool because during its operating cycle it is in contact with hot metal for only a short time.



FIGURE 15.11: Large, heavy-duty industrial cold saw. (Courtesy: Claus-Ing Industries, Inc.)

15.6 Circular Sawing

Circular sawing uses a rotating saw blade to provide a continuous motion of the tool past the work. Circular sawing is often used to cut long bars and tubes to specific lengths. The cutting action is similar to slot milling, except that the saw blade is thinner and contains more cutting teeth. Circular sawing machines have power spindles to rotate the saw blade and a feeding mechanism to drive the rotating blade into the work. Figure 15.8 shows a semi-automatic circular saw.

Band as well as circular saws have advanced to be highly automated and many of their functions are computer controlled as shown in Figure 15.9.

15.7 Cold Sawing:

Most cold saws, regardless of size, consist of a base; drive mechanism, blade arbor, vise, feed mechanism, and necessary guards and switches. On some small saws the blade is fed into the work by hand (Fig. 15.10).

On larger machines the feed mechanism is pneumatically or hydraulically operated. The operator controls the rate of feed. (Fig. 15.11).

The base of the machine or the vise can be swiveled to make angular cuts. In some cases two machines can be set up on a single work stand for production operations.

15.7.1 Cold Saw Blades

Blades smaller than 18 inches in diameter are cut directly in the rim of the saw disk. For cutting soft materials, the teeth are spaced farther apart, as in the case of bandsaw and power hacksaw blades, so that the gullet (the space between the teeth) will be large enough to accommodate large chips. When cutting thin tubing or other thin materials use saw blades with closely spaced teeth to avoid chattering and tooth breakage.

Cold saw blades with teeth cut directly on the periphery of the disk may be made of high carbon or high-speed steel.

Larger blades usually have segmented teeth. The body of the blade is made of rough, resilient alloy steel, and the inserted teeth are made of high-speed steel or tungsten carbide. The individual teeth or segments of three or four teeth are wedged or riveted to the blade and can be easily replaced if a tooth is damaged or broken. Larger cold saw blades can cut a kerf as wide as 1/4 inch and remove metal rapidly.

15.8 Abrasive Cut-Off Machines

Abrasive cut-off machines are used in many shops to cut metallic and nonmetallic materials. Because an abrasive - usually aluminum oxide - is used as the cutting tool, hardened steel can be cut without being annealed. The cutting action here is faster than on other types of cut-off machines.

Abrasive cut-off machines may be of



FIGURE 15.12: Abrasive cut-off operation (Courtesy: Norton Company)

the wet or dry type. The flow of coolant, usually water and an antirust chemical of some type are controlled by the operator. The coolant tank is separate or built into the base of the machine.

Some larger cut-off machines have power feed mechanisms and oscillators. The oscillator moves the abrasive disk back and forth in the cut as feed pressure is applied. This reduces the amount of blade in contact with the work at any given time and reduces the power input required to cut solid bar stock of a given cross-sectional area. An abrasive cut-off operation is shown in Figure 15.12.

The abrasive disks usually have a resinoid bonding agent, although rubber can be used on smaller wheels. Glass fiber is sometimes impregnated in the disk to increase its strength. Abrasive disks work efficiently at surface speeds of 12,000 to 15,000 surface feet per minute.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University
Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers
Former President
International Executive Board
Society of Carbide & Tool Engineers
Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

CHAPTER 16

Grinding Wheels

and Operations

16.1 Introduction

Grinding, or abrasive machining, is the process of removing metal in the form of minute chips by the action of irregularly shaped abrasive particles. These particles may be in bonded wheels, coated belts, or simply loose. The abrasive grains usually cut with a zero to negative rake angle and produce a large number of short, small, curly or wavy chips. The way an abrasive grain cuts material is shown in Fig. 16.1

16.2 Grinding Wheels

Grinding wheels are composed of thousands of small abrasive grains held together by a bonding material. Some typical grinding products are shown in Figure 16.2. Each abrasive grain is a cutting edge. As the grain passes over the workpiece it cuts a small chip, leaving a smooth, accurate surface. As each abrasive grain becomes dull, it breaks away from the bonding material because of machining forces and exposes new, sharp grains.

oxide, silicon carbide, cubic boron nitride, and diamond.

Aluminum Oxide: Refining bauxite ore in an electric furnace makes Aluminum oxide. The bauxite ore is first heated to eliminate any moisture, then mixed with coke and iron to form a furnace charge. The mixture is then fused and cooled. The fused mixture resembles a rocklike mass. It is washed, crushed, and screened to separate the various grain sizes.

Aluminum oxide wheels are manu-

16.2.1 Types of Abrasives

Two types of abrasives are used in grinding wheels: natural and manufactured. Except for diamonds, manufactured abrasives have almost totally replaced natural abrasive materials. Even natural diamonds have been replaced in some cases by synthetic diamonds.

The manufactured abrasives most commonly used in grinding wheels are aluminum

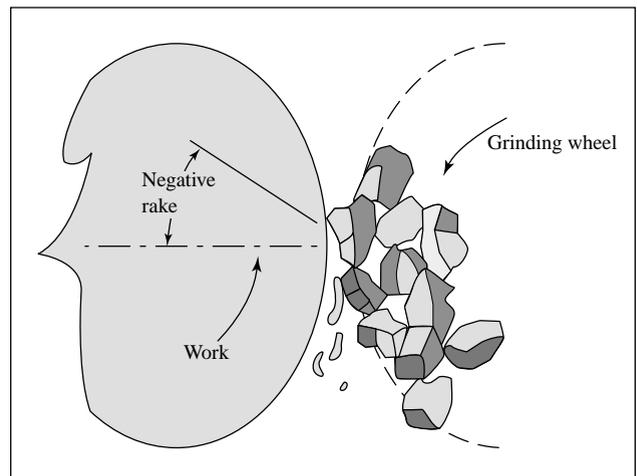


FIGURE 16.1: Abrasive grains cutting material during a grinding operation.



FIGURE 16.2: Typical grinding products. (Courtesy: Norton Company)

factured with abrasives of different degrees of purity to give them certain characteristics for different grinding operations and applications. The color and toughness of the wheel are influenced by the degree of purity.

General purpose aluminum oxide wheels, usually gray and 95 percent pure are the most popular abrasives used. They are used for grinding most steels and other ferrous alloys. White aluminum oxide wheels are nearly pure and are very friable (able to break away from the bonding material easily). They are used for grinding high strength, heat sensitive steels.

Silicon Carbide: Silicon carbide grinding wheels are made by mixing pure white quartz, petroleum coke, and small amounts of sawdust and salt, and

firing the mixture in an electric furnace. This process is called synthesizing the coke and sand. As in the making of aluminum oxide abrasive, the resulting crystalline mass is crushed and graded by particle size.

Silicon carbide wheels are harder and more brittle than aluminum oxide wheels. There are two principal types of silicon carbide wheels: black and green. Black wheels are used for grinding cast irons, non-ferrous metals like copper, brass, aluminum, and magnesium, and nonmetallics such as ceramics and gem stones.

Green silicon carbide wheels are more friable than the black wheels and used for tool and cutter grinding of cemented carbide.

Cubic Boron Nitride: Cubic boron nitride (CBN) is an extremely hard, sharp, and cool cutting abrasive. It is one of the newest manufactured abrasives and 2 1/2 times harder than aluminum oxide. It can withstand temperatures up to 2500 degrees Fahrenheit. CBN is produced by high temperature, high pressure processes similar to those used to produce manufactured diamond and is nearly as hard as diamond.

CBN is used for grinding super hard high-speed steels, tool and die steels, hardened cast irons, and stainless steels. Two types of cubic boron nitride wheels

are used in industry today. One type is metal coated to promote good bond adhesion and used in general purpose grinding. The second type is an uncoated abrasive for use in electroplated metal and vitrified bond systems.

Diamond: Two types of diamond are used in the production of grinding wheels: natural and manufactured. Natural diamond is a crystalline form of carbon and very expensive. In the form of bonded wheels, natural diamonds are used for grinding very hard materials such as cemented carbides, marble, granite, and stone.

Recent developments in the production of manufactured diamonds have brought their cost down and led to expanded use in grinding applications. Manufactured diamonds are now used for grinding tough and very hard steels, cemented carbide, and aluminum oxide cutting tools.

The synthetic diamond crystals shown in Figure 16.3a can be manufactured into polycrystalline tool blanks shown in Figure 16.3b and discussed in chapter 1, section 1.5 or pressed into diamond wheels shown in Figure 16.7.

16.2.2 Types of Bonds

Abrasive grains are held together in a grinding wheel by a bonding material. The bonding material does not cut during a grinding operation. Its main function is to hold the grains together with varying degrees of strength. Standard grinding wheel bonds are vitrified, resinoid, sillicate, shellac, rubber, and metal.

Vitrified Bond: Vitrified bonds are used on more than 75 percent of all

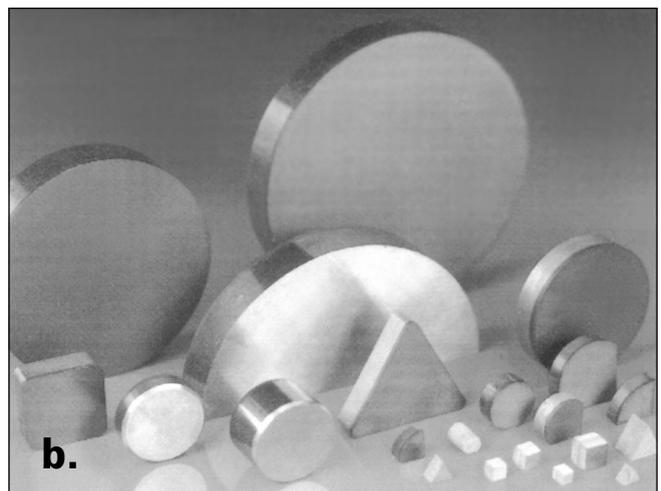


FIGURE 16.3: a) Synthetic diamond crystals; b) Polycrystalline tool blanks (Courtesy: Norton Company)

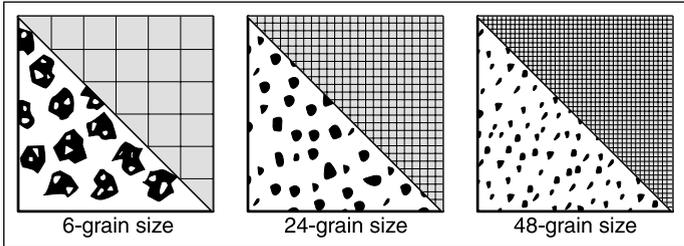


FIGURE 16.4: Comparison of three different grain sizes.

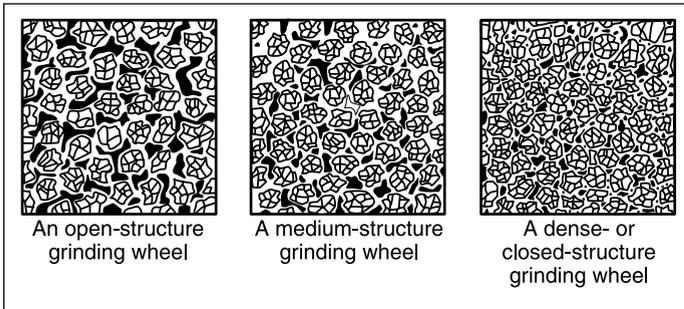


FIGURE 16.5: Comparison of three different grain structures.

grinding wheels. Vitrified bond material is comprised of finely ground clay and fluxes with which the abrasive is thoroughly mixed. The mixture of bonding agent and abrasive in the form of a wheel is then heated to 2400 degrees Fahrenheit to fuse the materials.

Vitrified wheels are strong and rigid. They retain high strength at elevated temperatures and are practically unaffected by water, oils, or acids. One disadvantage of vitrified bond wheels

is that they exhibit poor shock resistance. Therefore, their application is limited where impact and large temperature differentials occur. Resinoid Bond: Resinoid bonded grinding wheels are second in popularity to vitrified wheels. Phenolic resin in powdered or liquid form is mixed with the abrasive grains in a form and cured at about 360 degrees Fahrenheit. Resinoid wheels are used for grinding speeds up to 16,500 SFPM. Their main use is in rough grinding and cut-off operations. Care must be taken with resinoid bonded wheels since they will soften if they are exposed to water for extended periods of time.

Silicate Bond: This bonding material is used when heat generated by grinding must be kept to a minimum. Silicate bonding material releases the

abrasive grains more readily than other types of bonding agents. Speed is limited to below 4500 SFPM.

Shellac Bond: Shellac is an organic bond used for grinding wheels that produce very smooth finishes on parts such as rolls, cutlery, camshafts, and crankpins. They are not generally used on heavy duty grinding operations.

Rubber Bond: Rubber bonded wheels are extremely tough and strong. Their principal uses are as thin cut-off wheels and driving wheels in centerless grinding machines. They are also used when extremely fine finishes are required on bearing surfaces.

Metal Bond: Metal bonds are used primarily as bonding agents for diamond abrasives. They are also used in electrolytic grinding, where the bond must be electrically conductive.

16.2.3 Abrasive Grain Size

The size of an abrasive grain is important because it influences stock removal rate, chip clearance in the wheel, and surface finish obtained.

Abrasive grain size is determined by the size of the screen opening through which the abrasive grits pass. The number of the nominal size indicates the number of the openings per inch in the screen. For example, a 60 grit-sized grain will pass through a screen with 55 openings per inch, but it will not pass through a screen size of 65. A low grain size number indicates large grit, and a high number indicates a small grain.

Grain sizes vary from 6 (very coarse) to 1000 (very fine). Grain sizes are broadly defined as coarse (6 to 24), medium (30 to 60), fine (70 to 180), and very fine (220 to 1000). Figure 16.4 shows a comparison of three different grain sizes and the screens used for sizing. Very fine grits are used for polishing and lapping operations, fine grains for fine finish

STANDARD MARKING SYSTEM CHART												
ANSI STANDARD B74.13 – 1970												
Sequence: Prefix	1 Abrasive Type	2 Grain Size				3 Grade			4 Structure		5 Bond Type	6 Manufacturer's Record
51	A	36				L			5		V	23
Prefix	Abrasive Type	Abrasive (grain) Size				Grade			Structure		Bond Type	Manufacturer's Record
		Coarse	Medium	Fine	Very fine	Soft	Medium	Hard	Dense to Open			
Manufacturer's symbol indicating exact kind of abrasive (Use optional)	A • Aluminum Oxide C • Silicon Carbide	8	30	70	220	A E	I M	Q V	1	9	B • Resinoid	Manufacturer's private marking to identify when (Use optional)
		10	36	80	240	B F	J N	R W	2	10	BF • Resinoid Reinforced	
		12	46	90	280	C G	K O	S X	3	11	E • Shellac	
		14	54	100	320	D H	L P	T Y	4	12	O • Oxychloride	
		16	60	120	400			U Z	5	13	R • Rubber	
		20		150	500				6	14	RF • Rubber Reinforced	
		24		180	600				7	15	S • Silicate	
								8	16 etc.	V • Vitrified		
									(Use optional)			

FIGURE 16.6: ANSI standard marking system for abrasive grinding wheels. (United abrasives manufacturers' association.)



FIGURE 16.7: Typical Cubic Boron Nitride (CBN) and diamond grinding wheels. (Courtesy Norton Company)

and small diameter grinding operations. Medium grain sizes are used in high stock removal operations where some control of surface finish is required. Coarse grain sizes are used for billet conditioning and snagging operations in steel mills and foundries, where stock removal rates are important, and there is little concern about surface finish.

16.2.4 Grinding Wheel Grade

The grade of a grinding wheel is a measure of the strength of the bonding material holding the individual grains in the wheel. It is used to indicate the relative hardness of a grinding wheel. Grade or hardness refers to the amount of bonding material used in the wheel, not to the hardness of the abrasive. A soft wheel has less bonding material than a hard wheel.

The range used to indicate grade is A to Z, with A representing maximum softness and Z maximum hardness. The selection of the proper grade of wheel is very important. Wheels that are too soft tend to release grains too rapidly and wheel wear is great. Wheels that are too hard do not release the abrasive grains fast enough and the dull grains remain bonded to the wheel causing a condition known as ‘glazing’.

16.2.5 Grinding Wheel Structure

The structure of a grinding wheel refers to the relative spacing of the abrasive grains; it is the wheel’s density. There are fewer abrasive grains in an open-structure wheel than in a close-structure wheel. Figure 16.5 shows a

comparison of different structures used in a grinding wheel.

A number from 1 to 15 designates the structure of a wheel. The higher the number, the more open the structure; the lower the number, the more dense the structure.

16.3.2 Grinding Wheel Shapes and Faces

Most grinding wheel manufacturers have adopted eight standard wheel shapes and 12 standard wheel faces for general use. Figure 16.9 shows the most common standard wheel shapes used on all types of grinders. Figure 16.10 illustrates the standard wheel faces used on most grinding wheel shapes.

16.4 Electroplated Grinding Wheels

Of the several methods now used for fixing super abrasive particles of diamond or CBN to the working surface of an abrasive tool, electroplating is the fastest growing. More and more production operations involve combinations of hard-to-grind materials and complex wheel shapes that virtually dictate the use of electroplated super abrasive tools.

Characteristically, such tools consist of a precision tool form or mandrel with super abrasive particles deposited on the working surface and locked in place by electrodeposition of a bonding matrix, most frequently nickel. The particles so locked onto the tool surface may vary in size and dispersion to suit the purpose of the tool, but they should lie in a single layer.

Figure 16.11a shows a close-up view of a electroplated wheel; Figure 16.11b shows various size and shapes of electroplated wheels.

16.5 Wheel Balancing, Dressing and Truing

All grinding wheels are breakable, and some are extremely fragile. Great care should be taken in handling grinding wheels. New wheels should

16.3 Grinding Wheel Specifications

Grinding wheel manufacturers have agreed to a standardization system to describe wheel composition as well as wheel shapes and faces.

16.3.1 Grinding Wheel Markings

Abrasive grinding wheels have a different marking system than CBN and diamond wheels as discussed and shown below.

Abrasive Grinding Wheels: This marking system is used to describe the wheel composition as to type of abrasive, grain size, grade, structure, and bond type. Figure 16.6 illustrates this standard marking system.

CBN and Diamond Wheels: The same standardization is applicable to CBN and diamond wheels. Some typical CBN and diamond wheels are shown in Figure 16.7. Wheel markings are a combination of letters and numbers as shown in Figure 16.8.

STANDARD MARKING SYSTEM CHART							
FOR DIAMOND AND CBN WHEELS							
M	D	100 - P	100 - B			1/8	
Prefix	Abrasive Type	Grit Size	Grade	Diamond Concentration	Bond	Bond Modification	Diamond Depth (in.)
Manufacturer's symbol to indicate type of diamond	B Cubic boron nitride	20 1000	A (soft) to Z (hard)	25 (low) 50 75 100 (high)	B Resinoid M Metal V Vitriified	A letter or numeral or combination used here will indicate a variation from standard bond	1/16 1/8 1/4
	D Diamond						Absence of depth symbol indicates solid diamond

FIGURE 16.8: Standard marking system for Cubic Boron Nitride (CBN) and diamond grinding wheels.

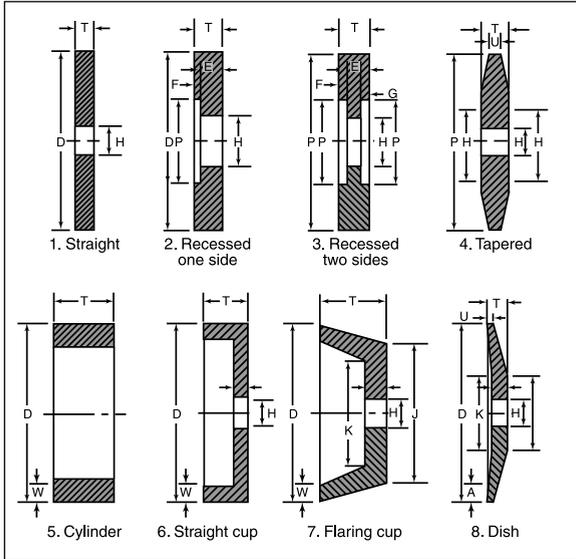


FIGURE 16.9: Eight standard grinding wheel shapes.

be closely inspected immediately after receipt to make sure they were not damaged during transit. Grinding wheels should also be inspected prior to being mounted on a machine.

To test for damage, suspend the wheel with a finger and gently tap the side with a screwdriver handle for small wheels, and a wooden mallet for larger wheels. An undamaged wheel will produce a clear ringing sound; a cracked wheel will not ring at all.

16.5.1 Wheel Balancing

It is important to balance wheels over 10 inches before they are mounted on a machine. The larger the grinding wheel, the more critical balancing becomes. Grinding wheel balance also becomes more critical as speed is increased. Out-of-balance wheels cause excessive vibration, produce faster wheel wear, and chatter, poor finishes, damage to spindle bearings, and can be dangerous.

The proper procedure for balancing wheels is to first statically balance the wheel. Next, mount the wheel on the grinding machine and dress. Then remove the wheel and rebalance it. Remount the wheel and dress slightly a second time.

Shifting weights on the wheel mount does balancing of wheels. The wheel is installed on a balancing arbor and placed on a balancing fixture. The weights

16.5.2 Dressing and Truing

Dressing is a process used to clean and restore a dulled or loaded grinding wheel-cutting surface to its original sharpness. In dressing, swarf is removed, as well as dulled abrasive grains and excess bonding material. In addition, dressing is used to customize a wheel face, so that it will give desired grinding results.

Truing is the process of removing material from the face of the wheel so that the resultant cutting surface runs absolutely true. This is very important in precision grinding, because an out of truth wheel will produce objectionable chatter marks on the workpiece. A new wheel should always be trued before being put to work. Also it is a good idea to true the wheel if it is

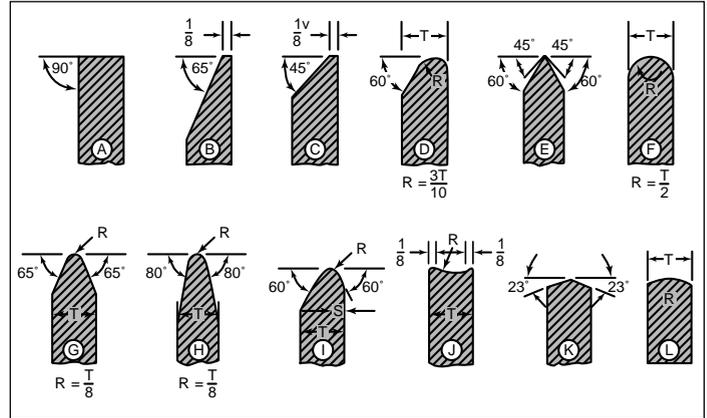


FIGURE 16.10: Twelve standard grinding wheel face contours.

are then shifted in a position to remove all heavy points on the wheel assembly.

being remounted on a machine.

Dressing and truing conventional grinding wheels are two separate and distinct operations, although they may sometimes be done with the same tool. The tools used for conventional grinding wheel dressing include the following:

Mechanical dressers - commonly called star dressers, are held against the wheel while it is running. The picking action of the points of the star shaped wheels in the tool remove dull grains, bond and other bits of swarf. Star dressers are used for relatively coarse-grained conventional wheels, generally in off-hand grinding jobs, where grinding accuracy is not the main consideration.

Dressing sticks - are used for off-hand dressing of smaller conventional wheels, especially cup and saucer shapes. Some of these sticks are made of an extremely hard abrasive called boron carbide. In use, a boron carbide stick is held against the wheel face to

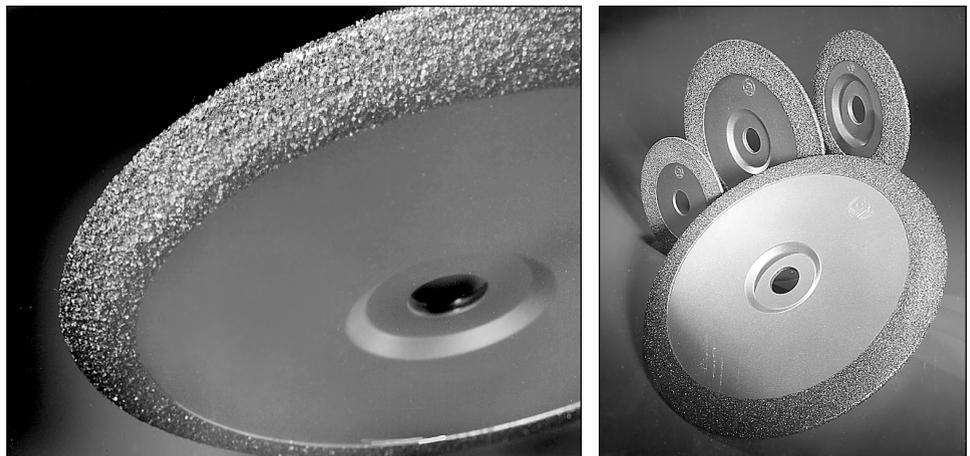


FIGURE 16.11: a) Close-up view of electroplated wheel b) Various sizes and shapes of electroplated wheels (Courtesy: Universal Superabrasives)

shear the dull abrasive grains and remove excess bond. Other dressing sticks contain coarse Crystolon or Alundum grains in a hard vitrified bond. Various dressing sticks are shown in Figure 16.12.

Diamond dressing tools - utilize the unsurpassed hardness of a diamond point to clean and restore the wheel grinding face. Although single point diamond tools were once the only products available for this kind of dressing, the increasing scarcity of diamonds has led to the development of multi-point diamond tools.

Multi-point diamond dressing tools use a number of small diamonds held in a matrix. In use, the tool is held securely in the tool holder and held flat against the face of the running wheel. As it dresses, the tool is traversed across the wheel face until the job is done. As diamonds on the surface of the tool wear away, fresh new diamond points are exposed to offer extended life and use. This type of tool produces a very consistent wheel face from dress to dress.

Multi-point diamond dressing tools are available in a wide range of shank diameters and face shapes, to meet the requirements of a broad variety of

grinding machines. Typical diamond tools used to dress grinding wheels are shown in Figure 16.13.

16.6 Grinding Wheel Selection

Before attempting to select a grinding wheel for a particular operation, the operator should consider the following six factors for maximum productivity and safe results:

Material to Be Ground: If the material to be ground is carbon steel or alloy steel, aluminum oxide wheels are usually selected. Extremely hard steels and exotic alloys should be ground with cubic boron nitride (CBN) or diamond. Nonferrous metals, most cast irons, nonmetallics, and cemented carbides require a silicon carbide wheel. A general rule on grain size is to use a fine grain wheel for hard materials, and a coarse grain wheel for soft and ductile materials. Close grain spacing and soft wheels should be used on harder materials, while open structure and harder wheels are preferable on soft materials.

Nature of the Grinding Operation: Finish required, accuracy, and amount of metal to be removed must be considered when selecting a wheel. Fine and accurate finishes are best obtained with small grain size and grinding wheels with resinoid, rubber, or shellac bonds. Heavy metal removal is obtained with coarse wheels with vitrified bonds.

Area of Contact: The area of contact between the wheel and workpiece is

also important. Close grain spacing, hard wheels, and small grain sizes are used when the area of contact is small. On the other hand, open structures, softer wheels, and larger grain sizes are recommended when the area of contact is large.

Condition of the Machine: Vibration influences the finish obtained on the part as well as wheel performance. Vibration is generally due to loose or worn spindle bearings, worn parts, out-of-balance



FIGURE 16.13: Single- and multipoint diamond dressing tools. (Courtesy Norton Company)

wheels, or insecure foundations.

Grinding Wheel Speed: Wheel speed affects the bond and grade selected for a given wheel. Wheel speeds are measured in surface feet per minute (SFPM). Vitrified bonds are commonly used to 6,500 SFPM or in selected operations up to 12,000 SFPM. Resinoid-bonded wheels may be used for speeds up to 16,500 SFPM.

Grinding Pressure: Grinding pressure is the rate of in-feed used during a grinding operation; it affects the grade of wheel. A general rule to follow is that as grinding pressures increase, harder wheels must be used.

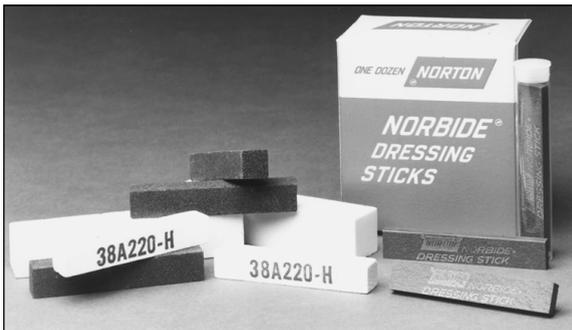


FIGURE 16.12: Various dressing sticks are shown. (Courtesy Norton Company)

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University
Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers
Former President
International Executive Board
Society of Carbide & Tool Engineers
Lawrence Tech. - www.ltu.edu
Prentice Hall - www.prenhall.com

CHAPTER 17

Grinding Methods and Machines

17.1 Introduction

Grinding, or abrasive machining, is one of the most rapidly growing metal removal processes in manufacturing. Many machining operations previously done on conventional milling machines, lathes and shapers, are now being performed on various types of grinding machines. Computer Numerical Control (CNC) resulting in greater productivity, improved accuracy, reliability, and rigid construction characterize today's industrial grinding machines. A typical internal grinding operation is shown in Figure 17.1.

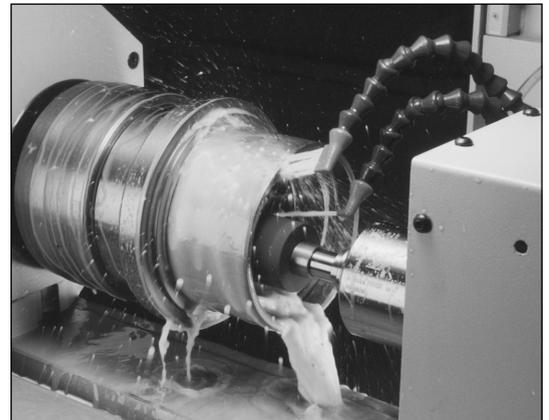


FIG. 17.1: Typical internal grinding operation. (Courtesy Kellenberger, A Hardinge Co.)

17.2 Grinding Processes

Grinding machines have advanced in design, construction, rigidity, and application far more in the last decade than any other standard machine tool in the manufacturing industry. Grinding machines fall into five categories:

- * Surface grinders
- * Cylindrical grinders
- * Centerless grinders
- * Internal grinders
- * Special types

of grinders.

17.2.1 Surface Grinding

Surface grinders are used to produce flat, angular, and irregular surfaces. A typical hand operated surface grinder is shown in Figure 17.2a. In the surface

grinding process, the grinding wheel revolves on a spindle and the work-piece, mounted on either a reciprocating or rotary table, is brought into

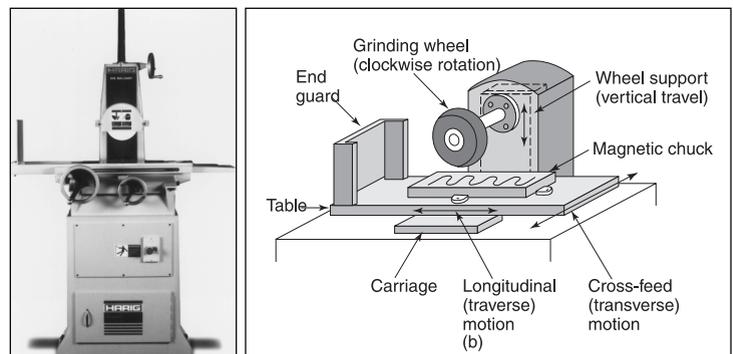


FIG. 17.2: (a) Typical standard surface grinder. (Courtesy Bridgeport Machine, Inc.) (b) Schematic illustration of the basic components and motions of a surface grinder.

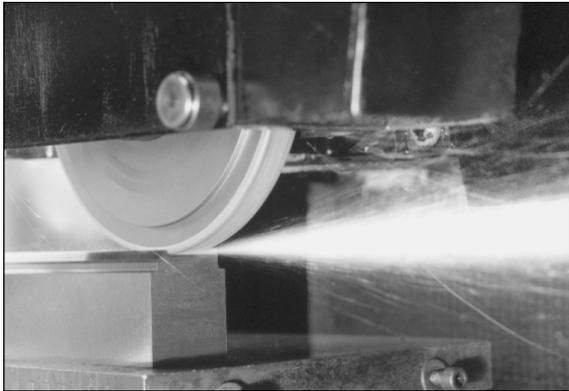


FIG. 17.3: Typical surface grinding operation. (Courtesy Norton Company)

contact with the grinding wheel (Fig. 17.2b).

A typical surface grinding operation is shown in Figure 17.3. Four types of surface grinders are commonly used in industry (Fig. 17.4).

Horizontal Spindle/Reciprocating Table: This surface grinder is the most commonly used type in industry. A manual surface grinder was shown in Figure 17.2a. A more sophisticated and automated surface grinder is shown in Figure 17.5. It is available in various sizes to accommodate large or small workpieces. With this type of surface grinder, the work moves back and forth

under the grinding wheel. The grinding wheel is mounted on a horizontal spindle and cuts on its periphery as it contacts the workpiece. The worktable is mounted on a saddle that provides cross feed movement of the workpiece. The wheelhead assembly moves vertically on a column to control the depth of cut required.

Horizontal Spindle/Rotary Table: This surface grinder also has a

horizontally mounted grinding wheel that cuts on its periphery. The workpiece rotates 360 degrees on a rotary table underneath the wheelhead. The wheelhead moves across the workpiece to provide the necessary cross feed movements. The metal removal rate is controlled by the amount of down-feed of the wheelhead assembly.

Vertical Spindle/ Reciprocating Table: This type of grinding machine is particularly suited for grinding long and narrow castings like the bedways of an engine lathe. It removes metal with the face of the

grinder wheel while the work reciprocates under the wheel. The wheelhead assembly, as on most other types of surface grinders, moves vertically to control the depth of cut. The table moving laterally accomplishes cross feed. The table is mounted on a saddle unit.

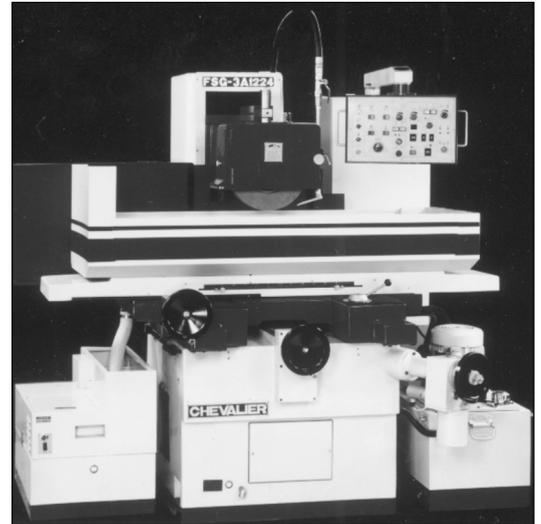


FIG. 17.5: Automated surface grinder with coolant system. (Courtesy Chevalier Machinery, Inc.)

Vertical Spindle/ Rotary Table:

This type of grinding machine (Fig. 17.6) is capable of heavy cuts and high metal removal rates. Vertical spindle machines use cup, cylinder, or segmented wheels. Many are equipped with multiple spindles to successively rough, semifinish, and finish large castings, forgings, and welded fabrications. These grinding machines are available in various sizes and have up to 225-HP motors to drive the spindle.

Work Holding Devices: Almost any work holding device used on a milling machine or drill press can be used on surface grinders. Vises, rotary tables, index centers, and other fixtures are used for special set-ups. However, the most common work holding device on surface grinders is the magnetic chuck.

Magnetic chucks hold the workpiece by exerting a magnetic attraction on the part. Only magnetic materials such as iron and steel may be mounted directly on the chuck. Two types of magnetic chucks are available for surface grinders: The permanent magnet and the electromagnetic chucks. Three types of magnetic chucks are shown in Figure 17.7.

On permanent magnet chucks, the

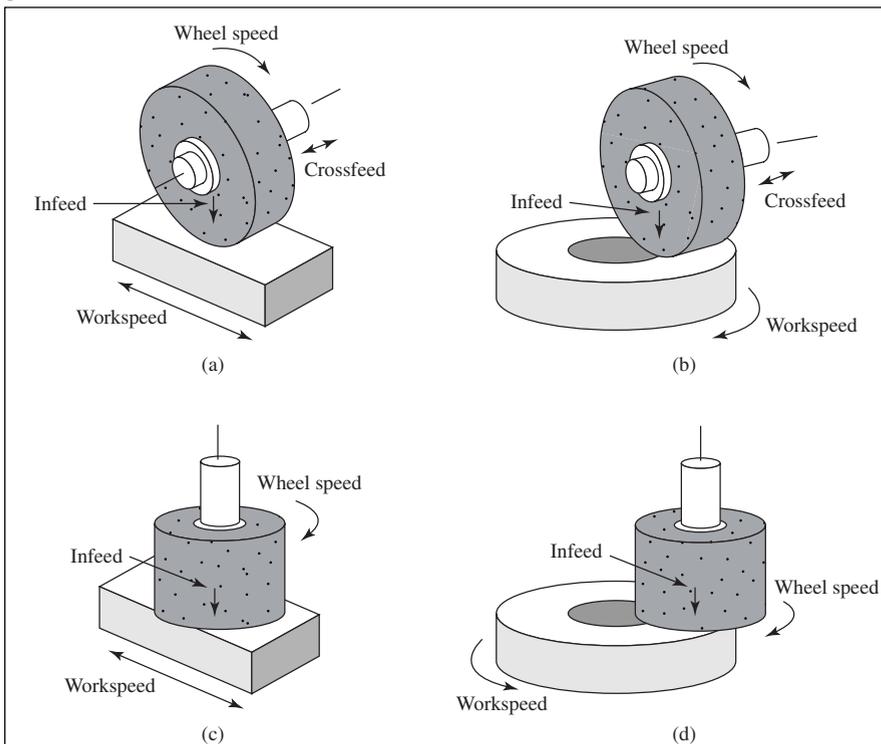


FIG. 17.4: Four types of surface grinders commonly used in industry: (a) horizontal spindle/reciprocating table, (b) horizontal spindle/rotary table, (c) vertical spindle/reciprocating table, (d) vertical spindle/rotary table.



FIG. 17.6: Vertical-spindle grinder with rotary table. (Courtesy WMW Machinery Co. Inc.)

holding power comes from permanent magnets. The work is placed onto the chuck and a hand lever is moved to energize the magnets. The electromagnetic chuck operates on 110 or 220 volts and is energized by a switch. This type of chuck has two advantages. First, the holding power may be adjusted to suit the area of contact of the workpiece; small amounts of current are used with smaller parts, large amounts with larger parts. A second

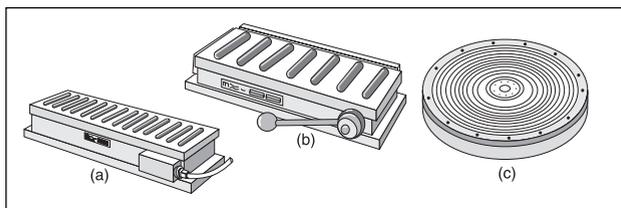


FIG. 17.7: Three magnetic chucks: (a) electromagnetic chuck, (b) permanent magnet chuck, (c) rotary electro-magnetic chuck.

advantage is the demagnetizer switch. It reverses the current flow momentarily and neutralizes the residual magnetism from the chuck and workpiece.

17.2.2 Cylindrical Grinding

Cylindrical grinding is the process of grinding the outside surfaces of a cylinder. These surfaces may be straight, tapered or contoured. Cylindrical grinding operations resemble lathe turning operations. They replace the lathe when the workpiece is hardened or when extreme accuracy and superior finish are required. Figure 17.8 illustrates the basic motion of the cylindrical grinding machine. As the workpiece revolves, the grinding wheel, rotating much faster in the opposite direction, is brought into con-

tact with the part. The workpiece and table reciprocate while in contact with the grinding wheel to remove material.

A CNC cylindrical grinder with a coolant system is shown in Figure 17.9; a very large roll grinder is shown in Figure 17.10.

Work Holding Devices: Work holding devices and accessories used on center-type cylindrical grinders are similar to those used on engine lathes.

The primary method of holding work is between centers as shown in Figure 17.9. The points on these centers may be high-speed steel or tungsten carbide (Fig. 4.12). A lubricant is used with either type and is applied between the point of the center and the center hole in the work.

Independent, universal and collet chucks can be used on cylindrical grinders when the work is odd-shaped or contains no center hole. They are used also for internal grinding operations.

17.2.3 Centerless Grinding

Centerless grinding machines eliminate the need to have center holes for the work or to use work-holding devices. In centerless grinding, the workpiece rests on a workrest blade and is

backed up by a second wheel, called the regulating wheel (Fig. 17.11). The rotation of the grinding wheel pushes the workpiece down on the workrest blade and against the regulating wheel. The regulating wheel, usually made of a rubber bonded abrasive, rotates in the same direction as the grinding wheel and controls the longitudinal feed of the work when set at a slight angle. By changing this angle and the speed of the wheel, the workpiece feed rate can be changed. The diameter of the workpiece is controlled by two factors: The distance between the grinding wheel and regulating wheel, and by changing the height of the workrest blade.

A typical centerless grinding opera-

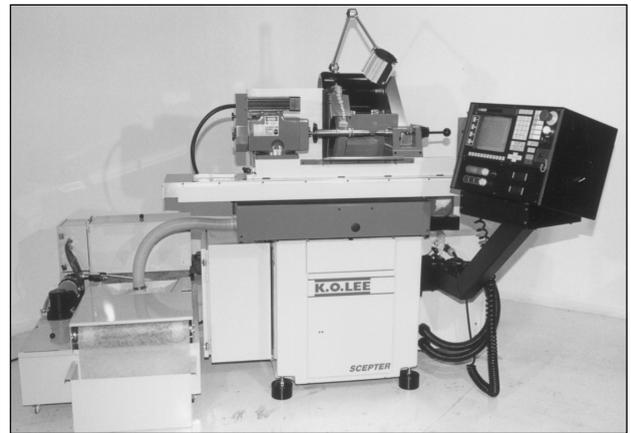


FIG. 17.9: CNC cylindrical grinder with coolant system. (Courtesy K. O. Lee Co.)

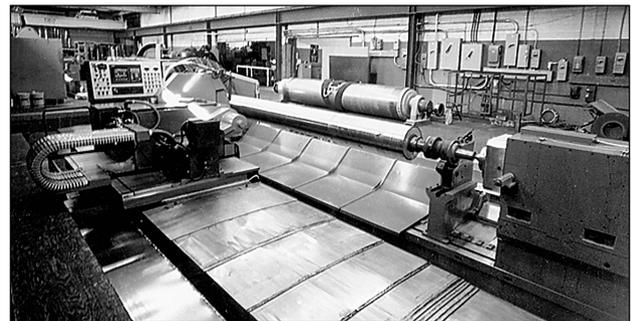


FIG. 17.10: Very large computer controlled roll grinder (Courtesy: WMW Machinery Co. Inc.)

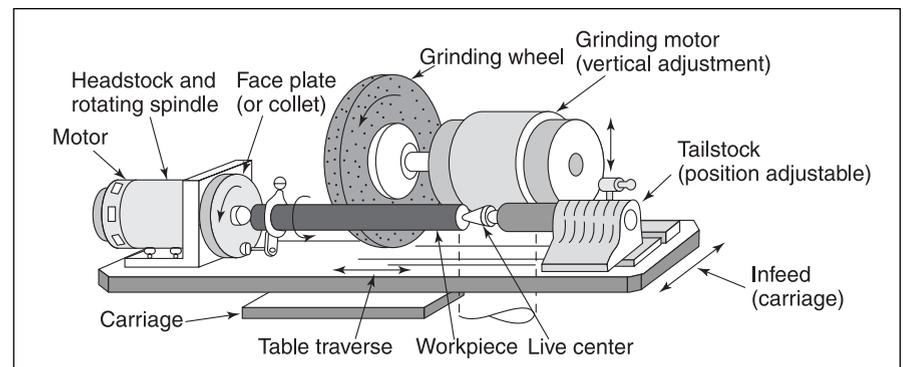


FIG. 17.8: Schematic illustration of the basic components and motions of a cylindrical grinder.

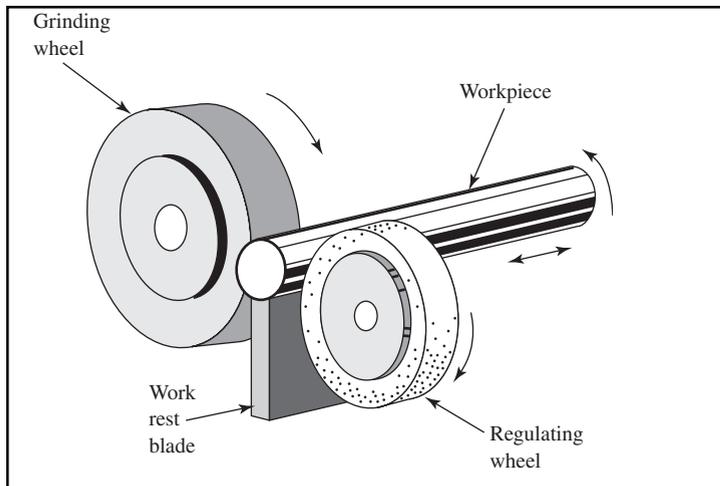


FIG. 17.11: Operating principle of a centerless grinder.

tion is shown in Figure 17.12 and centerless grinder is shown in Figure 17.13.

17.2.4 Internal Grinding

Internal grinders are used to accurately finish straight, tapered, or formed holes. The most popular internal grinder is similar in operation to a boring operation in a lathe. The workpiece is held by a work holding device, usually a chuck or collet, and revolved by a motorized headstock. A separate motor head in the same direction as the workpiece revolves the grinding wheel. It can be fed in and out of the work and also adjusted for depth of cut. An internal grinding operation with a steady rest is shown in Figure 17.14.

17.2.5 Special Grinding Processes

Special types of grinders are grinding machines made for specific types of work and operations. A brief de-

scription of the more commonly used special types follows:

Tool and Cutter Grinders: A tool and cutter grinder was introduced in Chapter 8 - Drilling Operations (Fig. 8.12). These grinding machines are designed to sharpen milling cutters, reamers, taps, and other machine tool cutters. A tabletop tool and cutter grinder is shown in Figure 17.15 and a 5-axis CNC cutter grinder is shown in Figure 17.16.

The general purpose cutter grinder is the most popular and versatile tool grinding machine. Various attachments are available for sharpening most types of cutting tools. Sharpening of a tap is shown in Fig. 17.17a and grinding of a milling cutter is shown in Fig. 17.17b. Figure 17.18 shows sharpening of a carbide milling cutter with a diamond cup-grinding wheel.

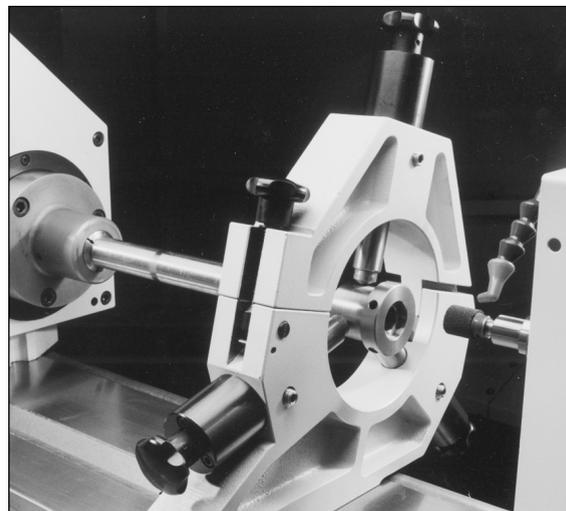


FIG. 17.14: Internal grinding operation; the workpiece is held by a collet and supported in a steady rest. (Courtesy kellenberger, A Hardinge Co.)

Jig Grinding Machines: Jig grinders were developed to locate and accurately grind tapered or straight holes. Jig grinders are equipped with a high speed vertical spindle for holding and driving the grinding wheel. They utilize the same precision locating system as do jig borers. A 5-axis continuous path jig grinder is shown in Figure 17.19.

Thread Grinding Machines: These are special grinders that resemble the cylindrical grinder. They must have a precision lead screw to produce the correct pitch, or lead, on a threaded part. Thread grinding machines also have a means of dressing or truing the cutting periphery of the grinding



FIG. 17.12: Typical centerless grinding operation. (Courtesy Cincinnati Machine)

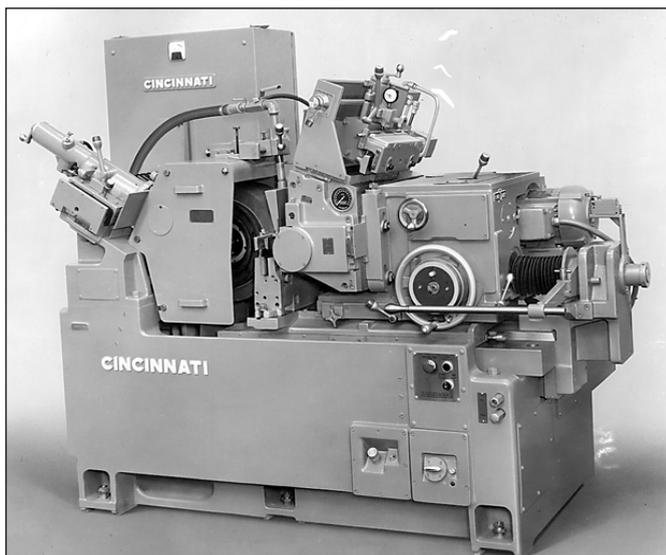


FIG. 17.13: A Centerless Grinder is shown (Courtesy: Cincinnati Machine)



FIG. 17.15: Table top tool and cutter grinder is shown sharpening an end milling cutter. (Courtesy Chevalier Machinery, Inc.)

wheel so that it will produce a precise thread form on the part. Figure 17.20 shows a CNC thread grinder with a robotic loading system and menu-driven software programs.

17.3 Creep-Feed Grinding

Grinding has traditionally been associated with small rates of metal removal and fine finishing operations. However, grinding can also be used for large-scale metal removal operations similar to milling, broaching, and planning. In creep-feed grinding, developed in the late 1950's, the wheel depth of cut is as much as 0.25 in., and the workpiece speed is low. The wheels are mostly softer grade



FIG. 17.16: 5-axis CNC cutter grinder. (Courtesy: Star Cutter Co.)

resin bonded with open structures to keep temperatures low and improve surface finish. The machines used for creep-feed grinding have special features, such as high power – up to 300hp – high stiffness, high damping capacity, variable spindle and worktable speeds, and ample capacity for grinding fluids.

Its overall competitive position with other material-removal processes indicate that creep-feed grinding can be economical for specific applications, such as in grinding shaped punches, twist-drill flutes, and various complex super alloy

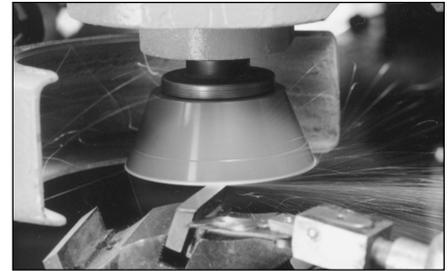


FIG. 17.18 Sharpening of a carbide milling cutter with a diamond cup grinding wheel. (Courtesy Norton Company)

grains

* Breaking of the bond

In most grinding processes, all three mechanisms are active to some extent. Attritions war is not desirable because the dulled grains reduce the efficiency of the process, resulting in increased power consumption, higher surface temperatures, and surface damage. However, attrition must go on to some

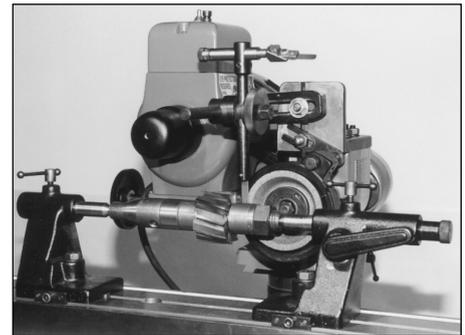
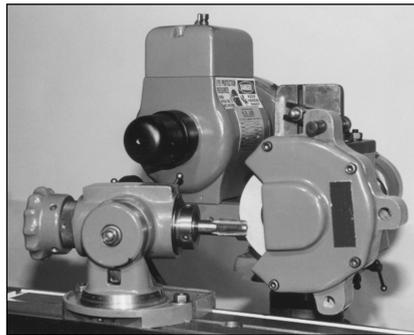


FIG. 17.17: Tool and cutter grinder setups: (a) sharpening of a tap, (b) sharpening of a milling cutter. (Courtesy K. O. Lee Co.)

parts. The wheel is dressed to the shape of the workpiece to be produced. Consequently, the workpiece does not have to be previously milled, shaped, or broached. Thus near-net shape castings and forgings are suitable parts for creep-feed grinding. Although generally one pass is sufficient, a second pass may be necessary for improved surface finish.

17.4 Grinding Wheel Wear

The wear of a grinding wheel can be caused by three actions:

- * Attrition or wearing down
- * Shattering of the

extent, with the forces on the grit being increased until they are high enough to shatter the grit or break the bond posts holding the dulled grit. The action of particles breaking away from the grains serves to keep the wheel sharp without excessive wear. However, the grains must eventually break from the bond or the wheel will have to be dressed. Rupturing the bond post that holds the grit allows dull grains to be sloughed off, exposing new sharp edges. If this occurs too readily, the wheel diameter wears down too fast. This raises wheel costs and prohibits close sizing on consecutive parts.

G-ratio: The G-ratio is the ratio of the amount of stock removed verses the amount of wear on the wheel, measured in cubic inches per minute. This ratio will vary from 1.0 to 5.0 in very rough grinding and up to 25.0 to

50.0 in finish grinding.

Even though grinding wheels are fairly expensive, a high G-ratio is not necessarily economical, as this may mean a slower rate of stock removal. It often takes some experimenting to find the wheel-metal combination, which is most economical for a job.

17.4.1 Attritions Wear

Attritions wear is responsible for the so-called ‘glazed’ wheel, which occurs when flat areas are worn on the abrasive grains but the forces are not high enough to break the dull grains out of the wheel face. Effective grinding ceases with a glazed wheel when the radial force becomes so high that the grit can no longer penetrate the workpiece surface to form chips. Attritions wear of the wheel occurs most often when fine cuts are taken on hard abrasive materials. Taking heavier cuts or using a softer wheel that will allow the grains to break out can often avoid it.

17.4.2 Grain Fracture

The forces that cause the grain to shatter may arise from the cutting forces acting on the wheel, thermal conditions, shock loading, welding action between the grit and the chip, or combinations of these factors. In finish grinding, this type of wheel wear is desirable, because it keeps sharp edges exposed, and still results in a low rate of wheel wear. In time, the wheel may become ‘loaded’ and noisy, and require dressing. A loaded wheel should be dressed by taking a few deep cuts with the diamond so that the metal charged layer is removed, and the chips are not just pushed further into the wheel. Then it should be finish dressed according to the application requirements.

17.4.3 Bond Fracture

It is desirable to have worn grit break out of the

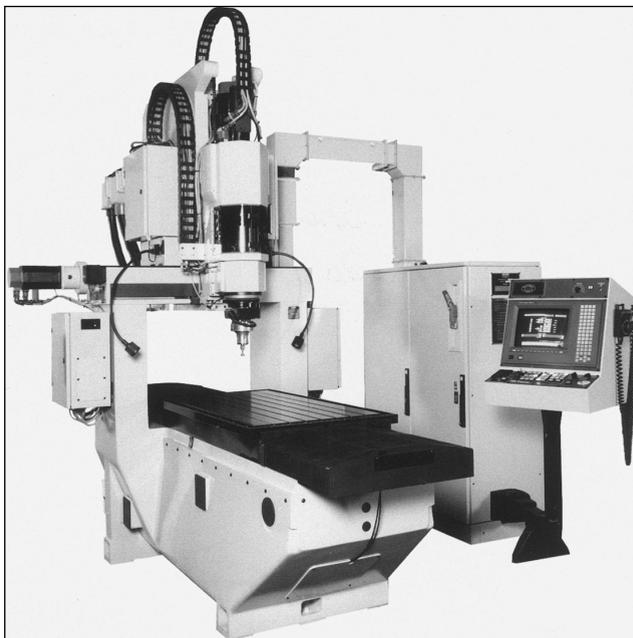


FIG. 17.19: Continuous path 5-axis jig grinder. (Courtesy Moore Tool Co., Inc.)

wheel so that new cutting edges will be exposed. This breaking down of the bond should progress fast enough so that heat generation is sufficiently low to avoid surface damage. On the other hand, bond breakdown should be slow enough so that wheel costs are not prohibitive. Normally, this means choosing the proper wheel grade for the job. Certain bond hardness is required to hold the grain in place. Softer wheels crumble too fast, while harder wheels hold the dull grit too long.

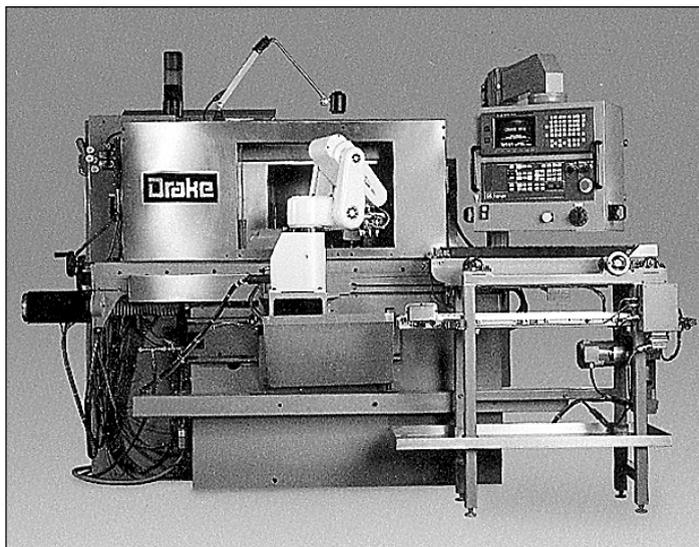


FIG. 17.20: CNC Thread Grinder with a robotic loading system (Courtesy: Drake Manufacturing Services)

17.5 Coated Abrasives

Typical examples of coated abrasives are sandpaper and emery cloth. The grains used in coated abrasives are more pointed than those used for grinding wheels. The grains are electrostatically deposited on flexible backing material, such as paper or cloth. The matrix or coating is made of resin.

Coated abrasives are available as sheets, belts, and disks and usually have a much more open structure than the abrasives on grinding wheels. Coated abrasives are used extensively in finishing flat or curved surfaces of metallic or nonmetallic parts, and in woodworking. The surface finishes obtained depend primarily on the grain sizes.

17.5.1 Abrasive Belt Machining

Coated abrasives are also used as belts for high-rate material removal. Belt grinding has become an important production process, in some cases replacing conventional grinding operations such as the grinding of camshafts. Belt speeds are usually in the range of 2500 to 6000 ft/min. Machines for abrasive-belt operations require proper belt support and rigid construction to minimize vibration. Figure 17.21 shows a multi-axis CNC double-station belt-grinding machine with menu-driven canned software programs.

17.6 Grindability

Grindability, in a like manner as machinability, may be thought of as the ease with which material can be removed from the workpiece by the action of the grinding wheel. Surface finish, power consumption, and tool (wheel) life can be considered as fundamental criteria of the grindability of metals. In addition, there are the important factors of chip formation and susceptibility to damage of the workpiece. Chip formation, which leads to a ‘loaded’

wheel, is detrimental.

The most important machine setting affecting machinability, the cutting speed, is not as important an influence on grindability because grinding is done at more or less constant speed. Instead, the important factor becomes the nature of the grinding wheel. The type of grit, grit size, bond material, hardness, and structure of the wheel, all influence the grindability of the workpiece. The problems of tool material and configuration variables were discussed in connection with machinability.

In grinding operations like snagging and cut-off work, the surface finish, and even the metallurgical damage to the workpiece, may become relatively unimportant. Wheel life and the rate of cut obtainable then become the criteria of grindability.

The best way to determine grindability is to start with the selection of the proper wheel. Beginning with the manufacturer's recommended grade for the conditions of the job, and then trying wheels on each side of this grade do this. Any improvement or deterioration in the grinding action, as evidenced by wheel wear, surface finish, or damage to the workpiece, can

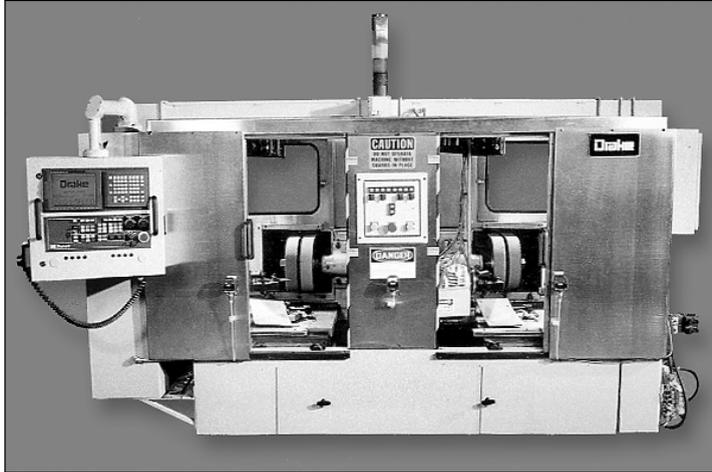


FIG.17.21 CNC double-station Belt Grinding Machine
(Courtesy: Drake Manufacturing Services)

be noted. After the proper wheel has been chosen, wheel life data may be obtained. Usually, this can be done during a production run.

Some of the factors to consider in establishing grindability ratings are discussed in the following examples of the grinding performance of metals:

Cemented Carbide: This material cannot be ground with aluminum oxide grit wheels. Although cemented carbide can be ground with pure silicon carbide wheels, the grinding ratio is very low and the material is easily damaged. Carbide is easily ground with diamond wheels if light cuts are taken to prevent damage to the workpiece material. However, diamond grit wheels are quite expensive. The overall grindability of this material is very low.

High Speed Steel: Hardened high speed steel can be ground quite successfully with aluminum oxide grit wheels. The grinding ratio is low, the relative power consumption high, and the possibility of damage to the workpiece is always present. Overall grindability is quite low.

Hardened Steel: Medium hard alloy or plain carbon steels are easily ground with aluminum oxide wheels. The grinding ratio is good, and damage to the workpiece is not a serious problem. Relative power consumption is moderate. The grindability rating is good.

Soft Steel: Annealed plain carbon steels grind with relatively low power consumption. Aluminum oxide wheels are satisfactory. The grinding ratio is quite high, but surface damage may be encountered. As a group, these materials are rated as having good grindability.

Aluminum Alloys: These soft alloys grind with quite low power consumption, but they tend to load the wheel quickly. Wheels with a very open structure are needed. Grinding ratios are good. Silicon carbide grit works well, and belt grinding outperforms wheel grinding in many cases.

Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. - www.ltu.edu

Prentice Hall - www.prenhall.com

CHAPTER 18

Lapping and Honing

18.1 Introduction

Lapping is a final abrasive finishing operation that produces extreme dimensional accuracy, corrects minor imperfections of shape, refines surface finish, and produces close fit between mating surfaces. Most lapping is done with a tooling plate or wheel (the lap), and fine-grained loose abrasive particles suspended in a viscous or liquid vehicle such as soluble oil, mineral oil, or grease. A typical lapping operation is shown in Figure 18.1.

Honing is a low velocity abrading process. Material removal is accomplished at lower cutting speeds than in grinding. Therefore, heat and pressure are minimized, resulting in excellent size and geometry control. The most common application of honing is on internal cylindrical surfaces. The cutting action is obtained using abrasive sticks mounted on a metal mandrel. Since the work is fixed in such a way as to allow floating, and no clamping or chucking, there is no distortion.

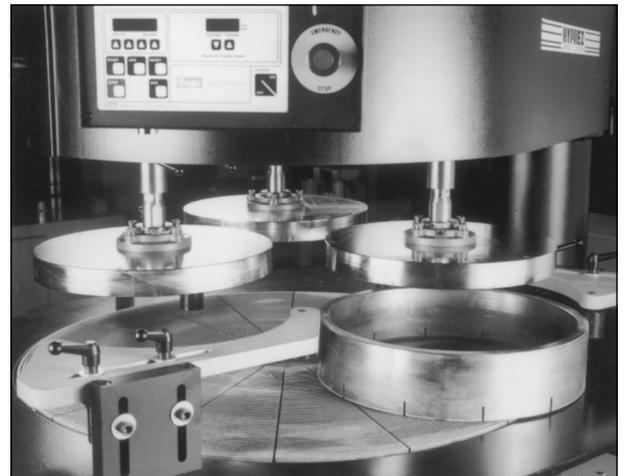


FIGURE 18.1: Typical lapping machine.
(Courtesy Engis Corp.)

18.2 Lapping Processes

The principal use of the lapping process is to obtain surfaces that are truly flat and smooth. Lapping is also used to finish round work, such as precision plug gages, to tolerances of 0.0005 to 0.00002 inches.

Work that is to be lapped should be previously finished close to the final size. While rough lapping can remove considerable metal, it is customary to leave only 0.0005 to 0.005 inches of stock to be removed.

Lapping, though it is an abrasive process, differs from grinding or honing because it uses a 'loose' abrasive instead of bonded abrasives like grinding wheels (Fig. 18.2).

These abrasives are often purchased 'ready mixed' in a 'vehicle' often made with an oil-soap or grease base. These vehicles hold the abrasive in suspension before and during use. The paste abrasives are generally used in hand-lapping operations. For machine lapping, light oil is mixed with dry abrasive so that it can be pumped onto the lapping surface during the lapping operation.

18.2.1 Lapping Machines

These machines are fairly simple pieces of equipment consisting of a rotating table, called a lapping plate, and three or four conditioning rings. Standard machines have lapping plates

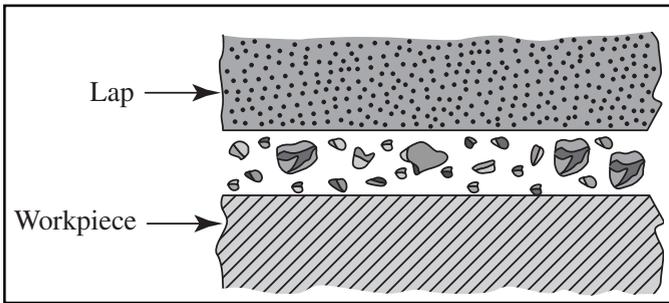


Figure 18.2. Abrasive grit must be uniformly graded to be effective in lapping.



Figure 18.3. Typical dual plate lapping machine. (Courtesy Engis Corporation)

from 12 to 48 inches in diameter. Large machines up to 144 inches are made. 1 to 20 HP motors run these tables. A typical lapping machine is shown in Figure 18.3.

The lapping plate is most frequently made of high-quality soft cast iron, though some are made of copper or other soft metals. This plate must be kept perfectly flat. The work is held in the conditioning rings. These rings rotate as shown in Fig. 18.4. This rotation performs two jobs. First it ‘conditions’ the plate, that is, it distributes the wear so that the lapping plate stays flat for a longer time. Secondly, it holds the workpiece in place. The speed at which the plate turns is determined by the job being done. In doing very critical parts, 10 to 15 RPM is used, and when polishing, up to 150 RPM is used.

A pressure of about 3 pounds per square inch (PSI) must be applied to the workpieces. Sometimes their own weight is sufficient. If not, a round, heavy pressure plate is placed in the conditioning ring. The larger machines use pneumatic or hydraulic lifts to place and remove the pressure plates. Figure 18.5 shows various lapping plates.

The workpiece must be at least as hard as the lapping plate, or the abrasive will be charged into the work. It will take from 1 to 20 minutes to complete the machining cycle. Time depends on the amount of stock removed, the abrasive used, and the quality required. Figure 18.6 shows a production-lapping machine.

The workpiece must be at least as hard as the lapping plate, or the abrasive will be charged into the work. It will take from 1 to 20 minutes to complete the machining cycle. Time depends on the amount of stock removed, the abrasive used, and the quality required. Figure 18.6 shows a production-lapping machine.

18.2.2 Grit and Plate Selection

Flatness, surface finish, and a polished surface are not necessarily achieved at the same time or in equal quality. For example, silicon carbide compound will cut fast and give good surface finish, but will always leave a ‘frosty’ or matte surface.

The grits used for lapping may occasionally be as coarse as 100 to 280 mesh. More often the ‘flour’ sizes of 320 to 800 mesh are used. The grits, mixed in slurry, are flowed onto the plate to replace worn-out grits as the machining process continues.

The case for using diamond super abrasives rather than conventional abrasives such as aluminum oxide or silicon carbide can be summed up in three words. Diamonds are faster, cleaner, and more cost-effective.

With diamond slurries, the lapping and polishing phases of a finishing operation can often be combined into

one step. Also, less time is required for cleaning parts and processing waste; throughput, along with overall productivity, is increased.

Lapping plates are manufactured from various materials as described below, and are available in standard sizes from 6 to 48 inches in diameter. Plates are supplied with square, spiral, and concentric and radial grooves as shown in Figure 18.5.

Iron - Aggressive Stock Removal:

- Excellent primary/roughing lap plate, with long service life
- Often used as an alternative to cast iron plates
- Produces a good surface finish on most materials, especially metals and ceramics.

Copper - Moderate to Aggressive Stock Removal:

- Most widely used, universal composite lap plate
- Excellent when primary and finishing lap are combined in a one step operation
- Suitable for virtually any solid material: metal, ceramic, glass, carbon, plastic, etc.

Ceramic - Moderate Stock Removal:

- Generally used to lap/polish ceramic parts and other stain-sensitive materials.
- Used in applications where metallic-type contamination cannot be tolerated
- Affordable, more machinable alternative to ‘natural’ ceramic plates.

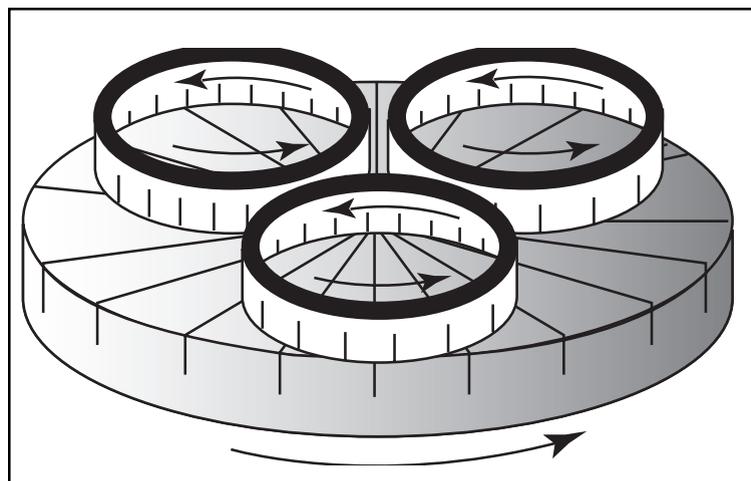


Figure 18.4. Conditioning rings used in lapping operations.



Figure 18.5. Typical lapping plates. (Courtesy Engis Corporation)

Tin/Lead - Fine Stock Removal:

- Most widely used finishing lap/polishing plate
- Often used in place of polishing pads
- Suitable for metal, ceramic and other materials.

Tin - Fine Stock Removal

- Often used where lead-type contamination cannot be tolerated
- Suitable for charging of extra-fine particulates.



Figure 18.6. Single plate lapping production machine equipped for diamond abrasive slurry use. (Courtesy Engis Corporation)

18.3 Advantages and Limitations

Any material, hard or soft, can be lapped, as well as any shape, as long as the surface is flat.

Advantages: There is no warping, since the parts are not clamped and very little heat is generated. No burrs are created. In fact, the process removes light burrs. Any size, diameter, and thickness from a few thousandths thick up to any height the machine will handle can be lapped. Various sizes and shapes of lapped parts are shown in Figure 18.7.

Limitations: Lapping is still somewhat of an art. There are so many variables that starting a new job requires experience and skill. Even though there are general recommendations and assistance from the manufacturers, and past experience is useful, trial and error may still be needed to get the optimum results.

18.4 Honing Processes

As stated earlier, honing is a low velocity abrading process. Material removal is accomplished at lower cutting speeds than in grinding. Therefore, heat and pressures are minimized, resulting in excellent size and geometry control. The most common application of honing is on internal cylindrical surfaces. A typical honing operation is shown in Figure 18.8.

Machining a hole to within less than 0.001 inch in diameter and maintaining true roundness and straightness with finishes less than 20 μ inches is one of the more difficult jobs in manufacturing.

Finish boring or internal grinding may do the job, but spindle deflection, variation in hardness of the material, and difficulties in precise work holding, make the work slow and the results uncertain. Honing, because it uses rectangular grinding stones instead of circular grinding wheels, as shown in Figures 18.9a and 18.9b, can correct these irregularities.

Honing can consistently produce finishes as fine as 4 μ inches and even finer finishes are possible. It can remove as little as 0.0001 inch of stock or as much as 0.125 inch of stock. However, usually only 0.002 to 0.020 inch stock is left on the diameter for honing. As shown in Figure 18.10, honing can correct a number of condi-

tions or irregularities, left by previous operations.

18.5 Honing Machines

For most work, honing machines are quite simple. The most used honing machines are made for machining internal diameters from 0.060 to 6 inches. However, large honing ma-



Figure 18.7. Various sizes and shapes of lapped parts. (Courtesy Engis Corporation)



Figure 18.8. Typical vertical honing operation. (Courtesy Sunnen Products Co.)

chines are made for diameters up to 48 inches. Larger machines are sometimes made for special jobs.

The length of the hole that can be honed may be anything from 1/2 inch to 6 or 8 inches on smaller machines, and up to 24 inches on larger machines. Special honing machines are made which will handle hole lengths up to 144

18.5.1 Horizontal Spindle Machines

Horizontal-spindle honing machines, for hand-held work with bores up to 6 inches, are among the most widely used. The machine rotates the hone at from 100 to 250 FPM.

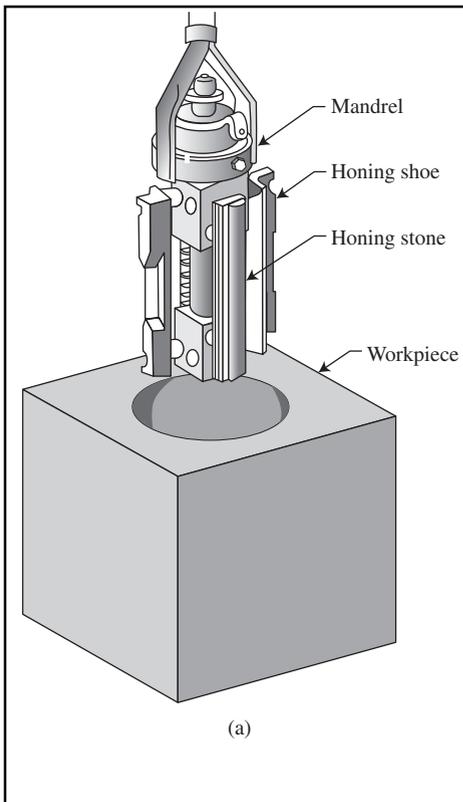


Figure 18.9a. Schematic illustration of the components of an internal hone.

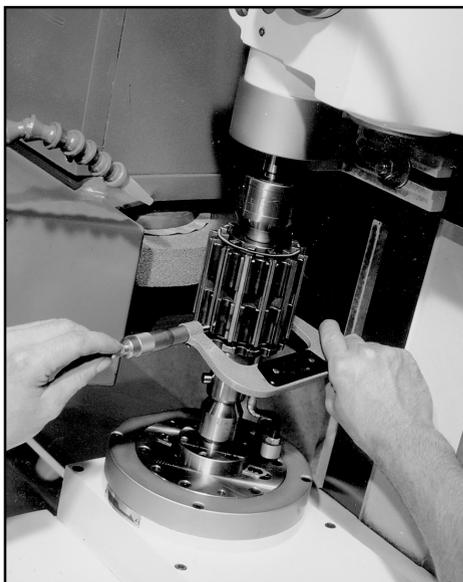


Figure 18.9b. Typical honing tool is shown being Checked (Courtesy: Gehring L.P.)

The machine operator moves the work back and forth (strokes it) over the rotating hone. The operator must ‘float’ the work, that is, not press it against the hone or the hole will be slightly oval. Sometimes the workpiece must be rotated.

Horizontal-spindle honing machines

are also made with ‘power stroking’. In these, the work is held in a self-aligning fixture and the speed and length of the stroke are regulated by controls on the machine.

As a hone is being used, it is expanded by hydraulic or mechanical means until the desired hole diameter is achieved. Various mechanical and electrical devices can be attached to the honing machine to control the rate of expansion, and stop it when final size is reached.

On the simplest hand-held machines, the operator may check the bore size with an air gage, continue honing, recheck, etc. until the size is correct. A horizontal-spindle honing machine is shown in Figure 18.11.

18.5.2 Vertical Spindle Machines

Vertical-spindle honing machines are used especially for larger, heavier work. These all have power stroking at speeds from 20 to 120 FPM. The length of the stroke is also machine controlled by stops set up by the operator.

Vertical honing machines are also made with multiple spindles so that several holes may be machined at once, as in automobile cylinders (Figure 18.8).

Hone Body: The hone body is made in several styles using a single stone for small holes, and two to eight stones as sizes get larger (Fig. 18.9b). The stones come in a wide variety of sizes and shapes. Frequently there are hardened metal guides between the stones to help start the hone cutting in a straight line.

Cutting Fluid: A fluid must be used with honing. This has several purposes: to clean the small chips from the stones and the workpiece, to cool the work and the hone, and to lubricate the cutting action.

A fine mesh filtering system must be used, since recirculated metal can spoil the finish.

A vertical honing operation was shown in Figure 18.8. A few of the parts honed on such a machine are shown in Figure 18.12.

18.6 Abrasive Tool Selection

The abrasive honing stone must be selected for the proper abrasive type, bond hardness and grit size to deliver the fastest stock removal and desired

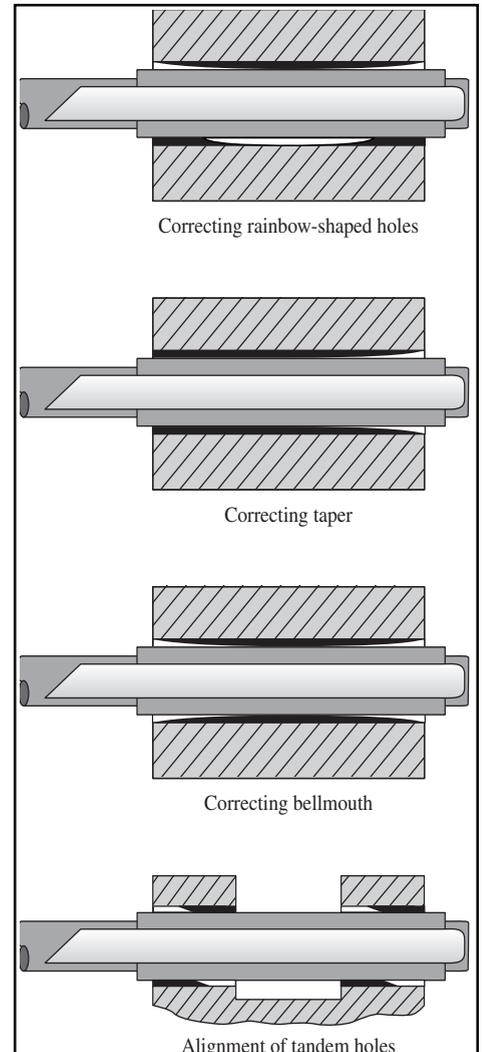


Figure 18.10. Undesirable conditions that can be corrected by honing.

surface finish. This selection is simple if done in the following three steps:

Step One: Select the abrasive type with respect to the material composition of the bore. There are four different types of abrasives: aluminum oxide, silicon carbide, diamond, and CBN. All four of these were discussed in the previous chapter. Each type has its own individual characteristics that make it best for honing certain materials. Some simplified guidelines for their use are:

- Mild steel hones best with aluminum oxide.
- Cast iron, brass, and aluminum hone best with silicon carbide.
- Glass, ceramic, and carbide hone best with diamond
- High speed tool steels, and super alloys hone best with CBN.



Figure 18.11. Horizontal-spindle honing machine (Courtesy Sunnen Products Co.)

Diamond and CBN are considered super abrasives because they are much harder than conventional abrasives. They cut easily and dull slowly, therefore allowing them to hone certain materials much faster and more efficiently than conventional abrasives.



Figure 18.12. Parts honed on a vertical honing machine. (Courtesy Sunnen Products Co.)

However, as shown above, super abrasives are not suited to honing all materials. For instance, diamond does not hone steel very well, and CBN may not be as economical as using aluminum oxide to hone soft steel.

Step Two: Use the stone hardness suggested in the manufacturer's catalog. If the stone does not cut, select the next softer stone; if the stone wears too fast, select the next harder stone. Stone hardness does not refer to the hardness of the abrasive grain, but to the strength of the bonding material holding the abrasive grains together, as discussed in the previous chapter. A bond must be strong enough to hold sharp abrasive grains in position to cut, but weak enough to allow dulled grains to be sloughed off to expose underly-

ing sharp grains. If the bond is too hard, the dulled abrasive grains will not be allowed to fall off, and the stock removal rate will be reduced. If the bond is too soft, the stone will wear excessively because sharp abrasive grains fall off before they are fully used.

Diamond and CBN abrasive grains dull so slowly that standard ceramic or resin bonds may not be strong enough when honing rough out-of-round bores in hard materials, or when CBN is used to hone soft steel. Metal bonds are best suited for these applications because the grains are held in a sintered metal matrix that is much stronger than standard bonds. As with choosing abrasive type, stone bond hardness must be matched to the application to maximize life and stock removal rates.

Step Three: Select the largest abrasive grit size that will still produce the desired surface finish. Surface finish is a function of the height of microscopic peaks and valleys on the bore surface and honing can produce almost any degree of roughness or smoothness through the use of different abrasive grit sizes.

Honing oil can improve stock removal rates by helping the cutting action of the abrasive grains. It prevents pickup (spot welding of tool to bore) and loading (chips coating the stone). Honing oil does this, not by acting as a coolant, but through chemical activity. The ingredients in the oil produce this chemical activity. Whenever the temperature rises at one of the microscopic cutting points, the sulfur in the oil combines with the iron in the steel to form iron sulfide, an unweldable compound, and welding is prevented. The antiwelding property of honing oil also prevents chips from sticking together and coating the stone. Water based coolants cannot produce this type of chemical activity. Use of water-based coolants will result in welding of metallic guide shoes

to the part and loading of vitrified abrasive honing stones.

18.7 Cylinder Block Honing

Bores sometime require a preliminary rough honing operation to remove stock, followed by finish honing to get the desired surface finish. A characteristic feature of a honed surface finish is crosshatch, which makes an excellent oil retention and bearing surface. The crosshatch pattern is generated in the bore surface as the workpiece is stroked back and forth over the rotating honing tool.

Plateau Honing: A few years ago a special surface finish generated interest in the engine rebuilding market. With this finish, the valleys are deep and the peaks have been removed to form plateaus, giving the name plateau honing or plateau finish as shown in Figure 18.13. A recent test by a ring manufacturer has shown that an engine with a true plateau finish consumed one-tenth the oil and had 80 percent less cylinder bore wear than the engines with conventional finishes.

Laser-Honing: With this process, considerably better results are achieved compared to traditional honing. Precisely defined surface structures can be obtained with Laser technology. Laser-honing is a combination of honing and Laser processing. This process generates Laser-produced lubricant reservoirs into a specifically defined area in order to achieve an ideal plateau surface finish. Such a hydrodynamic system can be produced exactly where it is required as shown in Figure 18.14.

Application of the Laser-honing process requires three steps. In the first step – rough honing – the macro-form

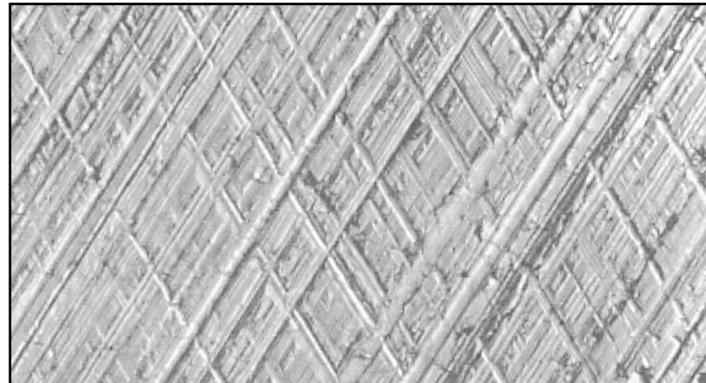


Figure 18.13. Plateau honed finish surface at 100x (Courtesy: Gehring L.P.)

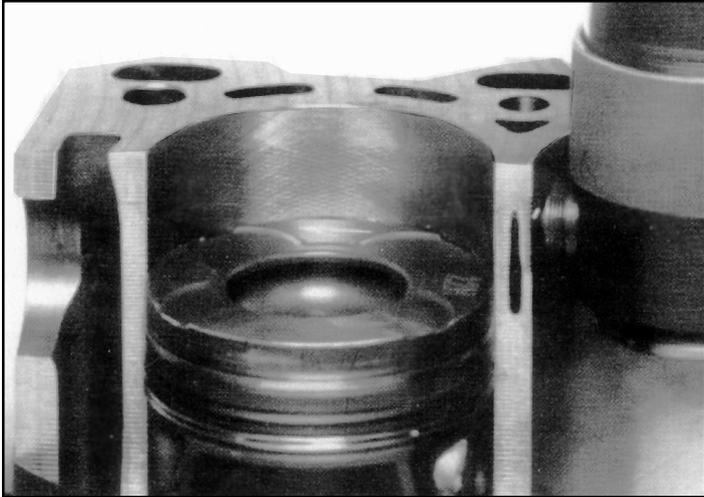


Figure 18.14. Laser generated honed finished surface (Courtesy: Gehring L.P.)

of the bore is produced. In the second step, precisely defined lubricant reservoirs are produced with the Laser. In step three –finish honing – an extremely fine surface finish is obtained, resulting in increased engine life by reduction of wear in the cylinder surface and on the piston rings.

18.8 Production Honing

Honing will not only remove stock rapidly, but it can also bring the bore to finish diameter within tight tolerances. This is especially true if the honing machine is equipped with automatic size control. With every stroke, the workpiece is pushed against a sensing tip that has been adjusted to the finish diameter of the bore. When the bore is to size, the sensing tip enters the bore and the machine stops honing. Size repetition from bore to bore is .0001 inch to .0002 inch. The operator simply loads and unloads the fixture and presses a button; everything else is automatic.

Single-Stroke Honing: A still faster and more accurate method of honing a bore to final size is Single-Stroke honing. The Single-Stroke tool (Fig. 18.15) is an expandable diamond plated sleeve on a tapered arbor. The sleeve is expanded only during set up, and no adjustments are necessary during honing. Unlike conventional hon-

ing, where the workpiece is stroked back and forth over the tool, in Single-Stroke honing the rotating tool is pushed through the bore one time, bringing the bore to size. The return stroke does nothing to the bore except get the workpiece off the tool. Single-Stroke honing is so accurate and consistent, that honed bores do not require gaging.

Although Single-Stroke honing has many advantages, it is limited in the types and volumes of material that can be removed. The size and overall volume of chip produced in one pass must be no more than the space between the diamond grits, or the tool will seize in the bore.

Workpieces are best suited for Single-Stroke honing when they are made of materials that produce small chips, such as cast iron, and that have interruptions that allow chips to be washed from the tool as the bore is being honed. Conventional honing should be used whenever the material to be honed produces long stringy chips, or the amount of stock to be removed is large.

18.9 Advantages and Limitations

Honing has developed into a productive manufacturing

Process, some advantages and limitations will be discussed below:

Advantages: The workpiece need not be rotated by power, there are no chucks, faceplates, or rotating tables needed, so there are no chucking or locating errors. The hone is driven from a central shaft, so bending of the shaft cannot cause tapered holes as it does when boring. The result is a truly round hole, with no taper or high or low spots, provided that the previous operations left enough stock so that the

hone can clean up all the irregularities.

Honing uses a large contact area at slow speed compared with grinding or fine boring, which use a small contact area at high speed. Because of the combined rotating and reciprocating motion used, a cross hatched pattern is created which is excellent for holding lubrication. Diameters with 0.001 to 0.0001 inch and closer accuracies can be repeatedly obtained in production work.

Honing can be done on most materials from aluminum or brass to hardened steel. Carbides, ceramics and glass can be honed by using diamond stones similar to diamond wheels.

Limitations: Honing is thought of as a slow process. However, new machines and stones have shortened honing times considerably. Horizontal honing may create oval holes unless the work is rotated or supported. If the workpiece is thin, even hand pressure may cause a slightly oval hole.

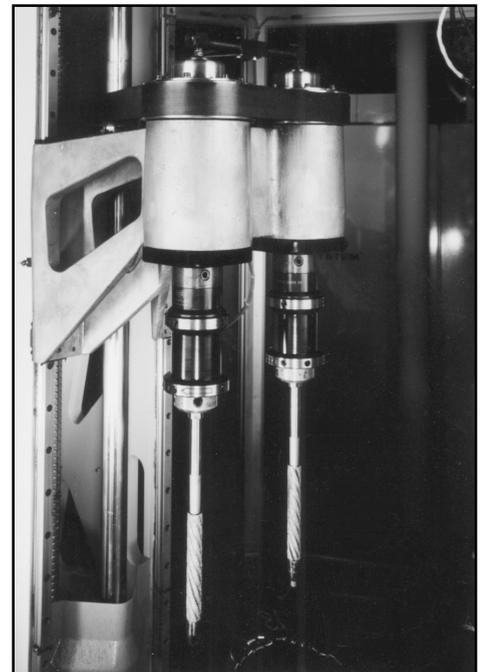


Figure 18.15. Single-stroke honing tools use expandable diamond-plated sleeves on a tapered arbor. (Courtesy Sunnen Products Co.)